



# Discovery AIM



ANSYS, Inc.  
Southpointe  
2600 ANSYS Drive  
Canonsburg, PA 15317  
[ansysinfo@ansys.com](mailto:ansysinfo@ansys.com)  
(T) 724-746-3304  
(F) 724-514-9494

Release 2020 R1  
January 2020

ANSYS, Inc. is certified to  
ISO 9001:2015



## i. Copyright and Trademark Information

© 2020 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEXlm and FLEXnet are trademarks of Flexera Software LLC.

## Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015

## U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

## Third-Party Software

See the legal information in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, contact ANSYS, Inc.

Published in the U.S.A.

Protected by US Patents 7,639,267, 7,733,340, 7,830,377, 7,969,435, 8,207,990, 8,244,508, 8,253,726, and 8,330,775

Copyright © 2003-2019 ANSYS, Inc. All Rights Reserved. SpaceClaim is a registered trademark of ANSYS, Inc.

Portions of this software Copyright © 2010 AcreSSO Software Inc. FlexLM and FLEXNET are trademarks of AcreSSO Software Inc.

Portions of this software Copyright © 2008 Adobe Systems Incorporated. All Rights Reserved. Adobe and Acrobat are either registered trademarks or trademarks of Adobe Systems Incorporated in the United States and/or other countries

ANSYS Workbench and GAMBIT and all other ANSYS, Inc. product names are trademarks or registered trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries.

Contains BCLS (Bound-Constrained Least Squares) Copyright (C) 2006 Michael P. Friedlander, Department of Computer Science, University of British Columbia, Canada, provided under a LGPL 3 license which is

i. Copyright and Trademark Information

included in the SpaceClaim installation directory (lgpl-3.0.txt). Derivative BCLS source code available upon request.

Contains SharpZipLib Copyright © 2009 C#Code

Anti-Grain Geometry Version 2.4 Copyright © 2002-2005 Maxim Shemanarev (McSeem).

Some SpaceClaim products may contain Autodesk® RealDWG by Autodesk, Inc., Copyright © 1998-2010 Autodesk, Inc. All rights reserved. Autodesk, AutoCAD, and Autodesk Inventor are registered trademarks and RealDWG is a trademark of Autodesk, Inc.

CATIA is a registered trademark of Dassault Systèmes.

Portions of this software Copyright © 2010 Google. SketchUp is a trademark of Google.

Portions of this software Copyright © 1999-2006 Intel Corporation. Licensed under the Apache License, Version 2.0. You may obtain a copy of the License at <http://www.apache.org/licenses/LICENSE-2.0>

Contains DotNetBar licensed from devcomponents.com.

KeyShot is a trademark of Luxion ApS.

MatWeb is a trademark of Automation Creations, Inc.

2008 Microsoft® Office System User Interface is licensed from Microsoft Corporation. Direct3D, DirectX, Microsoft PowerPoint, Excel, Windows, Windows Vista and the Windows Vista Start button are trademarks or registered trademarks of Microsoft Corporation in the United States and/or other countries.

Portions of this software Copyright © 2005 Novell, Inc. (<http://www.novell.com>)

Creo Parametric and PTC are registered trademarks of Parametric Technology Corporation.

Persistence of Vision Raytracer and POV-Ray are trademarks of Persistence of Vision Raytracer Pty. Ltd.

Portions of this software Copyright © 1993-2009 Robert McNeel & Associates. All Rights Reserved. openNURBS is a trademark of Robert McNeel & Associates. Rhinoceros is a registered trademark of Robert McNeel & Associates.

Portions of this software Copyright © 2005-2007, Sergey Bochkanov (ALGLIB project). \*

Portions of this software are owned by Siemens PLM © 1986-2011. All Rights Reserved. Parasolid and Unigraphics are registered trademarks and JT is a trademark of Siemens Product Lifecycle Management Software, Inc.

SOLIDWORKS is a registered trademark of SOLIDWORKS Corporation.

Portions of this software are owned by Spatial Corp. © 1986-2011. All Rights Reserved. ACIS and SAT are registered trademarks of Spatial Corp.

Contains Teigha for .dwg files licensed from the Open Design Alliance. Teigha is a trademark of the Open Design Alliance.

Development tools and related technology provided under license from 3Dconnexion. © 1992 – 2008 3Dconnexion. All rights reserved.

TraceParts is owned by TraceParts S.A. TraceParts is a registered trademark of TraceParts S.A.

Contains a modified version of source available from Unicode, Inc., copyright © 1991-2008 Unicode, Inc. All rights reserved. Distributed under the Terms of Use in <http://www.unicode.org/copyright.html>.

Portions of this software Copyright © 1992-2008 The University of Tennessee. All rights reserved. [1]

Portions of this software Copyright © XHEO INC. All Rights Reserved. DeployLX is a trademark of XHEO INC.

This software incorporates information provided by American Institute of Steel Construction (AISC) for shape data available at <http://www.aisc.org/shapesdatabase>.

## i. Copyright and Trademark Information

This software incorporates information provided by ArcelorMittal® for shape data available at <http://www.sections.arcelormittal.com/products-services/products-ranges.html>.

All other trademarks, trade names or company names referenced in SpaceClaim software, documentation and promotional materials are used for identification only and are the property of their respective owners.

\*Additional notice for LAPACK and ALGLIB Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:-Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.-Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer listed in this license in the documentation and/or other materials provided with the distribution.-Neither the name of the copyright holders nor the names of its contributors may be used to endorse promote products derived from this software without specific prior written permission.

BCLS is licensed under the GNU Lesser General Public License (GPL) Version 3, Copyright (C) 2006 Michael P. Friedlander, Department of Computer Science, University of British Columbia, Canada. A copy of the LGPL license is included in the installation directory (lGPL-3.0.txt).

Please contact [open.source@ansys.com](mailto:open.source@ansys.com) for a copy of the source code for BCLS.

Eigen is licensed under the Mozilla Public License (MPL) Version 2.0, the text of which can be found at: <https://www.mozilla.org/media/MPL/2.0/index.815ca599c9df.txt>. Please contact [open.source@ansys.com](mailto:open.source@ansys.com) for a copy of the Eigen source code.

HDF5 (Hierarchical Data Format 5) Software Library and Utilities

Copyright (c) 2006, The HDF Group.

NCSA HDF5 (Hierarchical Data Format 5) Software Library and Utilities

Copyright (c) 1998-2006, The Board of Trustees of the University of Illinois.

All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted for any purpose (including commercial purposes) provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions, and the following disclaimer.
2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions, and the following disclaimer in the documentation and/or materials provided with the distribution.
3. In addition, redistributions of modified forms of the source or binary code must carry prominent notices stating that the original code was changed and the date of the change.
4. All publications or advertising materials mentioning features or use of this software are asked, but not required, to acknowledge that it was developed by The HDF Group and by the National Center for Supercomputing Applications at the University of Illinois at Urbana-Champaign and credit the contributors.
5. Neither the name of The HDF Group, the name of the University, nor the name of any Contributor may be used to endorse or promote products derived

from this software without specific prior written permission from The HDF Group, the University, or the Contributor, respectively.

### DISCLAIMER:

THIS SOFTWARE IS PROVIDED BY THE HDF GROUP AND THE CONTRIBUTORS "AS IS" WITH NO WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED. In no

event shall The HDF Group or the Contributors be liable for any damages suffered by the users arising out of the use of this software, even if advised of the possibility of such damage. Anti-Grain Geometry - Version 2.4 Copyright (C) 2002-2004 Maxim Shemanarev (McSeem)

## i. Copyright and Trademark Information

Permission to copy, use, modify, sell and distribute this software is granted provided this copyright notice appears in all copies. This software is provided "as is" without express or implied warranty, and with no claim as to its suitability for any purpose.

Some ANSYS-SpaceClaim products may contain Autodesk® RealDWG by Autodesk, Inc., Copyright © 1998-2010 Autodesk, Inc. All rights reserved. Autodesk, AutoCAD, and Autodesk Inventor are registered trademarks and RealDWG is a trademark of Autodesk, Inc.

CATIA is a registered trademark of Dassault Systèmes.

Portions of this software Copyright © 2013 Trimble. SketchUp is a trademark of Trimble Navigation Limited.

This software is based in part on the work of the Independent JPEG Group.

Portions of this software Copyright © 1999-2006 Intel Corporation. Licensed under the Apache License, Version 2.0. You may obtain a copy of the License at <http://www.apache.org/licenses/LICENSE-2.0>

Contains DotNetBar licensed from devcomponents.com.

Portions of this software Copyright © 1988-1997 Sam Leffler and Copyright (c) 1991-1997 Silicon Graphics, Inc.

KeyShot is a trademark of Luxion ApS.

MatWeb is a trademark of Automation Creations, Inc.

2010 Microsoft® Office System User Interface is licensed from Microsoft Corporation. Direct3D, DirectX, Microsoft PowerPoint, Excel, Windows/Vista/Windows 7/Windows 8/Windows 10 and their respective Start Button designs are trademarks or registered trademarks of Microsoft Corporation in the United States and/or other countries.

Portions of this software Copyright © 2005 Novell, Inc. (Licensed at [http://stuff.mit.edu/afs/athena/software/mono\\_v3.0/arch/i386\\_linux26/mono/mcs/dlls/ManagedWindows.Forms/System.Windows.Forms.RTF/](http://stuff.mit.edu/afs/athena/software/mono_v3.0/arch/i386_linux26/mono/mcs/dlls/ManagedWindows.Forms/System.Windows.Forms.RTF/))

Pro/ENGINEER and PTC are registered trademarks of Parametric Technology Corporation.

POV-Ray is available without charge from <http://www.pov-ray.org>. No charge is being made for a grant of the license to POV-Ray.

POV-Ray License Agreement

### DISTRIBUTOR'S LICENCE AGREEMENT

Persistence of Vision Raytracer(tm) (POV-Ray(tm))

13 August 2004

Licensed Versions: Versions 3.5 and 3.6

Please read through the terms and conditions of this license carefully. This is a binding legal agreement between you, the "Distributor" and Persistence of Vision Raytracer Pty. Ltd. ACN 105 891 870 ("POV"), a company incorporated in the state of Victoria, Australia, for the product known as the "Persistence of Vision Raytracer(tm)", also referred to herein as "POV-Ray(tm)". The terms of this agreement are set out at <http://www.povray.org/distribution-license.html> ("Official Terms"). The Official Terms take precedence over this document to the extent of any inconsistency.

### 1. INTRODUCTION

1.1. In this agreement, except to the extent the context requires otherwise, the following capitalized terms have the following meanings:

(a) Distribution means:

(i) a single item of a distribution medium, including a CD Rom or DVD Rom, containing software programs and/or data;

(ii) a set of such items;

i. Copyright and Trademark Information

- (iii) a data file in a generally accepted data format from which such an item can be created using generally available standard tools;
  - (iv) a number of such data files from which a set of such items can be created; or
  - (v) a data file in a generally accepted data storage format which is an archive of software programs and/or data;
- (b) Derived Code means all software which is derived from or is an adaptation of any part of the Software other than a scene file;
- (c) Intellectual Rights means:
- (i) all copyright, patent, trade mark, trade secret, design, and circuit layout rights;
  - (ii) all rights to the registration of such rights; and
  - (iii) all rights of a similar nature which exist anywhere in the world;
- (d) Licensed Version means the version set out at the top of this agreement against the heading "Licensed Version" and all minor releases of this version (ie releases of the form x.y.z);
- (e) POV Associate means any person associated directly or indirectly with POV whether as a director, officer, employee, subcontractor, agent, representative, consultant, licensee or otherwise;
- (f) Modification Terms means the most recent version from time to time of the document of that name made available from the Site (g) Revocation List means the list of that name linked to from the Official Terms;
- (h) Site means [www.povray.org](http://www.povray.org);
- (i) Software means the Licensed Version of the Persistence of Vision Raytracer(tm) (also known as POV-Ray(tm)) (including all POV-Ray program source files, executable (binary) files, scene files, documentation files, help files, bitmaps and other POV-Ray files associated with the Licensed Version) in a form made available by
- POV on the Site;
- (j) User Licence means the most recent version from time to time of the document of that name made available from the Site.

## 2. OPEN SOURCE DISTRIBUTIONS

2.1. In return for the Distributor agreeing to be bound by the terms of this agreement, POV grants the Distributor permission to make a copy of the Software by including the Software in a generally recognised Distribution of a recognised operating system where the kernel of that operating system is made available under licensing terms:

- (a) which are approved by the Open Source Initiative ([www.opensource.org](http://www.opensource.org)) as complying with the "Open Source Definition" put forward by the Open Source Initiative; or
- (b) which comply with the "free software definition" of the Free Software Foundation ([www.fsf.org](http://www.fsf.org)). 2.2. As at June 2004, and without limiting the generality of the term, each of the following is a "generally recognised Distribution" for the purposes of clause 2.1: Debian, Red Hat (Enterprise and Fedora), SuSE, Mandrake, Xandros, Gentoo and Knoppix Linux distributions, and officially authorized distributions of the FreeBSD, OpenBSD, and NetBSD projects.

2.3. Clause 2.1 also applies to the Software being included in the above distributions 'package' and 'ports' systems, where such exist;

### 2.4. Where the Distributor reproduces the Software in accordance with clause 2.1:

- (a) the Distributor may rename, reorganise or repackage (without omission) the files comprising the Software where such renaming, reorganisation or repackaging is necessary to conform to the naming or organisation scheme of the target operating environment of the Distribution or of an established package management

### i. Copyright and Trademark Information

system of the target operating environment of the Distribution; and (b) the Distributor must not otherwise rename, reorganise or repackage the Software.

### 3. DISTRIBUTION LICENCE

3.1. Subject to the terms and conditions of this agreement, and in return for Distributor agreeing to be bound by the terms of this agreement, POV grants the Distributor permission to make a copy of the Software in any of the following circumstances:(a) in the course of providing a mirror of the POV-Ray Site (or part of it), which is made available generally over the internet to each person without requiring that person to identify themselves and without any other restriction other than restrictions designed to manage traffic flows;

(b) by placing it on a local area network accessible only by persons authorized by the Distributor whilst on the Distributor's premises;

(c) where that copy is provided to a staff member or student enrolled at a recognised educational institution;

(d) by including the Software as part of a Distribution where:

(i) neither the primary nor a substantial purpose of the distribution of the Distribution is the distribution of the Software. That is, the distribution of the Software

is merely incidental to the distribution of the Distribution; and

(ii) if the Software was not included in the Distribution, the remaining software and data included within the Distribution would continue to function effectively and

according to its advertised or intended purpose;

(e) by including the Software as part of a Distribution where:

(i) there is no data, program or other files apart from the Software on the Distribution;

(ii) the Distribution is distributed by a person to another person known to that person; or

(iii) the Distributor has obtained explicit written authority from POV to perform the distribution, citing this clause number, prior to the reproduction being

made.

3.2. In each case where the Distributor makes a copy of the Software in accordance with clause 3.1, the Distributor must, unless no payment or other consideration of any type is received by Distributor in relation to the Distribution:

(a) ensure that each person who receives a copy of the Software from the Distributor is aware prior to acquiring that copy:

(i) of the full name and contact details of the Distributor, including the Distributor's web site, street address, mail address, and working email address;

(ii) that the Software is available without charge from the Site;

(iii) that no charge is being made for the granting of a licence over the Software.

(b) include a copy of the User Licence and this Distribution License with the copy of the Software. These licences must be stored in the same subdirectory on the distribution medium as the Software and named in such a way as to prominently identify their purpose;

3.3. The Distributor must not rename, reorganise or repackage any of the files comprising the Software without the prior written authority of POV.

3.4. Except as explicitly set out in this agreement, nothing in this agreement permits Distributor to make any modification to any part of the Software.

### 4. RESTRICTIONS ON DISTRIBUTION

4.1. Nothing in this agreement gives the Distributor: (a) any ability to grant any licence in respect of the use of the Software or any part of it to any person;

i. Copyright and Trademark Information

- (b) any rights or permissions in respect of, including rights or permissions to distribute or permit the use of, any Derived Code;
- (c) any right to bundle a copy of the Software (or part thereof), whether or not as part of a Distribution, with any other items, including books and magazines. POV may, in response to a request, by notice in writing and in its absolute discretion, permit such bundling on a case by case basis. This clause 4.1(c) does not apply to Distributions permitted under clause 2;
- (d) any right, permission or authorisation to infringe any Intellectual Right held by any third party.

4.2. Distributor may charge a fee for the making or the provision of a copy of the Software.

4.3. Where the making, or the provision, of a copy of the Software is authorised under the terms of clause 3 but not under those of clause 2 of this agreement, the total of all fees charged in relation to such making or provision and including all fees (including shipping and handling fees) which are charged in respect

of any software, hardware or other material provided in conjunction with or in any manner which is reasonably connected with the making, or the provision, of a copy of the Software must not exceed the reasonable costs incurred by the Distributor in making the reproduction, or in the provision, of that copy for which the fee is charged.

4.4. Notwithstanding anything else in this agreement, nothing in this agreement permits the reproduction of any part of the Software by, or on behalf of:

- (a) Any person currently listed on the Revocation List from time to time;
- (b) Any related body corporate (as that term is defined in section 50 of the Corporations Law 2001 (Cth)) of any person referred to in clause 4.4(a);
- (c) Any person in the course of preparing any publication in any format (including books, magazines, CD Rom's or on the internet) for any of the persons identified in paragraph (a);
- (d) Any person who is, or has been, in breach of this Agreement and that breach has not been waived in writing signed by POV; or
- (e) Any person to whom POV has sent a notice in writing or by email stating that that person may not distribute the Software.

4.5. From the day two years after a version of the Software more recent than the Licensed Version is made available by POV on the Site clause 3 only permits reproduction of the Software where the Distributor ensures that each recipient of such a reproduction is aware, prior to obtaining that reproduction, that that reproduction of the Software is an old version of the Software and that a more recent version of the Software is available from the Site.

## 5. COPYRIGHT AND NO LITIGATION

5.1. Copyright subsists in the Software and is protected by Australian and international copyright laws.

5.2. Nothing in this agreement gives Distributor any rights in respect of any Intellectual Rights in respect of the Software or which are held by or on behalf of POV. Distributor acknowledges that it does not acquire any rights in respect of such Intellectual Rights.

5.3. Distributor acknowledges that if it performs out any act in respect of the Software without the permission of POV it will be liable to POV for all damages POV may suffer (and which Distributor acknowledges it may suffer) as well as statutory damages to the maximum extent permitted by law and that it may also be liable to criminal prosecution.

5.4. Distributor must not commence any action against any person alleging that the Software or the use or distribution of the Software infringes any rights, including Intellectual Rights of the Distributor or of any other person. If Distributor provides one or more copies of the Software to any other person in accordance with the agreement, Distributor waives all rights it has, or may have in the future, to bring any action, directly or

indirectly, against any person to the extent that such an action relates to an infringement of any rights, including Intellectual Rights of any person in any way arising from, or in relation to, the use, or distribution, (including through the authorisation of such use or distribution) of:(a) the Software;

(b) any earlier or later version of the Software; or

(c) any other software to the extent it incorporates elements of the software referred to in paragraphs (a) or (b) of this clause

5.4.

## 6. DISCLAIMER OF WARRANTY

6.1. To the extent permitted by law, all implied terms and conditions are excluded from this agreement. Where a term or condition is implied into this agreement and that term cannot be legally excluded, that term has effect as a term or condition of this agreement. However, to the extent permitted by law, the liability of POV for a breach of such an implied term or condition is limited to the fullest extent permitted by law.

6.2. To the extent permitted by law, this Software is provided on an "AS IS" basis, without warranty of any kind, express or implied, including without limitation, any implied warranties of merchantability, fitness for a particular purpose and non-infringement of intellectual property of any third party. The Software has inherent limitations including design faults and programming bugs.

6.3. The entire risk as to the quality and performance of the Software is borne by Distributor, and it is Distributor's responsibility to ensure that the Software fulfils Distributor's requirements prior to using it in any manner (other than testing it for the purposes of this paragraph in a non-critical and non-production environment), and prior to distributing it in any fashion.

6.4. This clause 6 is an essential and material term of, and cannot be severed from, this agreement. If Distributor does not or cannot agree to be bound by this clause, or if it is unenforceable, then Distributor must not, at any time, make any reproductions of the Software under this agreement and this agreement gives the

Distributor no rights to make any reproductions of any part of the Software.

## 7. NO LIABILITY

7.1. When you distribute or use the Software you acknowledge and accept that you do so at your sole risk. Distributor agrees that under no circumstances will it have any claim against POV or any POV Associate for any loss, damages, harm, injury, expense, work stoppage, loss of business information, business interruption, computer failure or malfunction which may be suffered by you or by any third party from any cause whatsoever, howsoever arising, in connection with your use or distribution of the Software even where POV was aware, or ought to have been aware, of the potential of such loss.

7.2. Neither POV nor any POV Associate has any liability to Distributor for any indirect, general, special, incidental, punitive and/or consequential damages arising as a result of a breach of this agreement by POV or which arises in any way related to the Software or the exercise of a licence granted to Distributor under this agreement.

7.3. POV's total aggregate liability to the Distributor for all loss or damage arising in any way related to this agreement is limited to the lesser of: (a) AU\$100, and (b) the amount received by POV from Distributor as payment for the grant of a licence under this agreement.

7.4. Distributor must bring any action against POV in any way related to this agreement or the Software within 3 months of the cause of action first arising. Distributor waives any right it has to bring any action against POV and releases POV from all liability in respect of a cause of action if initiating process in relation to that action is not served on POV within 3 months of the cause of action arising. Where a particular set of facts give rise to more than one cause of action this clause 7.4 applies as if all such causes of action arise at the time the first such cause of action arises.

7.5. This clause 7 is an essential and material term of, and cannot be severed from, this agreement. If Distributor does not or cannot agree to be bound by this clause, or if it is unenforceable, then Distributor must not, at any time, make any reproductions of the Software under this agreement and this agreement gives the Distributor no rights to make any reproductions of any part of the Software.

## 8. INDEMNITY

8.1. Distributor indemnifies POV and each POV Associate and holds each of them harmless against all claims which arise from any loss, damages, harm, injury, expense, work stoppage, loss of business information, business interruption, computer failure or malfunction, which may be suffered by Distributor or any other party whatsoever as a consequence of:

- (a) any act or omission of POV and/or any POV Associate, whether negligent or not;
- (b) Distributor's use and/or distribution of the Software; or
- (c) any other cause whatsoever, howsoever arising, in connection with the Software. This clause 8 is binding on Distributor's estate, heirs, executors, legal successors, administrators, parents and/or guardians.

8.2. Distributor indemnifies POV, each POV Associate and each of the authors of any part of the Software against all loss and damage and for every other consequence flowing from any breach by Distributor of any Intellectual Right held by POV.

8.3. This clause 8 constitutes an essential and material term of, and cannot be severed from, this agreement. If Distributor does not or cannot agree to be bound by this clause, or if it is unenforceable, then Distributor must not, at any time, make any reproductions of the Software under this agreement and this agreement gives the Distributor no rights to make any reproductions of any part of the Software.

## 9. HIGH RISK ACTIVITIES

9.1. This Software and the output produced by this Software is not fault-tolerant and is not designed, manufactured or intended for use as on-line control equipment in hazardous environments requiring fail-safe performance, in which the failure of the Software could lead or directly or indirectly to death, personal injury, or severe physical or environmental damage ("High Risk Activities"). POV specifically disclaims all express or implied warranty of fitness for High Risk Activities and, notwithstanding any other term of this agreement, explicitly prohibits the use or distribution of the Software for such purposes.

## 10. ENDORSEMENT PROHIBITION

10.1. Distributor must not, without explicit written permission from POV, claim or imply in any way that:

- (a) POV or any POV Associate officially endorses or supports the Distributor or any product (such as CD, book, or magazine) associated with the Distributor or any reproduction of the Software made in accordance with this agreement; or
- (b) POV derives any benefit from any reproduction made in accordance with this agreement.

## 11. TRADEMARKS

11.1. "POV-Ray(tm)", "Persistence of Vision Raytracer(tm)" and "POV-Team(tm)" are trademarks of Persistence of Vision Raytracer Pty. Ltd. Any other trademarks referred to in this agreement are the property of their respective holders. Distributor must not use, apply for, or register anywhere in the world, any word, name

(including domain names), trade mark or device which is substantially identical or deceptively or confusingly similar to any of Persistence of Vision Raytracer Pty. Ltd's trade marks.

## 12. MISCELLANEOUS

12.1. The Official Terms, including those documents incorporated by reference into the Official Terms, and the Modification Terms constitute the entire agreement between the parties relating to the distribution of the Software and, except where stated to the contrary in writing signed by POV, supersedes all previous negotiations and correspondence in relation to it.

12.2. POV may modify this agreement at any time by making a revised licence available from the Site at <http://www.povray.org/distribution-license.html>.

This agreement is modified by replacing the terms in this agreement with those of the revised licence from the time that the revised licence is so made available. It is your responsibility to ensure that you have read and agreed to the current version of this agreement prior to distributing the Software.

12.3. Except where explicitly stated otherwise herein, if any provision of this Agreement is found to be invalid or unenforceable, the invalidity or unenforceability of such provision shall not affect the other provisions of this agreement, and all provisions not affected by such invalidity or unenforceability shall remain in full force and effect. In such cases Distributor agrees to attempt to substitute for each invalid or unenforceable provision a valid or enforceable provision which achieves to the greatest extent possible, the objectives and intention of the invalid or unenforceable provision.

12.4. A waiver of a right under this agreement is not effective unless given in writing signed by the party granting that waiver. Unless otherwise stipulated in the waiver, a waiver is only effective in respect of the circumstances in which it is given and is not a waiver in respect of any other rights or a waiver in respect of future rights or actions.

12.5. The validity and interpretation of this agreement is governed by the laws in force in the State of Victoria, Australia. Distributor submits to the exclusive jurisdiction of the courts of that State and courts located within that State exercising federal jurisdiction.

12.6. References in this agreement to "written" and "writing" mean on paper or by fax and expressly exclude email and other forms of electronic communication.

### 13. CONTACT INFORMATION

13.1. This clause 13 does not form part of the agreement. License inquiries can be made via email; please use the following address (but see 13.2 below prior to emailing) : team-coord-[three-letter month]-[four-digit year]@povray.org. for example, team-coord-jun-2004@povray.org should be used if at the time you send the email it is the month of June 2004. The changing email addresses are necessary to combat spam. Old email addresses may be deleted at POV's discretion.

13.2. Note that the address referred to in 13.1 may change for reasons other than those referred to in that clause; please check the current version of this document at <http://www.povray.org/distribution-license.html>. for the current address. Your inability or failure to contact us is no excuse for violating the licence.

13.3. Do NOT send any email attachments of any sort other than by prior arrangement. Do not send email in HTML format. EMAIL MESSAGES INCLUDING ATTACHMENTS WILL BE DELETED UNREAD.

13.4. The following postal address is only for official license business. Please note that it is preferred that initial queries about licensing be made via email; postal mail should only be used when email is not possible, or when written documents are being exchanged by prior arrangement. While it is unlikely this address will change in the short term it would be advisable to check <http://www.povray.org/distribution-license.html> for the current one prior to sending postal mail.

Persistence of Vision Raytracer Pty. Ltd.

PO Box 407

Williamstown,

Victoria 3016

Australia

POV-Ray Licence Agreement

GENERAL LICENSE AGREEMENT

FOR PERSONAL USE

Persistence of Vision Ray Tracer (POV-Ray)

Version 3.6 License and Terms & Conditions of Use

version of 1 February 2005

(also known as POVLEGAL.DOC)

Please read through the terms and conditions of this license carefully. This license is a binding legal agreement between you, the 'User' (an individual or single entity) and Persistence of Vision Raytracer Pty. Ltd. ACN 105 891 870 (herein also referred to as the "Company"), a company incorporated in the state of Victoria, Australia, for the product known as the "Persistence of Vision Ray Tracer", also referred to herein as 'POV-Ray'.

YOUR ATTENTION IS PARTICULARLY DRAWN TO THE DISCLAIMER OF WARRANTY AND NO LIABILITY AND INDEMNITY PROVISIONS. TO USE THE PERSISTENCE OF VISION RAY TRACER ("POV-RAY") YOU MUST AGREE TO BE BOUND BY THE TERMS AND CONDITIONS SET OUT IN THIS DOCUMENT. IF YOU DO NOT AGREE TO ALL THE TERMS AND CONDITIONS OF USE OF POV-RAY SET OUT IN THIS LICENSE AGREEMENT, OR IF SUCH TERMS AND CONDITIONS ARE NOT BINDING ON YOU IN YOUR JURISDICTION, THEN YOU MAY NOT USE POV-RAY IN ANY MANNER. THIS GENERAL LICENSE AGREEMENT MUST ACCOMPANY ALL POV-RAY FILES WHETHER IN THEIR OFFICIAL OR CUSTOM VERSION FORM. IT MAY NOT BE REMOVED OR MODIFIED. THIS GENERAL LICENSE AGREEMENT GOVERNS THE USE OF

POV-RAY WORLDWIDE. THIS DOCUMENT SUPERSEDES AND REPLACES ALL PREVIOUS GENERAL LICENSES.

## INTRODUCTION

This document pertains to the use of the Persistence of Vision Ray Tracer (also known as POV-Ray). It applies to all POV-Ray program source files, executable (binary) files, scene files, documentation files, help files, bitmaps and other POV-Ray files contained in official Company archives, whether in full or any part thereof, and are herein referred to as the "Software". The Company reserves the right to revise these rules in future versions and to make additional rules to address new circumstances at any time. Such rules, when made, will be posted in a revised license file, the latest version of which is available from the Company website at

<http://www.povray.org/povlegal.html>.

## USAGE PROVISIONS

Subject to the terms and conditions of this agreement, permission is granted to the User to use the Software and its associated files to create and render images. The creator of a scene file retains all rights to any scene files they create, and any images generated by the Software from them. Subject to the other terms of this license, the User is permitted to use the Software in a profit-making enterprise, provided such profit arises primarily from use of the Software and not from distribution of the Software or a work including the Software in whole or part.

Please refer to <http://www.povray.org/povlegal.html> for licenses covering distribution of the Software and works including the Software. The User is also granted the right to use the scene files, fonts, bitmaps, and include files distributed in the INCLUDE and SCENES\INCDEMO sub-directories of the Software in their own scenes. Such permission does not extend to any other files in the SCENES directory or its sub-directories. The SCENES files are for the User's enjoyment and education but may not be the basis of any derivative works unless the file in question explicitly grants permission to do such.

This licence does not grant any right of re-distribution or use in any manner other than the above. The Company has separate license documents that apply to other uses (such as re-distribution via the internet or on CD); please visit <http://www.povray.org/povlegal.html> for links to these. In particular you are advised that the sale, lease, or rental of the Software in any form without written authority from the Company is explicitly prohibited. Notwithstanding anything in the balance of this licence agreement, nothing in this licence agreement permits the installation or use of the Software in conjunction with any product (including software)

## i. Copyright and Trademark Information

produced or distributed by any party who is, or has been, in violation of this licence agreement or of the distribution licence (<http://www.povray.org/distribution-license.html>)

(or any earlier or later versions of those documents) unless:

- a. the Company has explicitly released that party in writing from the consequences of their non compliance; or
- b. both of the following are true:
  - i. the installation or use of the Software is without the User being aware of the abovementioned violation; and
  - ii. the installation or use of the Software is not a result (whether direct or indirect) of any request or action of the abovementioned party (or any of its products), any agent of that party (or any of their products), or any person(s) involved in supplying any such product to the User.

## COPYRIGHT

Copyright © 1991-2003, Persistence of Vision Team.

Copyright © 2003-2004, Persistence of Vision Raytracer Pty. Ltd.

Windows version Copyright © 1996-2003, Christopher Cason.

Copyright subsists in this Software which is protected by Australian and international copyright laws. The Software is NOT PUBLIC DOMAIN. Nothing in this agreement shall give you any rights in respect of the intellectual property of the Company and you acknowledge that you do not acquire any rights in respect of such intellectual property rights. You acknowledge that the Software is the valuable intellectual property of the Company and that if you use, modify or distribute the Software for unauthorized purposes or in an unauthorized manner (or cause or allow the forgoing to occur), you will be liable to the Company for any damages it may suffer (and which you acknowledge it may suffer) as well as statutory damages to the maximum extent permitted by law and also that you may be liable to

criminal prosecution. You indemnify the Company and the authors of the Software for every single consequence flowing from the aforementioned events.

## DISCLAIMER OF WARRANTY

express or implied, including without limitation, any implied warranties of merchantability, fitness for a particular purpose and non-infringement of intellectual property of any third party. This Software has inherent limitations including design faults and programming bugs. The entire risk as to the quality and performance of the Software is borne by you, and it is your responsibility to ensure that it does what you require it to do prior to using it for any purpose (other than testing it), and prior to distributing it in any fashion. Should the Software prove defective, you agree that you alone assume the entire cost resulting in any way from such defect.

This disclaimer of warranty constitutes an essential and material term of this agreement. If you do not or cannot accept this, or if it is unenforceable in your jurisdiction, then you may not use the Software in any manner.

## NO LIABILITY

When you use the Software you acknowledge and accept that you do so at your sole risk. You agree that under no circumstances shall you have any claim against the Company or anyone associated directly or indirectly with the Company whether as employee, subcontractor, agent, representative, consultant, licensee or otherwise ("Company Associates") for any loss, damages, harm, injury, expense, work stoppage, loss of business information, business interruption, computer failure or malfunction which may be suffered by you or by any third party from any cause whatsoever, howsoever arising, in connection with your use or distribution of the Software even where the Company were aware, or ought to have been aware, of the potential of such loss. Damages referred to above shall include direct, indirect, general, special, incidental, punitive and/or consequential. This disclaimer of liability constitutes an essential and material term of this agreement. If you do not or cannot accept this, or if it is unenforceable in your jurisdiction, then you may not use the Software.

## INDEMNITY

You indemnify the Company and Company Associates and hold them harmless against any claims which may arise from any loss, damages, harm, injury, expense, work stoppage, loss of business information, business interruption, computer failure or malfunction, which may be suffered by you or any other party whatsoever as a consequence of any act or omission of the Company and/or Company Associates, whether negligent or not, arising out of your use and/or distribution of the Software, or from any other cause whatsoever, howsoever arising, in connection with the Software. These provisions are binding on your estate, heirs, executors, legal successors, administrators, parents and/or guardians.

This indemnification constitutes an essential and material term of this agreement. If you do not or cannot accept this, or if it is unenforceable in your jurisdiction, then you may not use the Software.**HIGH RISK ACTIVITIES**

This Software and the output produced by this Software is not fault-tolerant and is not designed, manufactured or intended for use as on-line control equipment in hazardous environments requiring fail-safe performance, in which the failure of the Software could lead or directly or indirectly to death, personal injury, or severe physical or environmental damage ("High Risk Activities"). The Company specifically disclaims any express or implied warranty of fitness for High Risk Activities and explicitly prohibits the use of the Software for such purposes.

## CRYPTOGRAPHIC SIGNING OF DOCUMENTS

Changes to this Agreement and documents issued under its authority may be cryptographically signed by the POV-Ray Team Co-ordinator's private PGP key.

In the absence of evidence to the contrary, such documents shall be considered, under the terms of this Agreement, to be authentic provided the signature is

valid. The master copy of this Agreement at <http://www.povray.org/povlegal.html> will also be signed by the current version of the team-coordinator's key.

The public key for the POV-Ray Team-coordinator can be retrieved from the location  
<https://secure.povray.org/keys/>. The current fingerprint for it is

B4DD 932A C080 C3A3 6EA2 9952 DB04 4A74 9901 4518.

## MISCELLANEOUS

This Agreement constitutes the complete agreement concerning this license. Any changes to this agreement must be in writing and may take the form of

notifications by the Company to you, or through posting notifications on the Company website. **THE USE OF THIS SOFTWARE BY ANY PERSON OR ENTITY IS**

**EXPRESSLY MADE CONDITIONAL ON THEIR ACCEPTANCE OF THE TERMS SET FORTH HEREIN.**  
Except where explicitly stated otherwise herein, if any provision of this

Agreement is found to be invalid or unenforceable, the invalidity or unenforceability of such provision shall not affect the other provisions of this agreement, and all provisions not affected by such invalidity or unenforceability shall remain in full force and effect. In such cases you agree to attempt to substitute for each invalid or unenforceable provision a valid or enforceable provision which achieves to the greatest extent possible, the objectives and intention of the invalid or unenforceable

provision. The validity and interpretation of this agreement will be governed by the laws of Australia in the state of Victoria (except for conflict of law provisions).

## CONTACT INFORMATION

License inquiries can be made via email; please use the following address (but see below prior to emailing) : team-coord-[three-letter month]-[four-digit year]@povray.org for example, team-coord-jun-2004@povray.org should be used if at the time you send the email it is the month of June 2004. The changing email addresses are necessary to combat spam and email viruses. Old email addresses may be deleted at our discretion.

## i. Copyright and Trademark Information

Note that the above address may change for reasons other than that given above; please check the version of this document at <http://www.povray.org/povlegal.html> for the current address. Note that your inability or failure to contact us for any reason is not an excuse for violating this licence.

Do NOT send any attachments of any sort other than by prior arrangement.

**EMAIL MESSAGES INCLUDING ATTACHMENTS WILL BE DELETED UNREAD.**

The following postal address is only for official license business. Please note that it is preferred that initial queries about licensing be made via email ; postal mail should only be used when email is not possible, or when written documents are being exchanged by prior arrangement.

Persistence of Vision Raytracer Pty. Ltd.

PO Box 407

Williamstown,

Victoria 3016

Australia

Portions of this software are owned by Siemens PLM © 1986-2013. All Rights Reserved. Parasolid, Unigraphics, and SolidEdge are registered trademarks and JT is a trademark of Siemens Product Lifecycle Management Software, Inc.

SolidWorks is a registered trademark of SolidWorks Corporation.

Portions of this software are owned by Spatial Corp. © 1986-2013. All Rights Reserved. ACIS, SAT and SAB are registered trademarks of Spatial Corp.

Contains Teigha for .dwg files licensed from the Open Design Alliance. Teigha is a trademark of the Open Design Alliance.

Development tools and related technology provided under license from 3Dconnexion. © 1992 – 2008 3Dconnexion. All rights reserved.

•TraceParts is owned by TraceParts S.A. TraceParts is a registered trademark of TraceParts S.A.

Copyright © 1991-2017 Unicode, Inc. All rights reserved.

Distributed under the Terms of Use in <http://www.unicode.org/copyright.html>. Permission is hereby granted, free of charge, to any person obtaining a copy of the Unicode data files and any associated documentation (the "Data Files") or Unicode software and any associated documentation (the "Software") to deal in the Data Files or Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, and/or sell copies of the Data Files or Software, and to permit persons to whom the Data Files or Software are furnished to do so, provided that either (a) this copyright and permission notice appear with all copies of the Data Files or Software, or

(b) this copyright and permission notice appear in associated Documentation.

**THE DATA FILES AND SOFTWARE ARE PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE**

**WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT OF THIRD PARTY RIGHTS. IN NO EVENT SHALL THE COPYRIGHT HOLDER OR HOLDERS INCLUDED IN THIS NOTICE BE LIABLE FOR ANY CLAIM, OR ANY SPECIAL INDIRECT OR CONSEQUENTIAL DAMAGES, OR ANY DAMAGES WHATSOEVER RESULTING FROM LOSS OF USE, DATA OR PROFITS, WHETHER IN AN ACTION OF CONTRACT, NEGLIGENCE OR OTHER TORTIOUS ACTION, ARISING OUT OF OR IN CONNECTION WITH THE USE OR PERFORMANCE OF THE DATA FILES OR SOFTWARE.**

Except as contained in this notice, the name of a copyright holder shall not be used in advertising or otherwise to promote the sale, use or other dealings in these Data Files or Software without prior written authorization of the copyright holder.

Portions of this software Copyright © 1992-2008 The University of Tennessee. All rights reserved.

This product includes software developed by XHEO INC (<http://xheo.com>).

Portions of this software are owned by Tech Soft 3D, Inc. Copyright © 1996-2013. All rights reserved. HOOPS is a registered trademark of Tech Soft 3D, Inc.

Portions of this software are owned by MachineWorks Limited. Copyright ©2013. All rights reserved. Polygonica is a registered trademark of MachineWorks Limited.

#### Apache License

Version 2.0, January 2004 <http://www.apache.org/licenses/>

#### TERMS AND CONDITIONS FOR USE, REPRODUCTION, AND DISTRIBUTION

##### 1. Definitions.

"License" shall mean the terms and conditions for use, reproduction, and distribution as defined by Sections 1 through 9 of this document.

"Licensor" shall mean the copyright owner or entity authorized by the copyright owner that is granting the License.

"Legal Entity" shall mean the union of the acting entity and all other entities that control, are controlled by, or are under common control with that entity. For the purposes of this definition,

"Control" means (i) the power, direct or indirect, to cause the direction or management of such entity, whether by contract or otherwise, or (ii) ownership of fifty percent (50%) or more of the outstanding shares, or (iii) beneficial ownership of such entity.

"You" (or "Your") shall mean an individual or Legal Entity exercising permissions granted by this License.

"Source" form shall mean the preferred form for making modifications, including but not limited to software source code, documentation source, and configuration files.

"Object" form shall mean any form resulting from mechanical transformation or translation of a Source form, including but not limited to compiled object code, generated documentation, and conversions to other media types.

"Work" shall mean the work of authorship, whether in Source or Object form, made available under the License, as indicated by a copyright notice that is included in or attached to the work (an example is provided in the Appendix below).

"Derivative Works" shall mean any work, whether in Source or Object form, that is based on (or derived from) the Work and for which the editorial revisions, annotations, elaborations, or other modifications represent, as a whole, an original work of authorship. For the purposes of this License, Derivative Works shall not include works that remain separable from, or merely link (or bind by name) to the interfaces of, the Work and Derivative Works thereof.

"Contribution" shall mean any work of authorship, including the original version of the Work and any modifications or additions to that Work or Derivative Works thereof, that is intentionally submitted to Licensor for inclusion in the Work by the copyright owner or by an individual or Legal Entity authorized to submit on behalf of the copyright owner. For the purposes of this definition,

"Submitted" means any form of electronic, verbal, or written communication sent to the Licensor or its representatives, including but not limited to communication on electronic mailing lists, source code control systems, and issue tracking systems that are managed by, or on behalf of, the Licensor for the purpose of discussing and improving the Work, but excluding communication that is conspicuously marked or otherwise designated in writing by the copyright owner as "Not a Contribution."

"Contributor" shall mean Licensor and any individual or Legal Entity on behalf of whom a Contribution has been received by Licensor and subsequently incorporated within the Work.

## i. Copyright and Trademark Information

2. Grant of Copyright License. Subject to the terms and conditions of this License, each Contributor hereby grants to You a perpetual, worldwide, non-exclusive, no-charge, royalty-free, irrevocable copyright license to reproduce, prepare Derivative Works of, publicly display, publicly perform, sublicense, and distribute the Work and such Derivative Works in Source or Object form.

3. Grant of Patent License. Subject to the terms and conditions of this License, each Contributor hereby grants to You a perpetual, worldwide, non-exclusive, no-charge, royalty-free, irrevocable (except as stated in this section) patent license to make, have made, use, offer to sell, sell, import, and otherwise transfer the Work, where such license applies only to those patent claims licensable by such Contributor that are necessarily infringed by their Contribution(s) alone or by combination of their Contribution(s) with the Work to which such Contribution(s) was submitted. If You institute patent litigation against any entity (including a cross-claim or counterclaim in a lawsuit) alleging that the Work or a Contribution incorporated within the Work constitutes direct or contributory patent infringement, then any patent licenses granted to You under this License for that Work shall terminate as of the date such litigation is filed.

4. Redistribution. You may reproduce and distribute copies of the Work or Derivative Works thereof in any medium, with or without modifications, and in Source or Object form, provided that You meet the following conditions:

- (a) You must give any other recipients of the Work or Derivative Works a copy of this License; and
- (b) You must cause any modified files to carry prominent notices stating that You changed the files; and
- (c) You must retain, in the Source form of any Derivative Works that You distribute, all copyright, patent, trademark, and attribution notices from the Source form of the Work, excluding those notices that do not pertain to any part of the Derivative Works; and
- (d) If the Work includes a "NOTICE" text file as part of its distribution, then any Derivative Works that You distribute must include a readable copy of the attribution notices contained within such NOTICE file, excluding those notices that do not pertain to any part of the Derivative Works, in at least one of the following places: within a NOTICE text file distributed as part of the Derivative Works; within the Source form or documentation, if provided along with the Derivative Works; or, within a display generated by the Derivative Works, if and wherever such third-party notices normally appear. The contents of the NOTICE file are for informational purposes only and do not modify the License. You may add Your own attribution notices within Derivative Works that You distribute, alongside or as an addendum to the NOTICE text from the Work, provided that such additional attribution notices cannot be construed as modifying the License.

You may add Your own copyright statement to Your modifications and may provide additional or different license terms and conditions for use, reproduction, or distribution of Your modifications, or for any such Derivative Works as a whole, provided Your use, reproduction, and distribution of the Work otherwise complies with the conditions stated in this License.

5. Submission of Contributions. Unless You explicitly state otherwise, any Contribution

intentionally submitted for inclusion in the Work by You to the Licensor shall be under the terms and conditions of this License, without any additional terms or conditions. Notwithstanding the above, nothing herein shall supersede or modify the terms of any separate license agreement you may have executed with Licensor regarding such Contributions.

6. Trademarks. This License does not grant permission to use the trade names, trademarks, service marks, or product names of the Licensor, except as required for reasonable and customary use in describing the origin of the Work and reproducing the content of the NOTICE file.

7. Disclaimer of Warranty. Unless required by applicable law or agreed to in writing, Licensor provides the Work (and each Contributor provides its Contributions) on an "AS IS" BASIS, WITHOUT WARRANTIES OR CONDITIONS OF ANY KIND, either express or implied, including, without limitation, any warranties or conditions of TITLE, NON-INFRINGEMENT, MERCHANTABILITY, or FITNESS FOR A PARTICULAR PURPOSE. You are solely

responsible for determining the appropriateness of using or redistributing the Work and assume any risks associated with Your exercise of permissions under this License.

i. Copyright and Trademark Information

8. Limitation of Liability. In no event and under no legal theory, whether in tort (including negligence), contract, or otherwise, unless required by applicable law (such as deliberate and grossly negligent acts) or agreed to in writing, shall any Contributor be liable to You for damages, including any direct, indirect, special, incidental, or consequential damages of any character arising as a result of this License or out of the use or inability to use the Work (including but not limited to damages for loss of goodwill, work stoppage, computer failure or malfunction, or any and all other commercial damages or losses), even if such Contributor has been advised of the possibility of such damages.

9. Accepting Warranty or Additional Liability. While redistributing the Work or Derivative Works thereof, You may choose to offer, and charge a fee for, acceptance of support, warranty, indemnity, or other liability obligations and/or rights consistent with this License. However, in accepting such obligations, You may act only on Your own behalf and on Your sole responsibility, not on behalf of any other Contributor, and only if You agree to indemnify, defend, and hold each Contributor harmless for any liability incurred by, or claims asserted against, such Contributor by reason of your accepting any such warranty or additional liability.

**END OF TERMS AND CONDITIONS**

# Contents

<b>i. Copyright and Trademark Information.....</b>	<b>iii</b>
<b>Chapter 1: Overview of Simulating Physics in ANSYS Discovery AIM.....</b>	<b>34</b>
1.1. The ANSYS Product Improvement Program.....	36
1.2. ANSYS Subscription Licensing.....	39
1.3. Creating Simulations Using Template Apps.....	39
1.3.1. Structural Template.....	40
1.3.2. Fluid Flow Template.....	41
1.3.2.1. Fluid Flow: Model.....	41
1.3.2.2. Fluid Flow: Physics.....	42
1.3.2.3. Fluid Physics Region.....	42
1.3.2.4. Solid Physics Region.....	43
1.3.2.5. Fluid Summary.....	43
1.3.3. Thermal Template.....	44
1.3.3.1. Thermal: Model.....	44
1.3.3.2. Thermal: Physics.....	44
1.3.3.3. Fluid Physics Region.....	45
1.3.3.4. Solid Physics Region.....	46
1.3.3.5. Thermal Summary.....	46
1.3.4. Electromagnetics Template.....	46
1.3.4.1. Electromagnetics: Model.....	47
1.3.4.2. Electromagnetics: Physics.....	47
1.3.4.3. Surrounding Region (Electromagnetic).....	49
1.3.4.4. Surrounding Region (Electrostatic).....	50
1.3.4.5. Solid Physics Region.....	50
1.3.4.6. Electromagnetic Summary.....	50
1.3.5. Topology Optimization Template.....	51
1.3.6. Fluid-Structure Interaction Template.....	52
1.3.6.1. Fluid-Structure Interaction: Model.....	52
1.3.6.2. Fluid-Structure Interaction: Physics.....	53
1.3.6.3. Fluid Physics Region.....	53
1.3.6.4. Solid Physics Region.....	53
1.3.6.5. Fluid-Structure Interaction Summary.....	54
1.3.7. Fluid-Solid Heat Transfer Template.....	54
1.3.7.1. Fluid-Solid Heat Transfer: Model.....	55
1.3.7.2. Fluid-Solid Heat Transfer: Physics.....	55
1.3.7.3. Fluid Physics Region.....	56
1.3.7.4. Solid Physics Region.....	56
1.3.7.5. Fluid-Solid Heat Transfer Summary.....	56
1.3.8. Polymer Extrusion Template.....	57
1.3.8.1. Simulating a Polymer Extrusion.....	58

1.3.9. Polymer Blow Molding Template.....	60
1.3.9.1. Blow Molding: Model.....	60
1.3.9.2. Blow Molding: Physics.....	61
1.3.9.3. Blow Molding: Physics Region.....	61
1.3.9.4. Simulating a Polymer Blow Molding.....	61
1.3.10. Geometry Modeling Template.....	64
1.3.11. Connect to Discovery Live Template.....	64
1.3.11.1. Connect to Discovery Live Template Considerations.....	66
1.3.12. Connect to Mechanical Template.....	69
1.3.13. Connect to Fluent Template.....	69
1.4. Performing Common Operations.....	70
1.4.1. Updating Tasks and Understanding their States.....	70
1.4.1.1. Interrupting an Update.....	71
1.4.1.2. Continuing from <b>Partially Updated</b> .....	71
1.4.2. Defining Locations.....	71
1.4.3. Defining Selection Sets.....	71
1.4.4. Manipulating a Geometric Entity.....	72
1.4.5. Understanding Reference Frames.....	73
1.4.5.1. Elements of a Reference Frame.....	73
1.4.5.2. Creating Reference Frames.....	77
1.4.5.3. Coordinate Systems.....	77
1.4.6. Using Construction Geometries.....	79
1.4.6.1. Points.....	80
1.4.6.2. Lines.....	82
1.4.6.3. Planes.....	83
1.4.7. Using Parameters and Design Points.....	84
1.4.8. Using Expressions.....	84
1.4.9. Defining Tabular Data.....	84
1.4.9.1. Tabular Data in Expressions.....	85
1.5. Managing Your Project.....	86
1.5.1. Associating Other Files with the Project.....	87
1.5.2. Archiving Projects for Sharing or Preserving Your Work.....	87
1.5.3. Accessing Your Fluent Project in AIM.....	87
1.6. Working with Project Reports.....	88
1.6.1. Generating a Project Report.....	88
1.6.2. Configuring Project Reports to Open Automatically.....	88
1.6.3. Editing a Project Report.....	88
1.7. Creating a Customized Workflow.....	89
1.7.1. Using Scripting Capabilities.....	89
1.7.2. Enabling Customization.....	89
<b>Chapter 2: Geometry.....</b>	<b>92</b>
2.1. Model Editing.....	92
2.2. Import.....	95
2.2.1. Usage Notes for Import Tasks.....	95

2.2.2. Geometry Import Source.....	95
2.2.2.1. Specifying a Geometry Import Source by Browsing.....	96
2.2.2.2. Specifying a Geometry Import Source by Active CAD Attach.....	96
2.2.2.3. Geometry Preferences.....	96
2.2.2.4. Geometry Dimensions.....	98
2.3. Configuration.....	99
2.3.1. Suppress Controls.....	99
2.3.1.1. Creating a Suppress Control.....	99
2.3.2. Move/Rotate Controls.....	100
2.3.2.1. Creating a Move/Rotate Control.....	100
2.3.3. Capping Surfaces.....	100
2.3.3.1. Creating Capping Surfaces.....	100
2.3.4. Creating a Box.....	101
2.3.4.1. Creating a Box Based on Geometry Selection.....	101
2.3.4.2. Creating a Box by Two Points.....	101
<b>Chapter 3: Meshing.....</b>	<b>104</b>
3.1. Overview of Meshing.....	104
3.2. Overview of Automatic Physics-aware Meshing.....	104
3.2.1. Adjusting the Solution Fidelity.....	105
3.2.2. Controlling the Mesh by Using Manual Meshing.....	106
3.2.3. Viewing the Mesh.....	106
3.3. Overview of Manual Meshing.....	106
3.3.1. Extracted Volume Workflow.....	106
3.3.2. Geometry Simplification Workflow.....	107
3.3.3. Flow Volume Meshing Workflow.....	108
3.3.4. Part-based Meshing Workflow.....	108
3.3.5. Best Practices.....	109
3.3.5.1. Structural Meshing Best Practices.....	109
3.3.5.2. Fluid Flow Meshing Best Practices.....	110
3.3.6. Mesh Size and Distribution.....	112
3.3.6.1. Proximity Size Function.....	114
3.3.6.2. Curvature Size Function.....	115
3.3.6.3. Fixed Size Function.....	115
3.3.6.4. Adaptive Size Function.....	115
3.3.6.5. Reuse of Computed Size Functions.....	116
3.3.7. Volume Creation.....	116
3.3.7.1. Volume Creation Limitations.....	117
3.3.7.2. Setting Defaults for Volume Creation.....	117
3.3.7.3. Global Sizing for Volume Creation.....	126
3.3.7.4. Additional Settings for Volume Creation.....	136
3.3.7.5. Troubleshooting Volume Creation Problems.....	137
3.3.8. Flow Volume Meshing.....	141
3.3.8.1. Common Boundary Layer Settings for Flow Volume Meshing.....	141
3.3.9. Part-based Meshing.....	147

3.3.9.1. Usage Notes for Part-based Meshing.....	147
3.3.9.2. Setting Defaults for Part-based Meshing.....	148
3.3.9.3. Parallel Part Meshing Best Practices.....	149
3.3.9.4. Global Sizing for Part-based Meshing.....	149
3.3.9.5. Common Boundary Layer Settings for Part-based Meshing.....	160
3.3.9.6. Additional Settings for Part-based Meshing.....	165
3.3.9.7. Troubleshooting Part-based Meshing Problems.....	170
3.3.10. Local Mesh Sizing.....	170
3.3.10.1. Defining Sizing on a Body.....	171
3.3.10.2. Defining Sizing on a Face.....	172
3.3.10.3. Defining Sizing on an Edge.....	172
3.3.10.4. Local Mesh Sizing Control Options.....	173
3.3.11. Volume Definitions.....	176
3.3.11.1. Defining a Volume - Extracted Volume.....	176
3.3.11.2. Defining a Volume - Geometry Simplification.....	176
3.3.11.3. Extracted Volume Properties.....	176
3.3.11.4. Geometry Simplification Properties.....	177
3.3.12. Boundary Layers.....	178
3.3.12.1. Defining Boundary Layers.....	178
3.3.12.2. Local Boundary Layer Control Options.....	179
3.3.13. Protected Topologies in a Mesh.....	183
3.3.13.1. Creating a Selection Set to Preserve Topologies in the Mesh.....	184
3.3.13.2. Protected Topologies and Mesh State.....	184
3.3.14. Element Shape.....	184
3.3.14.1. Defining the Element Shape of a Mesh.....	184
3.3.14.2. Element Shape Settings.....	185
3.3.15. Evaluating Mesh Quality.....	186
3.3.15.1. Viewing Mesh Diagnostics.....	187
3.3.15.2. Element Quality.....	187
3.3.15.3. Aspect Ratio Calculation for Triangles.....	188
3.3.15.4. Aspect Ratio Calculation for Quadrilaterals.....	188
3.3.15.5. Jacobian Ratio.....	189
3.3.15.6. Warping Factor.....	191
3.3.15.7. Parallel Deviation.....	193
3.3.15.8. Maximum Corner Angle.....	193
3.3.15.9. Skewness.....	194
3.3.15.10. Orthogonal Quality.....	195
3.3.15.11. Size Change.....	197
<b>Chapter 4: Physics.....</b>	<b>198</b>
4.1. Adjusting the Calculation Type Settings.....	198
4.1.1. Frequency Response Calculation Type Settings.....	198
4.1.1.1. Specifying a Single Frequency for Frequency Response Calculations.....	198
4.1.1.2. <b>Specifying a Range of Frequencies for Frequency Response Calculations.....</b>	<b>199</b>

4.1.1.3. Specifying a Tabular List of Frequencies for Frequency Response Calculations.....	199
4.2. Fidelity Capture.....	200
4.2.1. Controlling the Mesh by Using Fidelity Capture.....	200
4.3. Physics Regions.....	200
4.3.1. Fluid Flow Physics Regions.....	202
4.3.2. Structural Physics Regions.....	204
4.3.3. Electromagnetics Physics Region.....	205
4.3.4. Electric Conduction Physics Regions.....	206
4.3.5. Electrostatic Physics Regions.....	206
4.3.6. Thermal Physics Regions.....	207
4.3.7. Polymer Extrusion Physics Regions.....	209
4.3.8. Polymer Blow Molding Physics Regions.....	210
4.3.9. Changing a Region's Physics Type.....	212
4.3.10. Defining a New Physics Region.....	214
4.3.10.1. Region Connectivity and Polymer Extrusion.....	215
4.4. Material Assignments.....	216
4.4.1. Assigning Materials.....	216
4.4.1.1. Zero-Thermal-Strain Reference Temperature.....	218
4.4.1.2. Ideal Gas Law.....	218
4.4.1.3. Defining the Material Orientation for Orthotropic Elasticity.....	219
4.4.2. Creating Materials.....	219
4.4.3. Exporting Materials.....	219
4.4.4. Copy Material Assignments from Other Simulations.....	220
4.4.5. Adding a Library.....	220
4.4.6. Defining Material Properties.....	220
4.4.6.1. Bilinear Isotropic Hardening Model .....	221
4.4.6.2. Density.....	222
4.4.6.3. Elasticity.....	222
4.4.6.4. Isotropic Resistivity.....	223
4.4.6.5. Isotropic Seebeck Coefficient.....	223
4.4.6.6. Molar Mass.....	223
4.4.6.7. Specific Heat Capacity.....	223
4.4.6.8. Thermal Conductivity.....	223
4.4.6.9. Thermal Expansion.....	223
4.4.6.10. Viscosity.....	224
4.4.6.11. Viscosity Models for Polymer Extrusion Simulations.....	224
4.4.6.12. Viscosity Models for Polymer Blow Molding Simulations.....	242
4.4.6.13. Fatigue Material Properties.....	243
4.4.6.14. Electromagnetic Material Properties.....	246
4.4.7. Customizing the Appearance of Materials.....	250
4.5. Physics Options.....	251
4.5.1. Fluid Flow Physics Options.....	252
4.5.1.1. Flow Model Options.....	252

4.5.1.2. Turbulence Modeling Options.....	255
4.5.2. Structural Physics Options.....	259
4.5.2.1. Setting Solid Physics Options.....	259
4.5.2.2. Global Modeling Options.....	260
4.5.3. Electromagnetic Physics Options.....	260
4.5.3.1. Compute Inductance.....	260
4.5.3.2. Compute Impedance.....	260
4.5.3.3. Setting Ambient Temperature.....	261
4.5.3.4. Conductor Threshold Settings.....	261
4.5.3.5. Controlling Eddy Effect and Displacement Current Computations.....	261
4.5.3.6. Controlling Core Loss Computation.....	262
4.5.3.7. Setting Torque Computation.....	262
4.5.3.8. Setting Force Computation.....	263
4.5.4. Optimization Options.....	263
4.5.4.1. Specifying Optimization Options.....	263
4.5.5. Polymer Physics Options.....	264
4.5.5.1. Setting Polymer Physics Options.....	264
4.6. Physics Conditions.....	264
4.6.1. Fluid Flow Conditions.....	265
4.6.1.1. Far Field.....	265
4.6.1.2. Heat Source.....	266
4.6.1.3. Inlets.....	266
4.6.1.4. Momentum Source.....	269
4.6.1.5. Openings.....	271
4.6.1.6. Outlets.....	273
4.6.1.7. Particle Injections.....	276
4.6.1.8. Porous Media.....	280
4.6.1.9. Symmetry.....	282
4.6.1.10. Walls.....	283
4.6.1.11. Turbulence Specification Methods.....	286
4.6.2. Structural Loads and Constraints.....	287
4.6.2.1. Body Temperature.....	287
4.6.2.2. Bolt Pretension.....	288
4.6.2.3. Displacement.....	289
4.6.2.4. Distributed Mass.....	291
4.6.2.5. Inertia Load.....	292
4.6.2.6. Force.....	292
4.6.2.7. Moment.....	294
4.6.2.8. Pressure.....	295
4.6.2.9. Power Spectral Density (PSD).....	296
4.6.2.10. Point Mass.....	297
4.6.2.11. Support.....	297
4.6.3. Thermal Loads and Constraints.....	298
4.6.3.1. Convection.....	299

4.6.3.2. Convection and Radiation.....	300
4.6.3.3. Heat Flow.....	300
4.6.3.4. Heat Flux.....	301
4.6.3.5. Heat Generation.....	301
4.6.3.6. Heat Source.....	302
4.6.3.7. Insulated.....	302
4.6.3.8. Radiation.....	302
4.6.3.9. Temperature.....	303
4.6.4. Electromagnetics Conditions.....	303
4.6.4.1. Current.....	304
4.6.4.2. Terminal.....	306
4.6.4.3. Insulating.....	307
4.6.4.4. Body Temperature.....	308
4.6.5. Electric Conduction Loads and Constraints.....	308
4.6.5.1. Current.....	308
4.6.5.2. Equipotential.....	309
4.6.5.3. Body Temperature.....	309
4.6.5.4. Voltage.....	310
4.6.6. Electrostatic Loads and Constraints.....	310
4.6.6.1. Body Temperature.....	310
4.6.6.2. Electric Charge.....	310
4.6.6.3. Equipotential.....	311
4.6.6.4. Voltage.....	311
4.6.7. Optimization Conditions.....	311
4.6.7.1. Setting the Manufacturing Pull Direction.....	312
4.6.7.2. Specifying Size Limits.....	312
4.6.7.3. Setting the Stress Limit.....	312
4.6.7.4. Specifying Reflective Symmetry.....	312
4.6.7.5. Specifying Rotational Symmetry.....	313
4.6.8. Polymer Extrusion Conditions.....	313
4.6.8.1. Inlet Conditions.....	314
4.6.8.2. Outlet Conditions.....	316
4.6.8.3. Walls for Polymer Extrusion Simulations.....	317
4.6.8.4. Symmetry.....	322
4.6.8.5. Restrictors.....	322
4.6.8.6. Extrudate Deformation Conditions.....	323
4.6.8.7. Extrudate Exits.....	325
4.6.8.8. Free Surface Boundaries.....	325
4.6.8.9. Die Deformation Conditions.....	325
4.6.8.10. Energy Specification for Polymer Extrusion Flows.....	330
4.6.9. Polymer Blow Molding Conditions.....	330
4.6.9.1. Setting Pressure Conditions.....	331
4.6.9.2. Setting Polymer Edge Conditions.....	331
4.6.9.3. Setting Fixed Mold Conditions.....	331

4.6.9.4. Setting Force Driven Mold Conditions.....	331
4.6.9.5. Setting Velocity Driven Mold Conditions.....	332
4.6.9.6. Setting Symmetry Conditions.....	332
4.7. Initial Conditions.....	332
4.7.1. Initial Conditions for Fluid Flows.....	332
4.7.1.1. Initializing the Solution.....	333
4.7.2. Initial Conditions for Time-Dependent Fluid Flows.....	333
4.7.3. <b>Initial Conditions for Time Dependent Thermal</b> .....	334
4.7.3.1. Modifying the Initial Condition.....	334
4.7.4. Initial Conditions for Polymer Extrusion Flows .....	334
4.7.4.1. Setting Initial Temperatures for Polymer Extrusion Flows.....	334
4.7.5. Initial Conditions for Polymer Blow Molding .....	334
4.7.5.1. Setting Initial Temperatures for Polymer Blow Molding.....	334
4.7.5.2. Setting the Initial Layer Thickness for Polymer Blow Molding.....	335
4.8. Interface Conditions.....	335
4.8.1. Beams.....	336
4.8.1.1. Beam Workflow.....	336
4.8.1.2. Defining the Beam End Release.....	337
4.8.2. Specifying Contacts.....	337
4.8.2.1. Automatically Generating Contacts.....	338
4.8.2.2. Manually Setting Up a Contact.....	339
4.8.2.3. Defining Contact Behavior Properties.....	340
4.8.2.4. Contact Best Practices.....	353
4.8.3. Specifying Contacts for Blow Molding Simulations.....	360
4.8.3.1. Mold to Fluid Contact Behavior.....	360
4.8.4. Setting Up a Joint.....	361
4.8.4.1. Specifying Joint Behaviors.....	362
4.8.5. Specifying Structural Springs.....	367
4.8.5.1. Setting Up a Spring.....	368
4.8.5.2. Specifying the Behavior for a 1-D Spring.....	368
4.8.5.3. Defining the Behavior of a 3-D Spring.....	369
4.8.6. Identifying Connections.....	370
4.8.7. Unconstrained Body Detection.....	370
4.8.8. Region Interface Conditions.....	370
4.8.8.1. Adding Region Interface Conditions for Fluid Flow Simulations.....	372
4.8.8.2. Adding Region Interface Conditions for Polymer Extrusion Simulations.....	374
4.8.8.3. Automatically Generating Region Interfaces for Fluid Flow Simulations.....	379
4.8.8.4. Region Interface Statistics for Fluid Flow Simulations.....	380
4.8.9. Physics Coupling Interface.....	380
4.8.9.1. Data Transfers Possible using the Physics Coupling Interface.....	381
4.8.9.2. Control the Mapping on the Physics Coupling Interface.....	386
4.8.9.3. Physics Coupling Transcript.....	389
4.8.9.4. Mapping.....	394
4.9. Solver Options.....	408

4.9.1. Fluids Solver Options.....	409
4.9.1.1. Launching the Solver on Multiple Processes.....	409
4.9.1.2. Solving Steady-State Fluid Flow Simulations.....	410
4.9.1.3. Solving Time-Dependent Fluid Flow Simulations.....	410
4.9.1.4. Controlling Residuals.....	412
4.9.1.5. Adjusting Solver Meshing Controls.....	413
4.9.1.6. Solving Particle Tracks.....	415
4.9.1.7. Choosing a Gradient Method.....	416
4.9.1.8. Choosing an Advection Scheme.....	417
4.9.1.9. Choosing a Pressure Scheme.....	417
4.9.1.10. Applying Temporal Discretization.....	418
4.9.1.11. Choosing a Pressure-Velocity Coupling Scheme.....	419
4.9.1.12. Controlling Advancement.....	419
4.9.1.13. Setting Explicit Relaxation Factors.....	421
4.9.1.14. Applying High Order Term Relaxation.....	421
4.9.1.15. Using the Linear Solver.....	422
4.9.1.16. Setting Solution Limits.....	423
4.9.1.17. Setting Output Controls for Time-Dependent Fluid Flows.....	423
4.9.1.18. Judging Convergence.....	424
4.9.2. Structural Solver Options.....	430
4.9.2.1. Configuring the Solver for a Static Structural Solution.....	431
4.9.2.2. Configuring the Solver for a Modal Structural Solution.....	433
4.9.2.3. Configuring the Solver for an Eigenvalue Buckling Solution.....	435
4.9.2.4. Setting Solver Launch Options for a Structural Solution.....	437
4.9.2.5. Controlling Output Data for a Structural Solution.....	438
4.9.2.6. Setting Nonlinear Controls in a Static Structural Simulation.....	439
4.9.3. Thermal Solver Options.....	442
4.9.3.1. Configuring the Solver for a Thermal Solution.....	443
4.9.3.2. Setting Solver Launch Options for a Thermal Solution.....	444
4.9.3.3. Controlling Output Data for a Thermal Solution.....	445
4.9.3.4. Setting Nonlinear Controls in a Steady-State Thermal Simulation.....	446
4.9.3.5. Setting Nonlinear Controls in a Time-Dependent Thermal Simulation.....	449
4.9.4. Electric Conduction Solver Options.....	452
4.9.4.1. Configuring the Solver for an Electric Conduction Solution.....	453
4.9.4.2. Setting Solver Launch Options for an Electric Conduction Solution.....	454
4.9.4.3. Controlling Output Data for an Electric Conduction Solution.....	455
4.9.4.4. Setting Nonlinear Controls in an Electric Conduction Simulation.....	456
4.9.5. Troubleshooting Structural, Thermal, and Electric Conduction Simulations.....	459
4.9.5.1. Solver Unable to Converge.....	459
4.9.5.2. Element Distortions.....	460
4.9.5.3. Large Deformation.....	460
4.9.5.4. Insufficient Disk Space.....	460
4.9.5.5. Error While Starting Solver.....	460
4.9.5.6. Unable to Find Requested Modes.....	461

4.9.5.7. Internal Solution Magnitude Limit Was Exceeded.....	461
4.9.5.8. Large Number of Iterations Necessary for Convergence.....	461
4.9.5.9. Concerns with Active Large Deformation Effects.....	461
4.9.5.10. License Manager Server Is Down.....	461
4.9.6. User Commands for the MAPDL Solver.....	462
4.9.6.1. Specifying User Commands Options.....	462
4.9.6.2. User Commands Settings.....	463
4.9.7. Electromagnetic Solver Options.....	463
4.9.7.1. Launching the Solver on Multiple Processes.....	463
4.9.7.2. Controlling the Magnetics Solution.....	463
4.9.7.3. Specifying Skin Depth Resolution.....	464
4.9.7.4. Generating an Initial Mesh for Viewing on a Contour Plot.....	465
4.9.7.5. Judging Convergence for Electromagnetics Solutions.....	466
4.9.8. Electrostatic Solver Options.....	469
4.9.8.1. Configuring the Solver for an Electrostatic Solution.....	470
4.9.8.2. Setting Solver Launch Options for an Electrostatic Solution.....	471
4.9.8.3. Controlling Output Data for an Electrostatic Solution.....	472
4.9.8.4. Setting Nonlinear Controls in an Electrostatic Simulation.....	473
4.9.9. Topology Optimization Solver Options.....	476
4.9.9.1. Configuring the Solver for a Topology Optimization.....	476
4.9.10. Polymer Extrusion Solver Options.....	478
4.9.10.1. Specifying Launch Controls for the Polymer Solver.....	479
4.9.10.2. Setting Numerical Controls for Polymer Extrusion Flows.....	479
4.9.10.3. Setting Solution Controls for Polymer Extrusion Flows.....	483
4.9.10.4. Setting Output Controls for Polymer Extrusion Flows.....	488
4.9.11. Polymer Blow Molding Solver Options.....	489
4.9.11.1. Setting Solution Controls for Polymer Blow Molding Flows.....	489
4.9.11.2. Setting Output Controls for Polymer Blow Molding.....	491
4.9.12. Boundary Refinement.....	492
4.9.12.1. Specifying Boundary Refinement.....	492
4.9.13. Solution Fidelity Refinement.....	492
4.9.13.1. Specifying Solution Fidelity Refinement.....	493
4.9.13.2. Solution Fidelity Refinement Considerations.....	493
4.10. Simulation Steps.....	493
4.10.1. Defining Simulation Steps.....	493
4.10.2. Defining Steps using Status Groups.....	494
4.10.2.1. Creating Condition Groups.....	495
4.10.2.2. Defining Status Groups.....	495
4.10.2.3. Configuring Simulation Steps.....	496
4.11. Output Monitors.....	496
4.11.1. Creating a Monitor Chart.....	499
4.11.2. Viewing Monitor Charts.....	499
4.11.3. Exporting Solution Monitor Data.....	501
4.12. Creating Additional Physics Workflows.....	501

4.12.1. Time-Dependent Fluid Flow.....	502
4.12.2. Time-Dependent Thermal Workflow.....	502
4.12.3. Compressible Fluid Flows.....	503
4.12.3.1. Modeling Compressible Gas Flows.....	503
4.12.4. Particles in Fluid Flow.....	504
4.12.4.1. Particle Tracking Workflow.....	504
4.12.5. Pre-Stressed Modal Analysis.....	506
4.12.6. Random Vibration Analysis.....	506
4.12.6.1. Setting Up a Random Vibration Analysis.....	506
4.12.6.2. Random Vibration Results.....	507
4.12.7. Eigenvalue Buckling Analysis.....	508
4.12.7.1. Eigenvalue Buckling Workflow.....	509
4.12.8. Topology Optimization.....	509
4.12.8.1. Refining the Optimized Shape.....	510
4.12.8.2. Exporting the Optimized Geometry.....	510
4.13. Adjust a Simulation by Adding Physics (Guide Me).....	510
4.13.1. Adding a Fluid-Solid Heat Transfer (Guide Me).....	511
4.13.2. Adding a Structural Stress Computation (Guide Me).....	511
4.14. Suppressing a Structural Condition.....	512
<b>Chapter 5: Results.....</b>	<b>514</b>
5.1. Results Display Types.....	514
5.1.1. Contours.....	515
5.1.1.1. Specifying a Contour.....	516
5.1.1.2. Adding a New Contour.....	520
5.1.1.3. Contact Results.....	521
5.1.2. Vectors.....	521
5.1.2.1. Specifying a Vector.....	522
5.1.2.2. Adding a New Vector.....	524
5.1.2.3. Choosing a Vector Distribution Type.....	524
5.1.3. Isosurface.....	526
5.1.3.1. Specifying an Isosurface.....	527
5.1.3.2. Adding a New Isosurface.....	528
5.1.3.3. Choosing an Isovalue Specification.....	529
5.1.4. Streamlines.....	529
5.1.4.1. Specifying a Streamline.....	530
5.1.4.2. Adding a New Streamline.....	531
5.1.4.3. Choosing a Streamline Distribution Type.....	531
5.1.5. Particle Tracks.....	532
5.1.5.1. Specifying Particle Tracks.....	533
5.1.6. Line Chart.....	534
5.1.6.1. Specifying a Line Chart.....	534
5.1.6.2. Adding a New Line Chart.....	535
5.1.6.3. Viewing Line Charts.....	535
5.1.6.4. Exporting Line Chart Data.....	537

5.1.7. Shear Moment Diagrams.....	537
5.1.7.1. Specifying a Shear Moment Diagram.....	537
5.1.8. Linearized Stresses.....	538
5.1.8.1. Viewing Linearized Stresses as a Line Chart.....	538
5.1.8.2. Viewing Linearized Stresses as a Contour Result.....	538
5.2. Calculated Results.....	538
5.2.1. Calculated Value.....	539
5.2.1.1. Specifying a Calculated Value.....	539
5.2.1.2. Adding a New Calculated Value.....	540
5.2.1.3. Choosing a Calculated Value Method.....	541
5.2.1.4. Calculating a Pressure Drop.....	541
5.2.1.5. Adding a Calculated Value to a Monitor Chart.....	542
5.2.1.6. Examples of Calculated Value Monitors.....	542
5.2.2. Table Results.....	546
5.2.2.1. Matrix Table Results.....	547
5.2.2.2. Loss Data Table Results.....	548
5.2.2.3. Force Components Table Results.....	549
5.3. Exploring the Result Preview.....	551
5.3.1. Saving an Exploratory Contour Result as a High-fidelity Result Object.....	551
5.4. Exporting Results Data.....	552
5.5. Results Variables by Physics Type.....	552
5.5.1. Fluid Variables.....	552
5.5.2. Polymer Extrusion Variables.....	557
5.5.3. Polymer Blow Molding Variables.....	559
5.5.4. Structural Variables.....	561
5.5.5. Thermal Variables in Thermal/Structural Simulations.....	568
5.5.6. Electric Conduction Variables.....	569
5.5.7. Electrostatic Variables.....	570
5.5.8. Electromagnetic Variables.....	571
5.6. Graphics Setting and Display.....	575
5.6.1. Specifying a Variable Range.....	575
5.6.2. Specifying a Color Distribution.....	576
5.6.3. Specifying Deformation Scaling.....	576
5.6.4. Summary.....	577
5.6.4.1. Displaying Locations of Minimum and Maximum Values.....	578
5.7. Animation Results.....	578
5.7.1. Animating a Single Result.....	579
5.7.2. Animating Multiple Results.....	579
5.7.3. Animation Slider Behavior in Different Solution Types.....	579
<b>Chapter 6: Using Alternative Workflows.....</b>	<b>582</b>
6.1. Additional Methods for Creating Simulations.....	582
6.2. Efficient Modeling of Thin-Shell Structures.....	582
6.2.1. Workflow for Modeling Thin-Shell Structures.....	582
6.2.2. Defining the Thickness for Surface Bodies Representing Shells.....	583

6.2.2.1. Defining Thickness in Geometry Modeler.....	583
<b>6.2.2.2. Defining Thickness for Imported Geometry (No Model Editing)</b> .....	583
6.2.3. Creating Contacts for Surface Bodies Representing Shells.....	584
<b>6.3. Fatigue Analysis.....</b>	585
6.3.1. Fatigue Analysis using Stress Life.....	585
6.3.1.1. Computing Fatigue Results for Stress Life Using a Template.....	586
6.3.1.2. Analyzing Stress Life by Defining Materials and Properties.....	586
6.3.1.3. Analyzing Stress Life by Defining Stress Life Properties for All Materials....	588
6.3.2. Fatigue Analysis using Strain Life.....	589
6.3.2.1. Strain Life.....	589
6.3.2.2. Computing Fatigue Results for Strain Life Using a Template.....	592
6.3.2.3. Analyzing Strain Life by Defining Strain Material Properties.....	592
6.3.3. Fatigue Settings.....	593
<b>6.4. Modeling Trusses and Cables.....</b>	595
<b>6.5. Using Flagship Products with AIM-based Physics Definition .....</b>	595
6.5.1. Physics Transfer from AIM to Mechanical.....	595
6.5.2. Details for AIM to Mechanical Transfer.....	596
<b>Chapter 7: Parameters and Design Points.....</b>	<b>598</b>
7.1. Parameters.....	598
7.1.1. Defining Parameters.....	598
7.1.2. Viewing Parameters.....	600
7.2. Design Points.....	600
7.3. Creating Design Points Manually.....	601
7.4. Filtering Design Points.....	603
7.5. Modifying Design Points.....	607
7.6. Moving Parameters in the Dashboard.....	608
7.7. Design Point States.....	609
7.8. Design Point Updates.....	610
7.8.1. Updating Design Points.....	611
7.8.2. Monitoring Design Point Updates.....	612
7.9. Working with Design Point Update Data.....	612
7.9.1. Retaining Design Point Data within the Project.....	613
7.9.1.1. Marking Design Points to Retain Data.....	613
7.9.1.2. Marking Design Points to Not Retain Data.....	613
7.9.1.3. Retained Data Status.....	613
7.9.1.4. Setting a Design Point as Current.....	614
7.9.2. Exporting Design Points to New Projects.....	614
7.9.3. Exporting Design Point Parameter Values to a CSV File.....	615
7.9.3.1. Extended CSV File Format.....	615
7.10. Performing What-If Studies.....	616
7.10.1. Steps for Performing a What-If Study.....	616
7.10.2. Performing a Simple What-if Study.....	616
7.10.3. Performing a Large What-if Study.....	618
7.10.4. Preparing to Troubleshoot a Failing Design Point Update.....	620

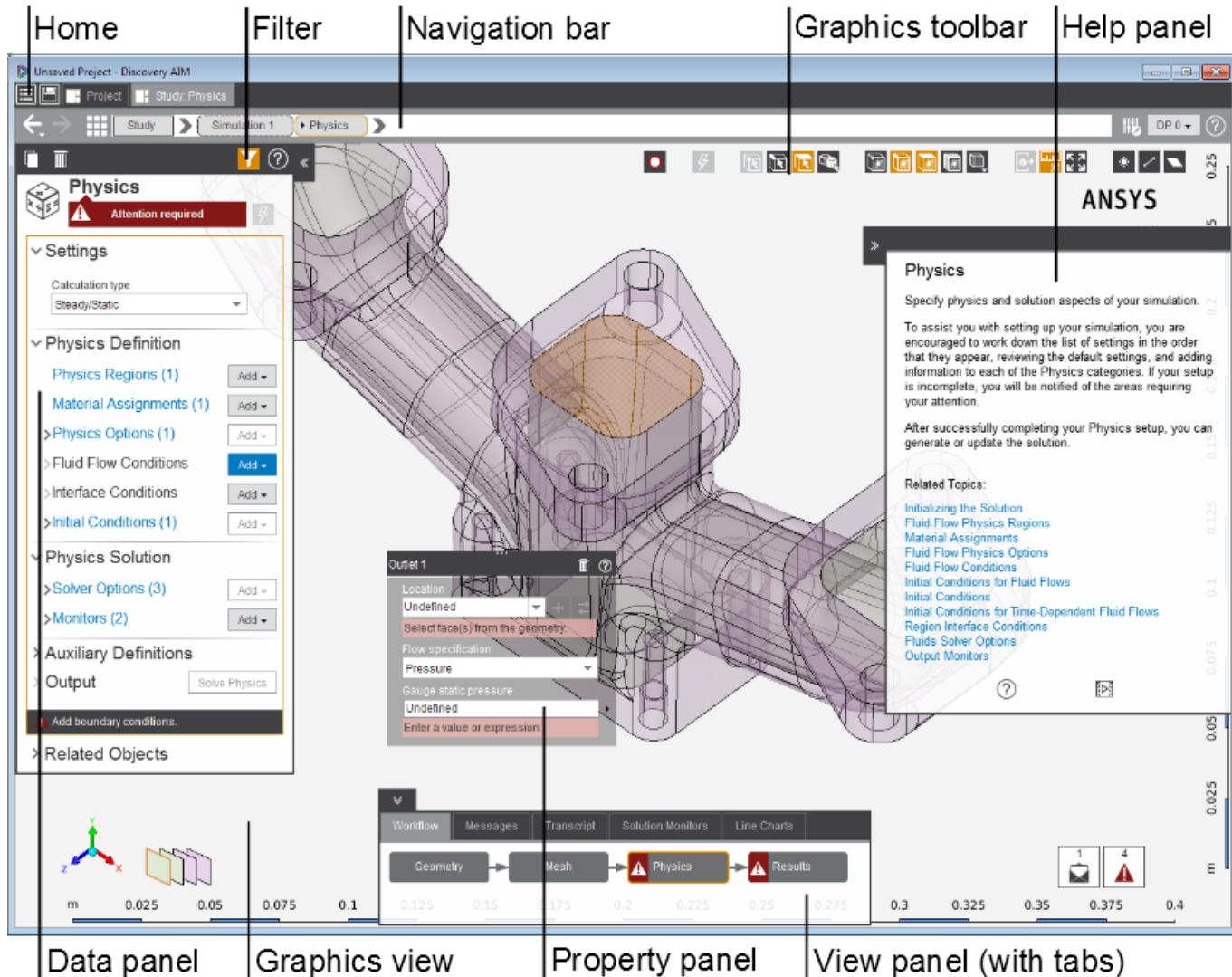
<b>Chapter 8: Expressions and Other Operations.....</b>	<b>622</b>
8.1. Expressions.....	622
8.1.1. Creating Expressions.....	622
8.1.2. Mathematical Constants.....	623
8.1.3. Variables.....	623
8.1.3.1. Variable Tensor Types.....	624
8.1.4. Units.....	625
8.1.4.1. Units in Expressions.....	627
8.1.4.2. Unit Multipliers.....	627
8.1.4.3. Unit Declaration Syntax.....	628
8.1.4.4. Temperature Unit Conversions.....	629
8.1.5. Single-Valued Expressions and Variables.....	629
8.1.6. Solution-Dependent Expressions.....	629
8.1.7. Location References.....	630
8.1.7.1. Specifying Locations in Expressions.....	630
8.1.7.2. Using Selection Sets in Expressions.....	631
8.1.7.3. About Location Functions.....	631
8.1.8. Expression Composition.....	637
8.1.9. Reusing Expressions and Values.....	638
8.1.9.1. Named Expressions and Values.....	639
8.1.9.2. Navigation to Named Expressions and Named Values.....	640
8.1.9.3. Creation of Named Expressions and Named Values.....	641
8.1.9.4. Editing a Named Expression Object.....	642
8.1.9.5. Copying a Named Expression or its Value.....	642
8.1.9.6. Duplication of Named Expressions.....	643
8.1.9.7. Deleting a Named Expression or Named Value.....	643
8.1.9.8. Named Expressions and Named Values in the Construction of New Expressions.....	644
8.1.10. Mathematical Operators and Functions.....	644
8.1.11. Quantity Functions.....	646
8.1.12. Object Retrieval Functions.....	648
8.1.13. Fluid Variables.....	649
8.1.14. Polymer Extrusion Variables.....	654
8.1.15. Polymer Blow Molding Variables.....	655
8.1.16. Structural Variables.....	657
8.1.17. Electric Conduction Variables.....	665
8.1.18. Electrostatic Variables.....	666
8.1.19. Thermal Variables in Thermal/Structural Simulations.....	666
8.1.20. Electromagnetics Variables.....	667
8.2. ANSYS Discovery AIM Journaling and Scripting Overview.....	671

# Chapter 1: Overview of Simulating Physics in ANSYS Discovery AIM

Numerical simulations of real-world multiphysics engineering problems are often comprised of distinct, but connected, steps that cover everything required for a complete investigation: from importing data, to defining physical models, to analyzing the results. A simulation is a collection of these connected steps that define a complete, end-to-end physics simulation.

ANSYS Discovery AIM (sometimes "AIM") provides a convenient workspace called a "study" for creating and defining one or more simulations. Many of the steps in a simulation correspond to a "task" in AIM. Each task is a container for everything involved within the step. Once a task is completed and **up-to-date**, the task generates data that is transferred to the next task in the workflow.

The following figure shows the ANSYS Discovery AIM workspace with an example of a simulation with four tasks:



The following table illustrates the general steps in a simulation and the corresponding task names in an AIM study:

General Steps for a Simulation	Tasks in an AIM Study
1. Importing or creating the geometry.	The <a href="#">Geometry Modeling</a> task (displayed as <b>Geometry</b> by default) is where you create or edit a geometry. Alternatively, the <a href="#">Data Import</a> task (displayed as <b>Geometry</b> by default) is where you import geometric data. This can be used for a more direct import of CAD files.
2. Optionally, suppressing parts of a geometry or repositioning multiple geometries.	This is generally recommended to be achieved through the Geometry Modeling task and defining physics regions in the Physics task. However, if a Data Import task was used, the optional <a href="#">Model Configuration</a> task (displayed as <b>Configuration</b> by default) is where you suppress parts of your geometry or reposition multiple geometries.
3. For some external-flow simulations, defining an external region in which you want to monitor the effects on the geometry.	This is generally recommended to be achieved through the Geometry Modeling task and defining physics regions in the Physics task. However, if a Data Import task was used, the optional <a href="#">Model Configuration</a> task (displayed as <b>Configuration</b> by default) is where you can define a box to represent an external region. You can also define such a region in your geometry-creation tool.
4. For some fluid-flow simulations, extracting a flow volume from a geometry.	This is generally recommended to be achieved through the Geometry Modeling task and defining physics regions in the Physics task. However, if a Data Import task was used and you have to extract a flow volume from a geometry in AIM, you do this in the <a href="#">Volume Creation</a> task (displayed as <b>Volu...tion</b> by default).
5. Creating a mesh.	The computational mesh is determined automatically based on the physics inputs and solution fidelity selection. If you decide to define the mesh manually, the <a href="#">Meshing</a> task (displayed as <b>Mesh</b> by default) is where you define mesh controls.
6. Defining the physical model, applying physics conditions, and generating a solution.	The <a href="#">Physics Solution</a> task (displayed as <b>Physics</b> by default) is where you define such things as the material properties, physics conditions, and solution settings.
7. Solving the physics.	When you <a href="#">update</a> an out-of-date Physics task, you start the solving process. You may <a href="#">monitor the progress</a> on the <b>Solution Monitor</b> view.
8. For a coupled simulation, transferring the physics data of the first process to the second.	For example, in a <a href="#">Fluid-Structure Interaction</a> simulation, transfer the <b>Flow</b> physics to the <b>Structural</b> physics.
9. Examining and saving the results.	The <a href="#">Results Evaluation</a> task (displayed as <b>Results Evaluation</b> by default) is where you display the results of the physics solution.

Having viewed the results, you have several options. You can:

- Edit your simulation input to refine your results.
- [Parameterize](#) some characteristics of the solution process and explore design alternatives by using [design points](#).

- Produce a report.
- Make the simulation available to others by [creating an archive](#).
- Use ANSYS ACT to [automate similar simulations](#).

## Planning a Simulation

Before you begin working in AIM, carefully consider your goals, such as obtaining a rough initial solution, obtaining a refined solution, or exploring design alternatives. Also think about providing [auxiliary definitions](#) for your simulation, such as defining selection sets or reference frames to make setup easier.

In addition, you can also learn [how AIM works within Workbench](#) and [how to use ANSYS ACT to create customized workflows](#) for process compression and automation.

## Creating a Simulation

Get started by using a [template app](#) for a common engineering physics simulation, such as structural or fluid flow simulations. When you use a template app, AIM will create tasks and associated objects for you, based on the initial information you provide.

As you work through the simulation, refer to the common tools and operations used throughout AIM, including [setting locations](#), and providing auxiliary definitions for your simulation, such as [setting selection sets](#), [defining reference frames](#), or [creating construction geometry](#).

The following videos also provide an introduction to using AIM:



## Running a Guided Simulation

When you use a guided simulation, AIM will walk you through the entire process of setting up, solving, and viewing results. You will not see the tasks and objects used for the simulation. Guided simulations have a special icon, so you can recognize them on the start page.

Guided simulation apps can be downloaded from the [ANSYS Store](#).

---

### 1.1. The ANSYS Product Improvement Program

This product is covered by the ANSYS Product Improvement Program, which enables ANSYS, Inc., to collect and analyze *anonymous* usage data reported by our software without affecting your work or product performance. Analyzing product usage data helps us to understand customer usage trends and patterns, interests, and quality or performance issues. The data enable us to develop or enhance product features that better address your needs.

#### **How to Participate**

The program is voluntary. To participate, select **Yes** when the Product Improvement Program dialog appears. Only then will collection of data for this product begin.

#### **How the Program Works**

After you agree to participate, the product collects anonymous usage data during each session. When you end the session, the collected data is sent to a secure server accessible only to authorized ANSYS employees.

After ANSYS receives the data, various statistical measures such as distributions, counts, means, medians, modes, etc., are used to understand and analyze the data.

## Data We Collect

The data we collect under the ANSYS Product Improvement Program are limited. The types and amounts of collected data vary from product to product. Typically, the data fall into the categories listed here:

*Hardware:* Information about the hardware on which the product is running, such as the:

- brand and type of CPU
- number of processors available
- amount of memory available
- brand and type of graphics card

*System:* Configuration information about the system the product is running on, such as the:

- operating system and version
- country code
- time zone
- language used
- values of environment variables used by the product

*Session:* Characteristics of the session, such as the:

- interactive or batch setting
- time duration
- total CPU time used
- product license and license settings being used
- product version and build identifiers
- command line options used
- number of processors used
- amount of memory used
- errors and warnings issued

*Session Actions:* Counts of certain user actions during a session, such as the number of:

- project saves
- restarts
- meshing, solving, postprocessing, etc., actions
- times the Help system is used
- times wizards are used
- toolbar selections

*Model:* Statistics of the model used in the simulation, such as the:

- number and types of entities used, such as nodes, elements, cells, surfaces, primitives, etc.
- number of material types, loading types, boundary conditions, species, etc.
- number and types of coordinate systems used
- system of units used
- dimensionality (1-D, 2-D, 3-D)

*Analysis:* Characteristics of the analysis, such as the:

- physics types used
- linear and nonlinear behaviors
- time and frequency domains (static, steady-state, transient, modal, harmonic, etc.)
- analysis options used

**Solution:** Characteristics of the solution performed, including:

- the choice of solvers and solver options
- the solution controls used, such as convergence criteria, precision settings, and tuning options
- solver statistics such as the number of equations, number of load steps, number of design points, etc.

**Specialty:** Special options or features used, such as:

- user-provided plug-ins and routines
- coupling of analyses with other ANSYS products

## Data We Do Not Collect

The Product Improvement Program does *not* collect any information that can identify you personally, your company, or your intellectual property. This includes, but is not limited to:

- names, addresses, or usernames
- file names, part names, or other user-supplied labels
- geometry- or design-specific inputs, such as coordinate values or locations, thicknesses, or other dimensional values
- actual values of material properties, loadings, or any other real-valued user-supplied data

In addition to collecting only anonymous data, we make no record of where we collect data from. We therefore cannot associate collected data with any specific customer, company, or location.

## Opting Out of the Program

You may stop your participation in the program any time you wish. To do so, select **ANSYS Product Improvement Program** from the Help menu. A dialog appears and asks if you want to continue participating in the program. Select **No** and then click **OK**. Data will no longer be collected or sent.

## The ANSYS, Inc., Privacy Policy

All ANSYS products are covered by the ANSYS, Inc., [Privacy Policy](#).

## Frequently Asked Questions

- *Am I required to participate in this program?*

No, your participation is voluntary. We encourage you to participate, however, as it helps us create products that will better meet your future needs.

- *Am I automatically enrolled in this program?*

No. You are not enrolled unless you explicitly agree to participate.

- *Does participating in this program put my intellectual property at risk of being collected or discovered by ANSYS?*

No. We do not collect any project-specific, company-specific, or model-specific information.

- *Can I stop participating even after I agree to participate?*

Yes, you can stop participating at any time. To do so, select **ANSYS Product Improvement Program** from the Help menu. A dialog appears and asks if you want to continue participating in the program. Select **No** and then click **OK**. Data will no longer be collected or sent.

- *Will participation in the program slow the performance of the product?*

No, the data collection does not affect the product performance in any significant way. The amount of data collected is very small.

- *How frequently is data collected and sent to ANSYS servers?*

The data is collected during each use session of the product. The collected data is sent to a secure server once per session, when you exit the product.

- *Is this program available in all ANSYS products?*

Not at this time, although we are adding it to more of our products at each release. The program is available in a product only if this *ANSYS Product Improvement Program* description appears in the product documentation, as it does here for this product.

- *If I enroll in the program for this product, am I automatically enrolled in the program for the other ANSYS products I use on the same machine?*

Yes. Your enrollment choice applies to all ANSYS products you use on the same machine. Similarly, if you end your enrollment in the program for one product, you end your enrollment for all ANSYS products on that machine.

- *How is enrollment in the Product Improvement Program determined if I use ANSYS products in a cluster?*

In a cluster configuration, the Product Improvement Program enrollment is determined by the host machine setting.

---

## 1.2. ANSYS Subscription Licensing

If you are using an ANSYS subscription to run ANSYS Discovery AIM, you can click **Manage subscription**  from within AIM to sign in to your ANSYS account.

Use **Manage subscription**  to:

- See when your subscription expires
  - Access and manage your ANSYS account
  - Release the subscription activation on the current device
  - Access the [ANSYS Discovery Forum](#), where you can find general information, tutorials, and support
- 

## 1.3. Creating Simulations Using Template Apps

Simulation templates are accessible when you start ANSYS Discovery AIM or add a new simulation on the **Study** panel. Each template defines a simulation with typical default settings so that you can configure your simulation quickly.

The installed templates are:

- [Structural](#), for a structural simulation that optionally solves thermal and electric conductance physics.
- [Fluid Flow](#), for an internal or external fluid-flow simulation that optionally solves thermal physics.
- [Thermal](#), for a thermal simulation that optionally solves structural, fluid flow, or electric conductance physics.
- [Electromagnetics](#), for an electromagnetic simulation, including electric conductance simulations that solve structural, fluid flow, or thermal physics.
- [Topology Optimization](#), to optimize the topology by automatically removing excess material.
- [Fluid-Structure Interaction](#), for a simulation that transfers fluid forces to a structure, without heat transfer.
- [Fluid-Solid Heat Transfer](#), for a fluid-solid simulation with heat transfer.
- [Polymer Extrusion](#), for a polymer extrusion simulation that optionally solves extrudate and die deformation physics.
- [Polymer Blow Molding](#), for a polymer blow molding simulation that models the shape of hollow parts from a cylindrical parison or a flat molten plastic sheet.
- [Geometry Modeling](#), for a geometry modeling simulation that contains only the geometric model for use with other physics templates.

- [Connect to Discovery Live Template](#) on page 64, to create a connection between AIM and Discovery Live, so that you can transfer data between them.
  - [Connect to Mechanical](#), if licensed, to connect to ANSYS Mechanical for additional structural physics simulation capabilities.
  - [Connect to Fluent](#), if licensed, to connect to ANSYS Fluent for additional fluid physics simulation capabilities.
- You can also create your own [customized workflows](#) and add more templates by visiting the [ANSYS Store](#).

### 1.3.1. Structural Template

#### Start > Structural

The **Structural** template app creates a structural simulation that can be used to simulate and evaluate structural displacements, stresses, strains, forces, and fatigue.

To create a structural simulation:

1. From the **Start** page, click **Structural** and click **Start**.
2. Specify model settings.
  - a) Specify the model source and editing options:
    - To create a new geometry, select **Define new geometry**.
    - To import a geometry and have the ability to edit it, select **Import geometry file** and enable **Allow geometry modeling** under **Model options**.
    - To import a geometry from an external CAD session, select **Connect to active CAD session**.
    - To [define the mesh manually](#) instead of using [automatic physics-aware meshing](#), select **Define mesh manually**.
    - If a task already exists that includes the desired geometry, select **Use existing task** and make the appropriate selection for **Task containing geometry**.
  - b) Specify other modeling options:
    - To [automatically generate contacts](#) between solid bodies, select **Detect contact automatically**.
    - Optionally, enable the suppression, movement, or rotation of parts of the geometry by selecting **Allow configuration**.
  - c) Click **Next**. If you have chosen to import the geometry, you are prompted to browse and select a geometry file.
3. Specify physics settings.
  - a) Specify the additional physics that you want to evaluate. To model situations where:
    - The temperature field introduces thermal strains in the structural field, which, in turn result in thermal expansion and stresses, select **Thermal**.
    - There are electric conduction loads on a system or component, select **Electric conduction**.
  - b) Specify the calculation type:
    - To model steady loads, select **Static**.
    - To model vibration characteristics, such as natural frequencies and mode shapes, select **Modal**.
    - To predict the buckling strength of your model, select **Eigenvalue buckling**.
  - c) Specify other physics options:
    - To automatically calculate [fatigue results](#) for your static structural analysis based on the S-n curve values defined for the default material, select **Compute fatigue results**.
    - To add [pre-stress effects to your modal analysis](#), select **Enable pre-stress**.
  - d) To add [random vibration analysis](#), select **Include random vibration**

4. When you have made your selections, click **Finish** to get started. Each of the tasks in the created simulation will have settings specific to the choices you make here in the template.

- If you selected **Define new geometry** from the Template panel, the Model Editor window opens where you can create your geometry.

For details on using the Model Editor, see [Model Editing](#) on page 92.

- If you have imported a geometry that you want to edit:

- a. Click the Geometry link in the **Simulation** panel.
- b. Click **Edit Geometry** in the **Geometry** panel.

- If you have chosen to allow configuration:

For details on using the Configuration task to suppress parts, see [Creating a Suppress Control](#).

For details on using the Configuration task to move and rotate parts, see [Creating a Move/Rotate Control](#).



[Static Structural Physics Overview](#)



[Modal Structural Physics Overview](#)

### 1.3.2. Fluid Flow Template

#### Start > Fluid Flow

The **Fluid Flow** template app creates a fluid flow simulation that can be used to simulate fluid motion and, optionally, heat transfer in and around structures or components.

On each panel of the template, review your options and make any necessary changes. The panels displayed depend on the options chosen.

1. Review the [model options](#) and make any necessary changes.
2. Specify the [physics to be simulated](#).
3. [Define the fluid region\(s\)](#) for your fluid flow simulation.
4. If your setup includes physics coupling options, [define the solid region\(s\)](#) for your simulation.
5. [Review your simulation inputs](#) and then click **Finish** to get started. Each of the tasks in the created simulation will have settings specific to the choices you make here in the template.



[Fluid Flow Physics Overview](#)

### 1.3.2.1. Fluid Flow: Model

In this step of the template, review the model options and make any necessary changes.

1. Specify how the geometry is brought into the simulation.

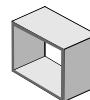
- To create a new geometry, select **Define new geometry**.
- To be able to edit a geometry, select **Import geometry file** and select the **Allow geometry modeling option**.
- To import a geometry from an external CAD session, select **Connect to active CAD session**.
- To import a geometry without the ability to edit it, select only **Import geometry file**.

- If a task already exists that includes the desired geometry, select **Use existing task** and make the appropriate selection for **Task containing geometry**.
2. To [define the mesh manually](#), rather than use [automatic physics-aware meshing](#), select **Define mesh manually**.
  3. If you are connecting to an active CAD session or you disabled **Allow geometry modeling**, you must specify how you would like the geometry to be treated:

**Represents the flow volume(s)**



**Requires the flow volume(s) to be created** (if geometry modeling is enabled, you must create the flow volume(s) in the Geometry Modeling task)



4. To enable the suppression, movement, or rotation of parts of the geometry, or to create an external flow (when a fluid region must be created around a solid body), select **Allow configuration**.
5. Click **Next**. If you have chosen to import the geometry, you are prompted to browse and select a geometry file.

You can then set the [physics options](#).

### 1.3.2.2. Fluid Flow: Physics

Specify the physics to be simulated.

1. To simulate turbulent flows with a significant amount of swirl, select **Swirling flow**.
2. Specify physics coupling options for the simulation.
  - When you want to model fluid motion and heat transfer effects in and around structures or components, select **Heat transfer between a fluid and a solid**.
  - When you want to transfer temperature from the solid body into the same body in a separate structural simulation, select **Structural stress due to solid thermal effects**. Thermal stress can then be calculated in the structural simulation.
  - When you want to transfer force from surfaces in the fluid onto surfaces on the solid body in a separate structural simulation, select **Structural stress due to fluid forces**.
3. Define your simulation behavior:
  - To model behavior that does not change over time, select **Steady-state fluid flow**.
  - To model time-varying behavior, select **Time-dependent fluid flow** and then specify the **Expected duration**.
4. Click **Next** to define the physics regions: [fluid regions](#) and/or [solid regions](#).

### 1.3.2.3. Fluid Physics Region

Define the fluid region(s) for your fluid flow simulation.

1. Specify the number of fluid regions.

Multiple fluid regions are required if you have different flow paths (for example, different fluid circuits in a heat exchanger). You can define up to five regions here; define any additional required regions in the Physics task.

2. Select the bodies in your geometry forming the fluid region or you can create the body representing the fluid flow region later, using the **Geometry Modeling** task to extract the volume. For fluid flow within a single fluid region, you can use all unassigned bodies to represent the fluid region.
3. Select a material for the fluid region.  
If your material is not listed, select **Other**. You will create or select the material later on the Material Assignments page.
4. Include any additional flow physics you would like to simulate within the region.
  - To include temperature variations within the fluid, select **Heat transfer**.
  - If any of the flow will experience [variations in the gas density](#), select **Compressible flow (ideal gas)**. If no selection is made, then incompressible flow will be modeled, where the effects of density variation in the flow field will be ignored.
  - To model discrete particles in a fluid, such as gas bubbles in a liquid, select **Particles**.
5. Click **Next**. You will either be prompted to [define your solid regions](#) or view a summary of your inputs, depending on the physics chosen.

### 1.3.2.4. Solid Physics Region

Define the solid region(s) for your fluid flow simulation.

1. Specify the number of solid regions.  
You will need separate solid regions if different bodies have different physics settings or initial conditions. You can define up to five regions here; define any additional required regions in the Physics task.
2. Specify the number of materials to assign within a solid region.  
A solid region may consist of several bodies and materials may be assigned to one or more bodies.
3. Select the location for the solid regions.  
If the solid region is made up of a single material you can select the bodies forming the solid region or use all bodies that were not assigned to a fluid region to represent the solid region. For multiple materials, select the bodies and the corresponding materials. All bodies selected will be part of this region.
4. Select the material.  
If your material is not listed, select **Other**. You will create or select the material later on the Material Assignments page.
5. After defining the materials for all solid regions, click **Next** to view a summary of your inputs.

### 1.3.2.5. Fluid Summary

This panel summarizes your inputs. Click **Finish** to apply your template settings or **Back** to change a setting.

- If you selected **Define new geometry** from the Template panel, the Model Editor window opens where you can create your geometry.

For details on using the Model Editor, see [Model Editing](#) on page 92.

- If you have imported a geometry that you want to edit:
  1. Click the Geometry link in the **Simulation** panel.
  2. Click **Edit Geometry** in the Geometry panel.
- If you have chosen to allow configuration:

For details on using the Configuration task to suppress parts, see [Creating a Suppress Control](#).

For details on using the Configuration task to move and rotate parts, see [Creating a Move/Rotate Control](#).

- For details on volume creation, see [Volume Creation](#).

### 1.3.3. Thermal Template

#### Start > Thermal

The Thermal template enables you to model steady state or time-dependent thermal effects in a solid and/or a fluid. You can simulate the effects of various applied thermal conditions. For a fluid, you can simulate inlet temperature variations and include buoyancy for thermal recirculation if required.

On each panel of the template, review your options and make any necessary changes. The panels displayed depend on the options chosen.

1. Review the [model options](#) and make any necessary changes.
2. Specify the [physics to be simulated](#).
3. If you chose to model thermal effects within fluids, [define the fluid region\(s\)](#) for your thermal simulation.
4. If you chose to model thermal effects within solids, [define the solid region\(s\)](#) for your thermal simulation.
5. [Review your simulation inputs](#) and then click **Finish** to get started. Each of the tasks in the created simulation will have settings specific to the choices you make here in the template.

---

 [Steady-State Thermal Physics Overview](#)

---

 [Steady-State Electric Conduction Thermal Physics Overview](#)

---

#### 1.3.3.1. Thermal: Model

In this step of the template, review the model options and make any necessary changes.

1. First, specify how the geometry is brought into the simulation.
  - To create a new geometry, select **Define new geometry**.
  - To be able to edit a geometry, select **Import geometry file** and select the **Allow geometry modeling option**.
  - To import a geometry from an external CAD session, select **Connect to active CAD session**.
  - To import a geometry without the ability to edit it, select only **Import geometry file**.
  - If a task already exists that includes the desired geometry, select **Use existing task** and make the appropriate selection for **Task containing geometry**.
2. To define the mesh manually, rather than use automatic physics-aware meshing, select **Define mesh manually**.
3. To enable the suppression, movement, or rotation of parts of the geometry, select **Allow configuration**.
4. Click **Next**. If you have chosen to import the geometry, you are prompted to browse and select a geometry file.

You can then set the [Physics options](#).

#### 1.3.3.2. Thermal: Physics

On this panel, you specify the physics to be simulated.

1. For **Heat transfer to be simulated**, select if you want to model thermal effects within solid bodies, within fluids, or within and between both solids and fluids (for conjugate heat transfer).
2. Specify any additional physics to simulate.

- If there is joule heating for resistive materials and contact electric conductance as well as Seebeck, Peltier, and Thomson effects for thermoelectricity, select **Solid heating via DC-conduction-induced thermal losses**.

**Note:** This option is only applicable when simulating heat transfer within solid bodies only and is only available for steady-state heat transfer.

- If you wish to include the calculation of stress and deformations due to the heat distribution within the solid bodies, select **Structural stress due to solid thermal effects**.

**Note:** In the case of time-varying heat transfer simulation, this option can only be used in conjunction with flow between a solid and a fluid and the stress will be computed based on the temperature distribution in the solid at the final time-point.

- If you wish to include any stresses or deformations due to pressure forces from the fluid being applied to the solid bodies, select **Structural stress due to fluid effects**.

**Note:** In the case of time-varying heat-transfer simulation, the stress will be computed due to applied pressures at the final time-point.

- If you want fatigue results for your static structural analysis based on the S-N curve values defined for the default material, select **Compute fatigue results**.

### 3. Define your simulation behavior:

- To model behavior that does not change over time, select **Steady-state heat transfer**.
- To model time-varying behavior, select **Time-dependent heat transfer** and then specify the **Expected duration**.

**Note:** This option is not compatible with some of the additional physics options. For compatible physics, stress will only be calculated based on the final time point.

### 4. Click **Next** to define the physics regions: [fluid regions](#) and/or [solid regions](#).

## 1.3.3.3. Fluid Physics Region

Define the fluid region(s) for your thermal simulation.

### 1. Specify the number of fluid regions.

Multiple fluid regions are required if you have different flow paths (for example, different fluid circuits in a heat exchanger). You can define up to five regions here; define any additional required regions in the Physics task.

### 2. Select the bodies in your geometry forming the fluid region or you can create the body representing the fluid flow region later, using the **Geometry Modeling** task to extract the volume.

### 3. Select a material for the fluid region.

If your material is not listed, select **Other**. You will create or select the material later on the Material Assignments page.

### 4. Include any additional flow physics you would like to simulate within the region.

- To include temperature variations within the fluid, select **Heat transfer**.
- If any of the flow will experience [variations in the gas density](#), select **Compressible flow (ideal gas)**. If no selection is made, then incompressible flow will be modeled, where the effects of density variation in the flow field will be ignored.
- To model discrete particles in a fluid, such as gas bubbles in a liquid, select **Particles**.

### 5. Click **Next**. You will either be prompted to [define your solid regions](#) or view a summary of your inputs, depending on the physics chosen.

## 1.3.3.4. Solid Physics Region

Define the solid region(s) for your thermal simulation.

1. Specify the number of solid regions.

You will need separate solid regions if different bodies have different physics settings or initial conditions.

You can define up to five regions here; define any additional required regions in the Physics task.

2. Specify the number of materials to assign within a solid region.

A solid region may consist of several bodies and materials may be assigned to one or more bodies.

3. Select the location for the solid regions.

If the solid region is made up of a single material you can select the bodies forming the solid region or use all bodies that were not assigned to a fluid region to represent the solid region. For multiple materials, select the bodies and the corresponding materials. All bodies selected will be part of this region.

4. Select the material.

If your material is not listed, select **Other**. You will create or select the material later on the Material Assignments page.

5. After defining the materials for all solid regions, click **Next** to view a summary of your inputs.

## 1.3.3.5. Thermal Summary

This panel summarizes your inputs. Click **Finish** to apply your template settings or **Back** to change a setting.

- If you selected **Define new geometry** from the Template panel, the Model Editor window opens where you can create your geometry.

For details on using the Model Editor, see [Model Editing](#) on page 92.

- If you have imported a geometry that you want to edit:

1. Click the Geometry link in the **Simulation** panel.
2. Click **Edit Geometry** in the Geometry panel.

- If you have chosen to allow configuration:

For details on using the Configuration task to suppress parts, see [Creating a Suppress Control](#).

For details on using the Configuration task to move and rotate parts, see [Creating a Move/Rotate Control](#).

## 1.3.4. Electromagnetics Template

### Start > Magnetics

The **Electromagnetics** template app creates an electric conduction or electrostatic simulation, or an electromagnetics simulation with AC or DC current sources, and/or permanent magnetic sources.

Select a **Electromagnetic** physics type with a source of Permanent magnets to simulate static magnetic fields produced by permanent magnets. You can also apply an AC current and specify the current frequency for designs in which sinusoidal time-varying currents flowing in conductors produce a time-varying magnetic field that induces Eddy currents. Or apply a DC current with magnetic effects to simulate a magnetostatic design or problem.

With the **Pure electric** physics type, you can apply a charge or voltage to simulate an electrostatic charge, or apply a DC current to determine the electric potential in a conducting body created by the external application of voltage or current loads.

You can then specify the impact of thermal and electromagnetic effects.

To create an electromagnetics or electric simulation:

1. From the **Start** page, click **Magnetics** and click **Start**.
  2. [Review the model options](#) and make any necessary changes.
  3. [Specify the physics to be simulated](#) and, if desired, the impact of thermal and electromagnetic effects.
  4. Define the surrounding region, for an [electromagnetic](#) or [electrostatic](#) simulation.
  5. [Define the solid region\(s\)](#) for your electric or electromagnetic simulation.
  6. [Review your selections](#), and then click **Finish** to get started. Each of the tasks in the created simulation will have settings specific to the choices you make here in the template.
    - If you selected **Define new geometry** from the Template panel, the Model Editor window opens where you can create your geometry.
- For details on using the Model Editor, see [Model Editing](#) on page 92.
- If you have imported a geometry that you want to edit:
    - a. Click the Geometry link in the **Simulation** panel.
    - b. Click **Edit Geometry** in the Geometry panel.
  - For details on physics, see [Physics Solution](#).
  - For details on physics coupling, see [Physics Coupling Interface](#).
  - For details on using the Configuration task to suppress parts, see [Creating a Suppress Control](#).
  - For details on using the Configuration task to move and rotate parts, see [Creating a Move/Rotate Control](#).



#### [Electromagnetics Physics Overview](#)

### 1.3.4.1. Electromagnetics: Model

In this step of the template, review the model options and make any necessary changes.

1. First, specify how the geometry is brought into the simulation.
  - To create a new geometry, select **Define new geometry**.
  - To be able to edit a geometry, select **Import geometry file** and select the **Allow geometry modeling option**.
  - To import a geometry from an external CAD session, select **Connect to active CAD session**.
  - To import a geometry without the ability to edit it, select only **Import geometry file**.
  - If a task already exists that includes the desired geometry, select **Use existing task** and make the appropriate selection for **Task containing geometry**.
2. To define the mesh manually, rather than use automatic physics-aware meshing, select **Define mesh manually**.
3. To enable the suppression, movement, or rotation of parts of the geometry, select **Allow configuration**.
4. Click **Next**. If you have chosen to import the geometry, you are prompted to browse and select a geometry file.

You can then set the Physics options.

### 1.3.4.2. Electromagnetics: Physics

**Start > Magnetics**

On the **Electromagnetics: Physics** panel of the Electromagnetics Template app, you specify the physics to be simulated. You can then specify the impact of thermal and electromagnetic effects.

1. Select your electromagnetic physics type.

Select an **Electromagnetic** physics type with a source of Permanent magnets to simulate static magnetic fields produced by permanent magnets. You can also apply an AC current and specify the current frequency for designs in which sinusoidal time-varying currents flowing in conductors produce a time-varying magnetic field that induces Eddy currents. Or apply a DC current with magnetic field to compute the current density and magnetic field.

With the **Pure electric** physics type, you can ignore magnetic field effects and simulate an electric field effect. Apply a charge or voltage to simulate an electrostatic field, or apply a DC current to determine the electric potential in a conducting body created by the external application of voltage or current loads.

2. Define your source.

For a Magnetic physics type:

- To simulate static magnetic fields produced by permanent magnets, select **Permanent magnets**.
- For designs in which sinusoidal time-varying currents flowing in conductors produce a time-varying magnetic field that induces Eddy currents, select **Applied current** and an **Applied current type** of **AC**, and then specify the **Current frequency**.
- To determine the current density in a conducting body created by the external application of voltage or current loads, select **Applied current** and then an **Applied current type** of **DC**.

For an Electric physics type, choose whether you are applying a charge or voltage to simulate an electrostatic field, or applying a current or voltage to simulate DC electric conduction.

3. Specify the **Electromagnetic-thermal behavior**. When adding thermal effects, a physics coupling is produced. Select one of the following:

- To maintain a constant temperature, select **Constant temperature**.
- To send heat rate data from the solid body in your electromagnetic simulation into one or more bodies in the thermal simulation, select **Electromagnetic effects influence temperature distribution**.
- If you want the solid body in your electromagnetic simulation to receive temperature data from one or more solid bodies in the thermal simulation, select **Applied thermal conditions influence electromagnetic effects**.
- For a bi-directional thermal-DC conduction simulation, select **Full interaction of thermal and DC current effects**.

4. For the Electromagnetic physics type, select any additional computations you want to run.

- For an applied direct current with magnetic effects, select **Compute inductance** to include the calculation of inductance in the simulation. Selecting this option will increase solution time.
- For an applied alternating current based electromagnetic source, select **Compute impedance** to include the calculation of impedance in the simulation. Selecting this option will increase solution time.
- To include the calculation of force in the simulation, select **Compute force**.
- To include the calculation of torque in the simulation, select **Compute torque**.
- To include the calculation of stress in the simulation, select **Compute stress**.

5. Click **Next**.

6. For a simulation that includes magnetic effects, specify how the surrounding region is defined and then select the surround material.

- a) From **Extent of the electromagnetic computation**, select an option:

- To create the surround automatically, select **Create surround automatically**.

- To create the surround manually, select **Create surround manually**. You will then need to edit the geometry to define a body for the surround, ensure the body is included in the physics region, and assign a material to it.
- To define the surround by geometry, select **Surround defined by geometry** and then **Select the surround bodies**.

b) Select the **Material for surround**. You can select **Air** or **Vacuum**, or **Select another material**.

7. Select **Next**.

8. Specify material assignments for solid regions:

a) Select the **Number of materials to assign within solid region**.

If assigning only one material:

- Select the **Location specification for solid region**, to **Use all unassigned bodies** or **Select bodies forming the solid region**.
- Select the **Material for the solid region**.

If assigning more than one material:

- Select the **Bodies to assign for Material 1**.
- Select the material for **Material 1**.
- Repeat for subsequent materials.

9. When you have made your selections, click **Finish** to get started. Each of the tasks in the created simulation will have settings specific to the choices you make here in the template.

- If you selected **Define new geometry** from the Template panel, the Model Editor window opens where you can create your geometry.

For details on using the Model Editor, see [Model Editing](#) on page 92.

- If you have imported a geometry that you want to edit:

- a. Click the Geometry link in the **Simulation** panel.
- b. Click **Edit Geometry** in the Geometry panel.

- For details on physics, see [Physics Solution](#).
- For details on physics coupling, see [Physics Coupling Interface](#).
- For details on using the Configuration task to suppress parts, see [Creating a Suppress Control](#).
- For details on using the Configuration task to move and rotate parts, see [Creating a Move/Rotate Control](#).



### [Electromagnetics Physics Overview](#)

## 1.3.4.3. Surrounding Region (Electromagnetic)

For a simulation that includes magnetic effects, you must define the surrounding region. By default, AIM creates the surrounding region automatically. You can also:

- Define the surrounding region manually. You will then need to edit the geometry to define a body for the surround, ensure the body is included in the physics region, and assign a material to it.
- Define the surround by selecting geometry.

You can then select the material to use for the surrounding region.

To define the surrounding region:

1. From **Extent of the electromagnetic computation**, select an option:

- To create the surround automatically, select **Create surround automatically**.
  - To create the surround manually, select **Create surround manually**.
  - To define the surround by geometry, select **Surround defined by geometry** and then **Select the surround bodies**.
2. Select the **Material for surround**. You can select **Air** or **Vacuum**, or **Select another material**.

### 1.3.4.4. Surrounding Region (Electrostatic)

For an electrostatic simulation, you may want to define the surrounding region. The surround is necessary if you want to know how the electric field and potential distribute in the air space and your geometry doesn't include a surround. For an electrostatic simulation, you may want to define the surrounding region, if one is not already a part of your geometry. The surround is necessary if you want to know how the electric field and potential distribute in the air space. By default, AIM assumes a surround is not required, but you can also:

- Direct AIM to create the surround automatically.
- Define the surrounding region manually. You will then need to edit the geometry to define a body for the surround, ensure the body is included in the physics region, and assign a material to it.
- Define the surround by selecting geometry.

You can then select the material to use for the surrounding region.

To define the surrounding region:

1. From **Extent of the electrostatic computation**, select an option:
  - To create the surround automatically, select **Create surround automatically**.
  - To create the surround manually, select **Create surround manually**.
  - To define the surround by geometry, select **Surround defined by geometry** and then **Select the surround bodies**.
2. Select the **Material for surround**. You can select **Air** or **Vacuum**, or **Specify another material**.

### 1.3.4.5. Solid Physics Region

Define the solid region(s) for your electric or electromagnetic simulation.

1. Specify the number of solid regions.  
You will need separate solid regions if different bodies have different physics settings or initial conditions. You can define up to five regions here; define any additional required regions in the Physics task.
2. Specify the number of materials to assign within a solid region.  
A solid region may consist of several bodies and materials may be assigned to one or more bodies.
3. Select the location for the solid regions.  
If the solid region is made up of a single material you can select the bodies forming the solid region or use all bodies that were not assigned to a fluid region to represent the solid region. For multiple materials, select the bodies and the corresponding materials. All bodies selected will be part of this region.
4. Select the material.  
If your material is not listed, select **Other**. You will create or select the material later on the Material Assignments page.
5. After defining the materials for all solid regions, click **Next** to view a summary of your inputs.

### 1.3.4.6. Electromagnetic Summary

This panel summarizes your inputs. Click **Finish** to apply your template settings or **Back** to change a setting.

- If you selected **Define new geometry** from the Template panel, the Model Editor window opens where you can create your geometry.

For details on using the Model Editor, see [Model Editing](#) on page 92.

- If you have imported a geometry that you want to edit:

1. Click the Geometry link in the **Simulation** panel.
2. Click **Edit Geometry** in the Geometry panel.

- If you have chosen to allow configuration:

For details on using the Configuration task to suppress parts, see [Creating a Suppress Control](#).

For details on using the Configuration task to move and rotate parts, see [Creating a Move/Rotate Control](#).

### 1.3.5. Topology Optimization Template

#### Start > Topology Optimization

The **Topology Optimization** template app automatically sets up a [topology optimization analysis](#), which is designed to optimize the topology by automatically removing excess material. Currently, you can either maximize the strength of your topology or minimize the response to free vibration.

To create a topology optimization simulation:

1. From the **Start** page, click **Topology Optimization** and click **Start**.

2. Specify model settings.

- a) Specify the model source and editing options:

- To create a new geometry, select **Define new geometry**.
- To import a geometry and have the ability to edit it, select **Import geometry file** and enable **Allow geometry modeling** under **Model options**.
- To import a geometry from an external CAD session, select **Connect to active CAD session**.
- To [define the mesh manually](#) instead of using [automatic physics-aware meshing](#), select **Define mesh manually**.
- If a task already exists that includes the desired geometry, select **Use existing task** and make the appropriate selection for **Task containing geometry**.

- b) Specify other modeling options:

- To [automatically generate contacts](#) between solid bodies, select **Detect contact automatically**.

- c) Click **Next**. If you have chosen to import the geometry, you are prompted to browse and select a geometry file.

3. When you have made your selections, click **Next**.

4. Set your design goals. By default, AIM optimizes your model to **Maximize strength**, or you can choose instead to **Minimize response to free vibration**.

5. Specify whether you want AIM to smooth your shape before export.

6. Click **Finish** to get started. Each of the tasks in the created simulation will have settings specific to the choices you make here in the template.

- If you selected **Define new geometry** from the Template panel, the Model Editor window opens where you can create your geometry.

For details on using the Model Editor, see [Model Editing](#) on page 92.

- If you have imported a geometry that you want to edit:

- a. Click the Geometry link in the **Simulation** panel.

- b. Click **Edit Geometry** in the **Geometry** panel.

### [Topology Optimization Overview](#)

## 1.3.6. Fluid-Structure Interaction Template

### **Start > Fluid-Structure Interaction**

The **Fluid-Structure Interaction** (FSI) template app creates a connected fluid flow and structural simulation. Fluid force data is transferred from the solved fluid simulation into the structural simulation using the [physics coupling interface](#).

On each panel of the template, review your options and make any necessary changes. The panels displayed depend on the options chosen.

1. Review the [model options](#) and make any necessary changes.
2. Specify the [physics to be simulated](#).
3. [Define the fluid region\(s\)](#) for your fluid-structure interaction simulation.
4. [Define the solid region\(s\)](#) for your fluid-structure interaction simulation.
5. [Review your simulation inputs](#) and then click **Finish** to get started. Each of the tasks in the created simulation will have settings specific to the choices you make here in the template.

### [Fluid-Structure Interaction Overview](#)

## 1.3.6.1. Fluid-Structure Interaction: Model

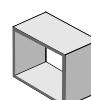
In this step of the template, review the model options and make any necessary changes.

1. Specify how the geometry is brought into the simulation.
  - To create a new geometry, select **Define new geometry**.
  - To be able to edit a geometry, select **Import geometry file** and select the **Allow geometry modeling option**.
  - To import a geometry from an external CAD session, select **Connect to active CAD session**.
  - To import a geometry without the ability to edit it, select only **Import geometry file**.
  - If a task already exists that includes the desired geometry, select **Use existing task** and make the appropriate selection for **Task containing geometry**.
2. To [define the mesh manually](#), rather than use [automatic physics-aware meshing](#), select **Define mesh manually**.
3. If you are connecting to an active CAD session or you disabled **Allow geometry modeling**, you must specify how you would like the geometry to be treated:

**Represents the flow volume(s)**



**Requires the flow volume(s) to be created** (if geometry modeling is enabled, you must create the flow volume(s) in the Geometry Modeling task)



4. Choose whether you want to [detect contacts automatically](#) between two solid, surface, or line bodies.
5. To enable the suppression, movement, or rotation of parts of the geometry, select **Allow configuration**.
6. Click **Next**. If you have chosen to import the geometry, you are prompted to browse and select a geometry file.

You can then set the [physics options](#).

## 1.3.6.2. Fluid-Structure Interaction: Physics

Specify the physics to be simulated.

1. To simulate turbulent flows with a significant amount of swirl, select **Swirling flow**.
2. To model fluid motion and heat transfer effects in and around structures or components, select **Heat transfer between a fluid and a solid**.
3. Define your simulation behavior:
  - To model behavior that does not change over time, select **Steady-state fluid flow**.
  - To model time-varying behavior, select **Time-dependent fluid flow** and then specify the **Expected duration**. Only the fluid calculation is time-dependent. Structural stress will be a static calculation based on the solution at the last time point.
4. Click **Next** to define the physics regions: [fluid regions](#) and [solid regions](#).

## 1.3.6.3. Fluid Physics Region

Define the fluid region(s) for your fluid-structure interaction simulation.

1. Specify the number of fluid regions.  
Multiple fluid regions are required if you have different flow paths (for example, different fluid circuits in a heat exchanger). You can define up to five regions here; define any additional required regions in the Physics task.
2. Select the bodies in your geometry forming the fluid region or you can create the body representing the fluid flow region later, using the **Geometry Modeling** task to extract the volume.
3. Select a material for the fluid region.  
If your material is not listed, select **Other**. You will create or select the material later on the Material Assignments page.
4. Include any additional flow physics you would like to simulate within the region.
  - To include temperature variations within the fluid, select **Heat transfer**.
  - If any of the flow will experience [variations in the gas density](#), select **Compressible flow (ideal gas)**. If no selection is made, then incompressible flow will be modeled, where the effects of density variation in the flow field will be ignored.
  - To model discrete particles in a fluid, such as gas bubbles in a liquid, select **Particles**.
5. Click **Next** to [define your solid regions](#).

## 1.3.6.4. Solid Physics Region

Define the solid region(s) for your fluid-structure interaction simulation.

1. Specify the number of solid regions.

You will need separate solid regions if different bodies have different physics settings or initial conditions. You can define up to five regions here; define any additional required regions in the Physics task.

2. Specify the number of materials to assign within a solid region.

A solid region may consist of several bodies and materials may be assigned to one or more bodies.

3. Select the location for the solid regions.

If the solid region is made up of a single material you can select the bodies forming the solid region or use all bodies that were not assigned to a fluid region to represent the solid region. For multiple materials, select the bodies and the corresponding materials. All bodies selected will be part of this region.

4. Select the material.

If your material is not listed, select **Other**. You will create or select the material later on the Material Assignments page.

5. After defining the materials for all solid regions, click **Next** to view a summary of your inputs.

## 1.3.6.5. Fluid-Structure Interaction Summary

This panel summarizes your inputs. Click **Finish** to apply your template settings or **Back** to change a setting.

- If you selected **Define new geometry** from the Template panel, the Model Editor window opens where you can create your geometry.

For details on using the Model Editor, see [Model Editing](#) on page 92.

- If you have imported a geometry that you want to edit:

1. Click the Geometry link in the **Simulation** panel.
2. Click **Edit Geometry** in the Geometry panel.

- If you have chosen to allow configuration:

For details on using the Configuration task to suppress parts, see [Creating a Suppress Control](#).

For details on using the Configuration task to move and rotate parts, see [Creating a Move/Rotate Control](#).

- For details on volume creation, see [Volume Creation](#).

## 1.3.7. Fluid-Solid Heat Transfer Template

### **Start > Fluid-Solid Heat Transfer**

The **Fluid-Solid Heat Transfer** template app defines simulations that solve conjugate heat transfer (CHT) where you can simulate fluid motion and heat transfer effects in and around structures or components.

On each panel of the template, review your options and make any necessary changes. The panels displayed depend on the options chosen.

1. Review the [model options](#) and make any necessary changes.
2. Specify the [physics to be simulated](#).
3. [Define the fluid region\(s\)](#) for your fluid-solid heat transfer simulation.
4. [Define the solid region\(s\)](#) for your fluid-solid heat transfer simulation.
5. [Review your simulation inputs](#) and then click **Finish** to get started. Each of the tasks in the created simulation will have settings specific to the choices you make here in the template.



### [Fluid-Solid Heat Transfer Overview](#)

## 1.3.7.1. Fluid-Solid Heat Transfer: Model

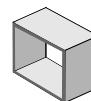
In this step of the template, review the model options and make any necessary changes.

1. Specify how the geometry is brought into the simulation.
  - To create a new geometry, select **Define new geometry**.
  - To be able to edit a geometry, select **Import geometry file** and select the **Allow geometry modeling option**.
  - To import a geometry from an external CAD session, select **Connect to active CAD session**.
  - To import a geometry without the ability to edit it, select only **Import geometry file**.
  - If a task already exists that includes the desired geometry, select **Use existing task** and make the appropriate selection for **Task containing geometry**.
2. To [define the mesh manually](#), rather than use [automatic physics-aware meshing](#), select **Define mesh manually**.
3. If you are connecting to an active CAD session or you disabled **Allow geometry modeling**, you must specify how you would like the geometry to be treated:

**Represents the flow volume(s)**



**Requires the flow volume(s) to be created** (if geometry modeling is enabled, you must create the flow volume(s) in the Geometry Modeling task)



4. Choose whether you want to [detect contacts automatically](#) between two solid, surface, or line bodies.
5. To enable the suppression, movement, or rotation of parts of the geometry, select **Allow configuration**.
6. Click **Next**. If you have chosen to import the geometry, you are prompted to browse and select a geometry file.

You can then set the [physics options](#).

## 1.3.7.2. Fluid-Solid Heat Transfer: Physics

Specify the physics to be simulated.

1. To simulate turbulent flows with a significant amount of swirl, select **Swirling flow**.
2. Specify physics coupling options for the simulation.
  - When you want to transfer temperature from the solid body into the same body in a separate structural simulation, select **Structural stress due to solid thermal effects**. Thermal stress can then be calculated in the structural simulation.
  - When you want to transfer force from surfaces in the fluid onto surfaces on the solid body in a separate structural simulation, select **Structural stress due to fluid forces**.
3. Define your simulation behavior:
  - To model behavior that does not change over time, select **Steady-state fluid-solid heat transfer**.
  - To model time-varying behavior, select **Time-dependent fluid-solid heat transfer** and then specify the **Expected duration**.
4. Click **Next** to define the physics regions: [fluid regions](#) and [solid regions](#).

## 1.3.7.3. Fluid Physics Region

Define the fluid region(s) for your fluid-solid heat transfer simulation.

- Specify the number of fluid regions.

Multiple fluid regions are required if you have different flow paths (for example, different fluid circuits in a heat exchanger). You can define up to five regions here; define any additional required regions in the Physics task.

- Select the bodies in your geometry forming the fluid region or you can create the body representing the fluid flow region later, using the **Geometry Modeling** task to extract the volume.

- Select a material for the fluid region.

If your material is not listed, select **Other**. You will create or select the material later on the Material Assignments page.

- Include any additional flow physics you would like to simulate within the region.

- To include temperature variations within the fluid, select **Heat transfer**.
- If any of the flow will experience [variations in the gas density](#), select **Compressible flow (ideal gas)**. If no selection is made, then incompressible flow will be modeled, where the effects of density variation in the flow field will be ignored.
- To model discrete particles in a fluid, such as gas bubbles in a liquid, select **Particles**.

- Click **Next** to [define your solid regions](#).

## 1.3.7.4. Solid Physics Region

Define the solid region(s) for your fluid-solid heat transfer simulation.

- Specify the number of solid regions.

You will need separate solid regions if different bodies have different physics settings or initial conditions. You can define up to five regions here; define any additional required regions in the Physics task.

- Specify the number of materials to assign within a solid region.

A solid region may consist of several bodies and materials may be assigned to one or more bodies.

- Select the location for the solid regions.

If the solid region is made up of a single material you can select the bodies forming the solid region or use all bodies that were not assigned to a fluid region to represent the solid region. For multiple materials, select the bodies and the corresponding materials. All bodies selected will be part of this region.

- Select the material.

If your material is not listed, select **Other**. You will create or select the material later on the Material Assignments page.

- After defining the materials for all solid regions, click **Next** to view a summary of your inputs.

## 1.3.7.5. Fluid-Solid Heat Transfer Summary

This panel summarizes your inputs. Click **Finish** to apply your template settings or **Back** to change a setting.

- If you selected **Define new geometry** from the Template panel, the Model Editor window opens where you can create your geometry.

For details on using the Model Editor, see [Model Editing](#) on page 92.

- If you have imported a geometry that you want to edit:

1. Click the Geometry link in the **Simulation** panel.
2. Click **Edit Geometry** in the Geometry panel.

- If you have chosen to allow configuration:

For details on using the Configuration task to suppress parts, see [Creating a Suppress Control](#).

For details on using the Configuration task to move and rotate parts, see [Creating a Move/Rotate Control](#).

- For details on volume creation, see [Volume Creation](#).
- For details on physics coupling, see [Physics Coupling Interface](#).

### 1.3.8. Polymer Extrusion Template

#### **Start > Polymer Extrusion**

The **Polymer Extrusion** template app defines simulations where you can investigate the extrusion of molten polymers.

To create a polymer extrusion simulation:

1. From the **Start** panel, click **Polymer Extrusion**.
2. Specify the model source and editing options:
  - To import a geometry without the ability to edit it, select **Import geometry file**.
  - To import a geometry from an external CAD session, select **Connect to active CAD session**.
3. Specify a simulation goal:
  - To investigate the flow of the molten polymer within just the die itself, select **Analyze flow within die**. With this option, several [conditions](#) are already provided for you as part of the template:
    - Inlet
    - Outlet
    - Wall
  - To determine the shape of the extrudate based on the geometry of the die, material characteristics, and operating conditions, select **Predict extrudate shape**. With this option, several [conditions](#) are already provided for you as part of the template:
    - Extrudate Deformation
    - Free Surface
    - Extrudate Exit
    - Inlet
    - Wall
  - To determine the shape of the die lip corresponding to the desired cross section of the extrudate, select **Determine die lip shape**. With this option, several [conditions](#) are already provided for you as part of the template:
    - Extrudate Deformation
    - Free Surface
    - Die Deformation
    - Extrudate Exit
    - Inlet
    - Wall

**Note:** You may need to [consider some issues](#) when setting up a die lip design simulation.

4. Specify additional fluid physics options.

- To obtain optimum melt flow distribution in polymer extrusion, include a **Restrictor**. Restrictors are solid structures placed in the polymer flow region. There are some [limitations](#) when using restrictors.
- To simulate more than one material simultaneously passing through different inlets, select **Co-extrusion of multiple materials**. With this option, several objects are already provided for you:
  - Two material assignments.
  - Two inlets, an outlet, and a wall flow physics condition.
  - Two contour result objects: one for each fraction of fluid entering through their assigned inlets.

The template assumes two materials and two inlets. If your simulation requires more materials or more inlets, you must define them manually.

- To simulate temperatures, thermal gradients, heat flow rates, and heat fluxes in the extrusion, select **Fluid heat transfer**.
- To simulate temperatures, thermal gradients, heat flow rates, and heat fluxes between the extrusion(s) and the die, select **Fluid-solid heat transfer**.

In addition to the objects that are created based on your simulation goal, with this option, the template automatically creates additional objects:

- Two physics regions: one for the fluid (Thermal Polymer Extrusion) and one for the solid (Thermal), where you must assign appropriate locations.
- Two material assignments with default materials (Generic Polymer and Structural Steel).
- A region interface condition between the fluid region(s) and solid region, instead of a wall condition.
- Two initial temperature objects: one for the fluid, and one for the solid.
- Contour results for temperature are available on all bodies, while other contour results (velocity, pressure, shear-rate, etc.) are defined on the fluid region(s).

## 5. Specify the other simulation options:

- To be able to suppress parts of the geometry, select **Allow configuration**.
- To be able to move or rotate the geometry, select **Allow configuration**.
- To [define the mesh manually](#) instead of using [automatic physics-aware meshing](#), select **Define mesh manually**.

## 6. When you have made your selections, click **Finish** to [get started](#). Each of the tasks in the created simulation will have settings specific to the choices you make here in the template.



[Predicting Extrudate Shape Using a Polymer Extrusion Simulation](#)



[Determining Die Lip Shape Using a Polymer Extrusion Simulation](#)

## 1.3.8.1. Simulating a Polymer Extrusion

Use the [Polymer extrusion template](#) to create a basic polymer extrusion simulation.

The template creates a basic workflow that you can adjust to suit your simulation requirements.

1. Open AIM.
2. On the Start panel, click **Polymer Extrusion**.
3. In the **Polymer Extrusion Template** panel:
  - a) Specify how you will load the geometry.
  - b) Set the goal of the simulation.
  - c) Optionally, specify if you are going to include restrictors (solid structures placed in the polymer flow region) to obtain optimum melt flow distribution.

- d) Optionally, specify if you are going to include the co-extrusion of multiple materials. The template automatically creates two material assignments, two inlets, and two contour result objects. If your simulation requires additional objects, you must define them manually.
- e) Optionally, select whether fluid thermal effects are to be included in the simulation.
- f) Optionally, select whether fluid-solid thermal effects are to be included in the simulation. The template automatically creates two physics regions (one for a fluid and one for a solid), two material assignments (a generic polymer and structural steel), a region interface condition, and two initial temperature condition objects. If your simulation requires additional objects, you must define them manually.
- g) Optionally, enable the suppression, movement, or rotation of parts of the geometry by selecting Allow configuration.
- h) Optionally, clear the **Define mesh manually** field to enable automatic physics-aware meshing.

**4. Click **Finish**.**

- For details on using the Configuration task to suppress parts, see [Creating a Suppress Control](#).
- For details on using the Configuration task to move and rotate parts, see [Creating a Move/Rotate Control](#).
- 5. (optional) Define any selection sets, if needed. Selection sets enable you to collect and combine topology to more easily manage location assignments to topology in a complex assembly. For example, selection sets can be useful for multiple selections in sophisticated CAD geometries.
- 6. (optional) Define any construction geometries, if needed. Construction geometries, such as points and planes, can be created for later use at any point in the simulation (such as Results).
- 7. (optional) Complete the Mesh task. If you cleared the **Define mesh manually** field to enable automatic physics-aware meshing, there is no Mesh task. Instead, you can set the **Solution fidelity** slider on the Extrusion physics task.

- a) Use the Mesh task to apply various settings for your computational domain.

When using the polymer extrusion template, the **Engineering intent** field of the Mesh task is automatically set to **Fluid flow or fluid-solid heat transfer**, which is applicable to a polymer extrusion simulation.

- b) Once the mesh is defined, you can generate the mesh and update the task. The up-to-date Mesh task can then be used in the Physics task.

**8. Complete the Extrusion task.**

- a) Specify the calculation type as **Steady/Static**.
- b) (optional) Move the **Solution fidelity** slider toward lower fidelity or higher fidelity.
- c) Define a [polymer extrusion physics region](#), or review the current settings. Select a relevant portion of the model and designate it to become a region where you want to simulate polymer extrusion. Polymer extrusion simulations can also include multi-body geometry models that can be applied to multiple physics regions for use with, for example, conjugate heat transfer.
- d) Define [material assignments](#), or review the current settings. Assign materials to specified regions. Use an existing material or create a new material assignment for all bodies participating in the solution. Several [material models](#) are available for polymer extrusion simulations to model polymer characteristics. You can choose from a Newtonian fluid, a Generalized Newtonian fluid, or a Simplified viscoelastic fluid.

For co-extrusion simulations, you can only use Newtonian or generalized Newtonian fluid materials. In addition, you must define your multiple material assignments for the polymer extrusion physics region before you can specify the incoming material for your particular inlet fluid flow conditions.

- e) (optional) Define [physics options](#), or review the current settings. For polymer extrusion simulations, options include inertia and gravitational effects.
- f) Define [fluid flow conditions](#), or review the current settings. Designate selected surfaces or bodies of your model to be an inlet, wall, extrudate, free surface, or extrudate exit. Assign extrusion-specific physics details (e.g., velocity, pressure, temperature) for each boundary that match your simulation requirements.

- g) Define **solid thermal conditions**, for fluid-solid conjugate heat transfer polymer simulations. Designate selected surfaces or bodies of your model to be an insulated, convection, or temperature conditions.
  - h) (optional) Define **symmetry conditions** for your polymer simulations when the physical geometry of interest, and the expected pattern of the flow/thermal solution, have mirror symmetry.
  - i) (optional) Define **region interfaces**, for conjugate heat transfer simulations with multiple physics regions.
  - j) (optional) Define **fluid-solid interface behaviors**, for fluid-solid conjugate heat transfer polymer simulations with multiple physics regions.
  - k) (optional) Define **initial conditions**, or review the current settings. Initial conditions provide an initial and approximate solution to the flow field and can be modified to improve convergence of the solution, however, default initial conditions will generally suffice.
  - l) (optional) Define **solver options**, or review the current settings. Solver options control how the flow field is calculated, and can be modified to improve the accuracy of your results, however, default solver options will generally suffice.
  - m) Once the physics is defined, you can generate a solution and update the task. The up-to-date Extrusion task can then be used in the Results task.
9. Complete the Results task.
- a) Proceed to the Results task to **evaluate the results** of the polymer extrusion simulation.
  - b) For polymer extrusion simulations, you are able to view **contour** or **vector** results for intermediate solutions.

### 1.3.9. Polymer Blow Molding Template

#### Start > Polymer Blow Molding

The **Polymer Blow Molding** template app allows you to simulate blow molding, a process in which air pressure is used to inflate soft molten plastic into a mold cavity. This process is used to create hollow plastic parts, such as bottles.

On each panel of the template, review your options and make any necessary changes. The panels displayed depend on the options chosen.

1. Review the **model options** and make any necessary changes.
2. Specify the **physics to be simulated**.
3. Define the **regions** for your blow molding simulation.
4. Click **Finish** to get started. Each of the tasks in the created simulation will have settings specific to the choices you make here in the template.

#### 1.3.9.1. Blow Molding: Model

In this step of the template, review the model options and make any necessary changes.

1. Specify how the geometry is brought into the simulation.
  - To create a new geometry, select **Define new geometry**.
  - To be able to edit a geometry, select **Import geometry file** and select the **Allow geometry modeling option**.
  - To import a geometry from an external CAD session, select **Connect to active CAD session**.
  - To import a geometry without the ability to edit it, select only **Import geometry file**.
  - If a task already exists that includes the desired geometry, select **Use existing task** and make the appropriate selection for **Task containing geometry**.
2. To **define the mesh manually**, rather than use **automatic physics-aware meshing**, select **Define mesh manually**.

3. To enable the suppression, movement, or rotation of parts of the geometry, or to create an external flow (when a fluid region must be created around a solid body), select **Allow configuration**.
4. Click **Next**. If you have chosen to import the geometry, you are prompted to browse and select a geometry file.

You can then set the [physics options](#).

## 1.3.9.2. Blow Molding: Physics

A blow molding simulation is time-dependent and therefore exhibits behavior that varies with time.

1. Enter the **Expected duration** for the calculation.
2. Specify any additional physics to be simulated.
  - Include a **Plunger** if you want to have more control on the stretching of the parison or if your process contains two molds: one fixed and one moving (the plunger).
  - Include **Heat transfer** if the material behavior varies significantly with temperature and you expect temperature variations, via heat exchange between the parison and the mold for example. You will specify the heat transfer coefficient when defining the [contact behavior](#). [Initial temperature conditions](#) for the parison, the mold, and the plunger (if selected) are created.
3. Click **Next** to define the blow molding [physics regions](#).

## 1.3.9.3. Blow Molding: Physics Region

Define the regions for the blow molding simulation. Note that regions need to be shell/surface bodies.

1. Select the bodies that form the parison.

2. Select the parison material.

If your material is not listed, select **Other**. You will create or select the material later on the Material Assignments page.

3. Select the bodies that form the mold.

Two material assignments are created, with structural steel assigned to the mold(s) by default and the material of your choice assigned to the parison.

4. If your simulation includes a plunger (or additional moving parts), select the bodies that form the plunger.

A [velocity driven mold](#) condition will be created under structural conditions, which you must define. In addition, a [second contact](#) (between the plunger and parison) is created that shares contact behavior with the first contact (between the mold and parison). Verify that the **Mold body side** of the contact has been correctly defined.

5. When you have made your selections, click **Finish** to get started.

Each of the tasks in the created simulation will have settings specific to the choices you make here in the template. By default, a single [polymer edge condition](#) is created that contains all the edges on the border of the parison. If the simulation contains other types of conditions (planes of symmetry, polymer free edges), you will have to create them after you have completed the template setup. You may also be required to remove edges from the polymer edge condition if needed.

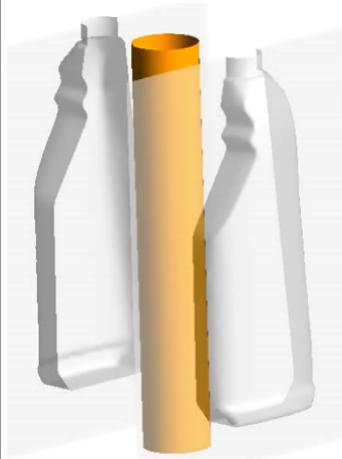
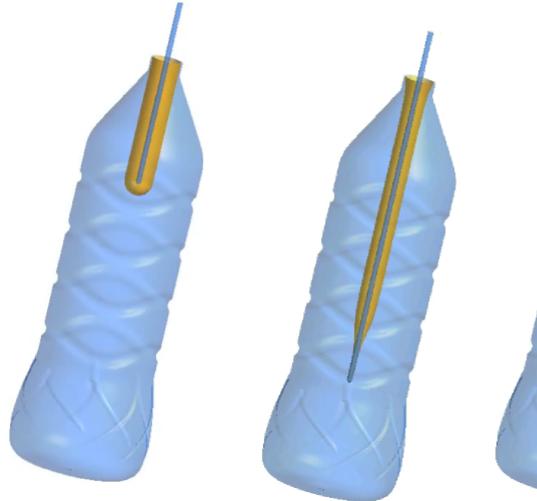
## 1.3.9.4. Simulating a Polymer Blow Molding

Use the [Polymer Blow Molding Template](#) on page 60 template to create a basic polymer blow molding simulation.

The template creates a basic workflow that you can adjust to suit your simulation requirements.

1. Open AIM.
2. On the Start panel, click **Polymer Blow Molding**.
3. In the **Polymer Blow Molding Template** panels:
  - a) Specify how you will load the geometry.
  - b) Optionally, specify if you are going to include the use of a plunger in the blow molding process. The template automatically creates several physics objects. If your simulation requires additional objects, you must define them manually.
  - c) Optionally, select whether thermal effects are to be included in the simulation.
  - d) Optionally, enable the suppression, movement, or rotation of parts of the geometry by selecting Allow configuration.
  - e) Optionally, clear the **Define mesh manually** field to enable automatic physics-aware meshing.

This template can be used to simulate a number of blow molding applications, described in the table below.

Blow molding simulation	Description	Example
Extrusion blow molding	Extrusion blow molding is the process where molten plastic is extruded into a parison (hollow tube). The parison is then enclosed in a cooled mold and air is blown into the parison, inflating it into the shape of the container.	
Injection stretch blow molding	Stretch blow molding is a method used for producing bottles, where the plastic is first molded into a "preform" then heated and blown using high-pressure air into bottles using metal blow molds. A plunger is used to stretch the preform before or during blowing.	

Blow molding simulation	Description	Example
Thermoforming	An example of thermoforming is the process where a plastic sheet is heated and molded into shape, then trimmed into blister packaging trays. A refrigerator cavity is another example of how thermoforming is used in manufacturing.	

4. Click **Finish**.
5. (optional) Define any selection sets, if needed. Selection sets enable you to collect and combine topology to more easily manage location assignments to topology in a complex assembly. For example, selection sets can be useful for multiple selections in sophisticated CAD geometries.
6. (optional) Define any construction geometries, if needed. Construction geometries, such as points and planes, can be created for later use at any point in the simulation (such as Results).
7. (optional) Complete the Mesh task. If you cleared the **Define mesh manually** field to enable automatic physics-aware meshing, there is no Mesh task. Instead, you can set the **Solution fidelity** slider on the Blow Molding physics task.
  - a) Use the Mesh task to apply various settings for your computational domain.  
When using the polymer blow molding template, the **Engineering intent** field of the Mesh task is automatically set to **Fluid flow or fluid-solid heat transfer**, which is applicable to a polymer blow molding simulation. It might be necessary to refine the mesh in the Polymer Blow Molding physics region; this region will indeed be stretched to cover the mold geometry.
  - b) Once the mesh is defined, you can generate the mesh and update the task. The up-to-date Mesh task can then be used in the Physics task.
8. Complete the Blow Molding physics task.
  - a) A blow molding simulation is time-dependent. Set the **Duration** to specify the length of time for the calculation.
  - b) (optional) Move the **Solution fidelity** slider toward lower fidelity or higher fidelity. This is only available when automatic physics-aware meshing is applied.
  - c) Define a [polymer blow molding physics region](#) and a structural physics region, or review the current settings. Select a relevant portion of the model and designate it to become a region where you want to simulate polymer blow molding. Note that both regions need to be shell bodies.
  - d) Define material assignments, or review the current settings. Assign materials to specified regions. Use an existing material or create a new material assignment for all bodies participating in the solution. Several [material models](#) are available for polymer blow molding simulations to model polymer characteristics. You can choose from a Newtonian fluid, a Generalized Newtonian fluid, or an Integral viscoelastic fluid.
  - e) Define [physics options](#), or review the current settings. For polymer blow molding simulations, options include gravitational and viscous dissipation effects.

- f) Define **fluid flow conditions**, or review the current settings. Designate selected surfaces or regions of your model to define the inflation pressure and edges to define polymer edges that can be fixed or free.
- g) Define the mold motion under **structural conditions**.
- h) Add **symmetry conditions** and define the plane(s) of symmetry.
- i) Define mold to fluid **contact interface conditions** and **contact behavior**.
- j) Define the **initial layer thickness** and initial temperature (if non-isothermal) with initial conditions. For the blow molding region, specify the thickness of each polymer layer.
- k) Define **solver options** or review the current settings. Solver options control how the flow field is calculated, and can be modified to improve the accuracy of your results, however, default solver options will generally suffice. In addition, an **adaptive meshing technique** is applied by default to handle the (sometimes complex and/or large) deformations of the parison.
- l) Once the physics is defined, you can generate a solution and update the task. The up-to-date Blow Molding task can then be used in the Results task.

**9.** Complete the Results task.

### 1.3.10. Geometry Modeling Template

#### Start > Geometry Modeling

The **Geometry Modeling** template app defines a simulation that contains only the geometry model for use with other physics template apps.

To create a geometry modeling simulation:

1. From the **Start** page, click **Geometry Modeling** and click **Start**.
2. Specify the model source and editing options:
  - To create a new geometry, select **Define new geometry**.
  - To import a geometry and have the ability to edit it, select **Import geometry file** and enable **Allow geometry modeling** under **Model options**.
3. When you have made your selections, click **Finish** to get started.

You can return to the **Start** page and instruct other physics template apps to use the geometry defined in this app through the **Use existing task** model option.

### 1.3.11. Connect to Discovery Live Template

#### Start > Connect to Discovery Live

The **Connect to Discovery Live** template app creates a connection between ANSYS Discovery AIM and ANSYS Discovery Live, so that you can transfer geometry and physics from Discovery Live to AIM, or geometry from AIM to Discovery Live. Typical applications include:

- You're an AIM user that received an .scdoc file from a Discovery Live user, and you want to import the file into AIM to refine the analysis.
- You're an AIM user that wants to export a geometry file to Discovery Live to perform design exploration. For this workflow, you must use a Geometry Modeling task rather than a Data Import task. Additionally, physics data cannot be exported from AIM to Discovery Live, so with this workflow, you must set up your physics again after you return to Discovery Live.
- You're a Discovery Live user that wants to export a preliminary design to AIM to refine it. For this workflow, you begin in Discovery Live. Refer to the Discovery Live help for more information.

When the data is transferred to AIM, AIM sets up a simulation process for each of the simulations contained in the Discovery Live .scdoc file. The transferred data includes materials and boundary conditions, so that the simulations are ready to update in AIM, where you can add more complex physics.

**Note:**

- If you are importing, browse and select a Discovery Live .scdoc file.
- If you are exporting, Discovery Live launches with the geometry displayed. To run Discovery Live, your graphics card must meet these requirements or an error is issued:
  - NVidia GPU
  - Kepler series or later
  - 4 GB minimum GPU dedicated RAM

To connect to Discovery Live:

1. From the **Start** page, click **Connect to Discovery Live** and click **Start**.
  - a) Choose a connection option.
    - To import, select **Import from Discovery Live**.
    - To export, select **Export to Discovery Live**.
  - b) Specify import or export options.
    - To import geometry and physics, select **Import physics setup**.
    - To import geometry only, clear **Import physics setup**.
    - To export, from the **Geometry to export** drop-down list, select the task containing the geometry you want to export.
  - c) Click **Next**. If you have chosen to **Import from Discovery Live**, you are prompted to browse and select the file.
2. Specify **Additional Physics**.
  - a) Select the physics task to which you want to apply the additional physics.
  - b) Specify the additional physics that you want to evaluate. To model situations where:
    - Fluid-solid heat transfer occurs, including temperature variations within the fluid, select **Fluid-solid heat transfer**.
    - The temperature field introduces thermal strains in the structural field, which then result in thermal expansion and stresses, select **Structural stress due to solid thermal effects**.
    - Fluid forces transfer to a structure without heat transfer, select **Compute fatigue results**.
    - Click **Next**.
  - c) To automatically calculate fatigue results for your static structural analysis based on the S-n curve values defined for the default material, select **Compute Fatigue results**.
  - d) Click **Next**.
3. Define the material(s) and solid region(s) to include in the fluid-solid heat transfer simulation in the **Solid Physics Region** panel.
  - a) To assign only one material:
    - Keep 1 from the dropdown.
    - Define the solid region location
      - automatically by selecting **Use all unassigned bodies**, or
      - manually by selecting the location with **Select bodies forming the solid region**.
    - Select the **Material for solid region** from the dropdown list.
    - If defining heat transfer from external surfaces, select the type of **Heat transfer option**.
  - b) For more than one material, define the location manually then select the material and the type of **Heat transfer option**.
  - c) Click **Next**.

4. Specify **Structural Information**.
  - a) Define the **Solid support locations** by selecting a face or selection set.
  - b) Click **Next**.
5. **Import Summary** is an opportunity to review your inputs and make changes. When you're satisfied, click **Finish**.

## 1.3.11.1. Connect to Discovery Live Template Considerations

There are differences between Discovery Live and AIM in areas such as model management, material properties, solvers, results, and so on. Awareness of these differences and their potential impact on data transfer enables you to use the **Connect to Discovery Live** template app efficiently.

## Unsupported Transfers

These types of data transfers from Discovery Live to AIM are not supported:

- Calculators or results.
- Multiple enclosures. Although the enclosures themselves will transfer, the mapping of boundary conditions onto the enclosures will be incomplete.
- Simulations involving faceted bodies.
- Topology optimization simulations.
- Swirl inlet conditions applied to more than one face.
- Gravity conditions for fluids solutions without thermal.
- Time-varying conditions including gravity, rotating wall, and mass flow when it contains both positive and negative values.
- Time stepping.
- Mass conditions (both distributed and point) for electro-thermal-stress and thermal-stress simulations.
- Time-varying convection and **Convection in Air** conditions for thermal-only simulations. In the case of time-varying convection, the temperature transfers but the coefficient does not.

## Manual Verification

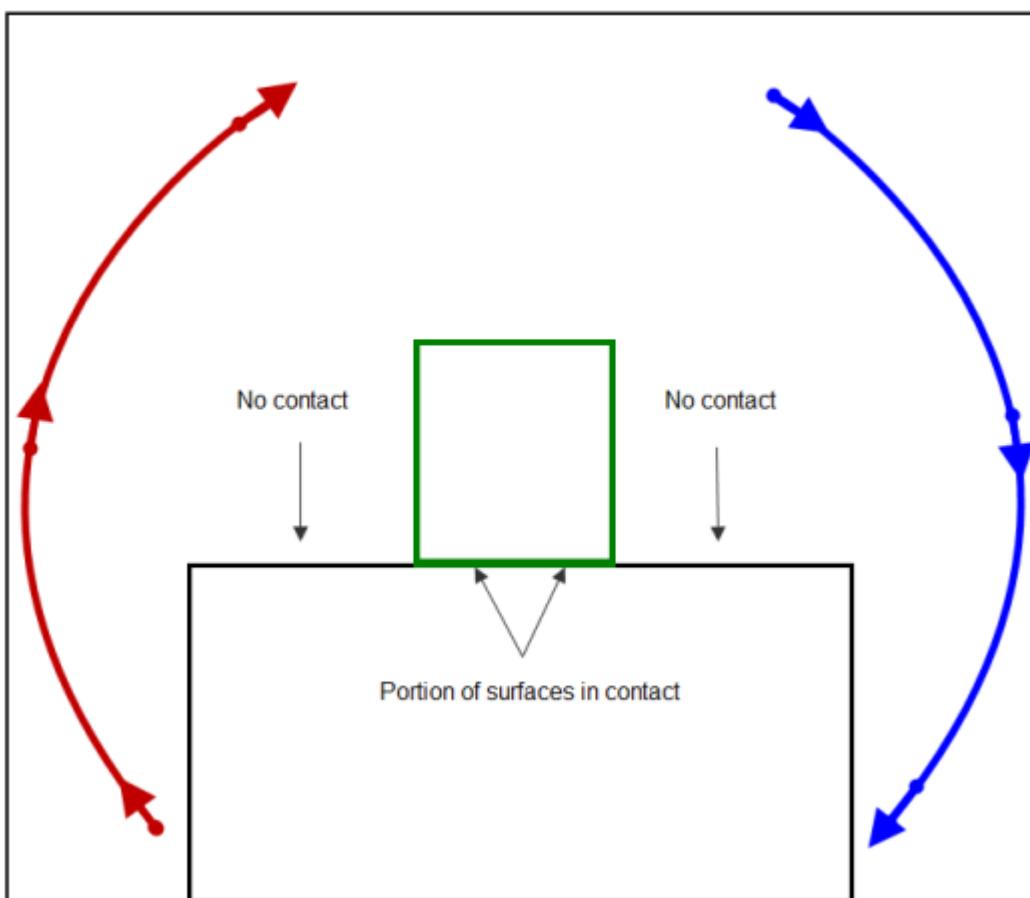
After transferring data from Discovery Live to AIM, there are cases in which you must perform manual verification:

- After geometry is transferred to AIM, additional geometry cleanup may be required in order to achieve a solution in AIM (for example, when the simulation involves external flow, complex structures, and so on).
- After transferring data to AIM, you must verify the setup (materials, initial conditions, and so on) to ensure your settings are as expected.
- It is especially important for you to check the properties of transferred materials that apply to state (solid, liquid, or gas). This is because in AIM, material properties are specific to the state of matter that the material will have in your simulation, but in Discovery Live they are not.
- Thermal expansion coefficient in a fluid context (liquid or gas) is volumetric, while in a structural context (solid), it is linear. If gravity is defined and a thermal expansion coefficient is specified, check its value to make sure that it is reasonable (**Physics Options > Buoyancy > Thermal expansion coefficient**).

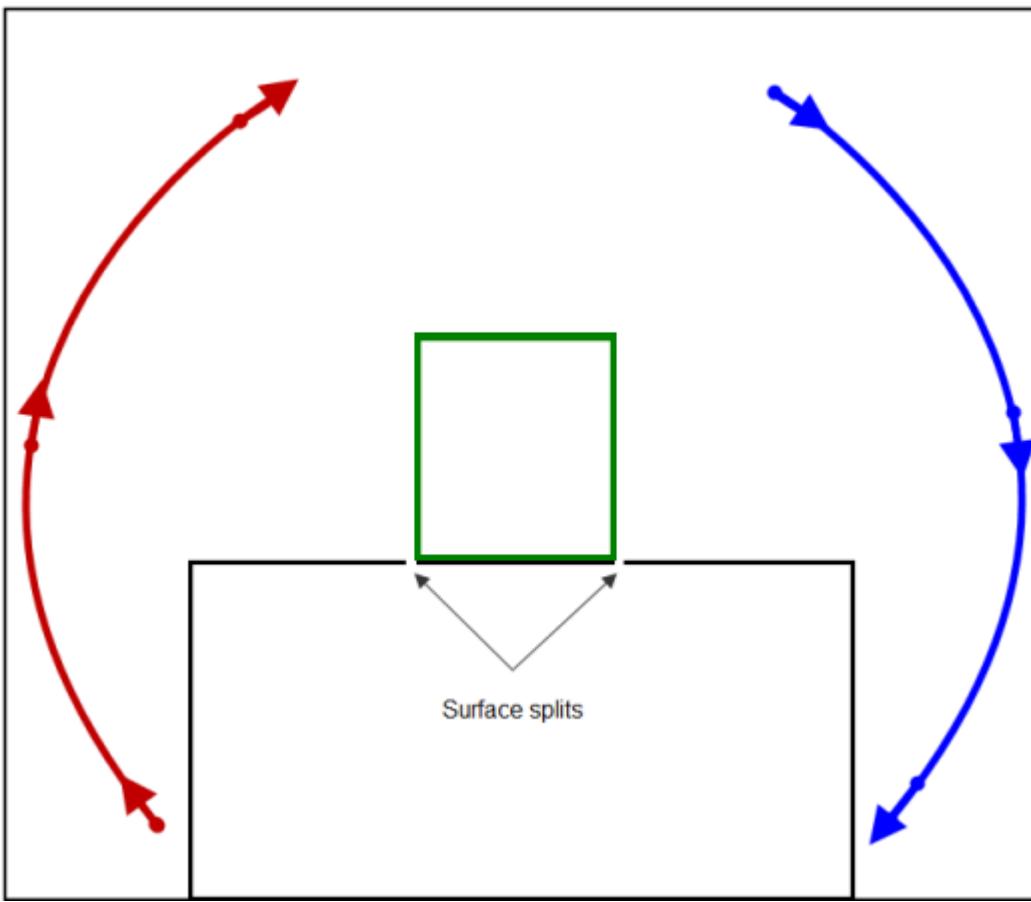
# Modeling Thermal Boundary Conditions and Coincident Surfaces

Some thermal boundary conditions are applied globally in Discovery Live. In other cases, you may specify a thermal boundary condition on an internal face or on a body that is adjacent to another body, thereby creating an internal face. In any of these cases involving thermal boundary conditions and coincident surfaces in Discovery Live, you may need to use Model Editing to model the external boundary conditions accurately for use in AIM.

For example, consider the **Convection in Air** case illustrated below. **Convection in Air** is a condition that is applied globally in Discovery Live. In the image of the two bodies below, the bottom (green) surface of the top body and the top (black) surface of the bottom body are in contact, but there are additional portions of the black surface that are not in contact with the green surface. If you add convection to the black surface, the convection is applied to the *entire* black surface, rather than to the external portions only. In most cases, the impact of this overlap is small, but in some cases you will need to modify the geometry in AIM as detailed below.



After importing the Discovery Live .scdoc file into AIM, right-click the Geometry task in the Workflow view and select **Edit Geometry**. If you need the bodies to remain separate (for example, because they are made of different materials, or you want to model sliding between them), use Model Editing to split the black surface into three portions as shown below, such that you can manually remove from the automatically created convection condition the portion of the black surface that is in contact with the green surface. The **Imprint** tool in Model Editing can identify these surfaces automatically.



Alternatively, if the two bodies are of the same material and there is no value in keeping them separate, you can use Model Editing to merge them.

Although the case above involves **Convection in Air**, you can use the same techniques for other boundary conditions.

## Other Considerations

Other differences that you need to be aware of include:

- The **Convection in Air** boundary condition is applied globally in Discovery Live. When this boundary condition is transferred to AIM, its location is set to the `DefaultConvectingSurfaces()` expression automatically. `DefaultConvectingSurfaces()` includes all surfaces except those to which another solid thermal boundary condition is applied (with exceptions discussed [here](#)).
- Custom materials defined in Discovery Live do not have any state, so there is no state to transfer to AIM. Instead, AIM uses the material properties from Discovery Live to set the material properties for all states (solid, liquid, and gas) in AIM that they could possibly apply to. For example, AIM uses the viscosity value from Discovery Live to set both liquid viscosity and gas viscosity in AIM. Viscosity does not apply to the solid state in AIM.
- The Discovery Live and AIM solvers handle bodies within enclosures differently. While the Discovery Live solver can extract volumes from enclosures automatically, the AIM solver cannot. When you transfer an enclosure from Discovery Live to AIM, the transfer process creates a new version of the enclosure from which the internal body has been subtracted. All of the solution data (for example, boundary conditions, material assignments, and so on) that is located on the internal body is translated to corresponding locations on the new enclosure.

- When a multi-load simulation is transferred from Discovery Live to AIM, each individual load case transfers into AIM as a separate Physics task.
- Fixed support conditions transfer from Discovery Live to AIM as support conditions, but cylindrical, hinged, planar, and spherical support conditions transfer to AIM as displacement conditions (with different degrees of freedom as appropriate).
- When a time-varying gravity boundary condition transfers from Discovery Live to AIM, its value transfers incorrectly as a single value rather than as a time-varying value.
- In Discovery Live, heat flow may be applied to a selected face, several faces, or an entire body. When transferred to AIM, the heat flow condition on faces transfers to the heat flow condition in AIM, but the heat flow condition on bodies transfers to the heat generation condition in AIM. Transfer of these conditions was handled differently prior to R19.2. To ensure the expected solver behavior if you are using an R19.1 .scdoc file, save the file in a later version of Discovery Live before importing it to AIM.
- For fluid simulations in Discovery Live, if an **End Time** is set or any time-varying boundary conditions are present and the simulation is transferred to AIM, the calculation type in AIM is set to time-dependent. Otherwise, the calculation type in AIM is set to steady-state.
- In Discovery Live, you do not specify a duration for transient physics, but AIM requires it. If you import a transient simulation from Discovery Live into AIM, you must specify a **Duration** on the Physics task in AIM.

### 1.3.12. Connect to Mechanical Template

#### **Start > Connect to Mechanical**

The **Connect to Mechanical** template app connects a geometry and mesh created in AIM to a Mechanical system using the template and the Workbench **Project Schematic**. You can also transfer selection sets.

1. From the **Start** page, click **Connect to Mechanical**.

If you select **Define new geometry** in the template, the Model Editor window opens where you can create your geometry. Create your geometry and then return to the AIM window by clicking the X in the upper right corner.

You will see Geometry, Mesh, and Mechanical tasks in the AIM **Workflow** view in the **Study**.

2. Select the Mesh task to add mesh controls and generate the mesh.
3. Right-click the Mechanical task in the Workflow view and select **Edit Physics** to launch Mechanical. You will see the imported geometry and mesh in Mechanical and can proceed with your analysis as you normally would.

If you return to the AIM model to make changes after you've transferred it to Mechanical, you can then update the model in Mechanical to capture those changes from AIM.

### 1.3.13. Connect to Fluent Template

#### **Start > Connect to Fluent**

The **Connect to Fluent** template app connects a mesh created in AIM to a Fluent system using the template and the Fluent **Project Schematic**. You can also transfer selection sets. Overlapping selection sets are not supported; you must define exclusive sets.

1. From the **Start** page, click **Connect to Fluent**.

If you select **Define new geometry** in the template, the Model Editor window opens where you can create your geometry. Create your geometry and then return to the AIM window by clicking the X in the upper right corner.

You will see Geometry, Mesh, and Fluent tasks in the AIM **Workflow** view in the **Study**.

2. Select the Mesh task to add mesh controls and generate the mesh.

3. Right-click the Fluent task in the Workflow view and select **Edit Physics** to launch Fluent. You will see the imported mesh in Fluent and can proceed with your analysis as you normally would.
- If you return to the AIM model to make changes after you've transferred it to Fluent, you can then update the model in Fluent to capture those changes from AIM.

## 1.4. Performing Common Operations

While working in AIM, there are several operations that you will commonly perform:

- [Updating tasks](#)
- [Defining locations](#)
- [Defining selection sets](#)
- [Manipulating a Geometric Entity](#) on page 72
- [Working with reference frames](#)
- [Creating construction geometry](#)
- [Using parameters and design points](#)
- [Using expressions](#)

**Note:** When typing Japanese characters in a text field, the user interface may be unresponsive and entering characters may not be possible. To recover, place the cursor in the text field, click the right mouse button and resume typing.

### 1.4.1. Updating Tasks and Understanding their States

As you progress through your simulation, you work within a given task, or an associated object, until your setup is complete. AIM provides you with visual cues that guide you and inform you as to a task's state of readiness.

Task State	Definition
 <b>Attention required</b>	The object/task has one or more fields that require your attention before it can be updated.
 <b>Out-of-date</b>	The object/task has not been updated.
 <b>Conditionally up-to-date</b>	The object/task may be valid and up-to-date but something that it depends on must be added or fixed. Correct the dependencies in order to move forward with defining the simulation.
 <b>Partially updated</b>	The object/task has been paused in the midst of being updated.
 <b>Up-to-date</b>	The object/task has been updated successfully.

When a task is complete and ready to be updated, (as indicated by the out-of-date state), you can update the task using either the blue Update icon (), or the corresponding blue update button at the bottom of the corresponding task's data panel (for example, ).

- [Interrupting an Update](#) on page 71
- [Continuing from Partially Updated](#) on page 71

## 1.4.1.1. Interrupting an Update

If you need to interrupt an update prior to completion, click the **Stop** button ( in the progress bar to interrupt the update at the next good stopping point. By waiting for a good stopping point, you may be able to review and use the output from the point the solution stopped.

When you have output from an interrupted update, the output is identified as **Partially updated**.

If you make no setup changes, a further update of the Physics task continues the solution process.

For some tasks, you may be able to *force stop* the update. A force stop immediately halts the update; it is unlikely that you will be able to review and use output from where it was stopped.

**Note:** There is no way to cancel the **Stop** operation once it is initiated.

- [Continuing from Partially Updated](#) on page 71

## 1.4.1.2. Continuing from Partially Updated

Any physics task in a partially updated state can be marked as **Up-to-date** so that you can view results. This can be useful when your solution is not converging but you think it's "good enough."

To continue from a partially updated state:

1. Right click on the physics task.
2. Select **Mark as Up-to-Date** from the context menu. A confirmation dialog box will appear.
3. Click **Yes** to confirm that you want to mark the physics task as up-to-date.

## 1.4.2. Defining Locations

Throughout AIM, you will need to assign a location to various objects in order to properly set up, solve, and analyze the results of your study.



### Defining Locations

## 1.4.3. Defining Selection Sets

Selection sets enable you to collect and combine objects, such as faces or bodies, in the graphics display so that you can use them more efficiently later in your study.



### Defining Selection Sets

Different selection sets defined within a simulation can share common objects (overlapping selection sets), however, when [connecting your work in AIM to Fluent](#), overlapping selection sets are not supported.

Selection sets can be created when a CAD geometry that uses Named Selections is imported into your simulation. Each imported named selection becomes a selection set and can become members of new selection sets you create in AIM. See [Geometry Preferences](#) on page 96 to learn how to control the importing of Named Selections.

#### 1.4.4. Manipulating a Geometric Entity

You can manipulate a geometric entity in several ways using a combination of keyboard and mouse.



In addition, you can click the **Enable exploded view** icon in the toolbar to create imaginary distance between bodies in your model. This functionality, available in Physics-related tasks for Fluid, Thermal, Electromagnetics, or Structural physics, enables you to more easily view the faces and bodies in your model. Use the slider to increase or decrease the exploded distance between the parts. You can select and show/hide faces and bodies in this view.

**Note:** In exploded view, springs are shown detached from the secondary location. In addition, geometry is not shown in exploded view when the mesh is displayed. Exploded view is not available in blow molding and extrusion physics.

Table 1.4.4.1. Keyboard and mouse combinations

Desired Manipulation	Action
Box select	LMB drag (left-right inside box, right-left intersected+ inside box)
Box select, add	Shift+LMB drag (left-right inside box, right-left intersected+ inside box)
Box select, remove	Ctrl+LMB drag (left-right inside box, right-left intersected+inside box)
Fit view	Alt+E
Fit view + reset center	Shift+RMB
Multi select	Ctrl+LMB
Multi "paint" select	Alt+LMB drag
Multi "paint" select, remove	Ctrl+Alt+LMB drag
Pan	Ctrl+RMB drag
Pan increments	Ctrl+Left arrow, Ctrl+Right arrow, Ctrl+Up arrow, Ctrl+Down arrow
Reset to center	Ctrl+RMB
Rotate about principal axes in 15°	Left arrow, Right arrow, Up arrow, Down arrow
Rotate free form	MMB drag
Rotate to axis	Alt+X, Alt+Shift X, Alt+Y, Alt+Shift Y, Alt+Z, Alt+Shift Z
Rotate to Iso-Y-up	Alt+I
Single select	LMB
Single select, toggle	Ctrl+LMB
Zoom	Shift+MMB drag
Zoom box	Shift+RMB drag (top-down in, bottom-up out)
Zoom increments	Wheel or Ctrl+, Ctrl-

## 1.4.5. Understanding Reference Frames

A reference frame provides a location and orientation relative to which spatially varying quantities are defined or measured. The frame's origin provides the reference location, while the frame's axes define the orientation. All of AIM's coordinate systems may be used to define coordinates and vectors within a given frame.

All studies include a single internal frame, the global reference frame, that provides a stationary, absolute location and orientation and cannot be modified. If alternative frames of reference are more convenient, you can create any number of additional reference frames relative to the global reference frame or any existing, independent reference frame.

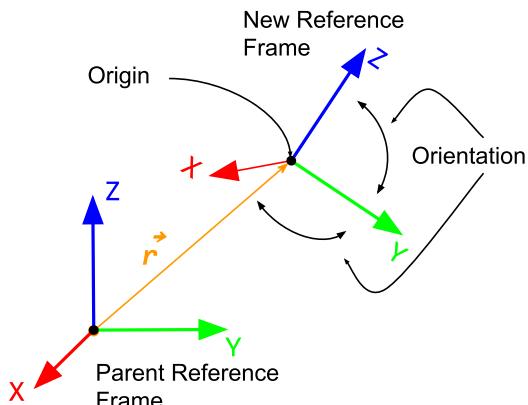
---

### Creating Reference Frames in AIM

---

## 1.4.5.1. Elements of a Reference Frame

A reference frame is defined relative to an existing reference frame by specifying the base location of its origin and orientation of axes. These may be further modified by accumulating a sequence of transformations, to bring the frame to its current location. The resulting origin location and axis orientations of the frame, relative to the Global Reference Frame, are reported in the Reference Frame data panel.



As there are many ways by which you might want to define a frame, the input requirements are as simple as possible without compromising your ability to create the frame you need.

## 1.4.5.1.1. Reference Frame Inputs

Table 1.4.5.1.1.1. Definition of reference frame inputs

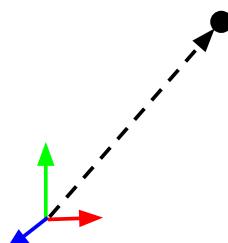
Input Field Name	Valid Inputs	Comments
Reference frame (a.k.a. <i>Parent frame</i> )	Any existing <i>independent</i> reference frame Default = "Global Reference Frame"	This is the frame of reference relative to which any user input coordinates and directions provided in the frame's base definition are defined, as well as transformations defined relative to the <i>parent frame</i> .  A frame can be thought of as being rigidly connected to its parent frame. It is this definition which is maintained—not the definition relative to the global frame. If the parent frame is modified, the dependent frames will transform with it.

Input Field Name	Valid Inputs	Comments
Preferred coordinate type	Cartesian, Cylindrical or Spherical  Default = Cartesian	The default coordinate type is a convenience option, having no effect on the frame itself.  When other AIM objects (such as physics conditions, points, planes, etc.) are defined relative to this frame, the preferred type determines the default coordinate type presented.
<b>Origin and Orientation</b>		The base origin and orientation are defined below.
Definition method	User defined, Coincident with parent  Default = User defined	The default <b>User defined</b> option leaves it up to you to define the origin and orientation.  The <b>Coincident with parent</b> option means the frame's origin is located at the parent frame's origin and the axes are aligned with the parent frame's axes. Transformations will still move the frame away from the parent frame.  This setting is useful if you want to create a second frame with specific transformations relative to the parent.
Origin		The origin is a point. The definition of the origin is exactly the same as defining a point, so the comments to follow only highlight some special considerations.
Define by	Entry, Location based  Default = Entry	When <b>Entry</b> is selected, the location of the origin is defined relative to the origin of the parent frame, in terms of the parent frame's coordinate systems. That is, the X, Y, Z coordinates specify offsets from the parent frame's origin along the parent frame's X, Y and Z axes.  If the parent system is changed or modified, the specified offset remains the same. That is, if the parent frame's origin is moved, the relative definition is preserved and the frame will move with the parent.  The Location based option allows you to specify the origin based on your geometry. The available calculation methods will depend on the objects selected. What's important to remember is that the selected location will no longer depend on the parent frame, unless the geometry itself is also transformed by a change in the parent or one of its other dependents. If the geometry is transformed based on a change to this frame or its dependents, a cyclic error will occur.
<b>Axis 1</b>		<b>Axis 1</b> allows you to specify the direction of one of the new frame's Cartesian axes.

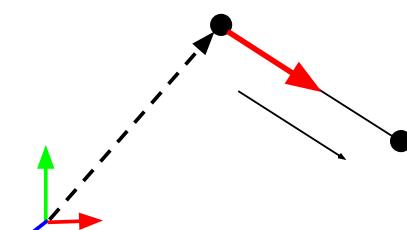
Input Field Name	Valid Inputs	Comments
Axis 1: Axis	X Axis, Y Axis, Z Axis	Local axis of the reference frame you are defining. The selected local axis will be exactly parallel to this direction. Use this option to specify the most important axis of your new frame. For example, if you plan to use cylindrical coordinates in this frame, you should define the direction of the local Z Axis in Axis 1.
Axis 1: Define by	Direction, Point Default = Direction	The <b>Direction</b> option allows you to specify the direction by entering a vector (relative to the parent frame, as with the origin), or by matching a direction based on selected geometry locations. As with the origin, the entry option retains the relative definition, while the <b>Location based</b> option will align (and update) with the geometry.
Axis 2	Automatic (on/off) Default = Automatic	Optional define the orientation of a second axis by specifying a direction or point which lies in the upper portion of the plane defined by Axis 1 and Axis 2. This can cause some confusion, so read carefully.

Table 1.4.5.1.1.2. Illustrative example of reference frame construction

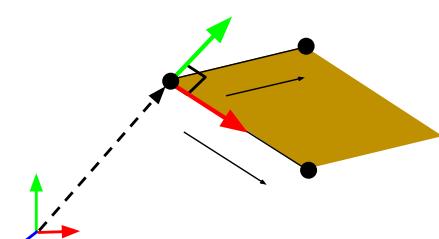
A single point defines the origin.



A second point or direction defines X axis (Axis 1).



To orient the Y axis, a third point or direction defines the XY plane. The Y axis lies in this plane, perpendicular to the X axis.



The Z axis is now oriented perpendicular to the XY plane.

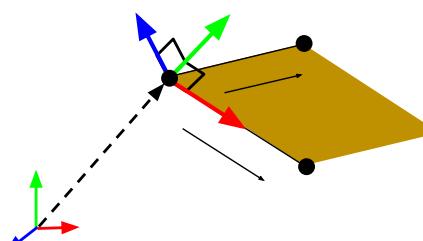


Table 1.4.5.1.1.3. Properties and definitions of transformations

Transformation	Comments
Translate	Translates the origin. The axes maintain the same orientation.
Rotate	Rotate the frame's a specified angle about the selected or defined rotation axis.
Reverse about X	Rotates the frame's axes 180 [degree] about the X axis. The X axis is unchanged, while the direction of the Y and Z axes is reversed.
Reverse about Y	Rotates the frame's axes 180 [degree] about the Y axis. The Y axis is unchanged, while the direction of the X and Z axes is reversed.
Reverse about Z	Rotates the frame's axes 180 [degree] about the Z axis. The Z axis is unchanged, while the direction of the X and Y axes are reversed.

## 1.4.5.2. Creating Reference Frames

A reference frame's final location and orientation are defined by its base origin and orientation (relative to its parent frame) and by any transformations that you apply.

To specify an independent reference frame:

1. In the Workflow view, create a Configuration task: right-click **Geometry** and select **Add Next > Model Configuration**.
2. From the **Configuration** task, select **Construction Geometry > Add > Reference Frame**. A Reference Frame data panel opens. You need to specify its position, orientation, and any transformations required.
3. For **Reference frame**, select an existing reference frame relative to which you wish to define your new frame. This is the *parent* frame, while the frame you are creating/editing is the *local* frame.
4. Under **Preferred coordinate type**, choose the type of coordinate system you would like to see by default when an object references this frame. For example, if you primarily need cylindrical coordinates in this frame, select **Cylindrical**. Note that all other coordinate types are always available.
5. Under **Origin and Orientation > Definition method**, indicate how you want to define the reference frame.
6. Under **Transformations**, you may further translate and rotate the frame from its base location. The transformations are *stacked*, meaning that they are applied in the order they are listed, from top to bottom. New transformations can be added manually by selecting **Add Transformation** at the bottom of the list, or interactively in the viewer if the **Define graphically** option is enabled. If the **Define graphically** option is enabled, the reference frame can be moved and oriented using mouse click-and-drag movements. Clicking and dragging an axis will translate the reference frame's origin, while dragging a ring around an axis will rotate the reference frame about that axis. When the mouse button is released, a new transform is added or modified. After releasing the mouse, the new transform is added to the stack or added to the last stack transform if possible. Sequential translations are independent, so if the last transform was a translation, subsequent interactive translations. As for rotations, sequential rotations about the same axis are independent and may be combined. Any translation or a rotation about a different axis will add a new transform to the stack. Transforms already in the stack may be deleted or modified manually. The transforms are always applied to the base location in the same order, so you may need to consider the effect of modifying an earlier transform.

One or more transforms can be added (stacked) onto the reference frame to alter the location and orientation. After the definition method has been applied, the stacked transforms are applied in order. There is no limit to the number of stacked transforms. By default, one stacked transform is provided with the type set to **None**, meaning it has no effect on the reference frame's location and orientation. If the type of the last transform is changed from **None** to anything else, another transform set to **None** is added at the end. A stacked transform can also be suppressed so that it has no effect on the reference frame's location and orientation, but the information is preserved so that the operation can be easily undone.

You have now defined a reference frame. Reference frames can be used elsewhere in your simulation, such as when coordinates are defined with respect to another frame of reference such as points, vectors, etc.

## 1.4.5.3. Coordinate Systems

A reference frame is essentially a local Cartesian coordinate system. However, in addition to the standard Cartesian coordinates, AIM also provides you with cylindrical and spherical coordinate systems in every frame. The additional cylindrical and spherical coordinate systems are related to the Cartesian coordinates in accordance with ISO 8000-2.

Coordinate systems are illustrated below with the definition of a position vector and its differential.



Figure Element	Definition
X, Y, Z	These are the axes of the underlying reference frame, relative to which coordinates are defined in the Cartesian, spherical and cylindrical coordinate systems.
$\vec{r}$	Position vector. From the origin of a reference frame, the position vector defines the location of a point relative to the reference frame's origin.
$x, y, z$	Distance (magnitude) from the origin measured parallel to the X, Y, and Z axis, respectively of the reference frame. Alternatively, can be viewed as the distance measured perpendicular to the YZ, ZX, and XY planes respectively.
$\rho$	(rho) Distance from the z axis (or the radius of a cylinder about which the z axis the point lies).
$\Phi$	(phi) Angular distance in the xy plane, starting from the positive x axis towards the positive y axis.
$\theta$	(theta) Angular distance from the z axis.
$r$	Distance from the origin. This is the magnitude of the position vector, $\vec{r}$
$e_{<..>}$	Direction vectors. These are unit vectors (for example, dimensionless) indicating the direction of greatest increase (for example, gradient) of each coordinate variable. Cartesian direction vectors are not position-dependent; they are parallel throughout a reference frame. However, direction vectors in cylindrical and spherical coordinate systems are position-dependent.
$e_x, e_y, e_z$	Unit direction vectors parallel to the positive x, y, and z axes, respectively. These may also be viewed as the directions normal or perpendicular to the yz, zx, and xy planes.
$e_\rho$	Unit vector directed towards the point starting from the nearest point on the z axis or normal to the surface of a cylinder about the z axis.
$e_\varphi$	Unit vector directed about the z axis, perpendicular to the $\rho Z$ plane
$e_\theta$	Unit vector directed about the origin and parallel to the -z axis. This can also be viewed as being normal to a cone created by revolving the position vector about the Z axis, with the positive direction of the normal pointing away from the Z axis.
$e_r$	Unit vector pointing from the origin to the point, or normal to the surface of a sphere about the origin.

The table below shows, in equation form, how a single position vector,  $\vec{r}$ , is described in terms of direction components and magnitudes in each coordinate system. It is also a useful reference for writing expressions in terms of these variables to describe physics conditions, etc.

Table 1.4.5.3.1. Description of direction components and magnitudes in each coordinate system of a single position vector

Coordinate System	Position Vector and its Differential	Coordinate Dimensions in SI Units
Cartesian $(x, y, z)$	$\vec{r} = x e_x + y e_y + z e_z$ $d\vec{r} = dx e_x + dy e_y + dz e_z$	directions: $(e_x, e_y, e_z) = ([], [], [])$ coordinates: $(x, y, z) = ([m], [m], [m])$

Coordinate System	Position Vector and its Differential	Coordinate Dimensions in SI Units
Cylindrical $(\rho, \varphi, z)$	$\vec{r} = \rho e_\rho + z e_z$ $d\vec{r} = d\rho e_\rho + \rho d\varphi e_\varphi + dz e_z$	directions: $(e_\rho, e_\varphi, e_z) = ([], [], [])$ coordinates: $(\rho, \varphi, z) = ([m], [rad], [m])$
Spherical $(r, \theta, \varphi)$	$\vec{r} = r e_r$ $d\vec{r} = dr e_r + r d\theta e_\theta + r \sin\theta d\varphi e_\varphi$	directions: $(e_\rho, e_\theta, e_z) = ([], [], [])$ coordinates: $(r, \theta, \varphi) = ([m], [rad], [rad])$

The following table provides a simple reference for relating variables in one system to another. As with the previous table, this provides a useful reference for writing expressions.

Table 1.4.5.3.2. Conversions of coordinate variables between Cartesian, cylindrical, and spherical coordinate systems

	Convert to Cartesian	Convert to Cylindrical	Convert to Spherical
Given a point defined in Cartesian coordinates	$x = x$ $y = y$ $z = z$	$\rho = \sqrt{x^2 + y^2}$ $\varphi = \arctan\left(\frac{y}{x}\right)$ $z = z$	$r = \sqrt{x^2 + y^2 + z^2}$ $\theta = \arccos\left(\frac{\sqrt{x^2 + y^2}}{\sqrt{x^2 + y^2 + z^2}}\right)$ $\varphi = \arctan\left(\frac{y}{x}\right)$
Given a point defined in Cylindrical coordinates	$x = \rho \cos(\varphi)$ $y = \rho \sin(\varphi)$ $z = z$	$\rho = \rho$ $\varphi = \varphi$ $z = z$	$r = \sqrt{\rho^2 + z^2}$ $\theta = \arctan\left(\frac{\rho}{z}\right)$ $\varphi = \varphi$
Given a point defined in Spherical coordinates	$x = r \sin(\theta) \cos(\varphi)$ $y = r \sin(\theta) \sin(\varphi)$ $z = r \cos(\theta)$	$\rho = r \sin(\theta)$ $\varphi = \varphi$ $z = r \cos(\theta)$	$r = r$ $\theta = \theta$ $\varphi = \varphi$

## 1.4.6. Using Construction Geometries

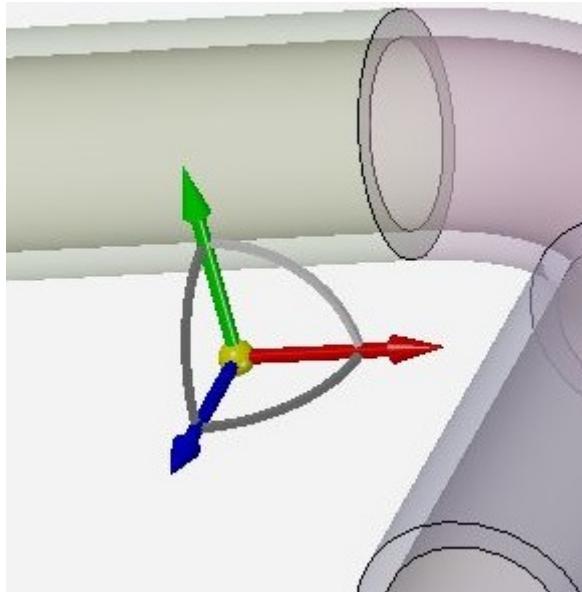
AIM supports the creation of basic construction geometries useful for defining various aspects of your simulations. These are found under **Auxiliary Definitions**:

- [Points](#) on page 80
- [Lines](#) on page 82
- [Planes](#) on page 83

## 1.4.6.1. Points

Points provide a method of abstraction between a model and a point in space that is external to the geometry in your simulation.

For fluid flow simulations, independent points can be used to identify flow volumes within a geometry that need to be extracted.

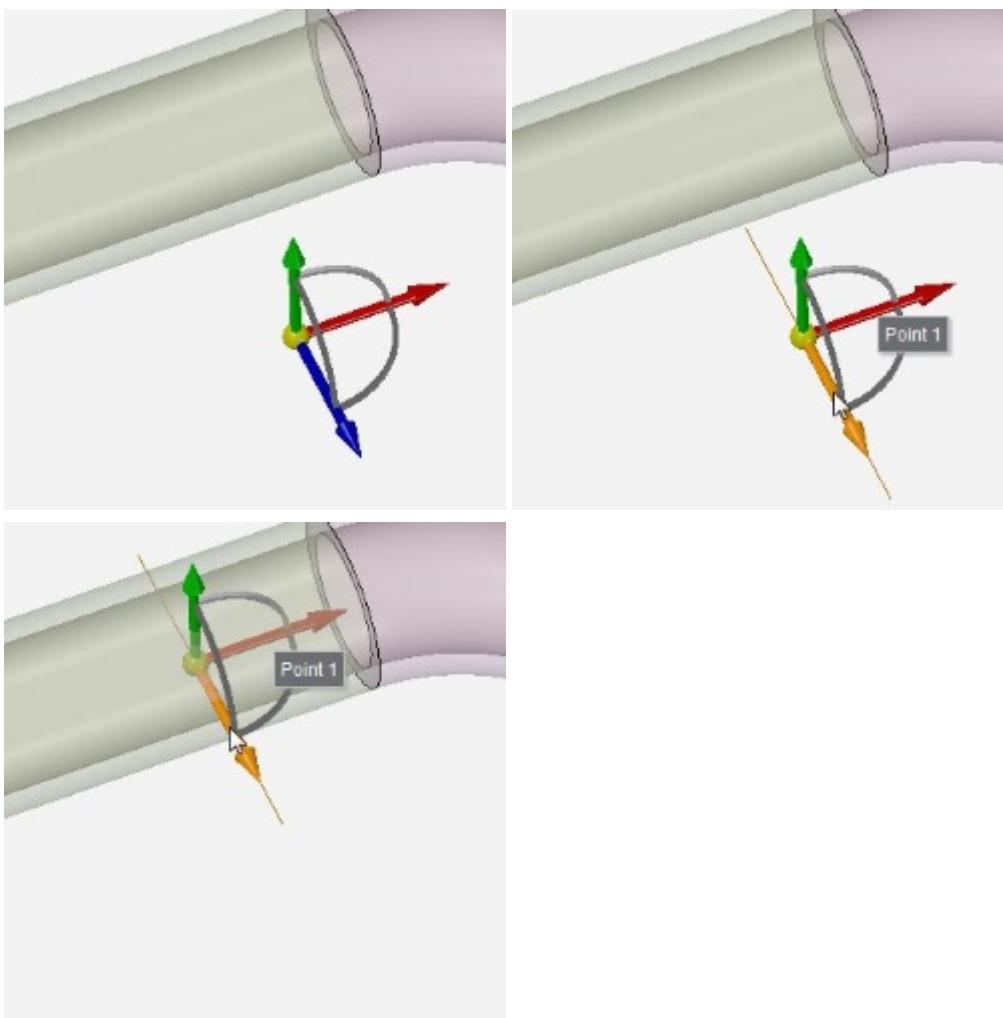


- [Specifying a Point Using the Point Control](#) on page 80
- [Specifying a Point by Geometric Selection](#) on page 81
- [Specifying a Point by Coordinates](#) on page 81
- [Manipulating a Geometric Entity](#) on page 72

### 1.4.6.1.1. Specifying a Point Using the Point Control

To specify a point graphically using the point control:

1. From **Auxiliary Definitions**, select **Construction Geometry > Add > Point**. You may have to zoom out to see the point.
2. Activate the point control by clicking on a directional arrow. The color will change to orange to indicate activation and you will see projection lines extending on either side.



3. Move the point along an axis by pulling the arrow along that axis.

### 1.4.6.1.2. Specifying a Point by Coordinates

To specify a point by coordinates:

1. From **Auxiliary Definitions**, select **Construction Geometry > Add > Point**. You may have to zoom out to see the point.
2. For **Reference frame**, leave the default **Global Reference Frame**, or click to select or define a reference frame.
3. For **Coordinate type**, select the type of coordinate system you want to use to define the point.
4. Specify the coordinates of the point.
  - For the Cartesian coordinate type, specify the coordinates in X, Y, and Z.
  - For the Cylindrical coordinate type, specify the coordinates in Radius, Azimuth angle, and Height.
  - For the Spherical coordinate type, specify the coordinates in Radius, Azimuth angle, and Polar angle.

You have now defined a point.

### 1.4.6.1.3. Specifying a Point by Geometric Selection

To specify a point by selection:

1. From **Auxiliary Definitions**, select **Construction Geometry > Add > Point**. You may have to zoom out to see the point.
2. For **Construction method**, select **Based on geometric selection**. The **Location** field is activated.
3. Select a topology and then click **Add selected entities** or **Replace with selected entities**.
4. For **Calculation method**, leave the default method of Centroid, or click to select the **Hit Point** method by which the point is calculated.

You have now defined a point.

**Note:** A geometry selection with a logical center point, such as a circular edge or cylindrical face, will use the center point of the geometry even if the geometry is not complete (for example, a circular arc).

## 1.4.6.2. Lines

A line can be added to visualize results; it is an abstract element separate from your geometry. A line is defined by start and end points; these points can be existing or you can create new ones. The position of the points can be changed by moving the point, or by modifying the coordinates directly in the **Point** or **Line** data panels.

Once defined, you can assign a variable to the line and view in the [Line Chart](#) on page 534 tab in the **View** panel.

The line can be moved and rotated interactively by dragging the point controls in the graphics view or by changing the coordinate values of the selected points in the **Line** data panel.

- [Adding and Positioning a Line Using the Point Control](#) on page 82
- [Adding and Positioning a Line by Coordinates](#) on page 82

### 1.4.6.2.1. Adding and Positioning a Line Using the Point Control

#### Geometry > Auxiliary Definition > Construction Geometry

To specify a line location by coordinates for the start and/or end point:

1. Select **Construction Geometry > Add > Line** or click on the **Create Line**  icon in the toolbar.
2. Select an existing **Start** point or **Create New** from the drop-downs.
3. Activate the point control by clicking on a directional arrow. The color will change to orange to indicate activation and you will see projection lines extending on either side.
4. Move the point along an axis by pulling the arrow along that axis.
5. Select an existing **End** point or **Create New** from the drop-downs. A line with a directional arrow will appear between the two points.

### 1.4.6.2.2. Adding and Positioning a Line by Coordinates

#### Geometry > Auxiliary Definition > Construction Geometry

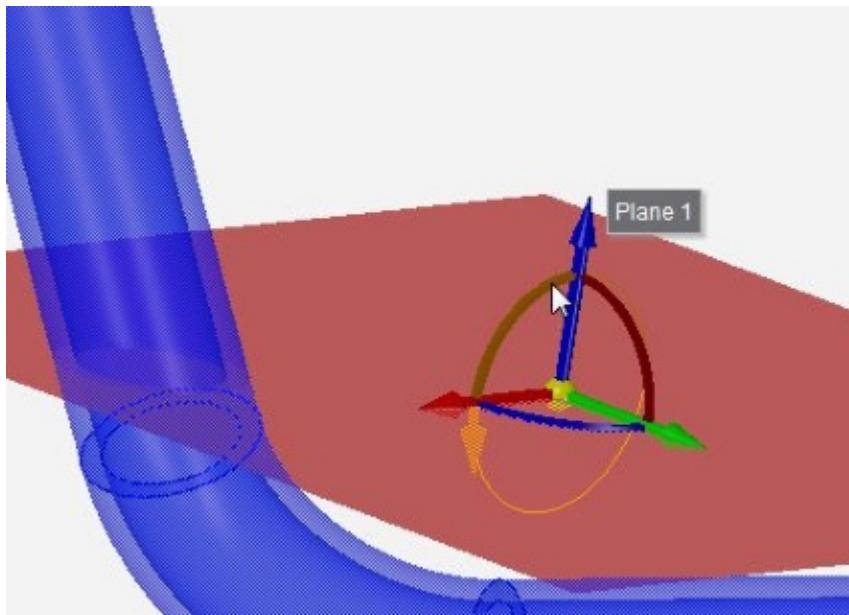
To specify a line location by coordinates for the start and/or end point:

1. Select **Construction Geometry > Add > Line** or click on the **Create Line**  icon in the toolbar.

2. Select an existing **Start** point or **End** point, or **Create New**, from the drop-downs.
3. Expand the point definition sections to directly edit the point.
4. Specify the new coordinates for the point.

## 1.4.6.3. Planes

Planes are abstract geometry constructs, separate from an imported geometry. A plane can be constructed and used to visualize the results of a simulation.



You may have to zoom out to see the plane. Position the plane by manipulating the plane control in the graphics view or through the Plane data panel.

- [Positioning a Plane Using the Plane Control](#) on page 83
- [Positioning a Plane Using the Data Panel](#) on page 84
- [Manipulating a Geometric Entity](#) on page 72

### 1.4.6.3.1. Positioning a Plane Using the Plane Control

Position the plane by manipulating the plane control in the graphics pane with the mouse or using a keyboard/mouse combination ([Manipulating a Geometric Entity](#) on page 72). You specify details of position and orientation according to your requirements.

To position the plane using the plane control:

1. From **Auxiliary Definitions**, select **Construction Geometry > Add > Plane**. *You may have to zoom out to see the plane.*
2. Activate the plane control by clicking on a directional arrow or circle. The color will change to orange to indicate activation.
  - move the plane along an axis by dragging the arrow along that axis or
  - rotate the plane about an axis by dragging the circle.

You have now defined a plane. Planes can be used elsewhere in any type of simulation. For example, planes can be used to display the results of your simulation.

## 1.4.6.3.2. Positioning a Plane Using the Data Panel

To position the plane using the data panel:

1. From **Auxiliary Definitions**, select **Construction Geometry > Add > Plane**. You may have to zoom out to see the plane.
2. For **Scoped bodies**, leave the default: AllBodies(), or select specific bodies. When you display results on your plane, results will appear on the portions of the plane that intersect the selected bodies.
3. Under **Construction method**, choose from the following:
  - **User specified**: exposes additional controls to specify the origin and orientation of the plane.
  - **Based on another plane**: exposes additional controls to specify a new plane from an existing plane.
  - **Based on a reference frame**: exposes additional controls to specify a new plane from defined reference frames.
4. Under **Transformation**, offset and rotational transformations can also be applied to the plane.

You have now defined a plane. Planes can be used elsewhere in any type of simulation. For example, planes can be used to display the results of your simulation.

## 1.4.7. Using Parameters and Design Points

AIM provides several means of extending your simulations using:

- *Parameters* - characteristics of a model or a simulation that you can quantify and vary to determine its effect on the results of the simulation.
- *Design points* - single sets of parameter values representing one design alternative.

For more information, see [Parameters and Design Points](#) on page 598.

## 1.4.8. Using Expressions

AIM provides a means of customizing your simulations using:

- *Expressions* - a string of characters used to define some quantity of interest that could be used for some purpose in an analysis.

For more information, see [Expressions](#) on page 622.

## 1.4.9. Defining Tabular Data

You can create a table to define certain material properties or boundary conditions. This is useful if you have property data that is dependent on a field variable (e.g. Temperature) and for time-dependent simulations.

If a field supports setting tabular data for more than one independent variable, you select the appropriate variables, then set the tabular data for each variable.

1. Click the  to the right of the property.



2. Select **Create tabular data** (). For fields supporting a single variable, a table appears.

If the field supports multiple variables, a list of those variables appears. Select the variables for which you want to define tabular data. The tables for the variables are listed below the field, under **Input for tables**, and the field becomes read-only. The table for the first independent variable appears.

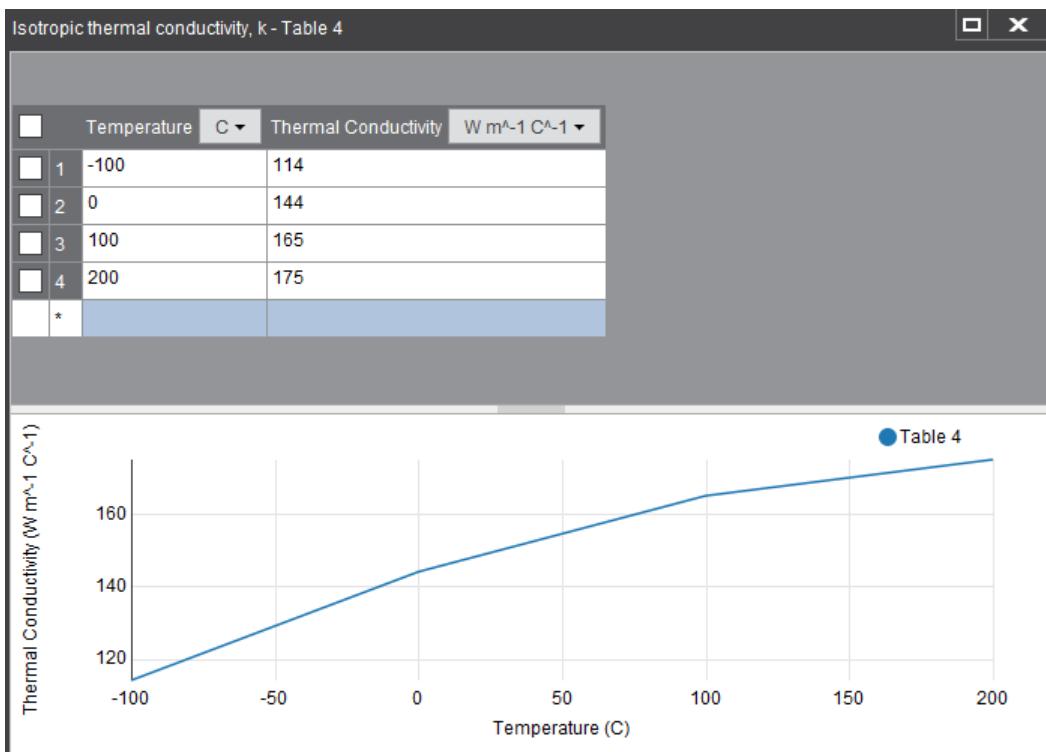
3. Fill out the rows of the table. You can add rows by filling in the empty bottom row of the table. You can

also delete rows by selecting the check box to the left of the row and clicking  at the top of the table.

In some cases, you can modify the units in each column using the dropdown box to the right of the column header. Any existing data in the table will automatically display in the chosen unit. The underlying data is not affected, only its display in the table.

	Temperature	C	Resistivity
1	20	K	
2	25	C	
3	30	R	
*		F	

As you enter data into the table, the chart below displays the resulting plot.



**Note:** The chart is in project units, which may differ from the selected unit in the table.

4. Click the at the top right of the window to close the table.
5. Repeat steps 2-4 as needed to set the tables for additional variables.

Once tabular data has been created for a property, you can edit the table by clicking the **Edit tabular data** button to the right of the property. For properties supporting multiple variables, you can remove one or more tables by clicking the **Edit tabular data** button, then de-selecting the independent variable you wish to remove.

Using tabular data in an expression is shown in [Tabular Data in Expressions](#) on page 85.

## 1.4.9.1. Tabular Data in Expressions

You can use tabular data in an expression to define properties as shown in following example.

Given the following tabular data:

Temperature [C]	Thermal Conductivity [W m^-1 C^-1]
100	114
200	144

You can use this table in an expression to define a material property in two ways:

Expression	Result
table(150[C])	This expression resolves to a constant value, interpolated from the table you have associated with the property. From the example, this will result in a value of 129 [W m^-1 C^-1].
table(Temperature)	Assuming Temperature is a valid field variable for this material property, this expression defines a non-constant material property (temperature dependent, in this case). The result will be evaluated during the solution based on the Temperature in the Physics Region.

## 1.5. Managing Your Project

To save your project, click **Home > File > Save**. The project is saved to the location specified by **Home > Tools > Options: Project Management > Default Folder for Permanent Files**.

When you choose your project's name, note that:

- If the project name contains characters that are not native to your operating system's locale setting, the project may fail to save and may corrupt the data.
- You must not use a percent sign (%) as a character in your project name--the project may fail to save.

The saved project is stored as a project file and corresponding folder.

You can view the files in the project using the Workbench Files view. Click the Project tab, then click **Home > View > Files**.

If you want to share or move your project, you need to archive both the project file and the project folder by using **Home > File > Archive**; see [Archiving Projects for Sharing or Preserving Your Work](#) on page 87 for details.

The project files are protected by AIM, and generally you should not modify them as that may corrupt the project. However, there are two exceptions:

- The `user_files` folder is where AIM stores the external files you import. See [Associating Other Files with the Project](#) on page 87 for details.
- The `session_files` folder contains a journal of the input that was used to create the study. You can replay this journal (and thus recreate the project) by browsing to it from the start page or **Home > File > Scripting > Run a script file**. However, you can also script changes to the journal to streamline and automate your workflow. See [Creating a Customized Workflow](#) on page 89 for details.

### 1.5.1. Associating Other Files with the Project

When you save your project ( **Home > File > Save**), the project file and project folder are saved to the location specified by **Home > Tools > Options: Project Management > Default Folder for Permanent Files**. Within the project folder there is a `user_files` folder that can contain:

- Any files (such as input files, referenced files, and so on) that you supply to a project. You are responsible for placing additional files (such as PowerPoint or Excel files, or other files from separate applications that are associated with this project) into *only* this folder.
- Any output (images, charts, and so on) generated by AIM that you want to have associated with the project. For example, the export of design point data to a CSV file produces a file that is written to this folder. See [Exporting Design Point Parameter Values to a CSV File](#) on page 615 for details.

AIM ensures that this folder is managed and archived appropriately with the rest of the project to prevent the risk of losing data.

### 1.5.2. Archiving Projects for Sharing or Preserving Your Work

If you want to preserve a project - perhaps to send it to a colleague or to ANSYS Technical Support - you need to create an **archive** (a zipped file that contains the project file and the associated directories):

1. Choose **Home > File > Archive**.
2. In the **Save Archive** dialog box:
  - a) Navigate to the folder where you want to save the file.
  - b) Name the archive.
  - c) Select the archive type: **Workbench Project Archive (.wbpz)** or a **Zip (.zip / .tar.gz)** file.

The Project Management tab of the **AIM Options** dialog allows you to specify the compression level for `.wbpz` archives. For more information, see [.](#)

3. Specify which optional items you want to archive:
  - Result/solution and retained design point files
  - Imported files external to the project directory

When you import an external file, AIM archives it by adding it to the `user_files` folder. However, if the external file refers to other files (for example, as when a CAD assembly is linked to the CAD parts), the system is not able to place all of the necessary referenced files in the `user_files` folder. In order to make the referenced files part of the archive, you must copy them manually into the `user_files` folder.

4. Click **Save**.
  - Items in the `user_files` folder.
  5. Click **Archive**.
- To restore an archived file, select **Open Other File** on the **Start** page, or **Home > File > Open** and select **Archive Files** from the file type dropdown. Navigate to the archived project and click **Open**. You are prompted for a project path to a folder where the archive will be extracted, and then the project opens.

### 1.5.3. Accessing Your Fluent Project in AIM

You can access the Fluent Schematic and Systems Toolbox from the Project tab. This enables you to:

- Work with parameters and design points as you would in Fluent (similar functionality is available in [AIM's Design Points Dashboard](#))
- Use DesignXplorer (a separate, licensed product)

The Project tab can contain only one Study system.

---

## 1.6. Working with Project Reports

AIM project reports reflect the layout of the Workbench **Project Schematic**, with sections for global project information, analysis system information, and system cell information.

- [Configuring Project Reports to Open Automatically on page 88](#)
- [Generating a Project Report on page 88](#)
- [Editing a Project Report on page 88](#)

### 1.6.1. Generating a Project Report

To generate a report of the current state of the project:

1. Select  **Home > File > Export Report**.
2. In the dialog box that appears, specify the name and location for the report file, then click **Save**.

Depending on the setting of  **Home > Tools > Options > Project Reporting**, either the report will open in your default browser or you will have to go to the directory where you stored the report and open it manually. For details, see [Configuring Project Reports to Open Automatically on page 88](#). If necessary, you can edit the report as described in [Editing a Project Report on page 88](#).

**Note:** The main report page appears using the settings on the machine where the Export Report is created; the design point report content is saved using the settings on the machine where the design point was executed.

This means that if you open a project that has multiple solved design points and was created by someone who worked in a different language, used a different unit system, or used a different decimal separator, the main report page of the report you export will display using the settings of your machine, but the linked design point report pages will display the settings from the original machine.

### 1.6.2. Configuring Project Reports to Open Automatically

To configure project reports to open automatically:

1. Open  **Home > Tools > Options > Project Reporting**.
2. Select the **After exporting report, automatically open in default browser** check box.
3. Click **OK**.

Upon generation of a report, it will immediately open in your default browser.

To open the report manually, navigate to the project's `user_files` directory and open the `projectname_report.html` file.

### 1.6.3. Editing a Project Report

Once you have generated a project report, you can edit its contents as needed.

1. Open the report file with an HTML-adapted editor by right-clicking the file and selecting the **Open with** menu option.
2. Edit the report contents and formatting as needed.
3. Save the file to a new location (to prevent it being overwritten the next time you generate a report).

## 1.7. Creating a Customized Workflow

Once you are familiar with using AIM to create simulations, you can optimize your own workflows in several different ways, depending on your needs.

- To rerun a simulation, or part of the definition, and then manually modify input if needed:

**Use scripting capabilities.** In AIM, you can record the actions that you perform in a script that you can edit and rerun as desired. [Learn more about using scripts](#). If you find yourself making frequent changes and reuse a script extensively, you can convert it to an app (custom template, guided setup, or guided simulation) for improved ease of use.

- To customize how AIM tasks are initially set up and defined:

**Create a custom template app.** Templates automate the definition of the simulation, prompting you for necessary inputs. Custom templates are available on the start page or when adding a new simulation on the **Study** panel. Learn more about creating custom template apps.

- To customize how AIM objects are defined and optimize repetitive simulation definition details:

**Create a guided setup app.** Similar to how a template or guided simulation app can automate or streamline the definition of an entire simulation, a guided setup optimizes the definition of a currently selected object based on its type and geometry. Guided setups appear in the **Guide Me** context (right-click) menu when a compatible object and geometry entity are selected. Learn more about creating guided setups.

- To define an end-to-end custom workflow:

**Create a guided simulation app.** Guided simulations constrain user inputs and results to only those that are defined in the app, so users are directed through their simulation. Guided simulation apps are available on the start page or when adding a new guided simulation on the **Study** panel. Learn more about creating guided simulation apps.

Learn more about the **Extension Manager** to [enable custom workflows](#).

### 1.7.1. Using Scripting Capabilities

When you set up and solve a simulation, your actions are recorded in a script and saved (by default) to a journal file (.wbjn) within your AIM project.

- To run a pre-existing journal file, select it by browsing to it from the start page or select **File > Scripting > Run a script file**. Then, select the journal file to load into AIM, where you can continue your analysis.
- To create a journal file containing a script with a set of specific commands, select **File > Scripting > Record journal**.

See [ANSYS Discovery AIM Journaling and Scripting Overview](#) on page 671 for more information about the format of journal files and how to edit them.

### 1.7.2. Enabling Customization

To install a custom app, workbench extension, or XML file, click the + on the start page (top right) and select **Install**. Browse to the location and choose the file to add the customization to the start page.

To make AIM aware of folders containing customized files:

- Click  **Home > Tools > Options** in the Extensions panel and include the folder containing the files.
- Use the **Extension Manager** to enable all custom workflows that you want available in a project. You start the **Extension Manager** from the **ACT Start Page**, which is accessed by selecting **File > Extensions > ACT Start Page**. To learn how to install, load, and unload extensions, see .

Once loaded, custom workflows are shown in AIM as follows:

- Template apps that create custom simulations can be selected from the start page.
- Guided setup app display as entries in the **Guide Me** context (right-click) menu when a compatible object and geometry entity are selected. This menu is present only if the object and geometry selection to which you've navigated matches the object or task and geometry selection defined in the extension.
- Guided simulations can be selected from the start page and are differentiated from templates by an icon on the bottom left of the tile.
- API-driven customizations display for features with which they are associated. For example, added structural boundary conditions display in every menu for adding a boundary condition and as a new subcategory. Similarly, an added solver displays as an option that can be selected.



# Chapter 2: Geometry

---

A simulation requires a geometry that represents a physical object that you want to apply to your engineering simulation. AIM enables you to import various geometric and CAD file types. The geometry you provide will be the basis for the subsequent assignment of properties and tasks used to define the physics in the simulation.

You can use AIM's [Model Editing](#) to create, import, and modify geometry. Alternatively, you can directly [import](#) one or more geometry files and optionally [configure](#) various aspects of the simulation.

You can import a single geometry that represents both fluid and structural bodies, and then use multiple physics tasks--one to simulate a fluid flow region, and the other to simulate a structural region.

**Note:** From 18.2 on, the electromagnetics solver only supports ASCII part names in geometry.

For beams, you must define cross-section properties. For more information, see [Defining the Cross Section Properties for Beams](#).

---

## 2.1. Model Editing

The Geometry Modeler and Geometry Modeling tasks in ANSYS Discovery AIM are based on ANSYS Discovery SpaceClaim technology.

You can access the Geometry Modeling Help from within the Geometry Modeler by clicking the question mark in the upper right-hand area of the window, or by hovering over the tool of interest and pressing F1.

# A Guide to Using Geometry Modeling in ANSYS Discovery AIM

A delay is expected when the Geometry Modeling task is created.

**Journaling and Batch Execution Support** Operations performed while editing geometry are not journaled.

**Scripting Support** Geometry editing operations can be automated via scripting.

You can execute modeling script commands by using either the Geometry Modeler script editor or AIM's standard Python script execution capabilities:

- To use the Geometry Modeler script editor: from within the Geometry Modeler, navigate to **Model > Show Script Editor**. For more information about the script editor, refer to **Customizing Geometry Modeling > Scripting** in the Geometry Modeling Help. You can access the Geometry Modeling Help from within the Geometry Modeler by clicking the question mark in the upper right-hand area of the window.
- To use AIM's standard capabilities: from within AIM, navigate to either **File > Scripting > Open Command Window** or **File > Scripting > Run Script File....**

When you execute modeling script commands using AIM's standard capabilities, the commands must be preceded by the text `with ModelingTransaction("label") :` and indented, where `label` is required and identifies the set of commands in the Geometry Modeler's undo/redo list. For example:

```
with ModelingTransaction("CreateCylinder") :
    CylinderBody.Create(
        Point.Create(0,0,0),
        Point.Create(0.01,0,0),
        Point.Create(0.01,0,0.005))
```

You can paste Geometry Modeler commands directly into AIM's command window after entering `with ModelingTransaction("label") :` into the window, but the commands will not be executed until you press **Enter**. If you paste a sequence of several commands, you may need to press **Enter** twice (once to complete your last command, and once on a blank line to execute the sequence).

Modeling script commands are documented in `<drive>:\Program Files\ANSYS Inc\<ansys-version>\scdm\SpaceClaim.Api.<Vnn>\API_Scripting_Class_Library.chm`, where `<drive>` is your installation drive, `<ansys-version>` is the ANSYS product version, and `<Vnn>` is the SpaceClaim API version.

<b>Detailing Tools Support</b>	Tools for detailing are located on the Geometry Modeler's <b>Detail</b> tab. The tools from the SpaceClaim Font group, as well as several tools from the Annotation group, are available.
<b>Shared Topology Support</b>	You can model Shared Topology features when using the Geometry Modeler in AIM. When doing so, keep the following recommendations and limitations in mind. For additional details about Shared Topology, refer to the section <b>Workbench &gt; Shared Topology</b> in the Geometry Modeling Help. <ul style="list-style-type: none"> <li>AIM supports the Share tool for basic and selective sharing operations. You can use the Share tool in conjunction with the Exclude Problem tool guide to exclude unwanted sharing before clicking Complete.</li> <li>AIM supports the Unshare tool for basic unsharing operations. When using the Unshare tool, Unshare must be applied on all bodies that are connected.</li> <li>If you intend to perform a series of operations in AIM that will change the model's topology and one of the operations involves the use of the Share tool, ANSYS recommends that you use the Share tool for the last operation in the series. This is because once topology is shared, no additional modeling operations that might alter the shared topology are supported (including basic sharing, selective sharing, and basic unsharing). If you must edit a model after applying shared topology, ANSYS recommends that you unshare, perform the edits, and then re-share.</li> </ul>
<b>Cannot Create Driving Dimensions for Adjacent Faces</b>	When you release the mouse button after pulling a face connected to an adjacent face which has a driving dimension (or ruler dimension or parameter), the pop-up tool (used to enter the dimension value and create a driving dimension for it) disappears immediately. You can still specify an exact value for the dimension by pressing the keyboard space bar before releasing the mouse button.
<b>Inserted Models Not Archived</b>	If you insert a model while creating/editing geometry (via <b>Assembly &gt; File</b> ), the imported model file(s) will not be included when you archive the AIM project, unless they are internalized before archiving. To internalize an imported model, right-click the model in the Structure tree and choose <b>Source &gt; Internalize All</b> .
<b>Design Point Update</b>	When dimensional geometry parameters are imported into AIM from SpaceClaim or a bi-directional CAD interface, design point updates may fail when dependencies exist between the parameters or when the parametric update in the CAD system or SpaceClaim cannot be realized with the parameter values specified in AIM.
	Attempting to perform design point updates when the units settings used in SpaceClaim and AIM differ may also fail. For example, if a SpaceClaim .scdoc file that was saved with its angular units set to degrees

is opened in an AIM session in which angular units is set to radians, attempting to update an angular parameter on the Design Points Dashboard may fail. In this case, open the .scdoc file in SpaceClaim, select **File > SpaceClaim Options > Units**, select **Radians** in the **Angle** drop-down menu, and click **OK**. Then re-open the .scdoc file in AIM.

<b>CAD Parameters Not Imported</b>	External CAD parameters are not imported into the Geometry Modeling environment. If you need to work with external CAD parameters in AIM, use a Data Import task instead of a Geometry Modeling task. To do so, either enable <b>Connect to active CAD session</b> or disable <b>Allow geometry modeling</b> when you make your selections in templates.
<b>Geometry Parameters</b>	<p>Driving Dimensions (also known as "Ruler Dimensions") defined in the Geometry Modeler are automatically synchronized with Geometry Parameters in AIM.</p> <ul style="list-style-type: none"> <li>While modifying geometry with the Pull or Move tool, you can create a Driving Dimension by clicking on the blue <b>[P]</b> button in the Graphics window. The button subsequently becomes gray, indicating that it cannot be clicked again. If you wish to delete the Driving Dimension, open the "Groups" panel, select the item under "Driving Dimensions" (by default, named "Group1", etc.), and click the <b>Delete</b> button in the panel.</li> <li>Driving Dimensions are synchronized with AIM Geometry Parameters when the Geometry Modeler is closed. You cannot switch to the Project or Parameter Set tab while editing geometry. To see Geometry Parameter changes in the Design Points Dashboard or Parameter Set tables, first close the Geometry Modeler.</li> <li>Creating new Geometry Parameters causes downstream tasks to become out-of-date. If you create a new Driving Dimension while editing the geometry, and then close the Geometry Modeler, a new Geometry Parameter is created and downstream AIM tasks must be updated.</li> <li>Driving Dimensions (Geometry Parameters) are deleted when the geometry associated with them is deleted. This is to be expected when directly deleting geometry. However, some Driving Dimension values can result in their geometry entities being indirectly deleted. For example, if the radius of a fillet, round, or cylinder is set to zero, the fillet/round/cylinder is deleted, along with its Driving Dimension and Geometry Parameter. Subsequent design point updates will fail if they expect that Geometry Parameter to exist. Try to avoid assigning Geometry Parameter values that would eliminate their associated geometry features. If such simulations are necessary, use multiple geometry models, some with features present and others without them.</li> </ul>
<b>Supported and Unsupported File Formats</b>	The supported and unsupported file formats for Geometry Modeling are the same as those for SpaceClaim Direct Modeler (SCDM). For details, refer to <b>File Operations &gt; Importing and exporting</b> in the Geometry Modeling Help. You can access the Geometry Modeling Help from within the Geometry Modeler by clicking the question mark in the upper right-hand area of the window.
<b>Parameters for Extracted Volumes Produce Unexpected Results</b>	Extracted volumes are regenerated when the surrounding geometry changes. If a geometry parameter (driving/ruler dimension) is defined on the extracted volume or on any of its faces or edges, redundant extracted volumes may result when that parameter's value is changed. Define your parameters on the surrounding geometry rather than the extracted volumes to avoid this.
<b>Selection Sets and Reference Frames Duplicated when Changed Downstream</b>	<p>Creating Selection Sets and Reference Frames in the embedded Geometry Modeling environment and subsequently modifying them in other parts of AIM results in duplicates with the same names when the Geometry task is updated.</p> <p>For example, if you create a Selection Set (Named Selection) in Geometry Modeling named "Group1" containing two surfaces, close Geometry Modeling, add another surface to it in the AIM data panel, and update the Geometry task, then a second "Group1" will be created. In the list of Selection Sets, the first "Group1" will have three surfaces, and the second "Group1" will have two surfaces.</p>

To avoid such duplication, always modify each Selection Set or Reference Frame in the same environment in which you created it. If the duplicates are intended, you can rename them to prevent confusion.

<b>Face and Edge Colors and Textures Not Transferred Downstream</b>	Only body colors are transferred from the embedded Geometry Modeling environment to downstream tasks. Face and edge colors and textures can be selected and viewed only in the Geometry Modeling environment.
<b>Suppressing and Activating Objects for Physics</b>	You can choose to suppress an object so that it will not be passed to downstream physics tasks. To do so, right-click an object in the Structure tree (or select multiple objects in the Structure tree and right-click), and then select <b>Suppress for Physics</b> from the context menu. Suppressed objects are still visible in the Design window. To reactivate an object, select it in the tree and then select <b>Activate for Physics</b> from the context menu. For related information, refer to the description of the Structure tree in the Geometry Modeling Help ( <b>Getting Started &gt; The Geometry Modeling interface &gt; Structure tree</b> ). You can access the Geometry Modeling Help from within the Geometry Modeler by clicking the question mark in the upper right-hand area of the window.
<b>Non-ASCII Part Names</b>	In Release 18.2 and later, the electromagnetics solver only supports ASCII part names in geometry.

## 2.2. Import

Import tasks are used to import one or more geometric or CAD file types for use in your simulation.

- [Usage Notes for Import Tasks](#) on page 95
- [Geometry Import Source](#) on page 95

**Note:** From 18.2 on, the electromagnetics solver only supports ASCII part names in geometry.

### 2.2.1. Usage Notes for Import Tasks

The Import task allows the selection and import of geometry files through the addition of one or more defined geometry import source objects. Each geometry import source object represents a single geometry source including its location and preferences.

**Note:** From 18.2 on, the electromagnetics solver only supports ASCII part names in geometry.

Additional geometry import sources can be added through the following actions:

<b>Browse Source</b>	Creates an empty geometry import source object that requires the file path to be set to an on-disk file location through use of a file browser dialog.
<b>Active CAD Attach</b>	Creates an empty geometry import source object that requires the import source location to be selected from the list of <b>Active CAD Attach</b> document sources.

Updating the **Geometry Import Task** executes the import for any geometry import sources that were not previously imported by the Task. Any previously imported geometry import sources that are not up-to-date are updated.

### 2.2.2. Geometry Import Source

The Geometry Import Source object represents a single geometry source including its location and specific preferences. Once a valid import source has been selected, the Import Type property displays the import source's type.

Source Selection is controlled through the following options defined by the **Source Selection Type** property:

<b>Browse Source</b>	Assigns the source <b>Location</b> property to an on-disk file location through the use of a file browser dialog.
<b>Active CAD Attach</b>	Assigns the source <b>Location</b> property to an Active CAD Attach document source selected from the list of the Active CAD Attach document sources.
The geometry import source can be Imported, Updated, and Synchronized according to its status.	
<b>Import</b>	Available prior to executing the source Import for the first time.
<b>Update</b>	Available once the source Import has been executed and a dependent property invalidates the source's status. Executing this action <i>pushes</i> any changed Dimensions to the source's originating application/CAD System.
<b>Synchronize</b>	Available once the source Import has been executed and the import source's status is up-to-date. Executing this action <i>pulls</i> all geometry and Dimension changes from the source's originating application/CAD System to the Import task.

## 2.2.2.1. Specifying a Geometry Import Source by Browsing

To specify a **Geometry Import Source** by browsing for the source:

1. For **Sources**, select **Add > Geometry Import Source**.
2. Under **Source selection type** the option will be set to **Browse Source**.
3. Select the "... file browse button for the **Location** property to launch the file browser dialog.
4. In the file browser dialog, set the file filter to the desired file format type(s).
5. Select a geometry file.
6. **Update** the Geometry Import Source object or Import task object to complete the import and generate the Model graphics.

## 2.2.2.2. Specifying a Geometry Import Source by Active CAD Attach

To specify a **Geometry Import Source** by **Active CAD Attach**:

1. For **Sources**, select **Add > Active CAD Attach**.
2. Under **Source selection type** the option will be set to **Active CAD Attach**.
3. Select an active/running CAD document from the **Location** property.
4. **Update** the Geometry Import Source object or Import task object to complete the import and generate the Model graphics.

For more information about CATIA V5 (CADNexus CAPRI CAE Gateway), see CATIA V5 Associative Geometry Interface (\*.CATPart, \*.CATProduct) in the CAD Integration User's Guide.

## 2.2.2.3. Geometry Preferences

Each Geometry Import Source provides a set of import preferences that control how the originating applications data is processed during import into AIM.

Changing a Geometry Preference will invalidate the Geometry Import Source and require an Update of either the source or the Import task. The changed preferences are applied to the originating applications data when the update is processed.

Option	Definition
<b>Import solid bodies</b>	Enables or disables the import of solid bodies. The default is enabled.
<b>Import surface bodies</b>	Enables or disables the import of surface bodies. The default is enabled.
<b>Import line bodies</b>	Enables or disables the import of line bodies. The default is disabled.
 <b>Mixed import resolution</b>	Allows parts of mixed dimension to be imported as components of assemblies that have parts of different dimensions. The following options control what is imported when there are bodies of mixed dimension in a multibody part:  <b>None</b> Nothing is imported. This is the default. <b>Solid</b> Only solids are imported. <b>Sheet</b> Only surfaces are imported. <b>Solid and Sheet</b> Only solids and surfaces are imported.
<b>Import dimensions</b>	Enables dimension processing. Dimension processing can slow down overall import performance.  <b>None</b> No parameters from the CAD source are processed. <b>Independent</b> Parameters from the CAD source which are independent are processed. These are input parameters which can be modified and promoted as design parameters. This is the default. <b>All</b> Both independent and dependent parameters are processed from the CAD source. Independent parameters can be modified and promoted as design parameters. Dependent parameters are read-only and cannot be modified or promoted.
<b>Dimension key</b>	Allows the specification of a key that is used to filter processed CAD system parameters during import. The key must be present at the beginning or the end of a CAD parameter's name to be valid for import. You may specify multiple prefixes/suffixes with each value separated by a semicolon. If the value is empty, all dimensions are imported. The default key is <b>ANS;DS</b> .
<b>Import selection sets</b>	Enables or disables processing of CAD system Named Selections that result in the creation of Selection Sets. The default is disabled.
<b>Selection set key</b>	Allows the specification of a key that is used to filter processed CAD system Named Selections during import. The key must be present at the beginning or end of a CAD Named Selection's name to be valid for import. You may specify multiple prefixes/suffixes with each value separated by a semicolon. If the value is empty, all Selection Sets are imported. The default key is <b>NS</b> .
 <b>Import coordinate systems</b>	Specifies whether coordinate systems created in the CAD system are imported as Reference Frame objects. The default is disabled.
 <b>Coordinate system key</b>	Allows specification of a key that is used to filter CAD system coordinate systems during import. You may specify multiple prefixes (including exclusion prefixes) with each value separated by a semicolon. The value is empty by default, which imports all CAD system coordinate systems. Geometry import preference keys (parameter, attribute, named selection, and coordinate system) all support the negation (-) option. You may enter multiple negation values each with their own negation (-) flag. Sample preference values include (DS;ANS;-ANS_X;-DS_Y)

Option	Definition
	and (-COLOR;-LAYER). Parameter keys are compared against the beginning and end of the parameter name while the others are compared only against the beginning of the name.
<input checked="" type="checkbox"/> <b>Process associativity</b>	Indicates whether action should be taken to allow associativity. Associativity processing can have a negative impact on import performance. The default is enabled.
<input checked="" type="checkbox"/> <b>Import using instances</b>	Determines whether a geometry import's part instances are recognized during processing to produce faster import times and allow smaller database sizes. The default is enabled.
<input checked="" type="checkbox"/> <b>Reader mode saves updated files</b>	Enables or disables saving of the internal part files generated from the geometry import during the import/update action. The default is disabled.
<input checked="" type="checkbox"/> <b>Enclosure and symmetry processing</b>	Enables or disables the processing of enclosure and symmetry CAD system Named Selections. The default is enabled.
<input checked="" type="checkbox"/> <b>Decompose disjoint geometry</b>	Enables or disables the decomposition of disjoint geometry into separate geometry entities. The default is enabled.
<input checked="" type="checkbox"/> <b>Clean Geometry On Import</b>	Remove unwanted features when importing geometry. The default is disabled.
<input checked="" type="checkbox"/> <b>Stitch Surface On Import</b>	Select the behavior for joining surfaces when importing geometry. The default is None.  The options are: <ul style="list-style-type: none"> <li>• <b>None:</b> Do not stitch surfaces on import.</li> <li>• <b>Program Tolerance:</b> Stitch surfaces on import using an internal algorithm for determining maximum stitch tolerance.</li> <li>• <b>User Tolerance:</b> Stitch surfaces on import using a user-defined <b>Stitch Tolerance</b>.</li> </ul> <b>Note:</b> Exercise caution when using this property. It is used for sewing neighboring faces together and for some healing operations. Too small a value will leave many unwanted gaps, while too large a value can end up making some faces disappear, and can also lead to unwanted gaps. A large tolerance value can also cause future modeling operations to fail.

## 2.2.2.4. Geometry Dimensions

### Using the Import Geometry Task

Any CAD System Parameters imported as AIM Dimensions using the **Import Dimensions** and **Dimension Key** Geometry Preferences are shown from the Geometry Import Source object. The imported Dimensions can be found in a separate **Dimensions** group along-side all other Geometry Import Source properties.

Geometry parameters containing a period in their names will not be imported. Only parameters with unique names formed of alphanumerical and the underscore characters are permitted. The parameters in the geometry file must be renamed and then they can be re-imported.

When dimensional geometry parameters are imported into AIM from SpaceClaim or a bi-directional CAD interface, design point updates may fail when dependencies exist between the parameters or when the parametric update in the CAD system or SpaceClaim cannot be realized with the parameter values specified in AIM.

Negative dimension values can invert the direction vector of SpaceClaim operations that they are associated with; this change is applied to the current and subsequent design point updates. As a result, when a Workbench input parameter is used as a driving dimension for a SpaceClaim geometry, negative dimension values may result in unexpected geometric changes.

Modification of an imported Dimension will invalidate the Geometry Input Source and require an Update of either the source or the Import task. The changed dimension values are pushed to the geometry import's originating application, modifying the geometry. The Geometry Import Source's model is then updated with the modified data.

For fluid flow simulations, independent points can be used to identify flow volumes within a geometry that need to be extracted.

## Using the Geometry Modeling Task

In the Geometry Modeling data panel and parameter tables, all driving dimensions are displayed in meters (for lengths) and radians (for angles), regardless of the units specified when creating them. For example, if you create a driving dimension of 10mm, it will be displayed as 0.01m in the Geometry Modeling data panel when you return from editing the geometry.

### 2.3. Configuration

Configuration objects and controls modify part components to produce a new part configuration. Each control object represents a single modification; multiple configure controls can be added to produce the desired output configuration.

You can add:

- [Suppress Controls](#)
- [Move/Rotate Controls](#)
- [Capping Surfaces](#)
- [Boxes](#)

#### 2.3.1. Suppress Controls

Suppress Control allows the removal of portions of the model for all subsequent tasks.

The Suppress Control location field can be composed of selection sets or specific topological selections. The parent part is identified for each selection item and the suppression is applied at the part level.

Selection of a body in a multi-body part will result in the suppression of all child bodies of the part. If more granular control over the suppression of bodies in a multi-body part is needed, modify the part/body relationships in the source import geometry file prior to applying the Suppress Control.

- [Creating a Suppress Control](#)

#### 2.3.1.1. Creating a Suppress Control

To create a Suppress Control:

1. For **Configure Controls**, select **Add > Suppress Control**
2. Assign the topological components to be suppressed

3. Update the **Suppress Control** by Updating the **Configuration task**
4. Parent parts for the selected topology are suppressed from the output configuration

### 2.3.2. Move/Rotate Controls

The Move/Rotate Control object allows the translation and rotation of a part component's origin within the output configuration. The translation and rotation transformation components are obtained from the control's *reference frame* property.

Part selections are obtained from traversing the topological hierarchy of each selected entity until their parent part is encountered.

Updating the Configuration task results in the replacement of each part component's origin transformation with the reference frame's transformation within the output configuration.

- [Creating a Move/Rotate Control](#)

## 2.3.2.1. Creating a Move/Rotate Control

To create a Move/Rotate Control:

1. For **Configure Controls**, select **Next Step > Move/Rotate Control**
2. Assign the topological components to be moved/rotated
3. Assign or create a **Reference Frame** specifying the replacement origin transformation for the selected parts
4. Update the **Move/Rotate Control** by Updating the **Configuration task**
5. Parent parts for the selected topology have their origin transformations replaced with the provided **Reference Frame** transformation within the output configuration

### 2.3.3. Capping Surfaces

Capping surfaces are an artificial geometric construct, separate from the imported geometry, that you can apply to the geometry in order to close off one or more openings. Capping surfaces are often required for extracting a flow volume for fluid flow simulations.

- [Creating a Capping Surface](#)

## 2.3.3.1. Creating Capping Surfaces

To specify an independent capping surface:

1. **Create Selection Sets** is enabled by default; this option is necessary to properly set up physics conditions in later tasks. In case of multiple openings in a geometry, in addition to creation of selection sets scoped to all openings, a separate selection set for each opening is created.
2. Under **Selection method**, choose from **Faces**, **Edges**, **Edge loops** or **Vertices** to expose additional controls to specify details for the selection technique. A construction algorithm exposes various algorithms available for constructing a capping surface which are usually hidden by a filter. Note that the order of the vertex selection can affect how the capping surface is generated. The vertices should be selected in order to define the outer boundary of the surface. For each method, specify the construction algorithm:
  - **Delaunay**: constructs capping surface using Delaunay triangulation
  - **Fill Hole**: constructs capping surface using hole filling algorithms
  - **Convex Hull**: constructs capping surface using convex envelope generated from selected loops
  - **Automatic**: software chooses the construction algorithm
3. After selecting a construction algorithm (or keeping the default: Automatic), add/replace any particular faces, edges, edge loop or vertices selections.

To cap multiple openings in a geometry, you can create a single capping surface by selecting multiple faces, edges, or edge loops. Each opening is patched with a unique surface.

You have now defined a capping surface. Capping surfaces are required for flow volume extraction with fluid flow simulations.

### 2.3.4. Creating a Box

In a fluid flow simulation, define the region where external flow is analyzed by creating a box that encloses all or part of the geometry.

To construct a new box:

1. From **Boxes**, select **Add > Box**.  
The **Box** data panel appears.
2. **Create Selection Sets** is enabled by default; this option is necessary to properly set up physics conditions in later tasks.
3. For **Construction Method**, choose from **Based on geometry selection** or **Two points**.
  - [Creating a Box by Entity Selection](#)
  - [Creating a Box by Coordinates](#)

#### 2.3.4.1. Creating a Box Based on Geometry Selection

Single or multiple entities can be directly enclosed within a box with provided cushion values.

1. **Create Selection Sets** is enabled by default; this option is necessary to properly set up physics conditions in later tasks.
2. For **Construction Method**, choose from **Based on geometry selection**.
3. Under **Location**, select the entities to be enclosed within a box with provided cushion values.
4. For **Cushion Type**, choose from **Uniform** or **Non-Uniform** to set the respective cushion values.

#### 2.3.4.2. Creating a Box by Two Points

Create a box by specifying **Point 1** and **Point 2** as its diagonal coordinates.

1. **Create Selection Sets** is enabled by default; this option is necessary to properly set up physics conditions in later tasks.
2. For **Construction Method**, choose from **Two points**.
3. Under **Point definition** choose from **Coordinates** or **Based on geometry selection** to specify Point 1 or Point 2.
4. For **Coordinates**:
  - a) For **Reference frame**, leave the default Global Reference Frame, or click to select or define a reference frame.
  - b) Under **Coordinate type**, select the type of coordinate system you want to use to define the point.
    - For the **Cartesian** coordinate type, specify the coordinates in X, Y, and Z.
    - For the **Cylindrical** coordinate type, specify the coordinates in radius, Azimuth angle, and height.
    - For the **Spherical** coordinate type, specify the coordinates in radius, Azimuth angle, and polar angle.
5. Under **Based on geometry selections**, the **Location** field is activated.
  - a) Select a topology and then click **Add selected entities** or **Replace with selected entities**.

- b) For **Calculation method**, leave the default method of **Centroid** or click to select the method by which the point is calculated.



# Chapter 3: Meshing

Discovery AIM provides meshing capabilities for various geometric models.

The mesh generation process may be either [automatic](#) or [manual](#). For the templates that support it, automatic physics-aware meshing is the default meshing approach, except for the Polymer Extrusion and Polymer Blow Molding templates.

## 3.1. Overview of Meshing

Discovery AIM provides meshing capabilities for various geometric models.

The mesh generation process may be either [automatic](#) or [manual](#). For the templates that support it, automatic physics-aware meshing is the default meshing approach, except for the Polymer Extrusion and Polymer Blow Molding templates.

## 3.2. Overview of Automatic Physics-aware Meshing

Automatic physics-aware meshing helps automate and simplify your problem setup. With automatic physics-aware meshing, the computational mesh is generated automatically based on the [solution fidelity](#) setting and the physics inputs. For the templates that support it, automatic physics-aware meshing is the default meshing approach, except for the Polymer Extrusion and Polymer Blow Molding templates.

Mesh characteristics that are determined automatically include:

Characteristic	Automatic Behavior
Element shapes	<ul style="list-style-type: none"> <li>If a model contains sweepable bodies (but no shells), a hexahedral mesh is generated automatically. If any body fails to mesh with hexahedral elements, all bodies are meshed with tetrahedral elements.</li> </ul>
Element sizes	<ul style="list-style-type: none"> <li>Body sizing controls are added for topology optimization and for sweepable bodies. For simulations that involve both, the mesher uses the smaller body size.</li> <li>Face sizing controls are added at nonlinear contact interfaces and structural condition locations for topology optimization simulation.</li> <li>Edge sizing controls are added for edges meshed with beam elements.</li> <li>An enhanced curvature size function is used and mesh resolution is increased for structural meshes containing shell elements or thin solid parts.</li> </ul>
Boundary layers	<ul style="list-style-type: none"> <li>Boundary layers are generated on the fluid side for wall conditions and solid/fluid region interfaces.</li> </ul>

With automatic physics-aware meshing:

- The mesh is generated when you update the Physics task. You can also select **Generate Mesh** from the RMB context menu on the Physics task to generate the mesh.
- Only bodies assigned to physics regions are meshed.
- No Mesh task is created.

- If you resume a database that was saved in AIM 19.0 or earlier and you want to use automatic physics-aware meshing, you must add a new Physics task downstream of the Geometry or Import task. Modifying the existing Physics task will not work.
- The preferred method for connecting AIM to Mechanical is to use the [Connect to Mechanical](#) template. If you use another template and plan to connect to Mechanical, use the [define mesh manually](#) option. To do so, you can either enable **Define mesh manually** on the template, or right-click the Physics task in the Workflow view and select **Define Mesh Manually**.
- Red color-coding of mesh grid lines provides visualization of mesh failure. If a mesh fails, click the right mouse button from the Messages panel and choose **Select Location** to highlight the geometry causing the failure.
- In general, when mesh generation is successful, mesh grid lines are black but change to yellow if the mesh state subsequently becomes out-of-date (for example, if you change the position of the **Solution fidelity** slider or make a change on the **Boundary Refinement** panel). However, if your geometry contains beams, the grid lines of the beam mesh remain black regardless of mesh state.
- Automatic physics-aware meshing does not support all physics. For example, physics-aware meshing is not available for Electromagnetics simulations.
- When there is an upstream Mesh task in your workflow, automatic physics-aware meshing is inactive.
- Automatic physics-aware meshing does not support all AIM workflow tasks. For example, physics-aware meshing is not available for simulations that contain Volume Creation tasks.
- If automatic physics-aware meshing fails to generate a satisfactory mesh, you can either:
  - Control the mesh by using [global fidelity capture](#).
  - Control the mesh by using [manual meshing](#).

### 3.2.1. Adjusting the Solution Fidelity

#### **Settings > Solution fidelity**

To adjust the solution fidelity:

1. Navigate to the Physics panel.
2. Move the **Solution fidelity** slider to the left for lower fidelity or to the right for higher fidelity.

Along with more accuracy, higher solution fidelity will capture more details of the geometry and consequently will require more time and system resources to solve. In general:

Position the slider...	If you want...
To the left	<ul style="list-style-type: none"> <li>• Rapid initial results</li> <li>• Low fidelity simulation</li> <li>• Design exploration facilitation</li> <li>• Results suitable for the designer</li> </ul>
In the middle	<ul style="list-style-type: none"> <li>• Reasonably accurate results</li> <li>• Good trade-off of time vs. accuracy</li> </ul>
To the right	<ul style="list-style-type: none"> <li>• Higher precision results</li> <li>• High fidelity simulation</li> <li>• Results suitable for the analyst</li> </ul>

**Note:** In addition to setting the **Solution fidelity** slider, which affects the solution globally, you can use [fidelity refinement](#) to identify localized geometric details of a specific size that you want to capture in the solution.

### 3.2.2. Controlling the Mesh by Using Manual Meshing

To enable [manual control of a mesh](#) that was generated using automatic physics-aware meshing:

1. Right-click the Physics task in the Workflow view.
2. Select **Define Mesh Manually**.

As a result, a Mesh task is inserted upstream of the Physics task. In such cases, information such as selection set definitions and boundary layers are transferred to the Mesh task. However, you may need to re-define some settings.

### 3.2.3. Viewing the Mesh

The mesh does not display automatically when you are working in automatic physics-aware meshing mode. To view a mesh that was generated using automatic physics-aware meshing:

Click **Show mesh**  in the toolbar.

## 3.3. Overview of Manual Meshing

Discovery AIM enables you to mesh various types of geometric models. To begin your study, you can set up a simulation by launching a [template](#) of pre-set tasks and values for a given physics type, or you can set up a simulation by adding the desired tasks manually. For most of the templates that support it, [automatic physics-aware meshing](#) is the default meshing approach. However, you can enable the **Define mesh manually** option on a template if you prefer.

If you are defining the mesh manually, the primary tools for meshing are **Meshing** tasks and **Volume Creation** tasks. Consider this information about which meshing task(s) to add:

- For structural, thermal, or electric conduction simulations, or for fluid flow simulations where the flow volume has already been defined (either in the provided CAD file or by creating it in the Geometry Modelling task), you will need a **Meshing** task in your workflow.

For these types of simulations, AIM uses a part-based meshing process, in which it meshes the entire part or assembly of parts in parallel. By default, it attempts to mesh sweepable bodies with hexahedrons and provides a tetrahedral mesh on bodies that are not sweepable or if the quality of the hexahedral mesh is poor. The mesh includes prism elements if boundary layers are generated.

- For other fluid flow simulations, you will need a **Volume Creation** task and a **Meshing** task. Use the **Volume Creation** task to:
  - Define ("extract") a flow volume
  - Group bodies into a single flow volume
  - Simplify a body that has many surface patches

Generating the mesh for a **Volume Creation** task results in a well-connected ("watertight") surface mesh, or "wrap" mesh. The quality and robustness of the wrap mesh is directly related to the quality of the facets used in display, and the sizing controls used for feature capturing. After generating the wrap, use the **Meshing** task to create a flow volume mesh for the wrapped geometry. The generated flow volume mesh consists of tetrahedral and prism elements.

### 3.3.1. Extracted Volume Workflow

You can extract a flow volume if your model does not contain a body to represent the flow region that you want to mesh. After you have imported your model, follow this workflow:

**Note:** This workflow assumes you are defining the mesh and simulation manually.



## Video: Workflow for Volume Creation and Volume Meshing

1. Add a **Volume Creation** task. In the Workflow view, right-click **Geometry** and select **Add Next > Volume Creation**.
2. On the **Volume Creation** panel, add a volume definition to represent the flow volume (**Objects > Volume Definitions > Add > Extracted Volume**).
3. Use the **Extracted Volume** panel to **define the flow volume**. For more information, refer to **Extracted Volume Properties** on page 176. You will need to create a point inside the flow region, and also create capping surfaces if they do not already exist in your model:
  - The **point** identifies the flow volume. One way to create the point is to select bodies, and then click **Create new** on the **Extracted Volume** panel, which places the point in the appropriate location inside the flow region automatically. You can also create a point from the **Volume Creation** panel (**Auxiliary Definitions > Construction Geometry > Add > Point**) or by clicking the **Add point** icon  on the graphics toolbar.
  - **Capping surfaces** close openings such as inlets and outlets, thereby ensuring the volume is watertight. On the **Volume Creation** panel, select **Objects > Capping Surfaces > Add > Capping Surface**. You can also create them by clicking the **Add capping surface** icon  on the graphics toolbar.
4. On the **Volume Creation** panel, enable **Use predefined settings** to automatically set the fineness of the mesh and optionally adjust the **Mesh resolution**, or disable **Use predefined settings** to set individual **Global Sizing** properties manually. In either case, set the **Global Sizing > Size function method** according to your preference. All of these controls affect **how the mesh size will be distributed** on the surface mesh when the wrap mesh is generated, as well as mesh size distribution within the volume when the flow volume mesh is generated. For more information, refer to **Global Sizing for Volume Creation** on page 126.
5. On the **Volume Creation** panel, specify  **Additional Settings** to control the wrap mesh generation. For more information, refer to **Additional Settings for Volume Creation** on page 136.
6. To define the mesh size at specified locations, add local mesh sizing controls (**Objects > Size Controls > Add**, and then choose face or edge sizing). For more information, refer to **Local Mesh Sizing** on page 170.
7. To **preserve topologies** in the mesh, add Selection Sets to create groupings of entities that will be protected by the mesher (**Auxiliary Definitions > Selection Sets > Add > Selection Set**).
8. Click **Create Volumes**.

Generating the mesh results in a well-connected surface mesh, which you can use as input to a **Meshing** task for **meshing the flow volume**.

After meshing, you can **evaluate the mesh quality** (**Output > Metrics > Add > Mesh Diagnostics**).

### 3.3.2. Geometry Simplification Workflow

You can unite multiple solids to create a single flow region, or simplify a body with many surface patches. After you have imported your model, follow this workflow:

**Note:** This workflow assumes you are defining the mesh and simulation manually.

1. Add a **Volume Creation** task. In the Workflow view, right-click **Geometry** and select **Add Next > Volume Creation**.
2. On the **Volume Creation** panel, add a volume definition for the bodies that you want to group (**Objects > Volume Definitions > Add > Geometry Simplification**).
3. Use the **Geometry Simplification** panel to **define the bodies to simplify or unite**. For more information, refer to **Geometry Simplification Properties** on page 177.

4. On the **Volume Creation** panel, enable **Use predefined settings** to automatically set the fineness of the mesh and optionally adjust the **Mesh resolution**, or disable **Use predefined settings** to set individual **Global Sizing** properties manually. In either case, set the **Global Sizing > Size function method** according to your preference. All of these controls affect **how the mesh size will be distributed** on the surface mesh when the wrap mesh is generated, as well as mesh size distribution within the volume when the flow volume mesh is generated. For more information, refer to [Global Sizing for Volume Creation](#) on page 126.

5. On the **Volume Creation** panel, specify  **Additional Settings** to control the wrap mesh generation. For more information, refer to [Additional Settings for Volume Creation](#) on page 136.
6. To define the mesh size at specified locations, add local mesh sizing controls (**Objects > Size Controls > Add**, and then choose body, face, or edge sizing). For more information, refer to [Local Mesh Sizing](#) on page 170.
7. To **preserve topologies** in the mesh, add Selection Sets to create groupings of entities that will be protected by the mesher (**Auxiliary Definitions > Selection Sets > Add > Selection Set**).

Voids present an exception to the use of Selection Sets to protect topology. If the imported geometry includes a fully-enclosed void, geometry simplification fills the void.

#### 8. Click **Create Volumes**.

Generating the mesh results in a surface mesh, which you can use as input to a **Meshing** task for **meshing the flow volume**.

After meshing, you can **evaluate the mesh quality** (**Output > Metrics > Add > Mesh Diagnostics**).

### 3.3.3. Flow Volume Meshing Workflow

You can use the surface mesh that resulted from a **Volume Creation** task to generate a flow volume mesh for a model. After creating the volume, follow this workflow to mesh the wrapped geometry:

**Note:** This workflow assumes you are defining the mesh and simulation manually.



[Video: Workflow for Volume Creation and Volume Meshing](#)

1. Add a **Meshing** task. In the Workflow view, right-click **Volume Creation** and select **Add Next > Meshing**.
2. On the **Mesh** panel, specify **Boundary Layer Settings** if your simulation will involve boundary layers. These settings provide global control over all boundary layers. For more information, refer to [Common Boundary Layer Settings for Flow Volume Meshing](#) on page 141.
3. To identify specific boundaries where you want a layered mesh, add local boundary layer controls (**Objects > Mesh Controls > Add > Boundary Layer**). Enable **Automatically defined** to apply a control to all faces except those that are used in Selection Sets, or disable it and use the **Location** field to select the desired faces manually. For more information, refer to [Local Boundary Layer Control Options](#) on page 179.
4. On the **Mesh** panel, click **Generate Mesh**.

After meshing, you can **evaluate the mesh quality** (**Output > Metrics > Add > Mesh Diagnostics**).

### 3.3.4. Part-based Meshing Workflow

You use part-based meshing for structural, thermal, or electric conduction simulations, or for fluid flow simulations where you have an existing flow volume. Part-based meshing generates a mesh for entire parts or assemblies of parts in parallel. Follow this workflow:

**Note:** This workflow assumes you are defining the mesh and simulation manually.

1. Add a **Meshing** task. In the Workflow view, right-click **Geometry** and select **Add Next > Meshing**.

- On the **Mesh** panel, set the **Engineering intent** control according to the type of physics you plan to simulate. This setting provides intelligent defaults based on your physics preference.

You must set the **Engineering intent** before you can generate the mesh for the **Meshing** task.

- On the **Mesh** panel, enable **Use predefined settings** to automatically set the fineness of the mesh and optionally adjust the **Mesh resolution**, or disable **Use predefined settings** to set individual **Global Sizing** properties manually. In either case, set the **Global Sizing > Size function method** according to your preference. All of these controls affect **how the mesh size will be distributed**. For more information, refer to [Global Sizing for Part-based Meshing](#) on page 149.
- On the **Mesh** panel, if **Use predefined settings** is enabled and **Engineering intent** is set to **Structural, thermal or electric conduction**, you can also enable **Model contains thin parts** to instruct the mesher to increase the mesh resolution for structural meshes containing shell elements or thin solid parts.
- On the **Mesh** panel, if **Use predefined settings** is enabled, you can also enable **Retry on mesh failures** to instruct the mesher to retry mesh generation automatically if a failure occurs due to poor mesh quality.
- On the **Mesh** panel, specify  **Additional Settings** to control the mesh generation. For more information, refer to [Additional Settings for Part-based Meshing](#) on page 165.
- On the **Mesh** panel, specify **Boundary Layer Settings** if you plan to define boundary layers. These settings provide global control over all boundary layers. For more information, refer to [Common Boundary Layer Settings for Part-based Meshing](#) on page 160.
- To specify whether a tetrahedral or hexahedral mesh is generated on a body, add local element shape controls (**Objects > Mesh Controls > Add > Element Shape**). Use the **Location** field of each control to select the desired bodies. For more information, refer to [Element Shape Settings](#) on page 185.
- To identify specific boundaries where you want a layered mesh, add local boundary layer controls (**Objects > Mesh Controls > Add > Boundary Layer**). Enable **Automatically defined** to apply a control to all faces except those that are used in Selection Sets, or disable it and use the **Location** field to select the desired faces manually. For more information, refer to [Local Boundary Layer Control Options](#) on page 179.
- To define the mesh size at specified locations, add local mesh sizing controls (**Objects > Size Controls > Add**, and then choose body, face, or edge sizing). For more information, refer to [Local Mesh Sizing](#) on page 170.

#### 11. Click **Generate Mesh**.

AIM meshes all the parts in parallel.

Refer to [Usage Notes for Part-based Meshing](#) on page 147 for more information.

After meshing, you can [evaluate the mesh quality](#) (**Output > Metrics > Add > Mesh Diagnostics**).

### 3.3.5. Best Practices

Best practices include methods and techniques for effective use of the meshing tools in AIM:

- [Structural Meshing Best Practices](#) on page 109
- [Fluid Flow Meshing Best Practices](#) on page 110

#### 3.3.5.1. Structural Meshing Best Practices

Best practices for structural meshing include:

# Part-based Meshing's Automatic Approach

By default, part-based meshing uses an automatic meshing approach; it meshes sweepable bodies with a hexahedral mesh and other bodies with a tetrahedral mesh. Alternatively, you can [specify element shape](#) to generate a tetrahedral or hexahedral mesh per body.

## 3.3.5.2. Fluid Flow Meshing Best Practices

Best practices for fluid flow meshing include:

### Choosing the Appropriate Workflow

If you set up a simulation by launching a flow [template](#), pre-set tasks and values for a fluid flow simulation are provided. If you start a simulation manually, you need to decide between using a **Volume Creation** task followed by a **Meshing** task, or using a **Meshing** task only. See [Overview of Manual Meshing](#) on page 106.

### Using Automatically Defined Boundary Layers

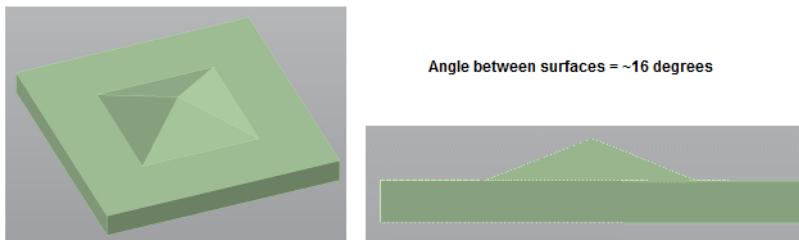
To automate [Location](#) specification when you are [defining boundary layer controls](#), enable **Automatically defined**. When **Automatically defined** is enabled, AIM associates the boundary layer control to the DefaultBoundaryLayer() function, which returns all faces except those that are used in Selection Sets.

Also see [Location References](#) on page 630.

### Ensuring Topology Is Preserved in a Wrap Mesh

This example uses the geometric model shown below, along with Selection Sets, to illustrate the effects of [topology protection](#) during the wrap generation that occurs when a **Volume Creation** task is processed. By adding Selection Sets directly to the **Volume Creation** task *before* initial generation of the task, you can create named groupings of entities that will be protected by the mesher. (For related information, see [Setting Optional Defaults for Volume Creation](#) on page 118.) Defining Selection Sets is just one method you can use to ensure topology is preserved in a mesh.

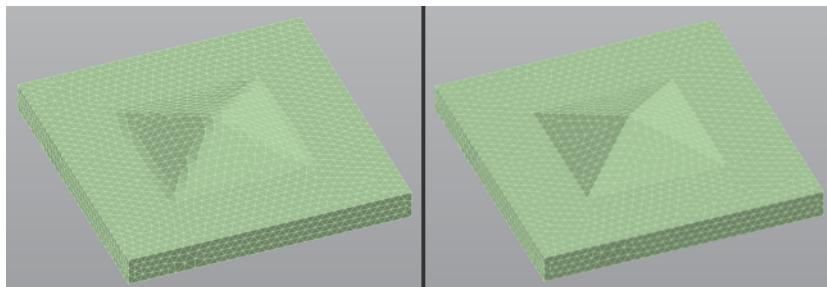
**Figure 3.3.5.2.1. Isometric and Front Views of the Geometric Model**



An alternative to defining Selection Sets is to use the value of the [Edge extraction angle](#) option. When no Selection Sets are defined, the value of the **Edge extraction angle** option determines which CAD features are captured in the mesh. An important distinction between the use of Selection Sets and the use of the **Edge extraction angle** option is that Selection Sets provide greater control over which specific topology is preserved, whereas the **Edge extraction angle** option has a global effect; any CAD features in the model that meet its criteria are captured in the mesh. The smaller the angle, the higher the number of features that are captured. In the figure below, the default **Edge extraction angle** of 40 degrees is retained for the mesh

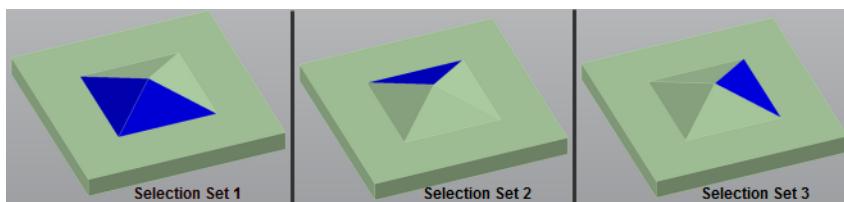
on the left, but it is set to 10 to obtain the mesh on the right. Because the angle between surfaces is approximately 16 degrees, the setting of 10 captures the edges in the mesh.

**Figure 3.3.5.2.2. Edge Extraction Angle Settings of 40 and 10 Degrees Respectively**



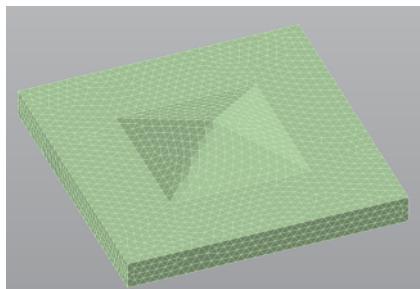
The remainder of this example illustrates the effects of defining the three Selection Sets shown below.

**Figure 3.3.5.2.3. Three Selection Sets Are Defined**



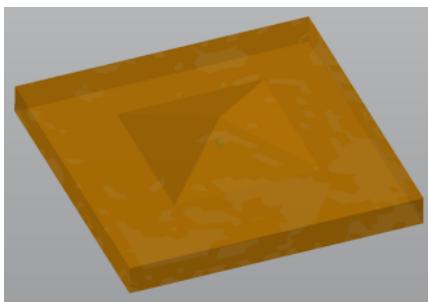
With the Selection Sets defined, the entities within them are protected and captured in the mesh as shown below even though the **Edge extraction angle** is still set to 40.

**Figure 3.3.5.2.4. Edge Extraction Angle of 40 with Selection Sets**



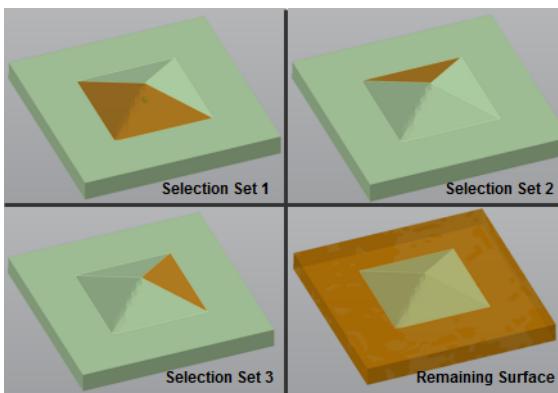
The presence of the Selection Sets also affects the selectable faces in the wrap mesh. When topology is protected, wrap generation separates out that topology. The value of **Edge extraction angle** does not affect this separation behavior. In the figure below, the **Edge extraction angle** is set to 10, but the wrap mesh results in only one selectable face because no Selection Sets are defined.

**Figure 3.3.5.2.5. No Selection Sets Results in One Selectable Face**



With the Selection Sets defined, the wrap mesh results in four selectable faces—one for each Selection Set and a fourth for the remaining surface. These faces will be available for selection in the next connected task. For example, if you add a **Meshing** task, you can select one or more of these faces to add boundary layer controls to specific boundaries.

**Figure 3.3.5.2.6. Selection Sets Result in Four Selectable Faces**



### 3.3.6. Mesh Size and Distribution

An important aspect of meshing in AIM is the size function, which controls how the mesh size is distributed on a face or within a body.

You can enable the **Settings > Use predefined settings** control to automatically set the fineness of the mesh, or disable it to set individual **Global Sizing** properties manually. In either case, you can set the **Global Sizing > Size function method** control according to your preference for mesh size distribution calculations. You determine which refinement mechanisms are activated by selecting **Curvature and proximity**, **Proximity**, **Curvature**, **Fixed**, or **Adaptive**. Depending on the selected **Size function method**, you can also set:

- The angles between normals for adjacent mesh elements (curvature size function methods)
- The number of mesh elements employed in the gaps between two geometric entities (proximity size function methods)
- The gradation between minimum and maximum sizes based on a specified growth rate (all size function methods)

## How the Mesher Computes the Size Function

The size function is computed when meshing begins. When the **Adaptive** size function method is used, the mesher uses the value of the **Element seed size** property to determine a starting point for the mesh size. The value of the **Element seed size** property can be user-defined, or it can be automatically computed by the mesher (defaulted based on the bounding box). When meshing begins, edges are meshed with this size initially, and then they are refined for curvature and 2D proximity. Next, mesh-based defeathering occurs. The final edge mesh is then passed into a least-squares fit size function, which guides face and volume meshing.

For all the other size function methods, the mesher examines the size sources, the distance to each source (based on the smallest size obtained at the location of the sources), and the growth rate. The smallest size at each point is selected and stored in a background grid. The mesher uses the sizes from the background grid to generate a mesh. The background grid is refined automatically to provide size distribution to the mesher.

The following factors contribute to the final mesh distribution:

- The size source, which can be any of the following:
  - Edge and face curvature, based on the normal angle variation between adjacent mesh elements in 2D (edge) or 3D (face)
  - Edge and face proximity, based on the number of element layers created in a gap between edges in 2D or between faces in 3D
  - Local element sizing on selected edges, faces, or bodies
  - Influence of a swept body
- The values of size function options, including **Minimum size/Proximity minimum size**, **Maximum size**, **Growth rate**, and **Maximum face size** (part-based meshing only)

The **Minimum size/Proximity minimum size** and **Maximum size** specifications represent, respectively, the global minimum and global maximum allowable element size. The **Maximum face size** specification represents the global maximum allowable size of the elements created by the free surface meshers when generating a part-based mesh. The **Growth rate** represents the increase in element edge length with each succeeding layer of elements from the edge or face. For example, a growth rate of 1.2 results in a 20% increase in element edge length with each succeeding layer of elements.

**Note:**

- The influences of scoped local element sizing and existing meshes are always active, but you have explicit control over whether to activate the influences of curvature, proximity, both, or neither (fixed).
- AIM saves the computed size functions to a [size field file](#) for reuse in certain meshing tasks.
- For part-based meshing, the size function works within parts, but not across parts. For volume creation and flow volume meshing, the size function takes the entire assembly into account.

## Setting Mesh Sizes to Optimize Performance

One of the most important values related to the size function is minimum size (**Minimum size** and **Proximity minimum size** controls):

- Setting a value for minimum size that is too large may mean that important features are not captured in the mesh.
- Setting a value for minimum size that is too small may lead to an unnecessarily fine mesh and longer meshing and solution time.
- Setting a minimum size that is smaller than the geometry tolerances or defeaturig tolerances may cause the mesher to refine the mesh smaller than holes/gaps in the geometry, thereby making it difficult for meshing to be successful.

Although minimum size is set to a certain value, a smaller sized mesh may be obtained locally if a **Hard** scoped size (on an edge, face, or body) has been assigned that is smaller than the minimum size. In such cases, the size function will automatically reduce the minimum size locally to ensure that the size transition between the scoped entity and the surrounding mesh follows the prescribed size function **Growth rate**.

However, there are times in which the size function will not reduce the minimum size locally:

- If a case involves a geometric feature (edge, face) that is smaller than the minimum size, you must either remove the small features or assign scoped sizes to them to ensure a smooth size transition.

- If a case involves a swept mesh in a multibody part where the boundary of the swept mesh is smaller than the minimum size due to a reduction of cross-sectional size in the swept region, you must add additional sizing controls to ensure a smooth size transition between swept regions.

Similarly, you can scope a **Hard** size that is larger than the values that are specified by the size function's **Maximum face size** and **Maximum size** options, as long as the size function produces a smooth size transition away from the scoped entity.

For all size function methods except **Adaptive**, **Hard** edge size information is injected into the background grid during the meshing process. Transitions may be abrupt between **Hard** edges (or any edge to which **Bias type/Bias growth rate** has been applied) and adjacent edges and face meshes. Edges with **Hard** edge sizing or bias will also override the **Maximum face size** and **Maximum size** properties.

### 3.3.6.1. Proximity Size Function

The proximity size function enables you to specify the minimum number of element layers created in regions that constitute "gaps" in the model. For the purposes of specifying a proximity size function, a "gap" is defined in one of two ways:

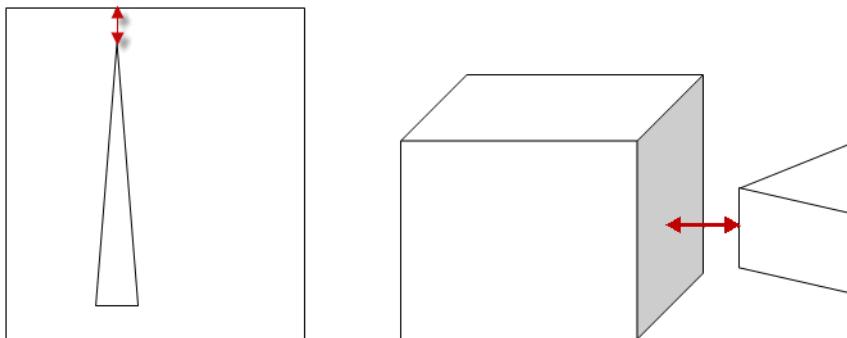
- The internal volumetric region between two faces
- The area between two opposing boundary edges of a face

The following properties define the proximity size function:

- Proximity size function sources**
- Minimum size**
- Proximity minimum size**
- Maximum face size** (part-based meshing only)
- Maximum size**
- Growth rate**
- Number of cells across gap**

The **Number of cells across gap** is the number of layers of elements to be generated in the gaps. **Proximity size function sources** determines which regions of proximity are considered in calculations. **Proximity minimum size** provides greater control over the size function, as it enables you to specify a minimum size for proximity size function calculations in addition to the minimum size that is specified for **Minimum size**.

**Note:** The proximity size function does not recognize the proximity between certain entities in cases involving voids in a model. For example, the proximity between a vertex and an edge on a face in 2D (below left) is ignored if the triangle is a void in the rectangle. Similarly, the proximity between a vertex or edge and a face in 3D (below right) is ignored if the prism and the block are voids in a larger domain. The two-headed arrows in the figure indicate the areas where proximity is ignored.



For more information, refer to the appropriate topic:

- [Global Sizing for Volume Creation with the Curvature and Proximity Size Function Method](#) on page 127

- [Global Sizing for Volume Creation with the Proximity Size Function Method on page 131](#)
- [Global Sizing for Part-based Meshing with the Curvature and Proximity Size Function Method on page 149](#)
- [Global Sizing for Part-based Meshing with the Proximity Size Function Method on page 155](#)

### 3.3.6.2. Curvature Size Function

The curvature size function examines curvature on edges and faces and computes element sizes on these entities such that the size does not violate the maximum size or the curvature normal angle, which are either computed automatically by the mesher or user-defined. The following properties define the curvature size function:

- **Minimum size**
- **Maximum face size** (part-based meshing only)
- **Maximum size**
- **Growth rate**
- **Curvature normal angle**

The **Curvature normal angle** is the maximum allowable angle that one element edge is allowed to span.

For more information, refer to the appropriate topic:

- [Global Sizing for Volume Creation with the Curvature and Proximity Size Function Method on page 127](#)
- [Global Sizing for Volume Creation with the Curvature Size Function Method on page 131](#)
- [Global Sizing for Part-based Meshing with the Curvature and Proximity Size Function Method on page 149](#)
- [Global Sizing for Part-based Meshing with the Curvature Size Function Method on page 154](#)

### 3.3.6.3. Fixed Size Function

The fixed size function does not refine the mesh based on curvature or proximity. Rather, you specify minimum and maximum sizes and gradation is provided between sizes based on the specified growth rate. The following properties define the fixed size function:

- **Minimum size**
- **Maximum face size** (part-based meshing only)
- **Maximum size**
- **Growth rate**

With the fixed size function, you must use size controls to mesh sizes locally, as opposed to the curvature and proximity size functions, which refine the sizes locally based on curvature and proximity of features in the geometry. Even if the specified local sizes are **Soft** sizes, they may override the global sizes when the fixed size function is used.

For more information, refer to the appropriate topic:

- [Global Sizing for Volume Creation with the Fixed Size Function Method on page 136](#)
- [Global Sizing for Part-based Meshing with the Fixed Size Function Method on page 159](#)

### 3.3.6.4. Adaptive Size Function

The adaptive size function is intended for meshes bound for structural solvers. The adaptive size function uses the value of the **Element seed size** property to determine a starting point for the mesh size. The value of the **Element seed size** property can be user-defined, or it can be automatically computed by the mesher (defaulted based on the bounding box). When meshing begins, edges are meshed with this size initially, and then they are refined for curvature and 2D proximity. Next, mesh-based defeathering occurs. The final edge mesh is then passed into a least-squares fit size function, which guides face and volume meshing. When the adaptive size function is used, the mesh may be much coarser with smaller node and element counts.

The following properties define the adaptive size function:

- **Element seed size**
- **Growth rate**
- **Adaptive resolution**

For more information, refer to [Global Sizing for Part-based Meshing with the Adaptive Size Function Method](#) on page 160.

### 3.3.6.5. Reuse of Computed Size Functions

AIM saves the size field from volume creation so it can reuse it in volume meshing.

When your workflow includes a **Volume Creation** task followed by a **Meshing** task, the volume creation process identifies the connected **Meshing** task that will inherit the sizing and updates the size values in that connected task.

#### 3.3.7. Volume Creation

You can use a **Volume Creation** task to:

- Extract a flow volume if your model does not contain a body to represent the flow region that you want to mesh; for example, to model fluid flowing within a pipe. This type of volume creation is called **Extracted Volume**.
- Group one or more bodies into a single flow volume or simplify a body with many surface patches. This type of volume creation is called **Geometry Simplification**.

When AIM processes a **Volume Creation** task, it uses the *shrink wrap* method. This method uses specialized boundary wrapping to extract a well-connected surface mesh and is useful for cases involving defeaturering or when you need to walk over features. To create a well-connected wrap object, the shrink wrap method:

1. Overlays a Cartesian grid on each geometry object and creates contiguous regions.
2. Refines the Cartesian grid based on size functions to better represent the geometry object.

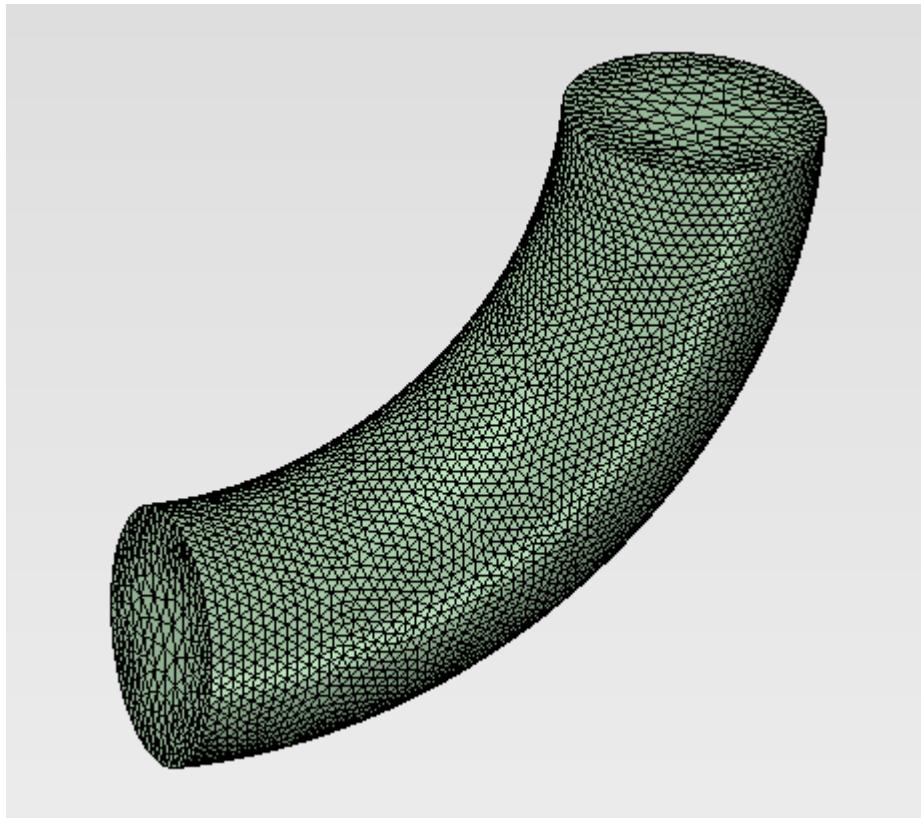
**Note:** Shrink wrap approximates the geometry using a staircase-like Cartesian grid. It requires finer cells to resolve thin gaps. In cases where a gap area is not curved and not aligned to the Cartesian axes, you may need to refine 3-4 times finer than the gap thickness. Take this into account when setting global and local minimum size values and the number of cells to be generated across gaps for the [proximity size function](#).

3. Extracts the interface on the boundary of the non-intersecting Cartesian volume region. When you create an extracted volume, this region encloses the point that identifies the flow region.
4. Projects the extracted interface onto the geometry.
5. Deletes degenerate and island regions, and intersects and remeshes them appropriately.
6. Imprints edges of the wrapped boundary region (which is a closed domain) and preserves the faces in the Selection Sets based on the original geometry.
7. Remeshes surfaces based on the size functions/size field.

**Note:**

- By default, the model that results from generating the **Volume Creation** task contains faces that reflect the bodies from the input model, but not the individual faces of those bodies. Adding Selection Sets to the **Volume Creation** task protects topology and obtains more topology granularity. For an example of protected topology and volume creation, see [Fluid Flow Meshing Best Practices](#) on page 110.
- For both types of volume creation, you need to define the volume that you want to extract or the bodies that you want to group, as appropriate. For related information, see [Extracted Volume Workflow](#) on page 106 and [Geometry Simplification Workflow](#) on page 107.

**Figure 3.3.7.1. Volume Creation**



### 3.3.7.1. Volume Creation Limitations

The shrink wrap method, which is used in **Volume Creation** tasks, has the following limitations:

- It does not support zero-thickness baffles. We recommend that you add thickness to any baffles in your model before attempting to wrap.
- It does not resolve acute interior angles (such as sharp corners) well. It usually skips the imprinting onto the edges at angles of less than 30 degrees. This imprinting is controlled by an internal relative imprinting threshold parameter. The parameter's value defaults to 2, which means it tries to imprint onto edges that are closer than two times the local size.

### 3.3.7.2. Setting Defaults for Volume Creation

The **Settings** menu for **Volume Creation** enables you to set the **Use predefined settings** control, which provides defaults for **Global Sizing** controls.

### Use Predefined Settings

If you enable **Use predefined settings**, you can use the **Mesh resolution slider** to set the fineness of the mesh for the model. You can set the slider toward lower resolution (**Low**) or higher resolution (**High**). A higher resolution provides more accuracy but uses more elements, more time, and more system resources.

The **Mesh resolution** automatically sets all the **Global Sizing** controls except **Size function method**. Therefore, when **Use predefined settings** is enabled, the **Global Sizing** controls become read-only and cannot be set as parameters. If you subsequently disable **Use predefined settings**, the settings are initially preserved but can be changed.

### 3.3.7.2.1. Setting Optional Defaults for Volume Creation



These settings are available for controlling **Volume Creation** if you turn Filtering off.

## Reassociate Reference IDs to Current Output Model

**Reassociate reference IDs to current output model** enables you to control whether reference IDs that AIM uses to keep track of entities are deleted and regenerated, or retained and reassigned when a **Volume Creation** task is updated and the input model has not been updated.

A task receives data (input) from the previous connected task. When a task is completed and up-to-date, it generates data (output) that is transferred to the next connected task in the simulation. AIM associates entities (such as faces) to reference IDs internally, and uses the associations to map the entities in the input model to the corresponding entities in the output model. These associations affect the handling of settings you assign directly to topological selections (such as boundary layers and physics settings assigned to faces). The associations that exist when a **Volume Creation** task is updated affect all tasks that follow it in the simulation.

The **Reassociate reference IDs to current output model** control is exposed after initial generation of the **Volume Creation** task. Its state (enabled or disabled) determines whether AIM retains the existing association when you update the **Volume Creation** task subsequently (for example, after you change a setting on the **Volume Creation** panel and update the task).

- When **Reassociate reference IDs to current output model** is enabled (the default), the behavior that occurs when you update the **Volume Creation** task depends on whether the input model was updated:
  - If the input model was updated, **Reassociate reference IDs to current output model** is disabled automatically and cannot be enabled. No reassociation occurs. New reference IDs are generated and associated to the model.
  - If the input model was not updated, AIM retains the reference IDs that were associated to the model when you generated the **Volume Creation** task the first time (that is, it reassociates the existing reference IDs to the current output model).
- When **Reassociate reference IDs to current output model** is disabled, no reassociation occurs. New reference IDs are generated and associated to the model each time you update the **Volume Creation** task.

## Reassociation Limitations

Reassociation has the following limitations:

- In some cases, considerable complexity in geometry and large variation in sizing controls or other settings between subsequent updates may cause inconsistency or inconstancy of reference IDs, which may lead to generation of new reference IDs. For example, a single face representing a topology in the previous output model may be broken down into multiple faces representing the same topology in the current output model or vice versa (n-to-1 or 1-to-n association).

To ensure controls (such as boundary layer controls or physics conditions assigned to faces) remain persistent and valid throughout task updates, we strongly recommend you define Selection Sets for entities to which you want to apply controls *before* initial generation of the **Volume Creation** task. Then when assigning a control to a **Location**, select a Selection Set rather than selecting topological entities directly. This practice protects the topological entities and keeps the entities and their reference IDs consistent

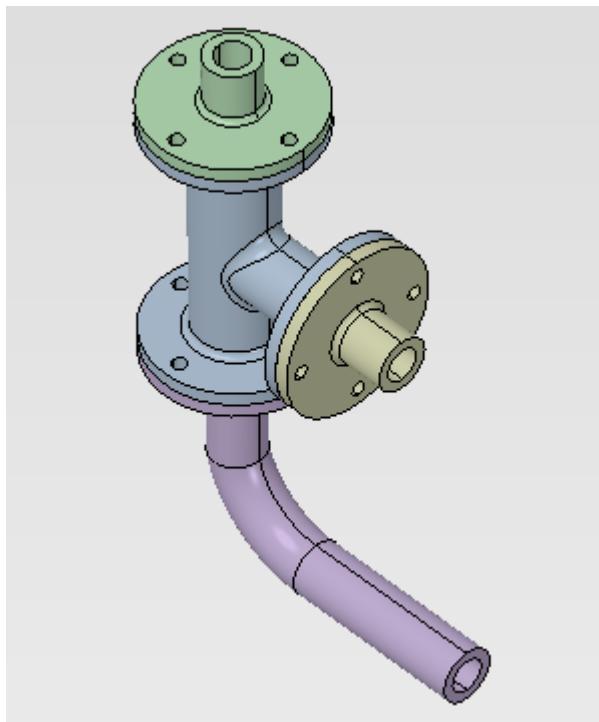
throughout the simulation. For related information, see [Ensuring Topology Is Preserved in a Wrap Mesh](#) on page 110.

- Reference IDs of bodies that represent the output of **Volume Creation** and **Meshing** tasks may also be affected by updates. For example, if you directly selected a body as the **Location** for a physics region or results contour, you may need to redefine the **Location** after an update.

## Reassociation Example

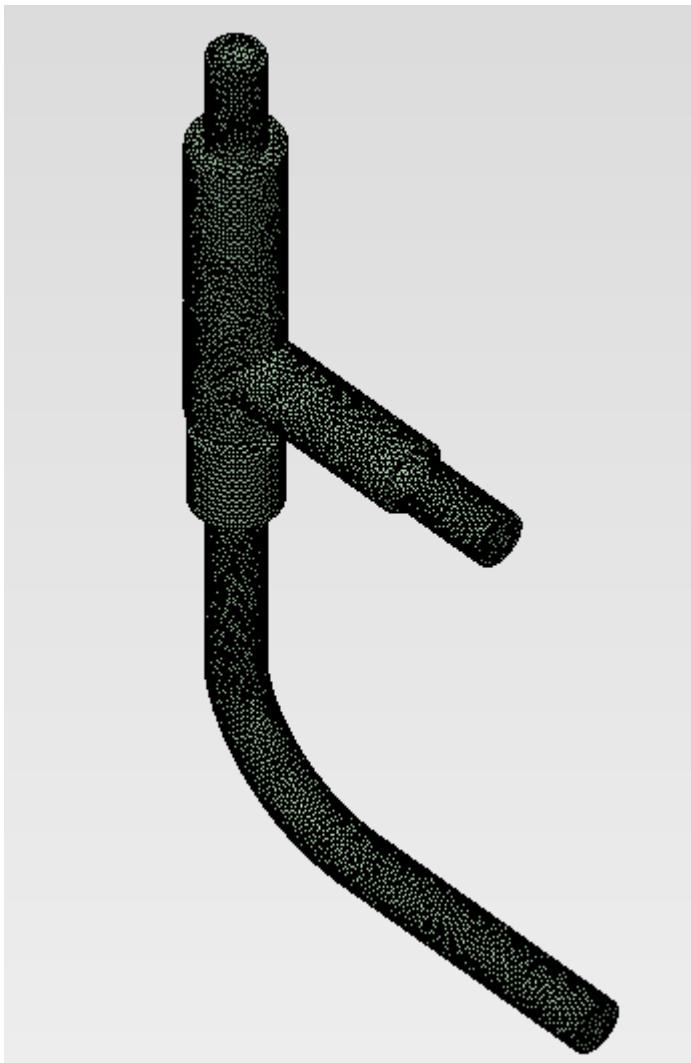
The following example uses this geometric model of a T-junction with a bent pipe to illustrate reassociation.

**Figure 3.3.7.2.1.1. T-junction with Bent Pipe**



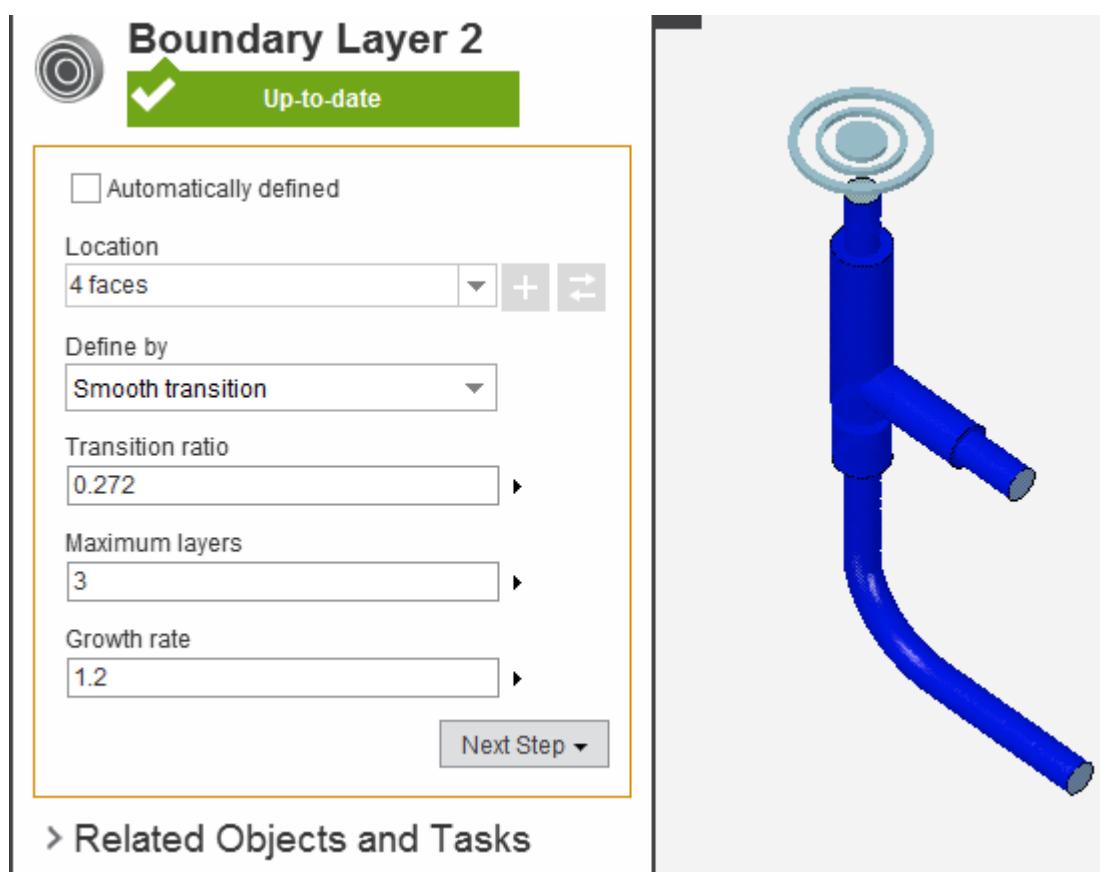
A **Volume Creation** task for flow volume extraction is generated, and **Show mesh** is clicked.

**Figure 3.3.7.2.1.2. Surface Mesh for Extracted Volume**



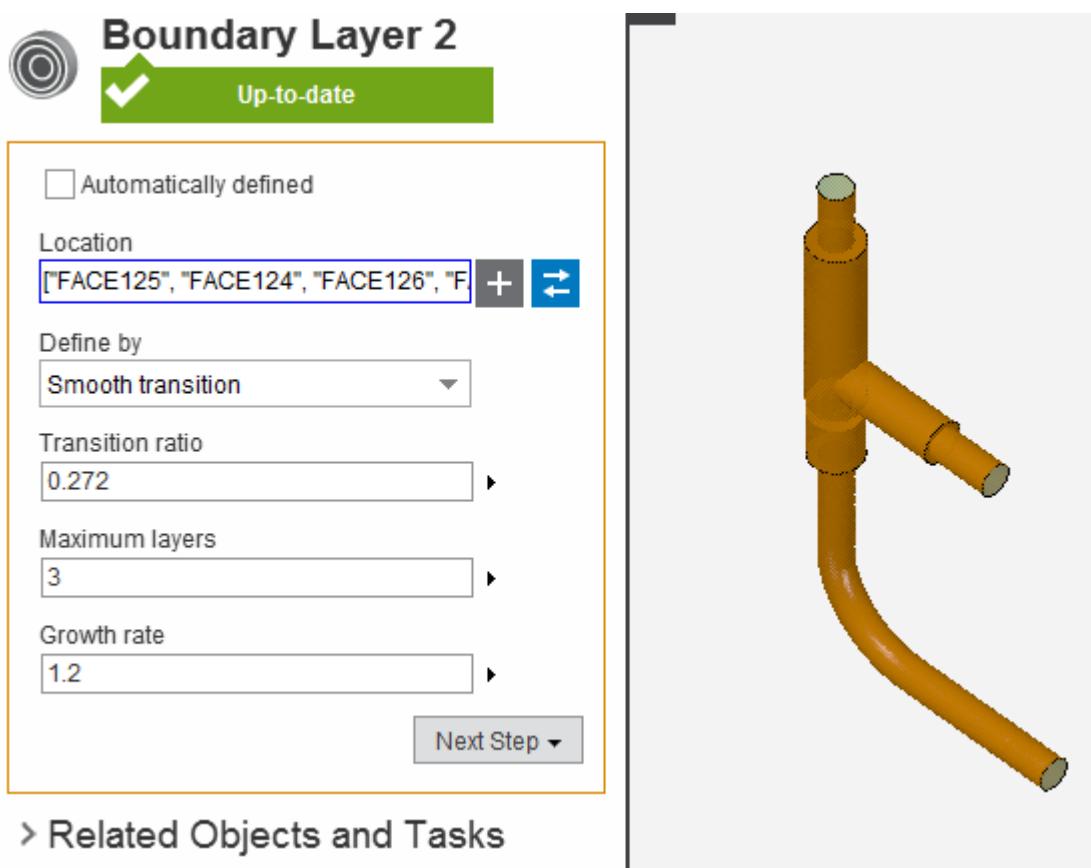
A connected **Meshing** task is added, and a **Boundary Layer** control is added to the **Meshing** task with four faces selected as boundary layers.

**Figure 3.3.7.2.1.3. Four Faces Are Selected as Boundary Layers**



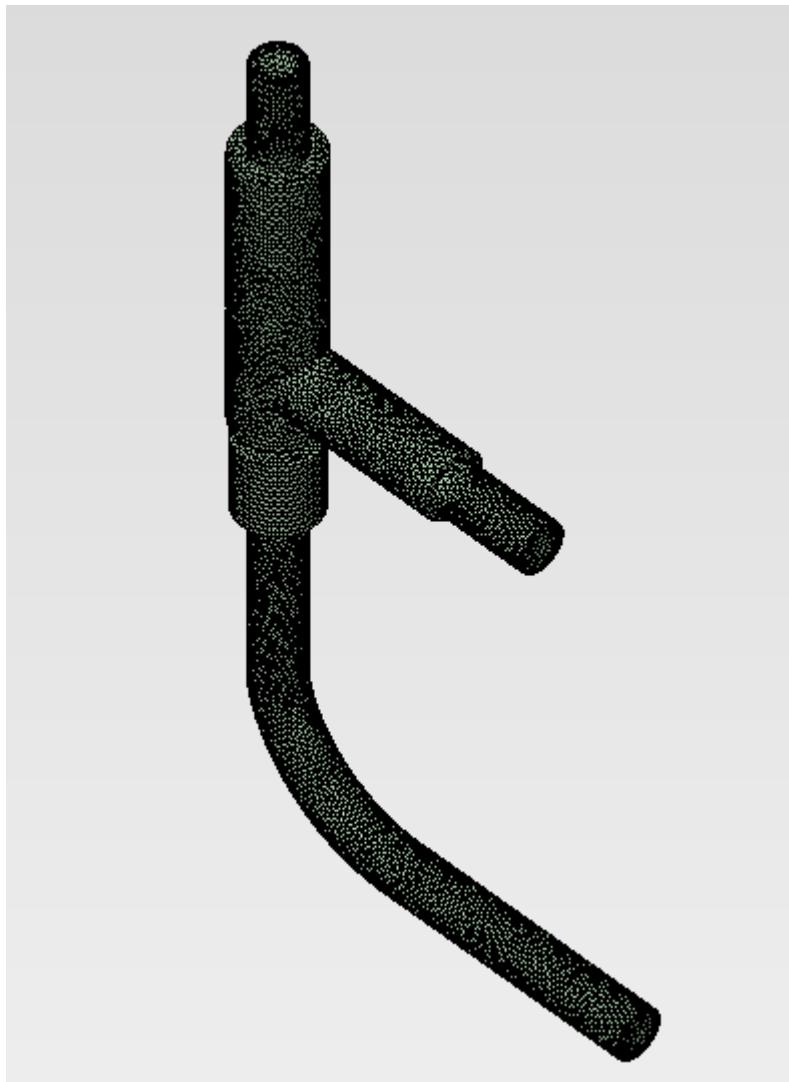
To check the reference IDs associated to the four faces, the **Location** field is clicked. As shown below, the IDs are FACE125, FACE124, FACE126, and FACE127.

**Figure 3.3.7.2.1.4. Reference IDs Are Checked**



The **Meshing** task is generated.

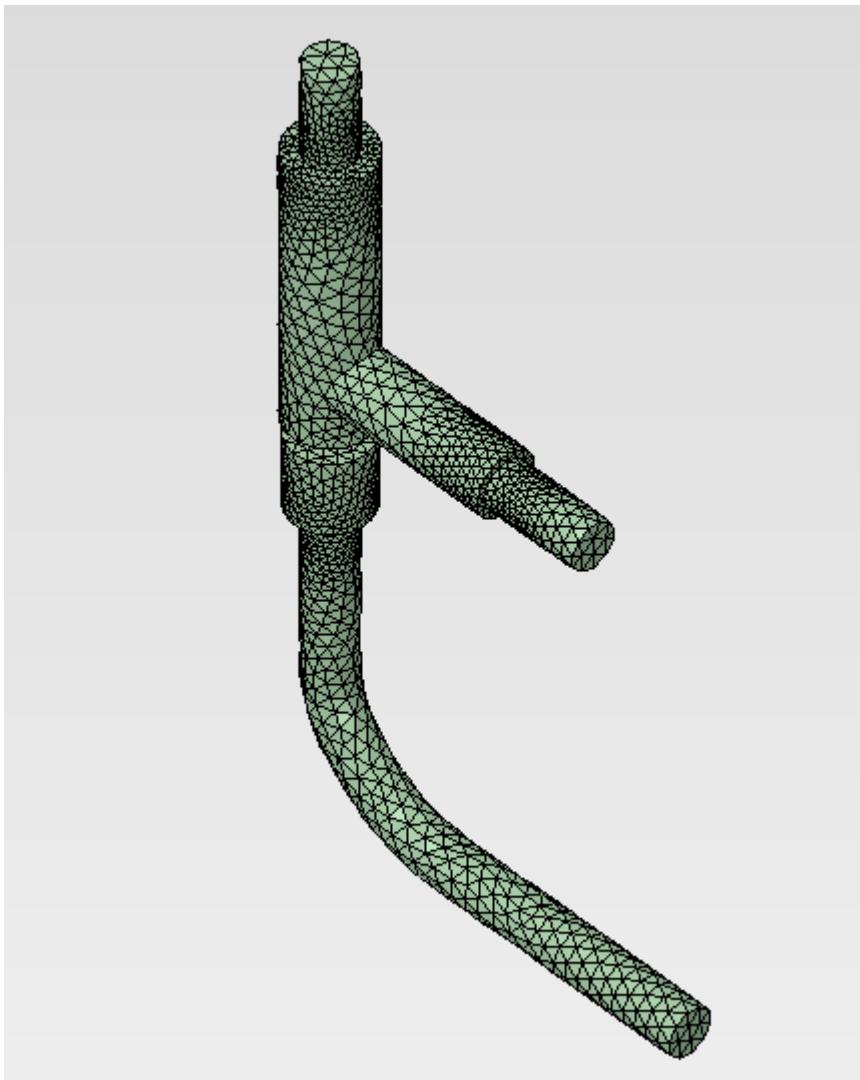
**Figure 3.3.7.2.1.5. Volume Mesh**



After returning to the **Volume Creation** panel, the **Mesh resolution** slider is moved toward the **Low** end to obtain a coarser mesh.

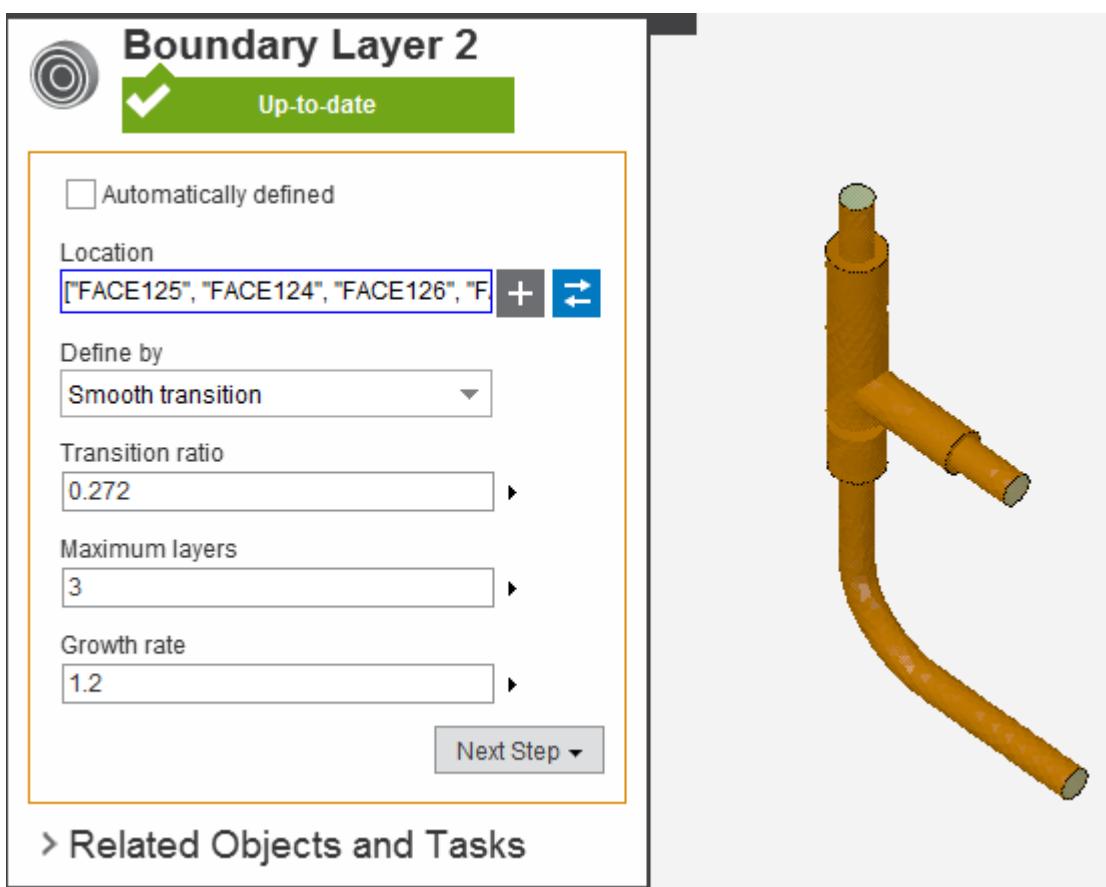
The **Volume Creation** task is now out-of-date and must be updated. The **Reassociate reference IDs to current output model** control is enabled by default. The **Volume Creation** task is updated.

**Figure 3.3.7.2.1.6. Coarser Surface Mesh for the Extracted Volume**



The **Meshing** task is also out-of-date. A return to the **Boundary Layer** control indicates that it is still valid. Clicking in the **Location** field confirms the reference IDs (FACE125, FACE124, FACE126, and FACE127) have been retained and reassigned to the faces.

**Figure 3.3.7.2.1.7. Existing Reference IDs Are Reassociated to the Boundary Layers**



The **Meshing** task is updated.

**Figure 3.3.7.2.1.8. Coarser Volume Mesh**



### 3.3.7.2.2. Setting Mesh Resolution

**Volume Creation > Settings**

**Mesh > Settings**

To control the fineness of the mesh for a model automatically, you must set the **Mesh resolution** by performing these steps.

1. Enable **Use predefined settings**.
2. Move the **Mesh resolution** slider to the left for lower resolution (**Low**) or to the right for higher resolution (**High**). A higher resolution setting provides more accuracy but may use more elements, more time, and more system resources.

### 3.3.7.3. Global Sizing for Volume Creation

The size function controls how the mesh size is distributed on the surface mesh when the wrap mesh is generated. These controls will also affect mesh size distribution within the volume when the flow volume mesh is generated. You can enable the **Settings > Use predefined settings** control to automatically set the fineness of the mesh, or disable it to set individual **Global Sizing** properties manually. In either case,

you can set the **Global Sizing > Size function method** control according to your preference for mesh size distribution calculations. You determine which refinement mechanisms are activated by selecting **Curvature and proximity**, **Proximity**, **Curvature**, or **Fixed**.

The **Size function method** for **Volume Creation** tasks is set to **Curvature and proximity** by default. We recommend that you retain this setting because it works best for preserving **protected topologies**.

For details about the properties you can set for each size function method, refer to the appropriate topic:

- [Global Sizing for Volume Creation with the Curvature and Proximity Size Function Method](#) on page 127
- [Global Sizing for Volume Creation with the Curvature Size Function Method](#) on page 131
- [Global Sizing for Volume Creation with the Proximity Size Function Method](#) on page 131
- [Global Sizing for Volume Creation with the Fixed Size Function Method](#) on page 136

### 3.3.7.3.1. Global Sizing for Volume Creation with the Curvature and Proximity Size Function Method

The size function controls how the mesh size is distributed on a surface or within a volume. The properties described here provide more precise control over mesh size distribution during **Volume Creation** tasks when **Size function method** is set to **Curvature and proximity**, which is the default. We recommend that you retain this setting because it works best for preserving **protected topologies**.

For information about overriding size function sizes, refer to [Setting Mesh Sizes to Optimize Performance](#) on page 113.

**Proximity** **Proximity size function sources** determines whether regions of proximity between faces and/or **Edges** **Faces** **Faces and edges** edges are considered when proximity size function calculations are performed. Choose one of the following:

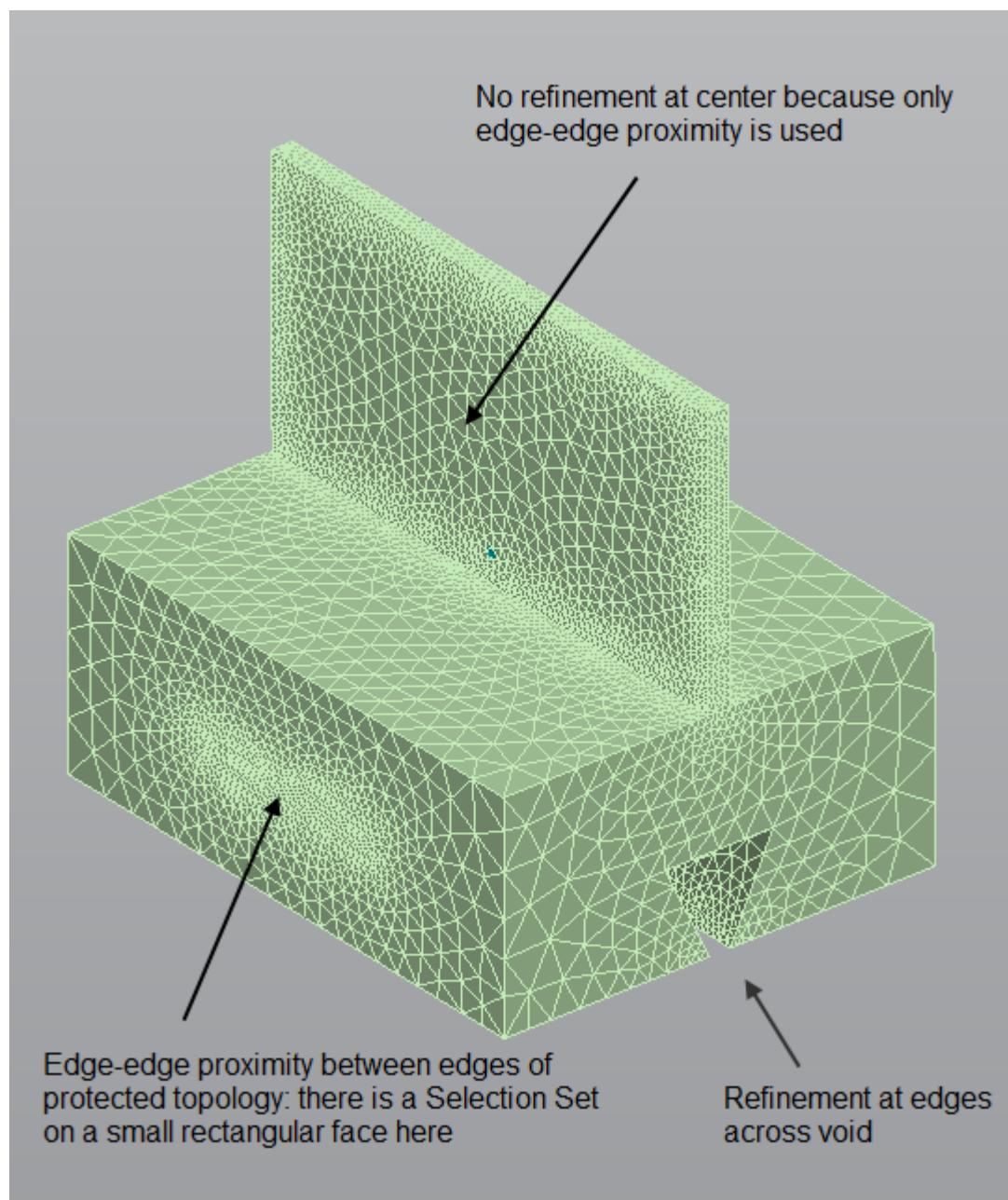
<b>Sources</b>	<b>Edges</b>	Considers edge-edge proximity. Face-face and face-edge proximity are not considered.
	<b>Faces</b>	This is the default. Considers face-face proximity between faces. Face-edge and edge-edge proximity are not considered (that is, the trailing edge of fluid around wings will not be captured with this setting).
	<b>Faces and edges</b>	Considers face-face and edge-edge proximity. Face-edge proximity is not considered.

#### Note:

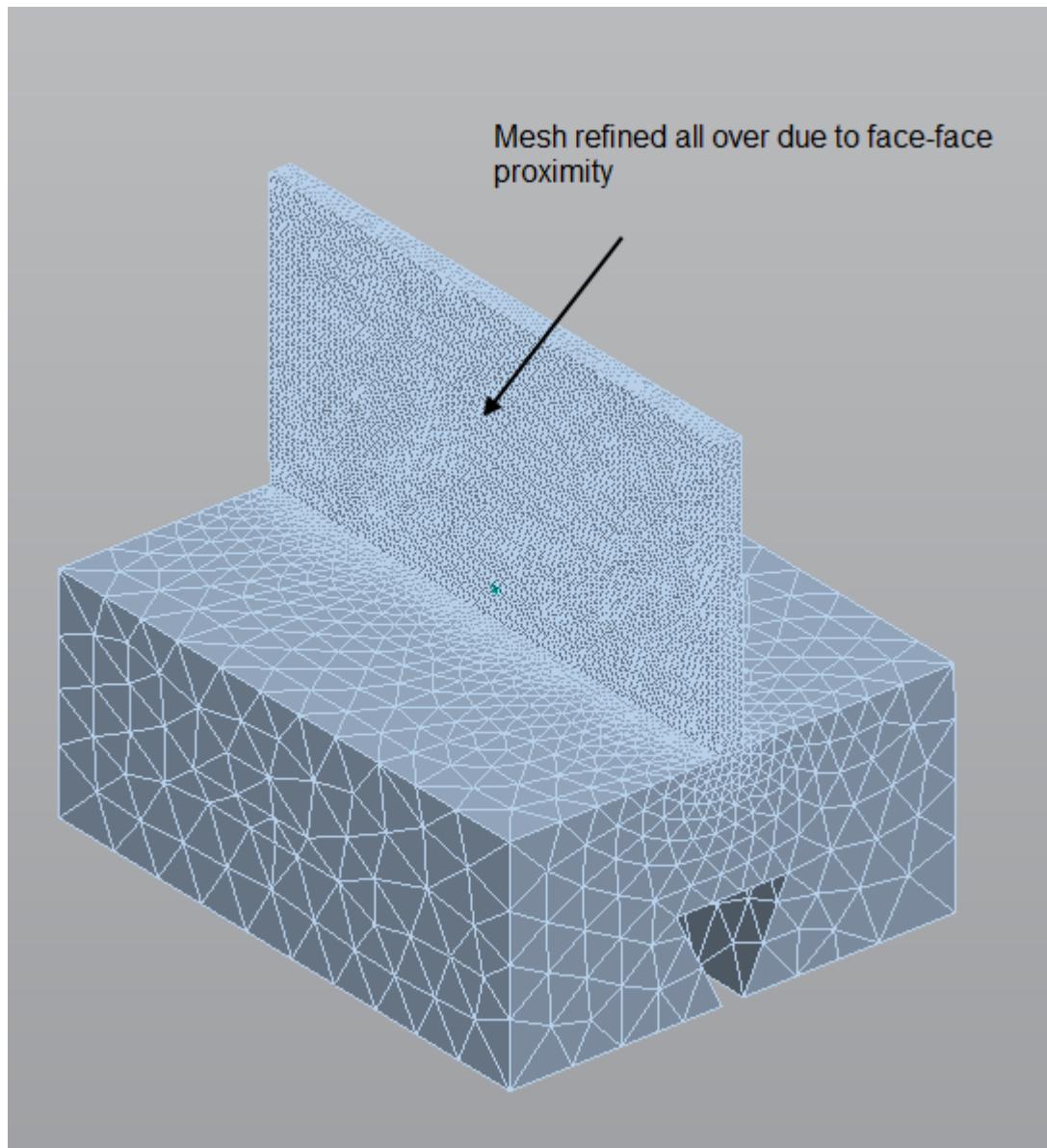
- In cases involving face-face proximity, the face normal orientation is ignored during the proximity calculation.
- In cases involving edge-edge proximity, edges across voids in a model are refined by wrapping because the volume exists at the time the refinement occurs.
- For many models, the **Edges** setting may be sufficient to resolve all proximity situations. For large complex models, using either the **Faces and edges** or **Faces** setting may result in longer computation time.

The figures below illustrate the effect of each **Proximity size function sources** setting.

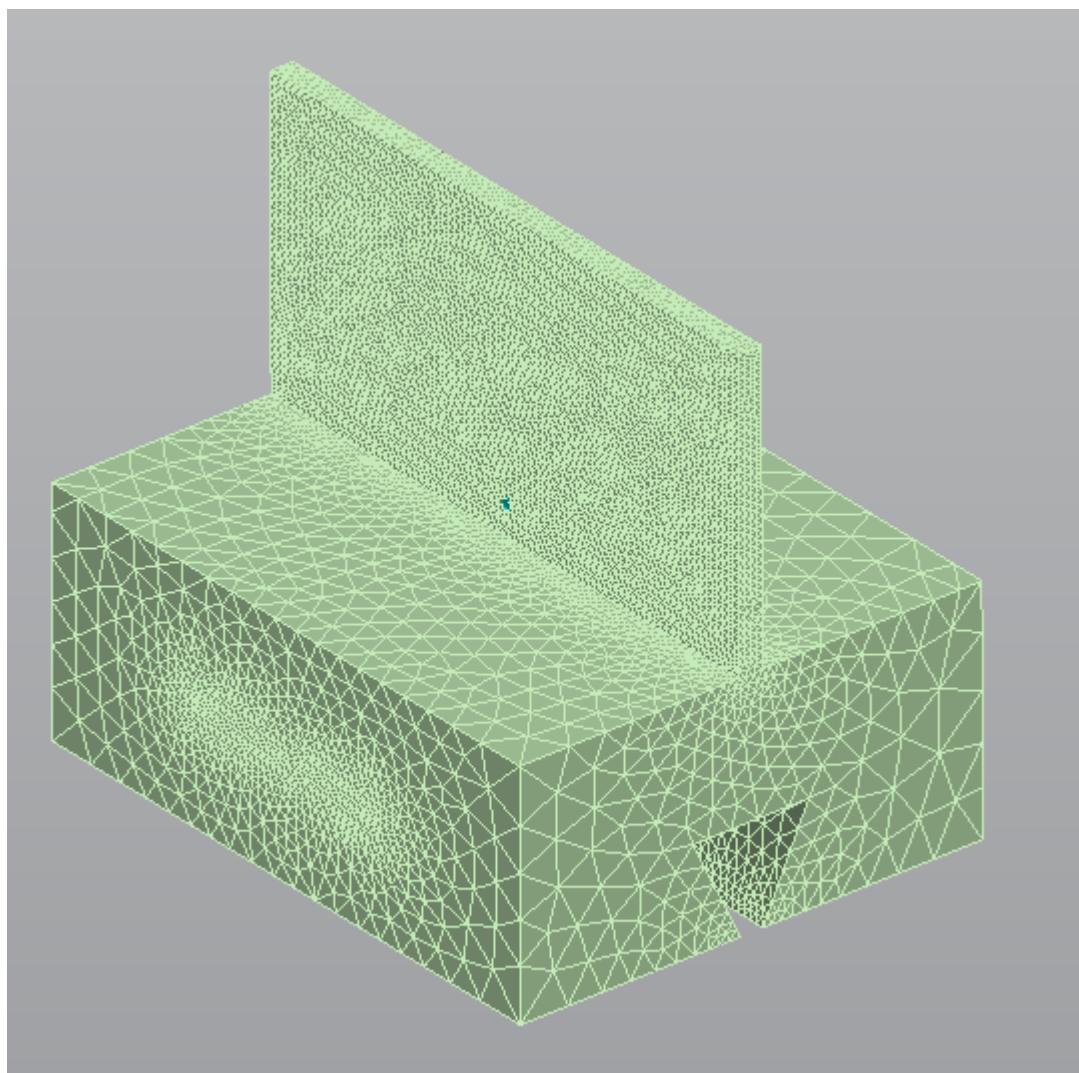
**Figure 3.3.7.3.1.1. Proximity Size Function Sources = Edges**



**Figure 3.3.7.3.1.2. Proximity Size Function Sources = Faces**



**Figure 3.3.7.3.1.3. Proximity Size Function Sources = Faces and Edges**



**Minimum Size** **Minimum size** is the minimum size that the size function returns to the mesher. Some element sizes may be smaller than this based on local feature sizes or other geometric anomalies. Specify a value greater than 0 or accept the default.

**Proximity Minimum Size** Use **Proximity minimum size** to specify the global minimum size to be used in proximity size function calculations. This is in addition to your specification of a global **Minimum size**, which is used in curvature size function calculations. Some element sizes may be smaller than this based on local feature sizes or other geometric anomalies. By default, **Proximity minimum size** is set equal to the default of **Minimum size**. You can accept the default or specify a value greater than 0.

Any feature that operates based on minimum element size will be based on the smaller of the two minimum size values; for example, the **Tessellation refinement** option that is used in **Volume Creation** tasks.

**Maximum Size** **Maximum size** is the maximum size that the size function returns to the mesher. Specify a value greater than 0 or accept the default.

**Growth Rate** **Growth rate** represents the increase in element edge length with each succeeding layer of elements. For example, a growth rate of 1.2 results in a 20% increase in element edge length with each succeeding layer of elements. Specify a value from 1.0 to 5.0 or accept the default. The default is 1.2.

<b>Curvature Normal Angle</b>	<b>Curvature normal angle</b> is the maximum allowable angle that one element edge is allowed to span. Enter a positive value of up to 180 degrees or 3.14 radians, or accept the default. The default is 18 degrees or 0.3142 radians.
<b>Number of Cells Across Gap</b>	<p><b>Number of cells across gap</b> is the minimum number of layers of elements to be generated in the gaps. You can specify a value from 1 to 100 or accept the default. The default is 3.</p> <p>The value of <b>Number of cells across gap</b> is an approximation:</p> <ul style="list-style-type: none"> <li>• If you specify 3 cells per gap on a narrow face, the final mesh may contain from 2–4 cells across the gap, depending on the orientation in relation to the global X, Y, Z axis.</li> <li>• You must define minimum size such that it allows the specified <b>Number of cells across gap</b> to be achieved. For example, if you have a gap of 1 mm, the minimum size is 0.5 mm, and the value of <b>Number of cells across gap</b> is 3, only 2 cells can be generated in the gap.</li> </ul>

### 3.3.7.3.2. Global Sizing for Volume Creation with the Curvature Size Function Method

The size function controls how the mesh size is distributed on a surface or within a volume. The properties described here provide more precise control over mesh size distribution during **Volume Creation** tasks when **Size function method** is set to **Curvature**.

For information about overriding size function sizes, refer to [Setting Mesh Sizes to Optimize Performance](#) on page 113.

<b>Minimum Size</b>	<b>Minimum size</b> is the minimum size that the size function returns to the mesher. Some element sizes may be smaller than this based on local feature sizes or other geometric anomalies. Specify a value greater than 0 or accept the default.
<b>Maximum Size</b>	<b>Maximum size</b> is the maximum size that the size function returns to the mesher. Specify a value greater than 0 or accept the default.
<b>Growth Rate</b>	<b>Growth rate</b> represents the increase in element edge length with each succeeding layer of elements. For example, a growth rate of 1.2 results in a 20% increase in element edge length with each succeeding layer of elements. Specify a value from 1.0 to 5.0 or accept the default. The default is 1.2.
<b>Curvature Normal Angle</b>	<b>Curvature normal angle</b> is the maximum allowable angle that one element edge is allowed to span. Enter a positive value of up to 180 degrees or 3.14 radians, or accept the default. The default is 18 degrees or 0.3142 radians.

### 3.3.7.3.3. Global Sizing for Volume Creation with the Proximity Size Function Method

The size function controls how the mesh size is distributed on a surface or within a volume. The properties described here provide more precise control over mesh size distribution during **Volume Creation** tasks when **Size function method** is set to **Proximity**.

For information about overriding size function sizes, refer to [Setting Mesh Sizes to Optimize Performance](#) on page 113.

**Proximity Size Function Sources** determines whether regions of proximity between faces and/or edges are considered when proximity size function calculations are performed. Choose one of the following:

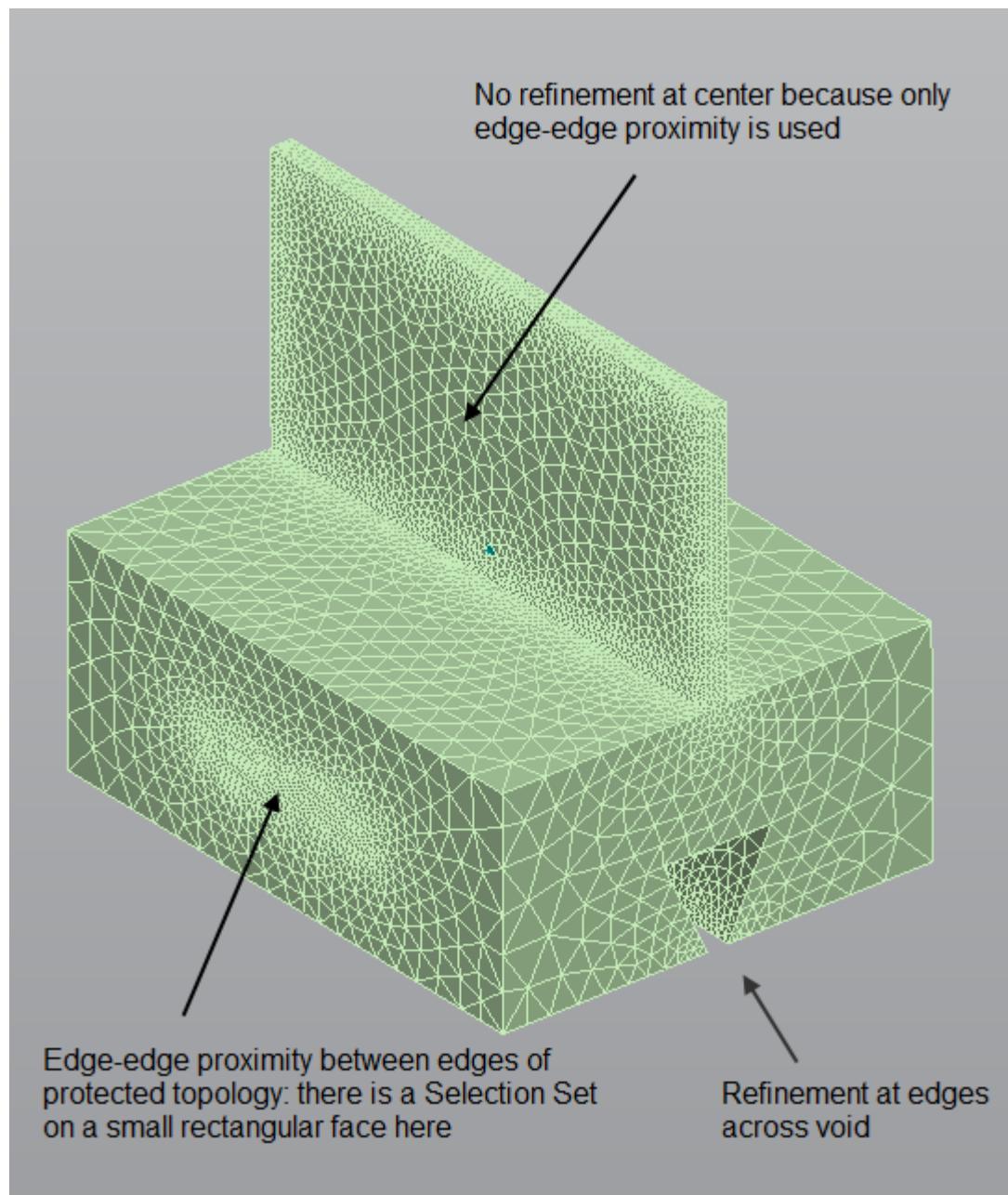
<b>Sources</b>	<b>Edges</b>	Considers edge-edge proximity. Face-face and face-edge proximity are not considered.
	<b>Faces</b>	This is the default. Considers face-face proximity between faces. Face-edge and edge-edge proximity are not considered (that is, the trailing edge of fluid around wings will not be captured with this setting).
	<b>Faces and edges</b>	Considers face-face and edge-edge proximity. Face-edge proximity is not considered.

**Note:**

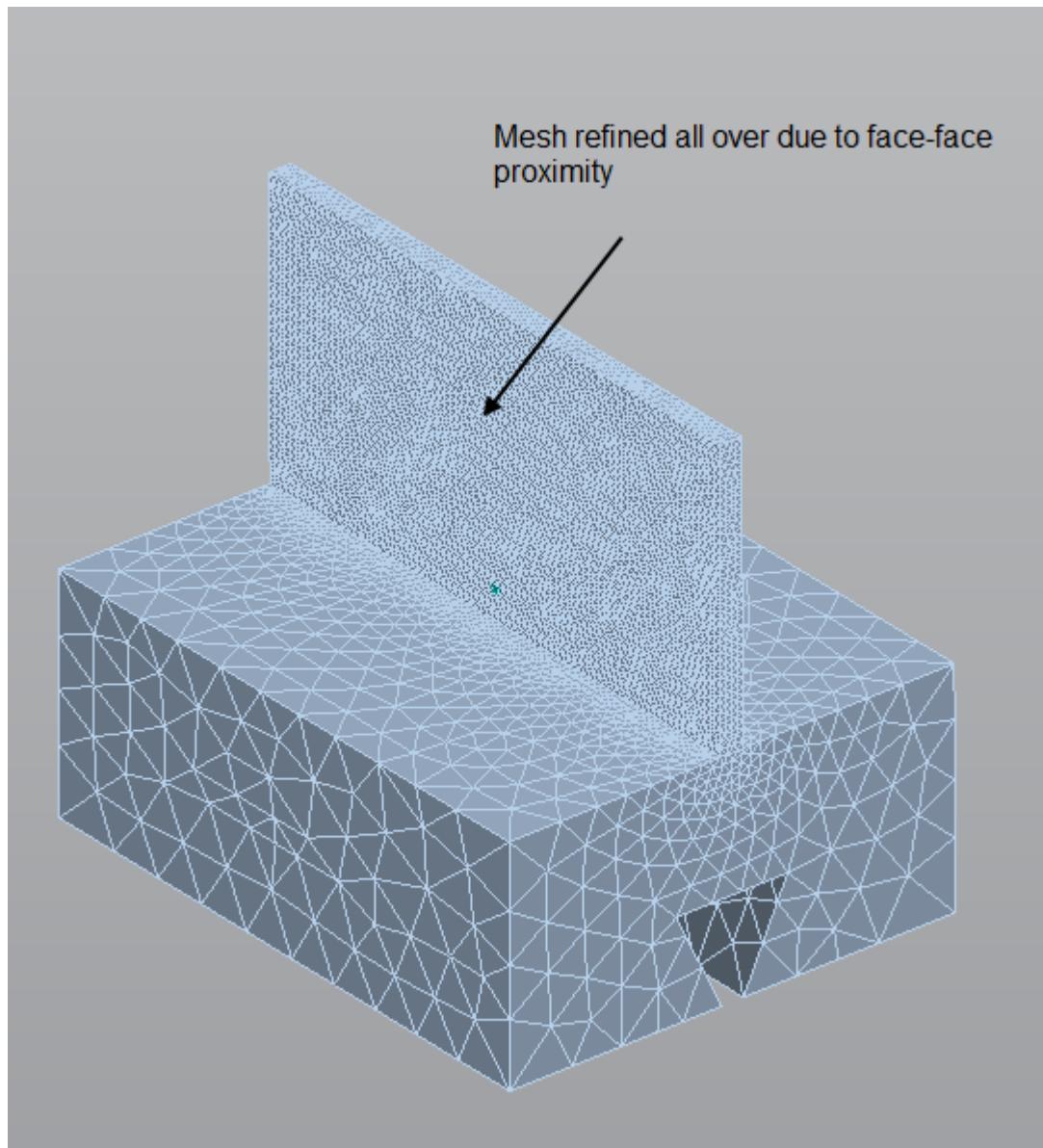
- In cases involving face-face proximity, the face normal orientation is ignored during the proximity calculation.
- In cases involving edge-edge proximity, edges across voids in a model are refined by wrapping because the volume exists at the time the refinement occurs.
- For many models, the **Edges** setting may be sufficient to resolve all proximity situations. For large complex models, using either the **Faces and edges** or **Faces** setting may result in longer computation time.

The figures below illustrate the effect of each **Proximity size function sources** setting.

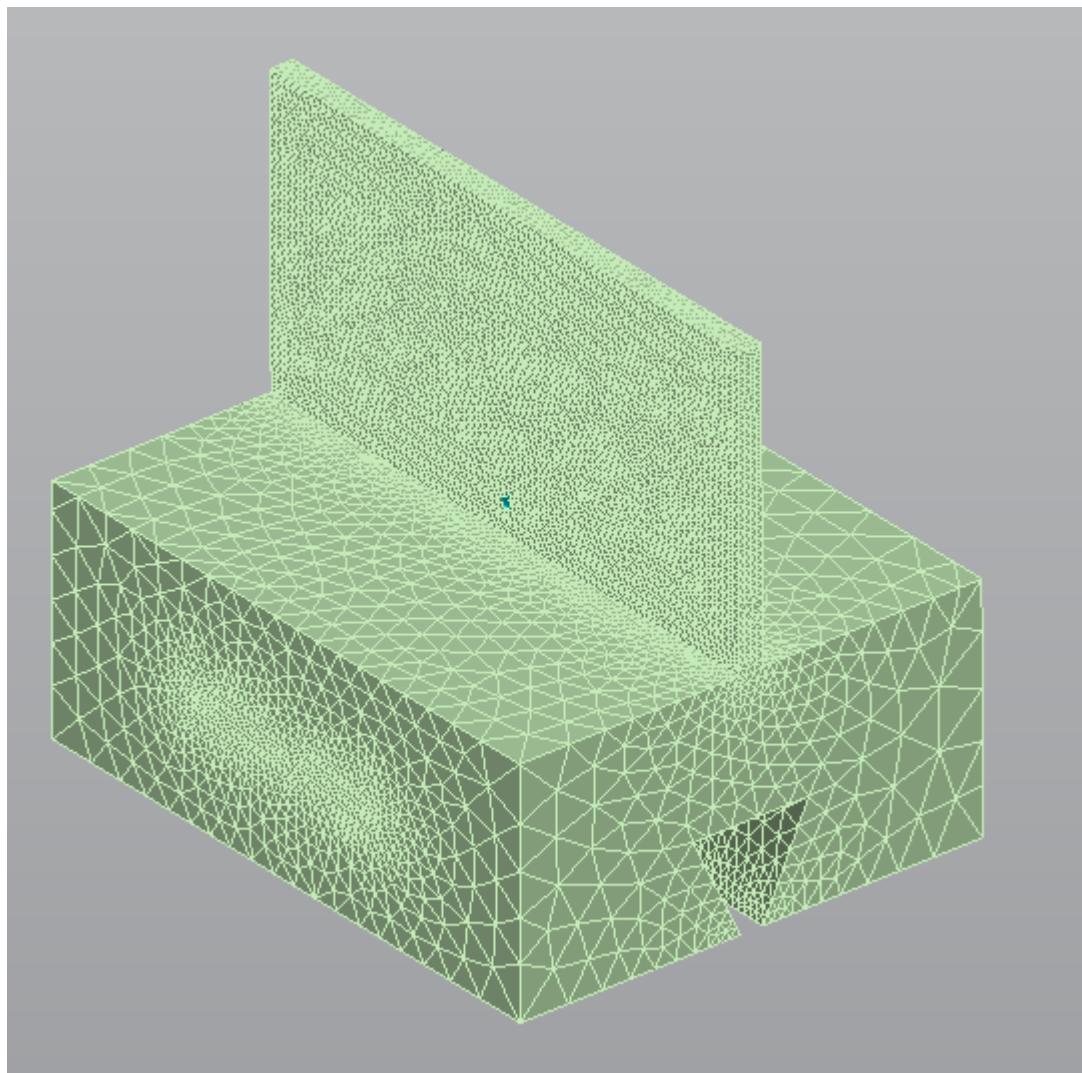
**Figure 3.3.7.3.3.1. Proximity Size Function Sources = Edges**



**Figure 3.3.7.3.3.2. Proximity Size Function Sources = Faces**



**Figure 3.3.7.3.3.3. Proximity Size Function Sources = Faces and Edges**



**Proximity Minimum Size** Use **Proximity minimum size** to specify the global minimum size to be used in proximity size function calculations. Some element sizes may be smaller than this based on local feature sizes or other geometric anomalies. You can accept the default or specify a value greater than 0.

**Maximum Size** **Maximum size** is the maximum size that the size function returns to the mesher. Specify a value greater than 0 or accept the default.

**Growth Rate** **Growth rate** represents the increase in element edge length with each succeeding layer of elements. For example, a growth rate of 1.2 results in a 20% increase in element edge length with each succeeding layer of elements. Specify a value from 1.0 to 5.0 or accept the default. The default is 1.2.

**Number of Cells Across Gap** **Number of cells across gap** is the minimum number of layers of elements to be generated in the gaps. You can specify a value from 1 to 100 or accept the default. The default is 3.

The value of **Number of cells across gap** is an approximation:

- If you specify 3 cells per gap on a narrow face, the final mesh may contain from 2–4 cells across the gap, depending on the orientation in relation to the global X, Y, Z axis.
- You must define minimum size such that it allows the specified **Number of cells across gap** to be achieved. For example, if you have a gap of 1 mm, the minimum size is 0.5 mm, and the value of **Number of cells across gap** is 3, only 2 cells can be generated in the gap.

### 3.3.7.3.4. Global Sizing for Volume Creation with the Fixed Size Function Method

The size function controls how the mesh size is distributed on a surface or within a volume. The properties described here provide more precise control over mesh size distribution during **Volume Creation** tasks when **Size function method** is set to **Fixed**.

For information about overriding size function sizes, refer to [Setting Mesh Sizes to Optimize Performance](#) on page 113.

<b>Minimum Size</b>	<b>Minimum size</b> is the minimum size that the size function returns to the mesher. Some element sizes may be smaller than this based on local feature sizes or other geometric anomalies. Specify a value greater than 0 or accept the default.
<b>Maximum Size</b>	<b>Maximum size</b> is the maximum size that the size function returns to the mesher. Specify a value greater than 0 or accept the default.
<b>Growth Rate</b>	<b>Growth rate</b> represents the increase in element edge length with each succeeding layer of elements. For example, a growth rate of 1.2 results in a 20% increase in element edge length with each succeeding layer of elements. Specify a value from 1.0 to 5.0 or accept the default. The default is 1.2.

### 3.3.7.4. Additional Settings for Volume Creation



These settings are available for **Volume Creation** tasks if you turn Filtering off.

Option	Definition
<b>Tessellation refinement</b>	Determines whether tessellation refinement occurs and to what extent. Tessellation refinement refines the display facets to try to get a better approximation of the true geometry. Choose one of the following: <ul style="list-style-type: none"> <li><b>Automatically determined</b> This is the default. Sets tessellation refinement to 10% of the value of <b>Minimum size/Proximity minimum size</b> (whichever is smaller). This is the recommended value for most cases. The value is recomputed automatically if you change the <b>Size function method</b>, <b>Minimum size</b>, or <b>Proximity minimum size</b> value.</li> <li><b>User defined</b> Enables you to specify a numerical <b>Refinement tolerance</b> value. The recommended range is between 5 and 10% of the value of <b>Minimum size/Proximity minimum size</b> (whichever is smaller). A value on the lower end of the range may work better if you have problems with gaps.</li> <li><b>Off</b> Uses the geometry tessellation, which may not accurately capture curved areas. This is the fastest option because no tessellation processing occurs.</li> </ul>
<b>Edge extraction angle</b>	Determines which CAD features are captured. Specify a value from 0 to 90 degrees or accept the default. The smaller the angle, the higher the number of features that are captured. If you specify a value greater than 90, an edge extraction angle of 90 is used. Setting the value to 0 captures all features, while removing any value reinitializes the default. The default is 40, such that if the shared faces on an edge form an angle smaller than (180 - 40) degrees, the edge is selected for meshing.
<b>Create intersecting</b>	Determines whether the intersection between faces is computed in cases where two parts/bodies overlap in space. When activated (set to <b>Yes</b> ), computes additional feature

Option	Definition
<b>edges for overlapping bodies</b>	edges to be respected during the snapping that occurs within wrapping. Activating this feature is very useful for avoiding zigzag boundaries at an intersection, because it ensures that the "real" intersection lines are respected. However, this operation can be computationally expensive, so you should deactivate it if you have many non-intersecting bodies in the model.

### 3.3.7.5. Troubleshooting Volume Creation Problems

This information provides tips and strategies for avoiding and handling problems that may occur when using **Volume Creation** tasks.

### Adjusting Tessellation Refinement

- If you receive a warning about missing tessellations, it may help to lower the [tessellation refinement tolerance](#) by 50%.
- In some cases, small defects in the faceting may lead to bad quality meshes. In many of these cases, a minor modification of the tessellation refinement tolerance can rectify the problem.

### Capturing Features and Resolving Gaps

The shrink wrap method employed by **Volume Creation** tasks uses a Cartesian grid. As detailed in [Volume Creation](#) on page 116, the method extracts the interface on the boundary of the non-intersecting Cartesian volume region and projects the extracted interface onto the geometry. For this region to be considered proper, it must cover all the required thin sections so that it can recover accurately those sections in the mesh. If the current size function settings would generate fewer than three elements across a thin section, then all of those elements may be intersected by the geometry surfaces and may be ignored, resulting in the thin section being ignored.

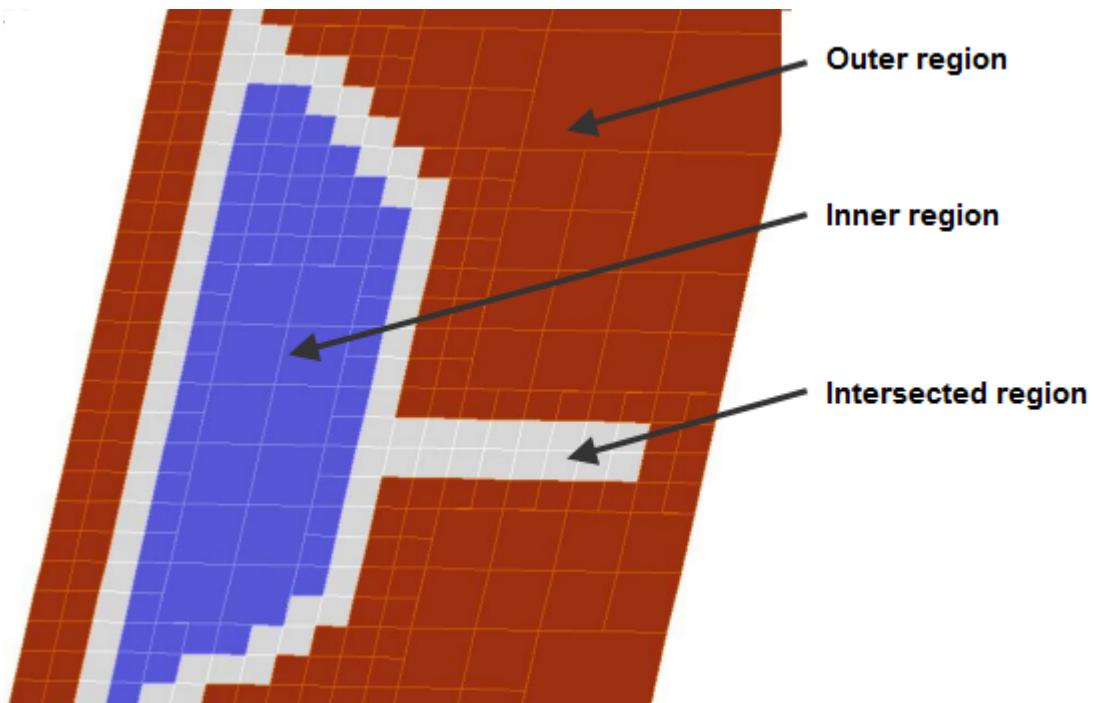
The figure below shows a model to be wrapped.

**Figure 3.3.7.5.1. Model to Be Wrapped**



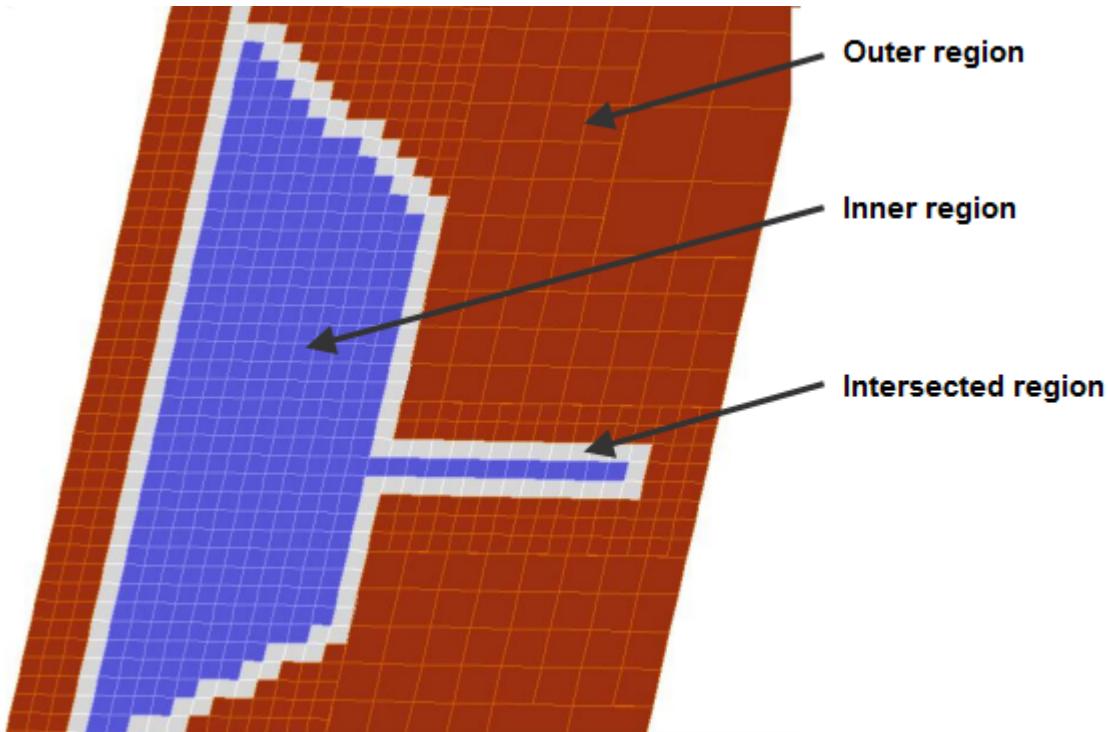
The figure below shows a grid for which coarse sizing is specified, where the purple elements make up the inner region, the gray elements make up the intersected region, and the red elements make up the outer region. Because there are fewer than three elements across the thin section, the thin section is ignored.

**Figure 3.3.7.5.2. Coarse Grid and Wrapping**



Finer sizing is specified for the grid in the figure below. There are three elements across the thin section, which in this case is preserved.

**Figure 3.3.7.5.3. Finer Grid and Wrapping**



After the shrink wrap method refines the background Cartesian grid based on the size function, it scales the grid according to the value of the **Feature resolution** option, which is set to 0.75 by default. You can set a smaller value for **Feature resolution** to adjust the background grid to preserve the thin sections. Use caution though, as a smaller value will generate more elements and impact memory. For example, changing the value from its default of 0.75 to 0.5 generates four times as many elements.

## Repairing Leaks

When the mesh for a **Volume Creation** task is generated, holes or gaps in surfaces or between bodies may result in “leaks” to the outside of a flow volume, such that the extracted flow volume would not be watertight. When a leak is present, the **Volume Creation** task may fail to extract the flow volume region for the specified **Point inside flow region**.

Leak tracing occurs from the bounding box to the **Point inside flow region**. When AIM detects a leak, it displays the leak path in the graphics view, an error message is issued, and the mesh is considered invalid. The leak path should indicate the leak's location.

**Note:** If a message indicates there is a leak but the leak path does not appear in the graphics view, you may need to click **View task input** to see the leak path.

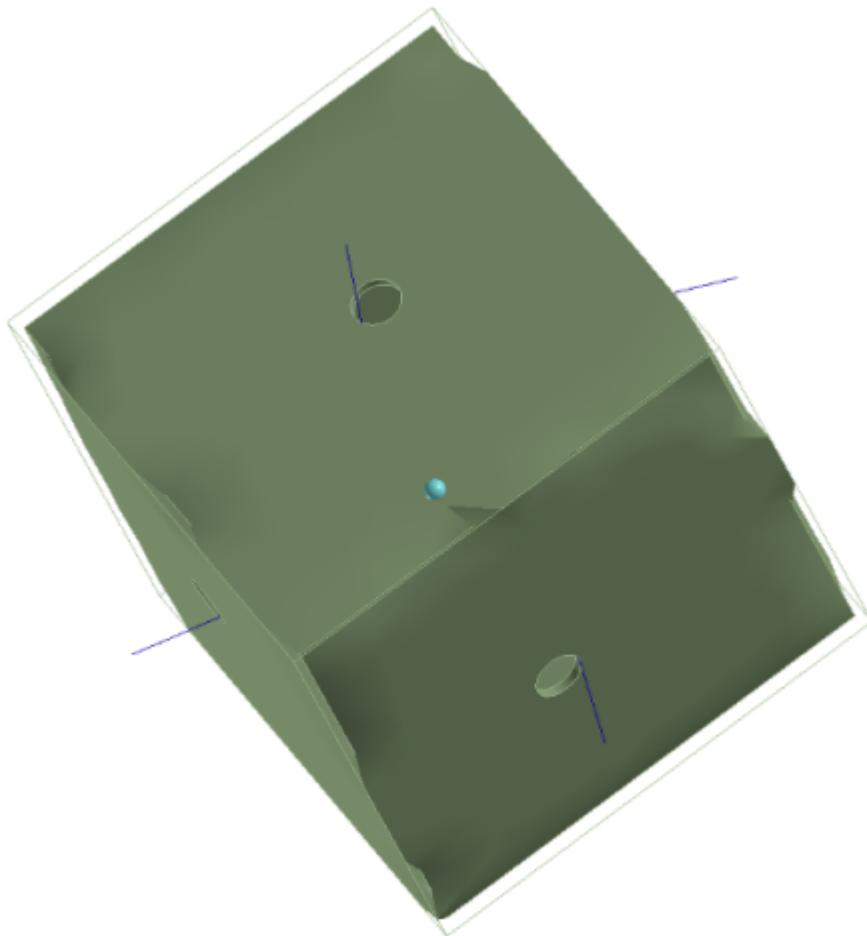
You must repair the leak before you can generate the mesh successfully. Common causes of leaks and recommended actions to repair them include:

- A capping surface is missing. Use **Objects > Construction Geometry > Add > Capping Surface** to add a capping face.
- There is a gap between bodies or faces. Return the model to the ANSYS DesignModeler application or your CAD package to close the gap (for example, by moving a body or adding a face).

- Low facet quality introduces a higher probability for leaking. Increasing facet quality may help reduce leakage.

Leak paths are displayed graphically as line bodies, as illustrated by the figure below.

**Figure 3.3.7.5.4. Leak Path**



## Understanding Volume Creation and Mesh State

When your simulation contains **Volume Creation** tasks, making certain changes causes the mesh state of the entire configuration to become underdefined and require a remesh (assuming the configuration was meshed prior to the change). Changes that affect mesh state include:

- Changes to **Volume Creation** task properties
- Creation or deletion of a **Selection Set** in a **Volume Creation** task
- Changes to volume definition properties (either **Extracted Volume** or **Geometry Simplification**)
- Changes to a **Point inside flow region** that is used by an **Extracted Volume**
- Changes to a **boundary layer control** applied to a connected **Meshing** task
- Changes to topology scoped to a local mesh sizing, local boundary layer, or volume definition, including suppressing or unsuppressing topology
- Changes to **size controls**
- Changes to capping surface location
- Deletions of capping surfaces

- Additions or deletions of mesh controls
- Deletion, suppression, or unsuppression of a body or part that is a member of a Selection Set
- Updates to geometry from source

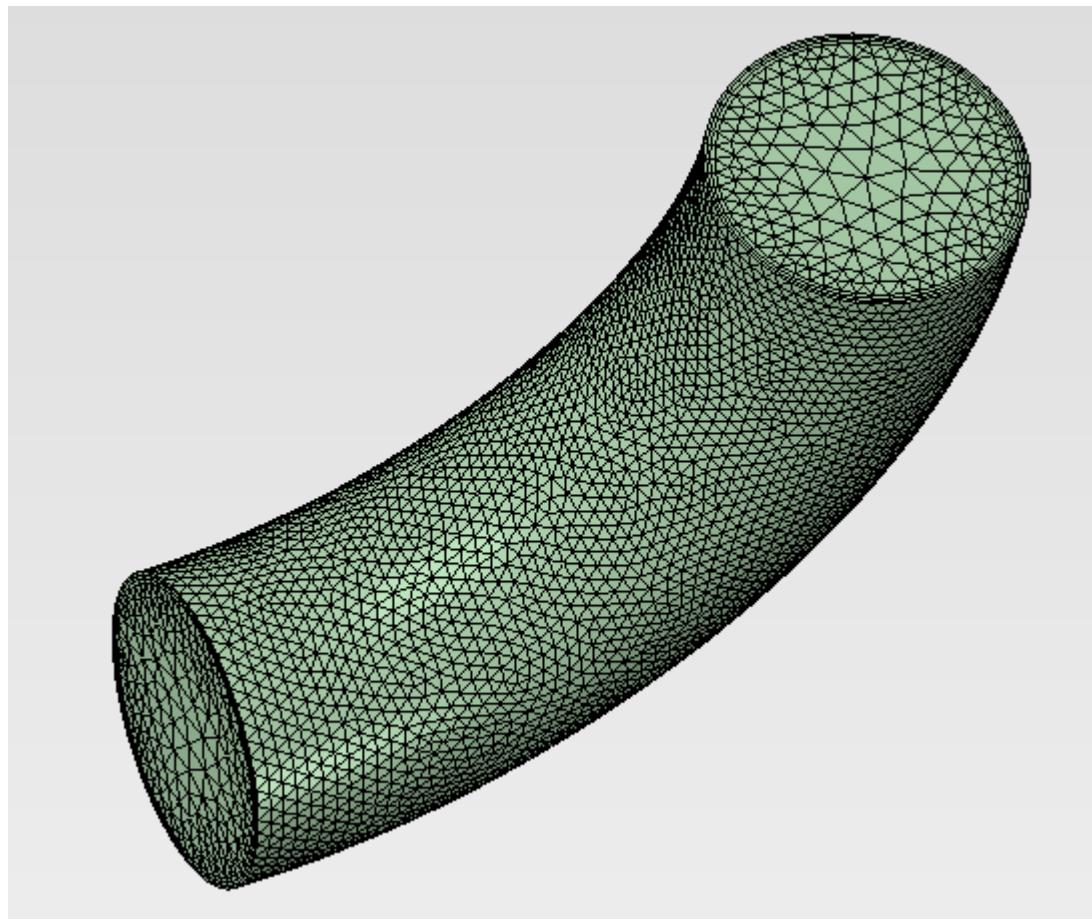
A warning message is issued if a geometry change results in invalid scope.

### 3.3.8. Flow Volume Meshing

To generate a flow volume mesh based on the surface mesh that was output from a [Volume Creation](#) task, add a **Meshing** task for meshing the wrapped geometry. AIM generates a volume mesh consisting of tetrahedral and prism elements.

The **Meshing** task inherits a [size field file](#) from the previous connected task. The size field file contains computed size functions for size properties (such as **Minimum size**, **Maximum size**, and **Growth rate**). You cannot further refine the sizing in the **Meshing** task.

**Figure 3.3.8.1. Flow Volume Mesh with Boundary Layers**



### 3.3.8.1. Common Boundary Layer Settings for Flow Volume Meshing

For a **Meshing** task that is being used to mesh the output of a [Volume Creation](#) task, these options provide global control over all boundary layers.

# Collision Avoidance

The **Collision avoidance** control determines the approach that is to be taken in areas of proximity to avoid collisions that may occur when surface meshes are layered from opposite sides into each other.

The option that you choose for **Collision avoidance** is used *only in areas of proximity*. In areas of proximity, if the option is set to **Layer compression**, layer compression is performed; if it is set to **Stair stepping**, stair stepping is performed; if it is set to **None**, no collision checking is performed. However, in all other problematic scenarios (for example, invalid normals, quality failure, bad surface mesh, and so on), local stair stepping is performed regardless of which option you choose.

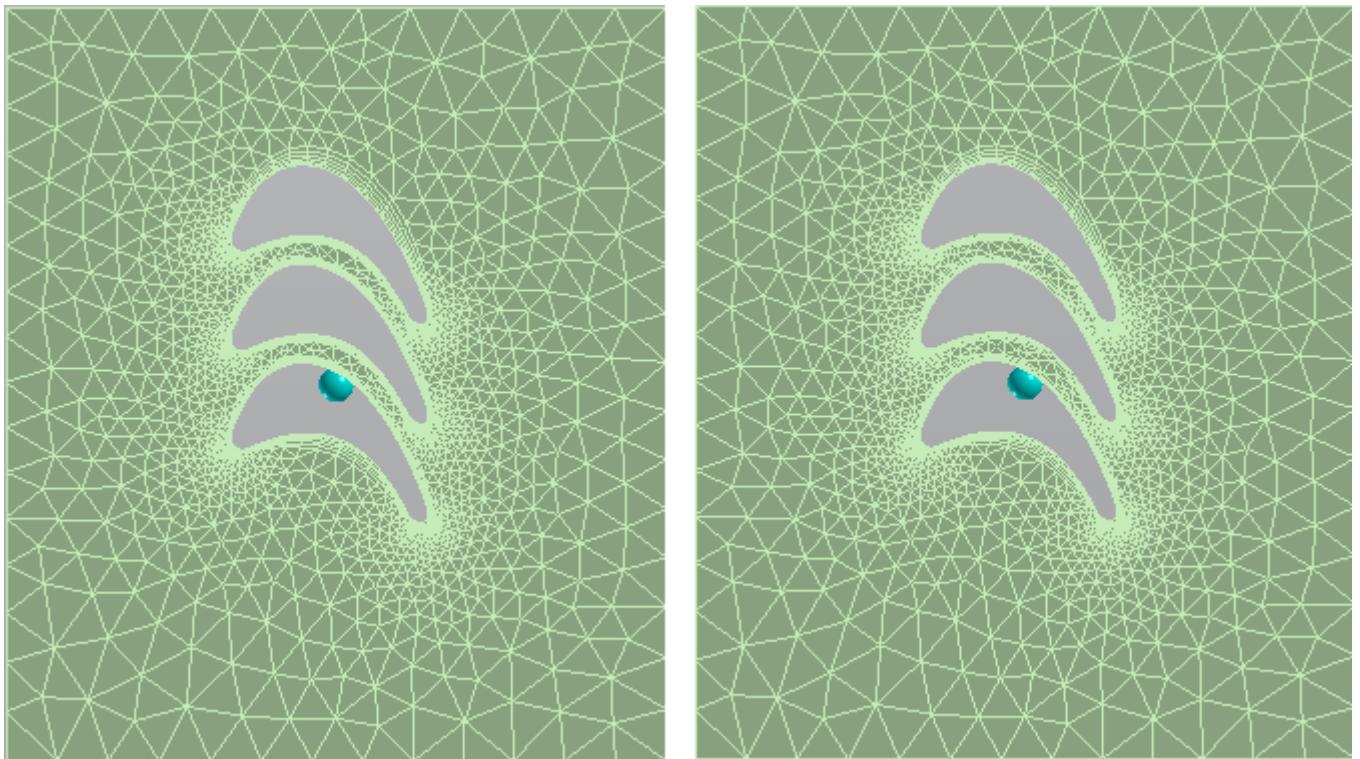
**Note:** When **Collision avoidance** is set to **Layer compression** and local stair stepping occurs after compression, poor quality pyramids may be introduced into the mesh. Because of this possibility, a warning message will appear whenever stair stepping occurs after compression.

Choose one of the following:

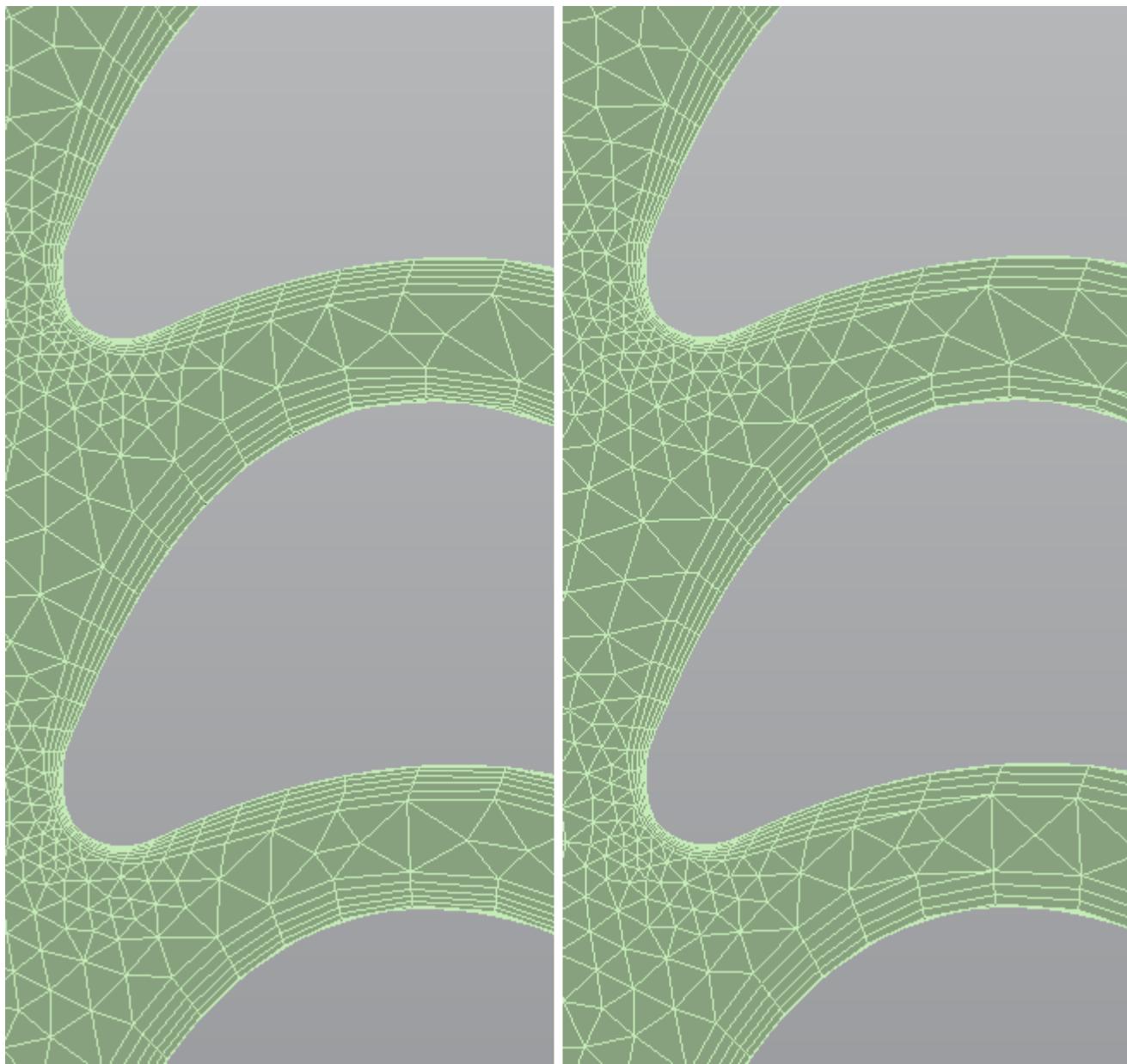
- |                          |  |
|--------------------------|--|
| <b>None</b>              | The <b>None</b> option does not check for layer collisions. Selecting this option speeds up layer computation time; however, it can result in an invalid mesh and mesh failures. For these reasons, this option is not recommended.  |
| <b>Layer compression</b> | The <b>Layer compression</b> option compresses layers in areas of collision. In these areas, the defined heights and ratios are reduced to ensure the same number of layers throughout the entire boundary layer region. Generally, this option is best for avoiding the creation of pyramids in the mesh.   |
| <b>Stair stepping</b>    | This is the default. Rather than compressing the prism layers, with <b>Stair stepping</b> the prism layers are “stair stepped” in the proximity region to avoid collision and to maintain the gap defined by the <b>Gap factor</b> control. The <b>Stair stepping</b> approach to layer growth locally reduces layers to avoid collisions, as well as bad quality elements in sharp or tight corners. The term “stair stepping” refers to the steps created between one layer and the next. Using this approach, special logic is used to fill the steps with pyramid and tetrahedron elements for prism steps, or prism, pyramid, and tetrahedron elements for hex steps. This special logic helps the mesher obtain a high-quality transition to the tetrahedral mesh. |

The graphics below illustrate how **Layer compression** and **Stair stepping** differ.

**Figure 3.3.8.1.1. Layer Compression vs. Stair Stepping (View of Entire Mesh)**



**Figure 3.3.8.1.2. Layer Compression vs. Stair Stepping (View of Details in Mesh)**



### 3.3.8.1.1. Optional Boundary Layer Settings for Flow Volume Meshing



These settings are available if you turn Filtering off. For a **Meshing** task that is being used to mesh the output of a **Volume Creation** task, these settings provide greater control over all boundary layers.

#### Fix First Layer

The **Fix first layer** control determines whether the heights or ratios of the first layer will be modified to avoid collision. Valid values are **Yes** and **No**. The default is **No**. This option will not allow the value that is set for the **First layer height** control to be changed.

The **Fix first layer** control is applicable only when **Collision avoidance** is set to **Layer compression**.

## Gap Factor

The **Gap factor** control enables maintenance of the gap between intersecting prisms. Valid values are from 0 to 2. The default is **0.5**. A value of **1** means a gap equal to the ideal tet cell height based on base face size in proximity to each other is maintained.

The **Gap factor** control is applicable only when **Collision avoidance** is set to **Layer compression** or **Stair stepping**.

## Maximum Height Over Base

The **Maximum height over base** control sets the maximum allowable prism aspect ratio (that is, the ratio of height over base of the base triangle). When the prism aspect ratio reaches this value, the height of the prisms stops growing. That is, new prisms continue to form, but the heights of the prisms will not increase. Valid values are from 0.1 to 5. The default is **1**.

## Growth Rate Type

The **Growth rate type** control determines the height of the layers given the initial height and height ratio. Choose one of the following:

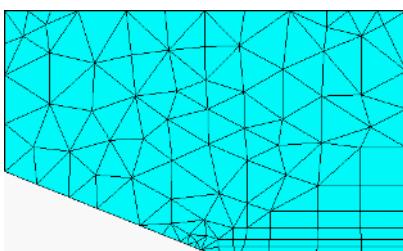
<b>Geometric</b>	This is the default. With this option, the prism height of a particular layer is defined by $h * r^{(n-1)}$ , where $h$ = initial height, $r$ = height ratio, and $n$ = layer number. The total height at layer $n$ is: $h(1-r^n)/(1-r)$ .
<b>Exponential</b>	With this option, the prism height of a particular layer is defined by $h * e^{(n-1)p}$ , where $h$ = initial height, $p$ = exponent, and $n$ = layer number.
<b>Linear</b>	With this option, the prism height of a particular layer is defined by $h(1+(n-1)(r-1))$ , where $h$ = initial height, $r$ = height ratio, and $n$ = layer number. The total height at layer $n$ is $nh((n-1)(r-1)+2)/2$ .

## Maximum Angle

The **Maximum angle** control determines prism layer growth around angles and when prisms will adhere (project) to adjacent surfaces/walls. If the layered mesh involves extruding from one surface and not its neighbor, and the angle between the two surfaces is less than the specified value, the prisms (sides) will adhere (project) to the adjacent wall. Valid values are from 90 to 180 (degrees). Typically, a value between 120 and 180 is desirable. The default is **140** degrees or **2.44** radian. Refer to the figures below for examples of maximum angle.

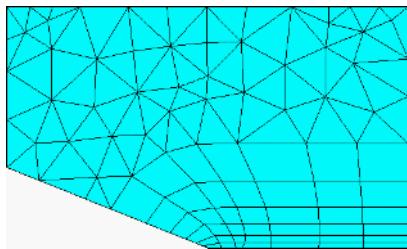
In the figure below, the angle between the planes is 158.2 (21.8) degrees. Since the maximum angle is less than the angle between the walls, the prism layers are capped with pyramids.

**Figure 3.3.8.1.1.1. Maximum Angle = 140**



In the figure below, the maximum angle exceeds the separation angle between the surfaces, so the prism remains attached to the adjacent surface.

**Figure 3.3.8.1.1.2. Maximum Angle = 180**

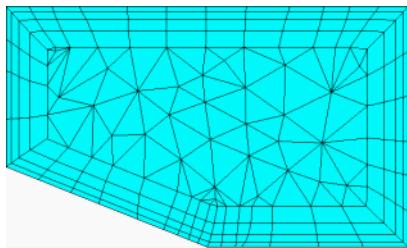


## Fillet Ratio

The **Fillet ratio** control determines whether a fillet proportional to the total height of a prism element will be created when a prism element is generated in the corner region of a tetrahedral mesh. Creating a fillet proportional to the total height of the prism makes it possible to control the smoothness of the prism layer. Valid values are from 0 to 1 (decimal values are allowed). A value of 0 means no fillets. The default is 1. Refer to the figures below for examples of fillet ratio.

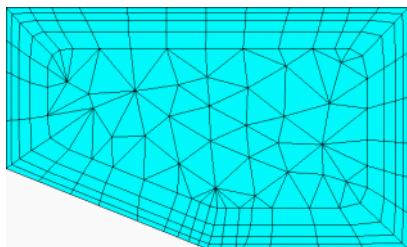
**Note:** For meshing corners with angles less than 60 degrees, there may not be space for a fillet.

**Figure 3.3.8.1.1.3. Fillet Ratio = 0.0**

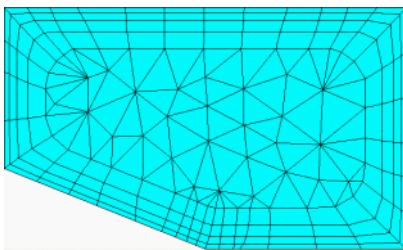


In the figure below, the radius of the inner prism fillet is 0.5 times the height of the total prism thickness.

**Figure 3.3.8.1.1.4. Fillet Ratio = 0.5**



**Figure 3.3.8.1.1.5. Fillet Ratio = 1.0**



## Use Post Smoothing

The **Use post smoothing** control determines whether smoothing will be performed after layers are created. Smoothing attempts to improve element quality by moving the locations of nodes with respect to surrounding nodes and elements. Valid values are **Yes** and **No**. The default is **Yes**. When this control is set to **Yes**, the **Number of iterations** control appears.

## Number of Iterations

The **Number of iterations** control determines the number of smoothing iterations that will be performed to improve the mesh. Valid values are from 1 to 20. The default is **5**.

**Number of iterations** appears only when **Use post smoothing** is set to **Yes**.

### 3.3.9. Part-based Meshing

For structural, thermal, or electric conduction simulations, or for fluid flow simulations where you have an existing flow volume, AIM uses a part-based meshing process in which it meshes the entire part or assembly of parts in **parallel**. Within the **Meshing** task, you have access to all the properties and controls you need to define the mesh.

By default, AIM attempts to mesh sweepable bodies with hexahedrons and provides a tetrahedral mesh on bodies that are not sweepable or if the quality of the hexahedral mesh is poor. Alternatively, you can **specify element shape** to generate a tetrahedral or hexahedral mesh per body. The mesh includes prism elements if boundary layers are generated.

### 3.3.9.1. Usage Notes for Part-based Meshing

- AIM calculates default sizing control values, such as those related to size function methods.

After this calculation occurs, changing the **Input** model to a **Meshing** task does not cause AIM to recalculate these values. Furthermore, any user-defined values are retained for use with the new **Input** model.

Similarly, default sizing values are not recalculated upon a geometry refresh. If you refresh geometry, all global and local mesh sizing controls in the **Meshing** task are retained. If you suppress a body or part and then refresh, any controls associated with the suppressed topology are invalidated.

- To see the effect of specifying multiple sizing controls of the same type on the same entity, use separate **Meshing** tasks. For example, if you want to compare the mesh you will obtain if you specify an **Element size** of 4 on BODY1 to the mesh you will obtain if you specify an **Element size** of 5 on BODY1, use two **Meshing** tasks.
- If you generate the mesh and then change the **Input** model, the **Meshing** task, which includes any associated mesh controls, becomes underdefined.

## 3.3.9.2. Setting Defaults for Part-based Meshing

The **Settings** menu for **Mesh** enables you to set:

- **Use predefined settings**, which provides defaults for **Global Sizing** controls.
- **Engineering intent**, which provides intelligent defaults for the **Shape checking**, **Midside nodes**, and **Size function method** controls based on your physics preference.
- **Model contains thin parts**, which determines whether the mesher uses a curvature size function that has been enhanced for shell meshing and thin solids.
- **Retry on mesh failures**, which determines whether the mesher tries to remesh automatically after a meshing failure.

## Use Predefined Settings

If you enable **Use predefined settings**, you can use the **Mesh resolution slider** to set the fineness of the mesh for the model. You can set the slider toward lower resolution (**Low**) or higher resolution (**High**). A higher resolution provides more accuracy but uses more elements, more time, and more system resources.

The **Mesh resolution** automatically sets all the **Global Sizing** controls except **Size function method**.

Therefore, when **Use predefined settings** is enabled, the **Global Sizing** controls become read-only and cannot be set as parameters. If you subsequently disable **Use predefined settings**, the settings are initially preserved but can be changed.

## Engineering Intent

You must set the **Engineering intent** before you can generate the mesh for the **Meshing** task.

**Note:** The **Structural** and **Fluid Flow** templates create simulations that are populated with physics and solver settings appropriate for the simulation type. If you start your simulation by selecting one of these templates, the **Shape checking**, **Midside nodes**, and **Size function methods** controls for your **Meshing** task are set accordingly.

Choose one of the following:

<b>Structural, thermal or electric conduction</b>	Defaults are optimized for a structural, thermal, or electric conduction simulation. The default for <b>Shape checking</b> is <b>Standard mechanical</b> , the default for <b>Midside nodes</b> is <b>Kept</b> , and the default for <b>Size function method</b> is <b>Curvature</b> .
<b>Fluid flow or fluid-solid heat transfer</b>	Defaults are optimized for a fluid flow or fluid-solid heat transfer simulation. The default for <b>Shape checking</b> is <b>Fluid flow</b> , the default for <b>Midside nodes</b> is <b>Dropped</b> , and the default for <b>Size function method</b> is <b>Curvature</b> .

## Model Contains Thin Parts

If **Use predefined settings** is enabled and **Engineering intent** is set to **Structural, thermal or electric conduction**, you can also enable **Model contains thin parts** to instruct the mesher to increase the mesh resolution for structural meshes containing shell elements or thin solid parts. With this setting enabled, the mesher uses a curvature size function that has been enhanced for shell meshing and thin solids.

## Retry on Mesh Failures

If **Use predefined settings** is enabled, you can also enable **Retry on mesh failures** to instruct the mesher to retry mesh generation automatically if there is a failure due to poor mesh quality. With each retry, the mesher increases the fineness of the mesh in an effort to obtain a good mesh.

### 3.3.9.3. Parallel Part Meshing Best Practices

For part-based meshing, you can control parallel processing operations through the **Additional Settings > Use all processors** option. See [Additional Settings for Part-based Meshing](#) on page 165 for more information.

For the most efficient use of machine resources, it is important that the running processes do not oversaturate the processing cores or the available memory. You must allocate processing cores to each of these mechanisms in a way that provides the most benefit for your workflow. When parallel part meshing is invoked with the default number of CPUs, it automatically uses the cores of all available CPUs with the inherent limitation of 2 gigabytes per CPU core.

Best practices include:

- Know how many physical processing cores are available.
- For Design Point updates, meshing is done serially by default.
- For parallel part meshing we recommend turning off hyper-threading as this may lead to degradation of parallel performance.

### 3.3.9.4. Global Sizing for Part-based Meshing

The size function controls how the mesh size is distributed when a mesh is generated. You can enable the **Settings > Use predefined settings** control to automatically set the fineness of the mesh, or disable it to set individual **Global Sizing** properties manually. In either case, you can set the **Global Sizing > Size function method** control according to your preference for mesh size distribution calculations. You determine which refinement mechanisms are activated by selecting **Curvature and proximity**, **Proximity**, **Curvature**, **Fixed**, or **Adaptive**. For details about the properties you can set for each size function method, refer to the appropriate topic:

- [Setting Defaults for Part-based Meshing](#) on page 148
- [Global Sizing for Part-based Meshing with the Curvature and Proximity Size Function Method](#) on page 149
- [Global Sizing for Part-based Meshing with the Curvature Size Function Method](#) on page 154
- [Global Sizing for Part-based Meshing with the Proximity Size Function Method](#) on page 155
- [Global Sizing for Part-based Meshing with the Fixed Size Function Method](#) on page 159
- [Global Sizing for Part-based Meshing with the Adaptive Size Function Method](#) on page 160

#### 3.3.9.4.1. Global Sizing for Part-based Meshing with the Curvature and Proximity Size Function Method

The size function controls how the mesh size is distributed on a surface or within a volume. For **part-based meshing**, the default **Size function method** is based on the value of the **Engineering intent** control. These properties provide more precise control over mesh size distribution during **Meshing** tasks when **Size function method** is set to **Curvature and proximity**.

For information about overriding size function sizes, refer to [Setting Mesh Sizes to Optimize Performance](#) on page 113.

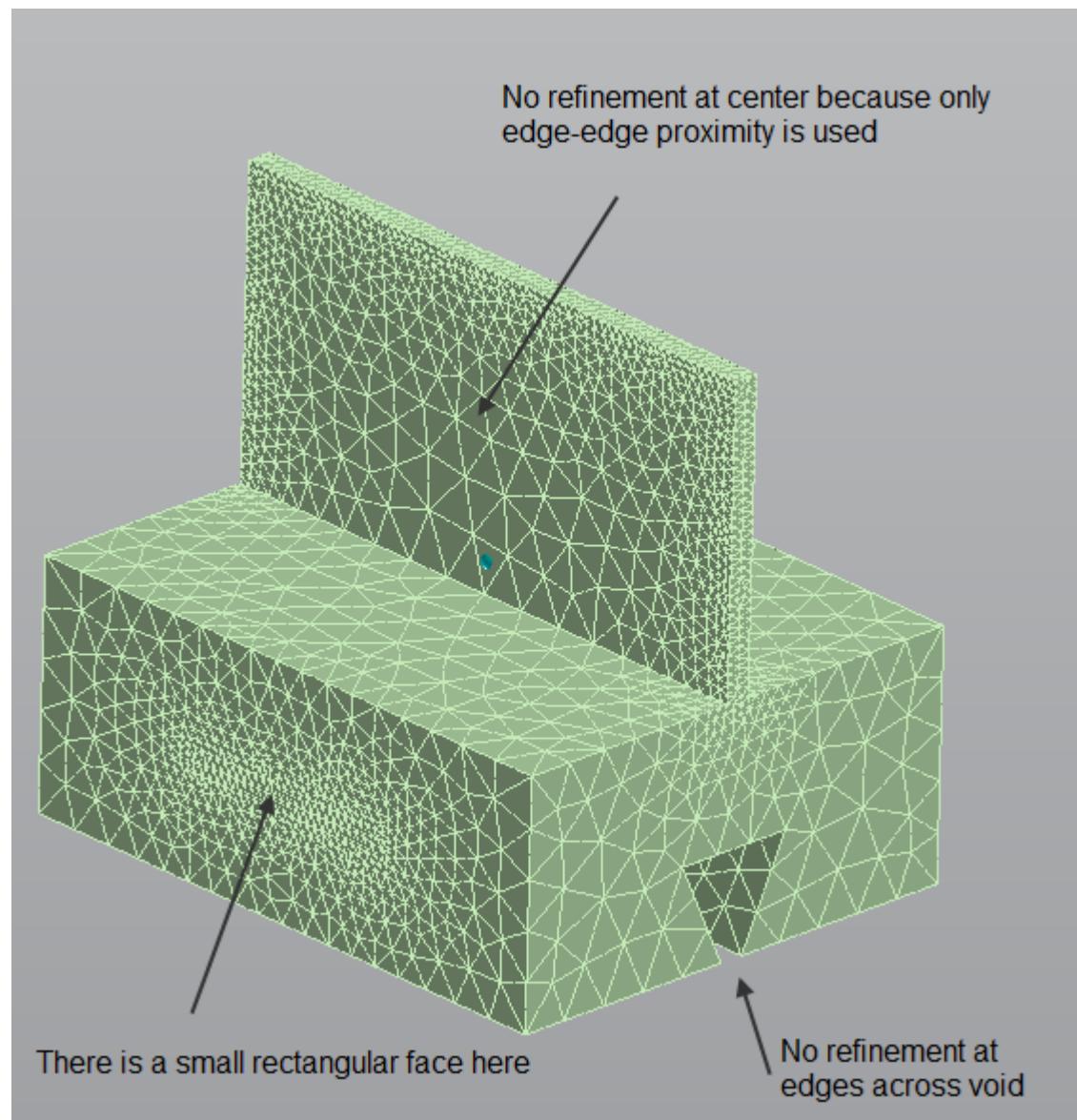
<b>Proximity Size Function Sources</b>	<b>Proximity size function sources</b> determines whether regions of proximity between faces and/or edges are considered when proximity size function calculations are performed. Choose one of the following:
<b>Edges</b>	Considers edge-edge proximity. Face-face and face-edge proximity are not considered.
<b>Faces</b>	This is the default. Considers face-face proximity between faces. Face-edge and edge-edge proximity are not considered (that is, the trailing edge of fluid around wings will not be captured with this setting).
<b>Faces and edges</b>	Considers face-face and edge-edge proximity. Face-edge proximity is not considered.

**Note:**

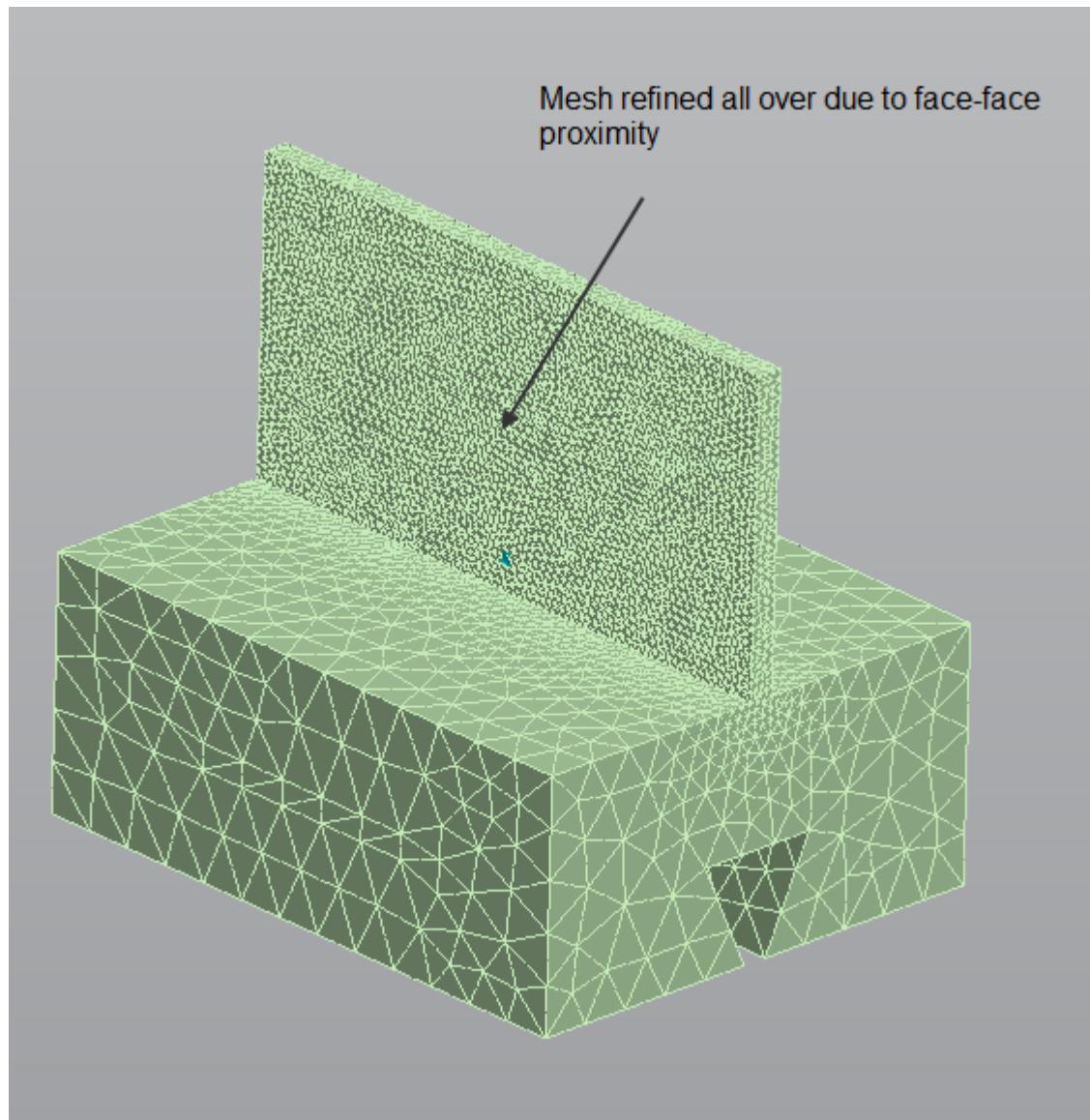
- In cases involving face-face proximity, the face normal orientation is ignored during the proximity calculation.
- In cases involving edge-edge proximity, edges across voids in a model are not refined by part-based meshing.
- For many models, the **Edges** setting may be sufficient to resolve all proximity situations. For large complex models, using either the **Faces and edges** or **Faces** setting may result in longer computation time.

The figures below illustrate the effect of each **Proximity size function sources** setting.

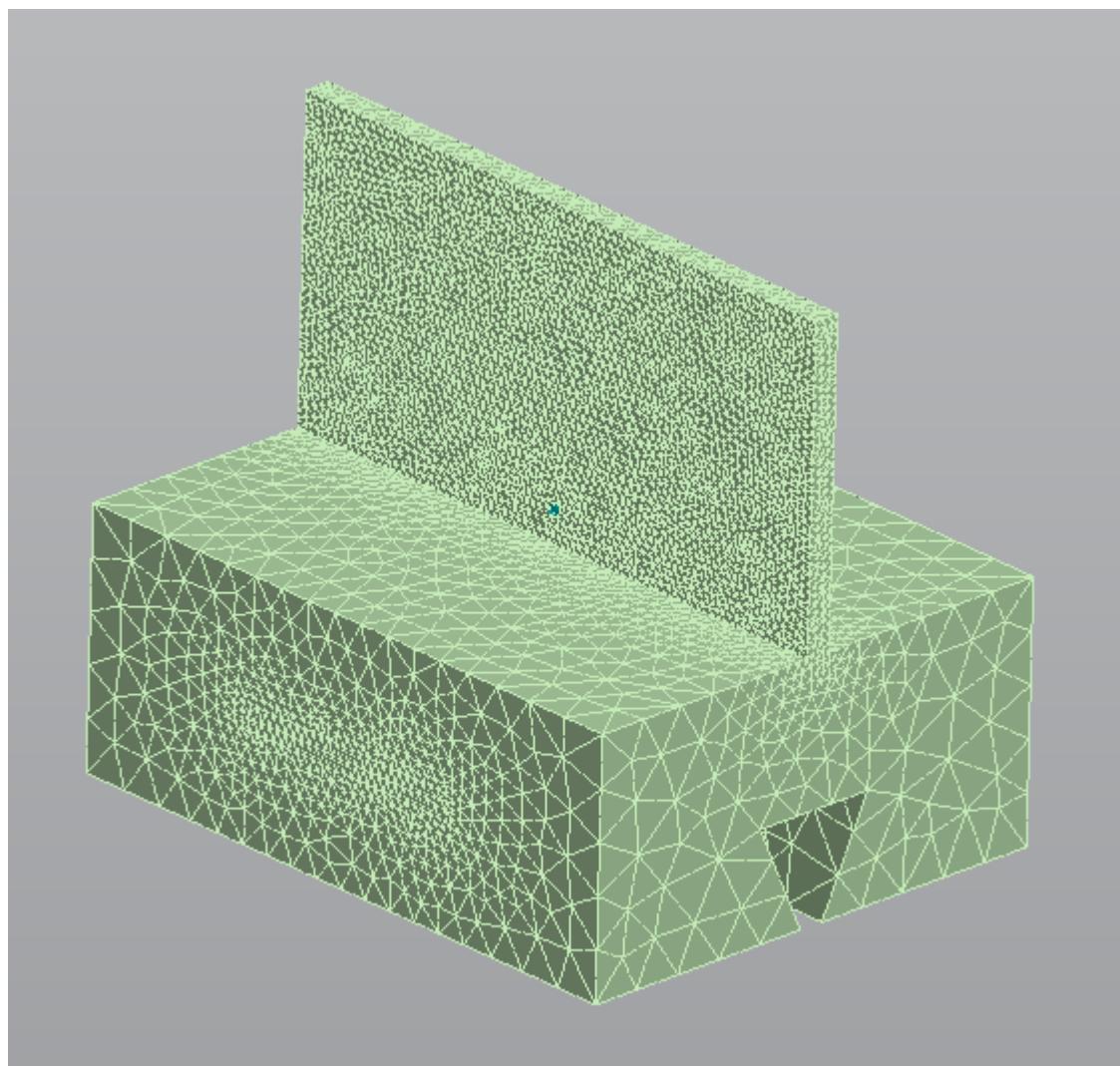
**Figure 3.3.9.4.1.1. Proximity Size Function Sources = Edges**



**Figure 3.3.9.4.1.2. Proximity Size Function Sources = Faces**



**Figure 3.3.9.4.1.3. Proximity Size Function Sources = Faces and Edges**



**Use Fixed Size Function for Sheets** Enable if you want to use fixed sizing on sheet bodies while you are using other sizing options to refine the mesh for the rest of the model.

#### Function for Sheets

**Minimum Size** **Minimum size** is the minimum size that the size function returns to the mesher. Some element sizes may be smaller than this based on local feature sizes or other geometric anomalies. Specify a value greater than 0 or accept the default.

**Proximity Minimum Size** Use **Proximity minimum size** to specify the global minimum size to be used in proximity size function calculations. This is in addition to your specification of a global **Minimum size**, which is used in curvature size function calculations. Some element sizes may be smaller than this based on local feature sizes or other geometric anomalies. By default, **Proximity minimum size** is set equal to the default of **Minimum size**. You can accept the default or specify a value greater than 0.

Any feature that operates based on minimum element size will be based on the smaller of the two minimum size values; for example, the **Defeaturing tolerance** option that is used in part-based meshing.

**Maximum Face Size** **Maximum face size** is the maximum size that the size function will return to the surface mesher. Element faces may be larger than this size based on hard edge sizes or floating point arithmetic. Specify a value greater than 0 or accept the default.

**Note:** If you parameterize the **Maximum size** or **Maximum face size** control and then change the **Size function method** from **Adaptive** to any other method (or vice versa), the parameterization of one control will have no effect on the other. However, if you have modified the default size value, your user-defined value will be copied to the other control.

**Maximum Size** **Maximum size** is the maximum size that the size function returns to the mesher. Specify a value greater than 0 or accept the default.

**Growth Rate** **Growth rate** represents the increase in element edge length with each succeeding layer of elements. For example, a growth rate of 1.2 results in a 20% increase in element edge length with each succeeding layer of elements. Specify a value from 1.0 to 5.0 or accept the default. The default is 1.2.

**Curvature Normal Angle** **Curvature normal angle** is the maximum allowable angle that one element edge is allowed to span. Enter a positive value of up to 180 degrees or 3.14 radians, or accept the default. The default is 18 degrees or 0.3142 radians.

**Number of Cells Across Gap** **Number of cells across gap** is the minimum number of layers of elements to be generated in the gaps. You can specify a value from 1 to 100 or accept the default. The default is 3. The value of **Number of cells across gap** is an approximation:

- The value may not be satisfied exactly in every gap. When sweeping, interval assignment may change the number of divisions (that is, elements or cells) in a gap.
- You must define minimum size such that it allows the specified **Number of cells across gap** to be achieved. For example, if you have a gap of 1 mm, the minimum size is 0.5 mm, and the value of **Number of cells across gap** is 3, only 2 cells can be generated in the gap.

**Note:** In cases involving patch conforming tetra meshing and swept meshing, the proximity size function drives the surface mesh size distribution as follows. The value of **Number of cells across gap** is applicable to both 3D proximity (that is, the number of 3D elements/cells between two faces in a body) and 2D proximity (that is, the number of 2D elements/cells between two edges on a face), and the global **Growth rate** value is taken into account in the gap automatically. However, the 3D proximity size function affects only the surface mesh in the gap, and assumes the volume mesh will use the global settings. Hence, if you define local mesh sizing on a body and specify local **Element size** or local **Growth rate** settings that differ drastically from the global size function settings (or if boundary layers are specified in conjunction with patch conforming tetra meshing), the final number of cells across a 3D gap may deviate from the specified **Number of cells across gap** value.

**Smoothing** **Smoothing** attempts to improve element quality by moving locations of nodes with respect to surrounding nodes and elements. The **Low**, **Medium**, or **High** option controls the number of smoothing iterations along with the threshold metric where the mesher will start smoothing. The default is **Medium**.

### 3.3.9.4.2. Global Sizing for Part-based Meshing with the Curvature Size Function Method

The size function controls how the mesh size is distributed on a surface or within a volume. For [part-based meshing](#), the default **Size function method** is based on the value of the [Engineering intent](#) control. The properties described here provide more precise control over mesh size distribution during **Meshing** tasks when **Size function method** is set to [Curvature](#).

For information about overriding size function sizes, refer to [Setting Mesh Sizes to Optimize Performance](#) on page 113.

<b>Use Fixed Size Function for Sheets</b>	Enable if you want to use fixed sizing on sheet bodies while you are using other sizing options to refine the mesh for the rest of the model.
<b>Minimum Size</b>	<b>Minimum size</b> is the minimum size that the size function returns to the mesher. Some element sizes may be smaller than this based on local feature sizes or other geometric anomalies. Specify a value greater than 0 or accept the default.
<b>Maximum Face Size</b>	<b>Maximum face size</b> is the maximum size that the size function will return to the surface mesher. Element faces may be larger than this size based on hard edge sizes or floating point arithmetic. Specify a value greater than 0 or accept the default.
	<b>Note:</b> If you parameterize the <b>Maximum size</b> or <b>Maximum face size</b> control and then change the <b>Size function method</b> from <b>Adaptive</b> to any other method (or vice versa), the parameterization of one control will have no effect on the other. However, if you have modified the default size value, your user-defined value will be copied to the other control.
<b>Maximum Size</b>	<b>Maximum size</b> is the maximum size that the size function returns to the mesher. Specify a value greater than 0 or accept the default.
<b>Growth Rate</b>	<b>Growth rate</b> represents the increase in element edge length with each succeeding layer of elements. For example, a growth rate of 1.2 results in a 20% increase in element edge length with each succeeding layer of elements. Specify a value from 1.0 to 5.0 or accept the default. The default is 1.2.
<b>Curvature Normal Angle</b>	<b>Curvature normal angle</b> is the maximum allowable angle that one element edge is allowed to span. Enter a positive value of up to 180 degrees or 3.14 radians, or accept the default. The default is 18 degrees or 0.3142 radians.
<b>Smoothing</b>	<b>Smoothing</b> attempts to improve element quality by moving locations of nodes with respect to surrounding nodes and elements. The <b>Low</b> , <b>Medium</b> , or <b>High</b> option controls the number of smoothing iterations along with the threshold metric where the mesher will start smoothing. The default is <b>Medium</b> .

### 3.3.9.4.3. Global Sizing for Part-based Meshing with the Proximity Size Function Method

The size function controls how the mesh size is distributed on a surface or within a volume. For [part-based meshing](#), the default **Size function method** is based on the value of the [Engineering intent](#) control. The properties described here provide more precise control over mesh size distribution during **Meshing** tasks when **Size function method** is set to **Proximity**.

For information about overriding size function sizes, refer to [Setting Mesh Sizes to Optimize Performance](#) on page 113.

<b>Proximity Size Function Sources</b>	<b>Proximity size function sources</b> determines whether regions of proximity between faces and/or edges are considered when proximity size function calculations are performed. Choose one of the following:
<b>Edges</b>	Considers edge-edge proximity. Face-face and face-edge proximity are not considered.
<b>Faces</b>	This is the default. Considers face-face proximity between faces. Face-edge and edge-face proximity are not considered (that is, the trailing edge of fluid around wings will not be captured with this setting).

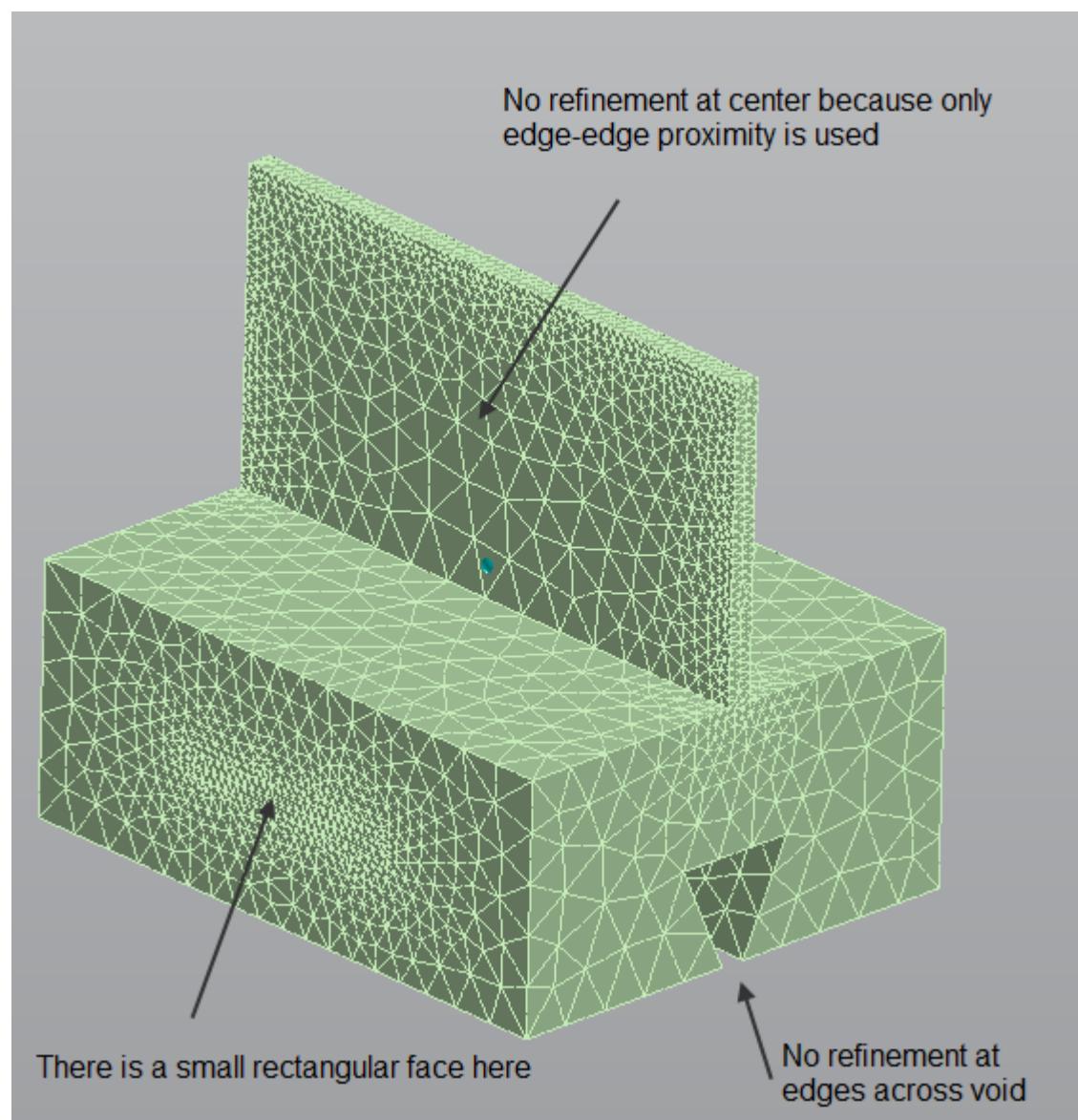
**Faces and edges** Consider face-face and edge-edge proximity. Face-edge proximity is not considered.

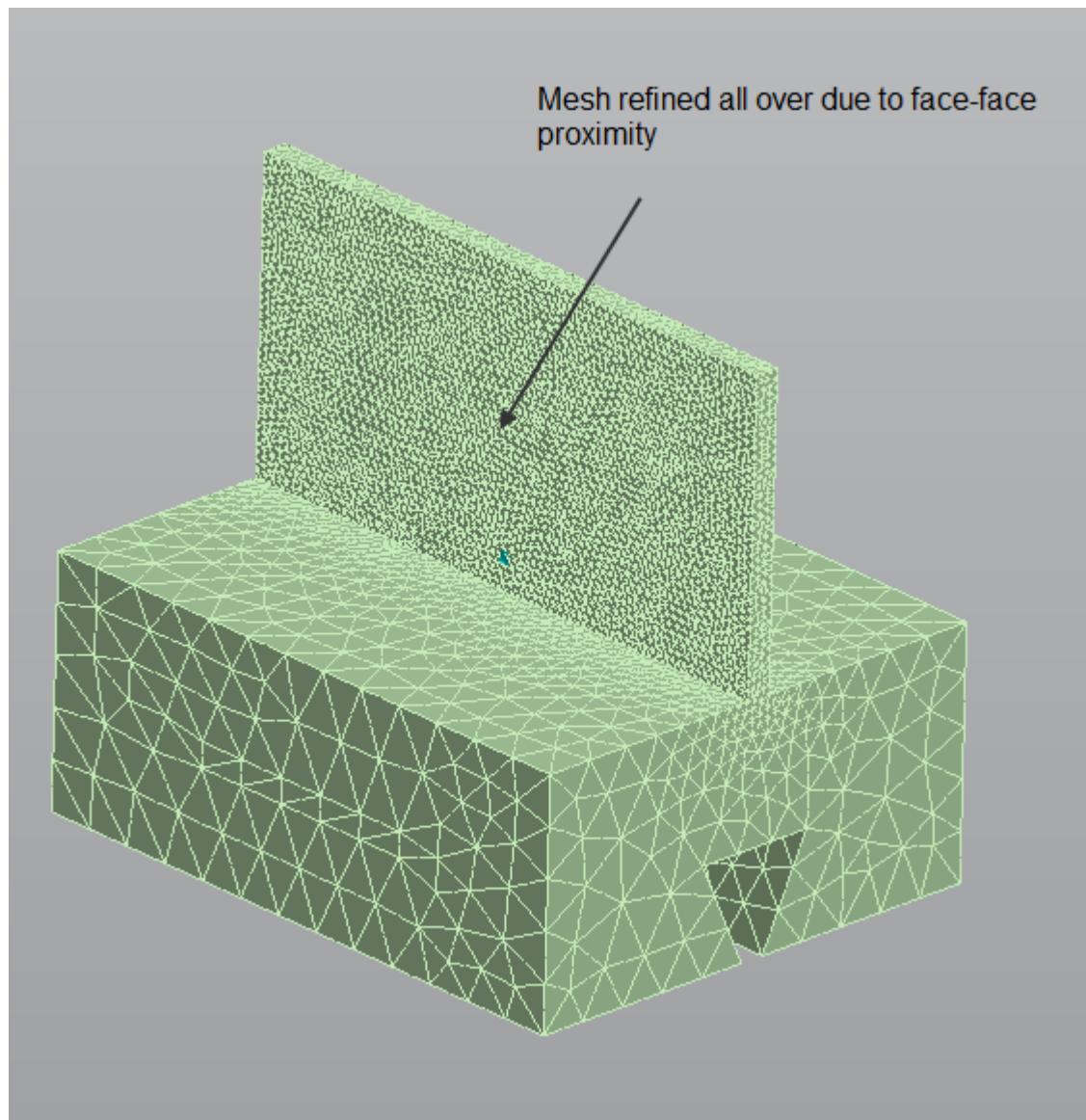
**Note:**

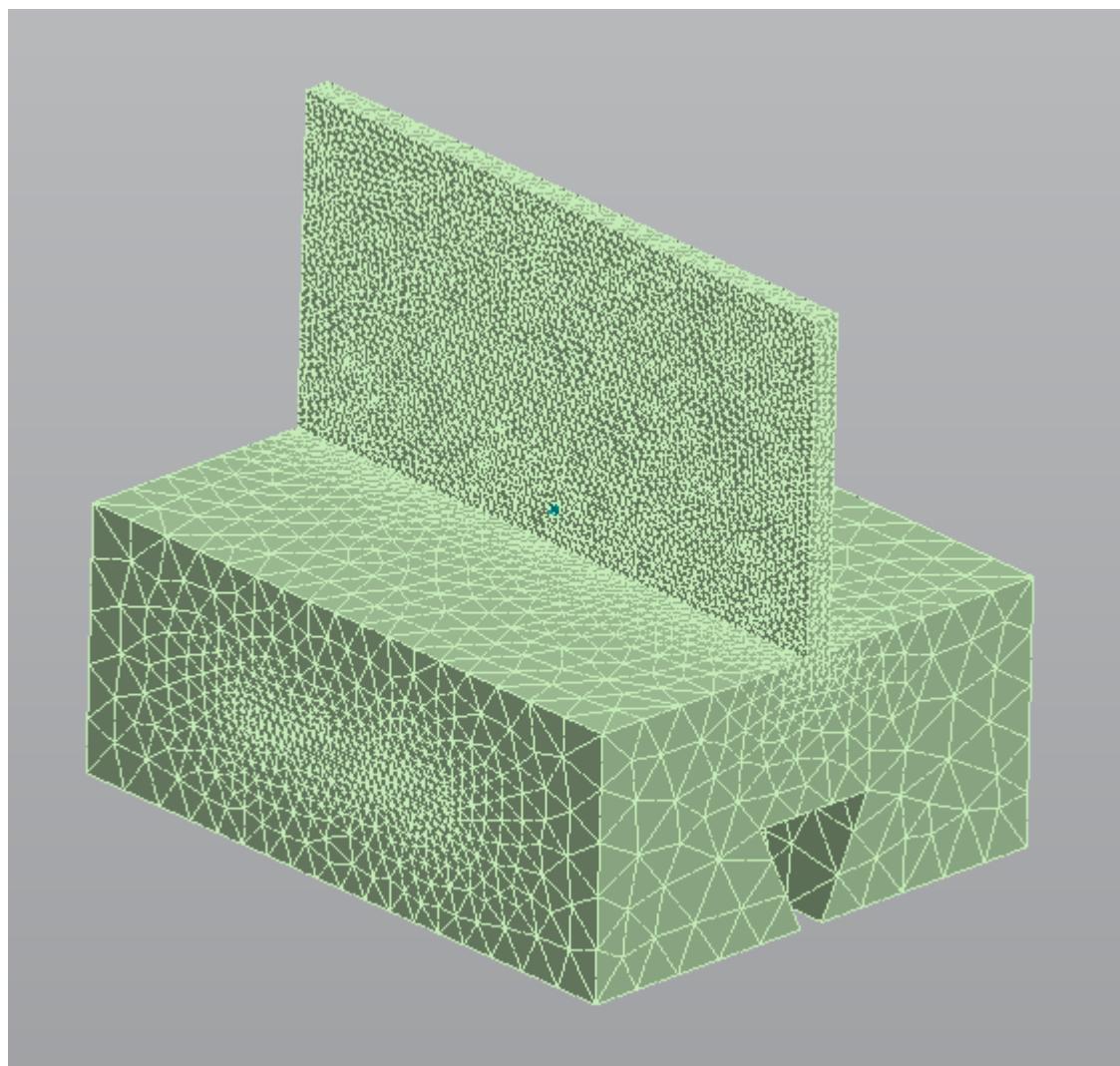
- In cases involving face-face proximity, the face normal orientation is ignored during the proximity calculation.
- In cases involving edge-edge proximity, edges across voids in a model are not refined by part-based meshing.
- For many models, the **Edges** setting may be sufficient to resolve all proximity situations. For large complex models, using either the **Faces and edges** or **Faces** setting may result in longer computation time.

The figures below illustrate the effect of each **Proximity size function sources** setting.

**Figure 3.3.9.4.3.1. Proximity Size Function Sources = Edges**



**Figure 3.3.9.4.3.2. Proximity Size Function Sources = Faces****Figure 3.3.9.4.3.3. Proximity Size Function Sources = Faces and Edges**



**Use Fixed Size** Enable if you want to use fixed sizing on sheet bodies while you are using other sizing options to refine the mesh for the rest of the model.

#### Function for Sheets

**Proximity Minimum Size** Use **Proximity minimum size** to specify the global minimum size to be used in proximity size function calculations. Some element sizes may be smaller than this based on local feature sizes or other geometric anomalies. You can accept the default or specify a value greater than 0.

**Maximum Face Size** **Maximum face size** is the maximum size that the size function will return to the surface mesher. Element faces may be larger than this size based on hard edge sizes or floating point arithmetic. Specify a value greater than 0 or accept the default.

**Note:** If you parameterize the **Maximum size** or **Maximum face size** control and then change the **Size function method** from **Adaptive** to any other method (or vice versa), the parameterization of one control will have no effect on the other. However, if you have modified the default size value, your user-defined value will be copied to the other control.

**Maximum Size** **Maximum size** is the maximum size that the size function returns to the mesher. Specify a value greater than 0 or accept the default.

<b>Growth Rate</b>	<b>Growth rate</b> represents the increase in element edge length with each succeeding layer of elements. For example, a growth rate of 1.2 results in a 20% increase in element edge length with each succeeding layer of elements. Specify a value from 1.0 to 5.0 or accept the default. The default is 1.2.
<b>Number of Cells Across Gap</b>	<p><b>Number of cells across gap</b> is the minimum number of layers of elements to be generated in the gaps. You can specify a value from 1 to 100 or accept the default. The default is 3.</p> <p>The value of <b>Number of cells across gap</b> is an approximation:</p> <ul style="list-style-type: none"> <li>• The value may not be satisfied exactly in every gap. When sweeping, interval assignment may change the number of divisions (that is, elements or cells) in a gap.</li> <li>• You must define minimum size such that it allows the specified <b>Number of cells across gap</b> to be achieved. For example, if you have a gap of 1 mm, the minimum size is 0.5 mm, and the value of <b>Number of cells across gap</b> is 3, only 2 cells can be generated in the gap.</li> </ul> <p><b>Note:</b> In cases involving patch conforming tetra meshing and swept meshing, the proximity size function drives the surface mesh size distribution as follows. The value of <b>Number of cells across gap</b> is applicable to both 3D proximity (that is, the number of 3D elements/cells between two faces in a body) and 2D proximity (that is, the number of 2D elements/cells between two edges on a face), and the global <b>Growth rate</b> value is taken into account in the gap automatically. However, the 3D proximity size function affects only the surface mesh in the gap, and assumes the volume mesh will use the global settings. Hence, if you define local mesh sizing on a body and specify local <b>Element size</b> or local <b>Growth rate</b> settings that differ drastically from the global size function settings (or if boundary layers are specified in conjunction with patch conforming tetra meshing), the final number of cells across a 3D gap may deviate from the specified <b>Number of cells across gap</b> value.</p>

**Smoothing** **Smoothing** attempts to improve element quality by moving locations of nodes with respect to surrounding nodes and elements. The **Low**, **Medium**, or **High** option controls the number of smoothing iterations along with the threshold metric where the mesher will start smoothing. The default is **Medium**.

### 3.3.9.4.4. Global Sizing for Part-based Meshing with the Fixed Size Function Method

The size function controls how the mesh size is distributed on a surface or within a volume. For [part-based meshing](#), the default **Size function method** is based on the value of the [Engineering intent](#) control. The properties described here provide more precise control over mesh size distribution during **Meshing** tasks when **Size function method** is set to [Fixed](#).

For information about overriding size function sizes, refer to [Setting Mesh Sizes to Optimize Performance](#) on page 113.

**Minimum Size** **Minimum size** is the minimum size that the size function returns to the mesher. Some element sizes may be smaller than this based on local feature sizes or other geometric anomalies. Specify a value greater than 0 or accept the default.

**Maximum Face Size** **Maximum face size** is the maximum size that the size function will return to the surface mesher. Element faces may be larger than this size based on hard edge sizes or floating point arithmetic. Specify a value greater than 0 or accept the default.

**Note:** If you parameterize the **Maximum size** or **Maximum face size** control and then change the **Size function method** from **Adaptive** to any other method (or vice versa), the parameterization of one control will have no effect on the other. However, if you

have modified the default size value, your user-defined value will be copied to the other control.

<b>Maximum Size</b>	<b>Maximum size</b> is the maximum size that the size function returns to the mesher. Specify a value greater than 0 or accept the default.
<b>Growth Rate</b>	<b>Growth rate</b> represents the increase in element edge length with each succeeding layer of elements. For example, a growth rate of 1.2 results in a 20% increase in element edge length with each succeeding layer of elements. Specify a value from 1.0 to 5.0 or accept the default. The default is 1.2.
<b>Smoothing</b>	<b>Smoothing</b> attempts to improve element quality by moving locations of nodes with respect to surrounding nodes and elements. The <b>Low</b> , <b>Medium</b> , or <b>High</b> option controls the number of smoothing iterations along with the threshold metric where the mesher will start smoothing. The default is <b>Medium</b> .

### 3.3.9.4.5. Global Sizing for Part-based Meshing with the Adaptive Size Function Method

The size function controls how the mesh size is distributed on a surface or within a volume. For [part-based meshing](#), the default **Size function method** is based on the value of the [Engineering intent](#) control. The properties described here provide more precise control over mesh size distribution during **Meshing** tasks when **Size function method** is set to [Adaptive](#).

For information about overriding size function sizes, refer to [Setting Mesh Sizes to Optimize Performance](#) on page 113.

<b>Element Seed Size</b>	<b>Element seed size</b> determines the starting point of the mesh size. Specify a value greater than 0 or accept the default.
<b>Growth Rate</b>	<b>Growth rate</b> represents the increase in element edge length with each succeeding layer of elements. For example, a growth rate of 1.2 results in a 20% increase in element edge length with each succeeding layer of elements. Specify a value from 1.0 to 5.0 or accept the default. The default is 1.5.
<b>Adaptive Resolution</b>	The <b>Adaptive resolution</b> is a scaling value for various functions defined in the adaptive size function method. Specify a value from 0.0 to 2.0 or accept the default. The default is 1.0.

### 3.3.9.5. Common Boundary Layer Settings for Part-based Meshing

For [part-based meshing](#), these options provide global control over all boundary layers.

## Collision Avoidance

The **Collision avoidance** control determines the approach that is to be taken in areas of proximity to avoid collisions that may occur when surface meshes are layered from opposite sides into each other.

The option that you choose for **Collision avoidance** is used *only in areas of proximity*. In areas of proximity, if the option is set to **Layer compression**, layer compression is performed; if it is set to **Stair stepping**, stair stepping is performed; if it is set to **None**, no collision checking is performed. However, in all other problematic

scenarios (for example, invalid normals, quality failure, bad surface mesh, and so on), local stair stepping is performed regardless of which option you choose.

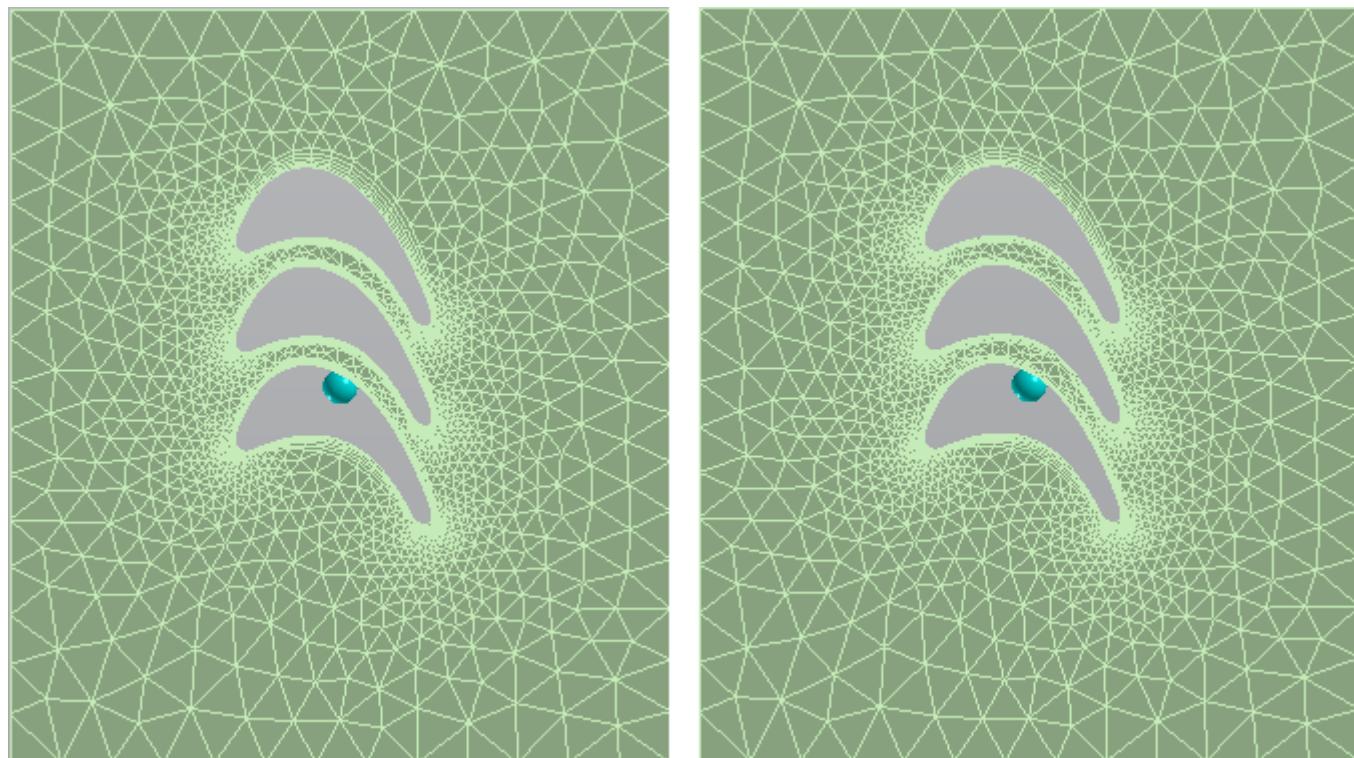
**Note:** When **Collision avoidance** is set to **Layer compression** and local stair stepping occurs after compression, poor quality pyramids may be introduced into the mesh. Because of this possibility, a warning message will appear whenever stair stepping occurs after compression.

Choose one of the following:

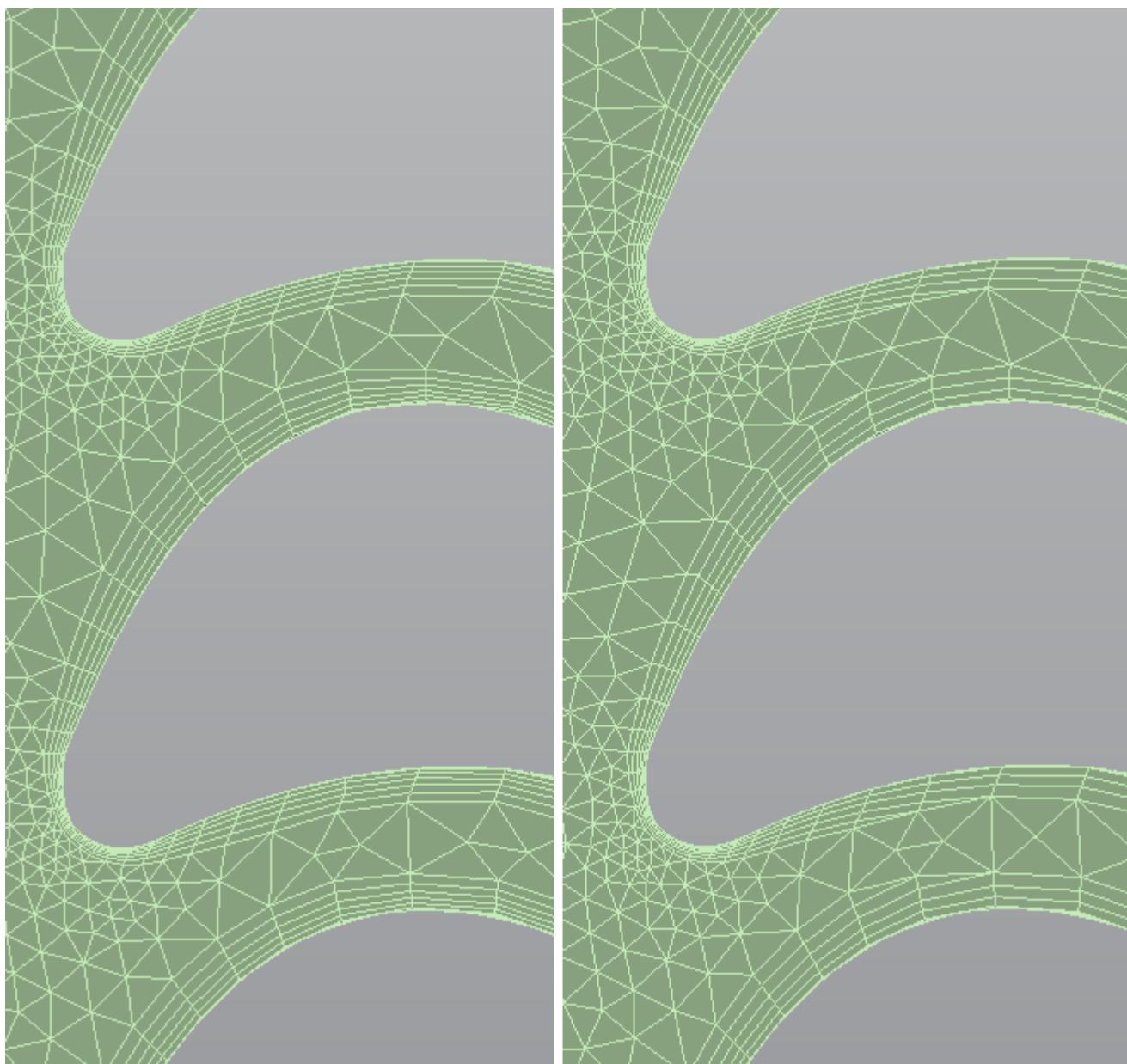
- |                          |  |
|--------------------------|--|
| <b>None</b>              | The <b>None</b> option does not check for layer collisions. Selecting this option speeds up layer computation time; however, it can result in an invalid mesh and mesh failures. For these reasons, this option is not recommended.  |
| <b>Layer compression</b> | The <b>Layer compression</b> option compresses layers in areas of collision. In these areas, the defined heights and ratios are reduced to ensure the same number of layers throughout the entire boundary layer region. Generally, this option is best for avoiding the creation of pyramids in the mesh.   |
| <b>Stair stepping</b>    | This is the default. Rather than compressing the prism layers, with <b>Stair stepping</b> the prism layers are “stair stepped” in the proximity region to avoid collision and to maintain the gap defined by the <b>Gap factor</b> control. The <b>Stair stepping</b> approach to layer growth locally reduces layers to avoid collisions, as well as bad quality elements in sharp or tight corners. The term “stair stepping” refers to the steps created between one layer and the next. Using this approach, special logic is used to fill the steps with pyramid and tetrahedron elements for prism steps, or prism, pyramid, and tetrahedron elements for hex steps. This special logic helps the mesher obtain a high-quality transition to the tetrahedral mesh. |

The graphics below illustrate how **Layer compression** and **Stair stepping** differ.

**Figure 3.3.9.5.1. Layer Compression vs. Stair Stepping (View of Entire Mesh)**



**Figure 3.3.9.5.2. Layer Compression vs. Stair Stepping (View of Details in Mesh)**



### 3.3.9.5.1. Optional Boundary Layer Settings for Part-based Meshing



These settings are available if you turn Filtering off. For [part-based meshing](#), these settings provide greater control over all boundary layers.

#### Fix First Layer

The **Fix first layer** control determines whether the heights or ratios of the first layer will be modified to avoid collision. Valid values are **Yes** and **No**. The default is **No**. This option will not allow the value that is set for the **First layer height** control to be changed.

The **Fix first layer** control is applicable only when **Collision avoidance** is set to **Layer compression**.

## Gap Factor

The **Gap factor** control enables maintenance of the gap between intersecting prisms. Valid values are from 0 to 2. The default is **0.5**. A value of **1** means a gap equal to the ideal tet cell height based on base face size in proximity to each other is maintained.

The **Gap factor** control is applicable only when **Collision avoidance** is set to **Layer compression** or **Stair stepping**.

## Maximum Height Over Base

The **Maximum height over base** control sets the maximum allowable prism aspect ratio (that is, the ratio of height over base of the base triangle). When the prism aspect ratio reaches this value, the height of the prisms stops growing. That is, new prisms continue to form, but the heights of the prisms will not increase. Valid values are from 0.1 to 5. The default is **1**.

## Growth Rate Type

The **Growth rate type** control determines the height of the layers given the initial height and height ratio. Choose one of the following:

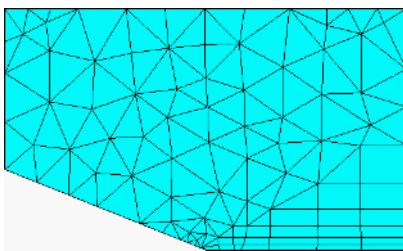
<b>Geometric</b>	This is the default. With this option, the prism height of a particular layer is defined by $h * r^{(n-1)}$ , where $h$ = initial height, $r$ = height ratio, and $n$ = layer number. The total height at layer $n$ is: $h(1-r^n)/(1-r)$ .
<b>Exponential</b>	With this option, the prism height of a particular layer is defined by $h * e^{(n-1)p}$ , where $h$ = initial height, $p$ = exponent, and $n$ = layer number.
<b>Linear</b>	With this option, the prism height of a particular layer is defined by $h(1+(n-1)(r-1))$ , where $h$ = initial height, $r$ = height ratio, and $n$ = layer number. The total height at layer $n$ is $nh((n-1)(r-1)+2)/2$ .

## Maximum Angle

The **Maximum angle** control determines prism layer growth around angles and when prisms will adhere (project) to adjacent surfaces/walls. If the layered mesh involves extruding from one surface and not its neighbor, and the angle between the two surfaces is less than the specified value, the prisms (sides) will adhere (project) to the adjacent wall. Valid values are from 90 to 180 (degrees). Typically, a value between 120 and 180 is desirable. The default is **140** degrees or **2.44** radian. Refer to the figures below for examples of maximum angle.

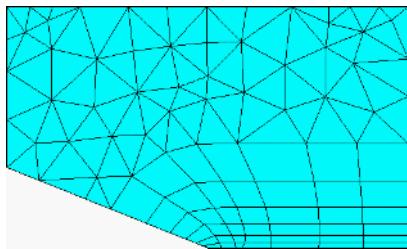
In the figure below, the angle between the planes is 158.2 (21.8) degrees. Since the maximum angle is less than the angle between the walls, the prism layers are capped with pyramids.

**Figure 3.3.9.5.1.1. Maximum Angle = 140**



In the figure below, the maximum angle exceeds the separation angle between the surfaces, so the prism remains attached to the adjacent surface.

**Figure 3.3.9.5.1.2. Maximum Angle = 180**

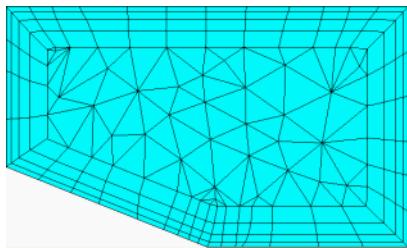


## Fillet Ratio

The **Fillet ratio** control determines whether a fillet proportional to the total height of a prism element will be created when a prism element is generated in the corner region of a tetrahedral mesh. Creating a fillet proportional to the total height of the prism makes it possible to control the smoothness of the prism layer. Valid values are from 0 to 1 (decimal values are allowed). A value of 0 means no fillets. The default is 1. Refer to the figures below for examples of fillet ratio.

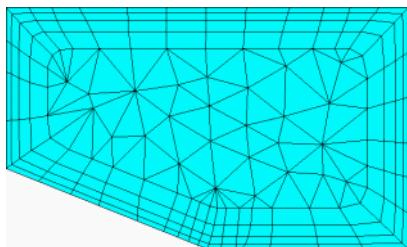
**Note:** For meshing corners with angles less than 60 degrees, there may not be space for a fillet.

**Figure 3.3.9.5.1.3. Fillet Ratio = 0.0**

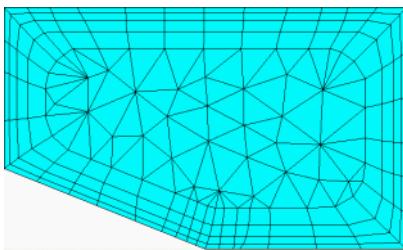


In the figure below, the radius of the inner prism fillet is 0.5 times the height of the total prism thickness.

**Figure 3.3.9.5.1.4. Fillet Ratio = 0.5**



**Figure 3.3.9.5.1.5. Fillet Ratio = 1.0**



## Use Post Smoothing

The **Use post smoothing** control determines whether smoothing will be performed after layers are created. Smoothing attempts to improve element quality by moving the locations of nodes with respect to surrounding nodes and elements. Valid values are **Yes** and **No**. The default is **Yes**. When this control is set to **Yes**, the **Number of iterations** control appears.

## Number of Iterations

The **Number of iterations** control determines the number of smoothing iterations that will be performed to improve the mesh. Valid values are from 1 to 20. The default is **5**.

**Number of iterations** appears only when **Use post smoothing** is set to **Yes**.

## 3.3.9.6. Additional Settings for Part-based Meshing

 These settings are available if you turn Filtering off. You can use them to set preferences for [part-based meshing](#).

## Shape Checking

The global **Shape checking** control enables you to set the level of shape checking that AIM performs in generating the mesh. Some shape checks have a stricter set of criterion than others. Choose one of the following:

<b>Use engineering intent</b>	This is the default. The mesher determines the shape checking criterion based on the value of the <a href="#">Engineering intent</a> control.
<b>Standard mechanical</b>	Uses the default ANSYS Workbench shape checking criterion. This criterion has proven to be effective for linear, modal, stress, and thermal problems. A unitless metric is used comparing volume to edge <sup>3</sup> for solid elements and area to edge length <sup>2</sup> for shell elements. <a href="#">Jacobian ratio</a> is computed at integration points.
<b>Aggressive mechanical</b>	Uses a shape checking criterion based on <a href="#">Jacobian ratio</a> at nodes with a tighter limit on both Jacobian ratio and ANSYS Workbench metric. This option is much more restrictive than the ANSYS Workbench criterion used for the <b>Standard mechanical</b> option. If the Jacobian ratio is less than 0 or greater than the limit, then the metric is -1.0 and will cause shape failure. <b>Aggressive mechanical</b> will usually produce more elements, longer meshing times, and possibly mesh failures. This option is recommended if the mesh is intended for large deformation or material nonlinear analysis.
<b>Fluid flow</b>	Uses a shape checking criterion based on element volume. This option is recommended for all fluid flow simulations.

<b>None</b>	Turns off most shape checking. Minimal shape checking still occurs so there is no guarantee that a mesh will be returned. This option is intended for troubleshooting; use it with caution as it could lead to solver failures or incorrect solution results.
-------------	---

For automatic meshing, shape checking ensures a quality mesh based on specific error limits. The following table presents the shape checking limit criterion for each option. If the results of any test are outside of the limits shown, the mesh fails. For example, a **Jacobian ratio** greater than 40 computed at element nodal points will cause a mesh failure when using the **Aggressive mechanical** option.

Criterion	Standard mechanical	Aggressive mechanical	Fluid flow
Aspect Ratio	Quality metric not used	Quality metric not used	Quality metric not used
Element Volume	< 0	< 0	< $10^{-32}$ ; a warning is delivered
Face Angle	Quality metric not used	Quality metric not used	Quality metric not used
Face Warping	Quality metric not used	Quality metric not used	Quality metric not used
Jacobian Ratio	> 40 for 3D and > 10 for 2D; computed at element integration points	> 40 for both 3D and 2D; computed at element nodal points	Quality metric not used
Orthogonal Quality	Quality metric not used	Quality metric not used	Quality metric not used
ANSYS Workbench Shape Metric	< 0.75 for one-dimensional elements  < 0.01 for two-dimensional elements  < $10^{-4}$ for three-dimensional elements	< 0.85 for one-dimensional elements  < 0.02 for two-dimensional elements  < 0.01 for three-dimensional elements	Quality metric not used

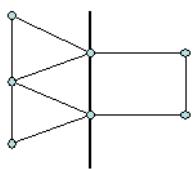
## Midside Nodes

The global **Midside nodes** control determines whether meshes are created with midside nodes (quadratic elements) or without midside nodes (linear elements). Reducing the number of midside nodes reduces the number of degrees of freedom. Choose one of the following:

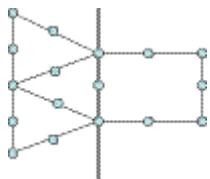
<b>Use engineering intent</b>	This is the default. The mesher determines whether to create quadratic or linear elements based on the value of the <b>Engineering intent</b> control.
<b>Dropped</b>	No elements have midside nodes.
<b>Kept</b>	All elements have midside nodes.

The figures below illustrate the difference between the **Dropped** and **Kept** options. These examples are for a solid body. The heavy vertical line in each figure represents the body boundary.

**Figure 3.3.9.6.1. Midside Nodes = Dropped**



**Figure 3.3.9.6.2. Midside Nodes = Kept**



**Note:** Both linear and quadratic elements are supported for 2D axisymmetric models. This includes support for 2D linear and quadratic boundary layers on axisymmetric parts. You must set **Midside nodes** to **Dropped** to obtain linear elements in such cases.

## Force Straight Edge Sides

If you want elements to be meshed with straight edges, set the **Force straight edge sides** control to **Yes**. You must set the control to **Yes** for electromagnetic simulations.

## Patch Conforming Triangular Surface Mesher

The **Patch conforming triangular surface mesher** control determines which triangle surface meshing strategy is used by patch conforming meshers. In general, the advancing front algorithm provides a smoother size variation and better results for skewness and orthogonal quality. Choose one of the following:

### Automatically determined

This is the default. The mesher determines whether to use the Delaunay or advancing front algorithm based on a variety of factors such as surface type, face topology, and defeatured boundaries.

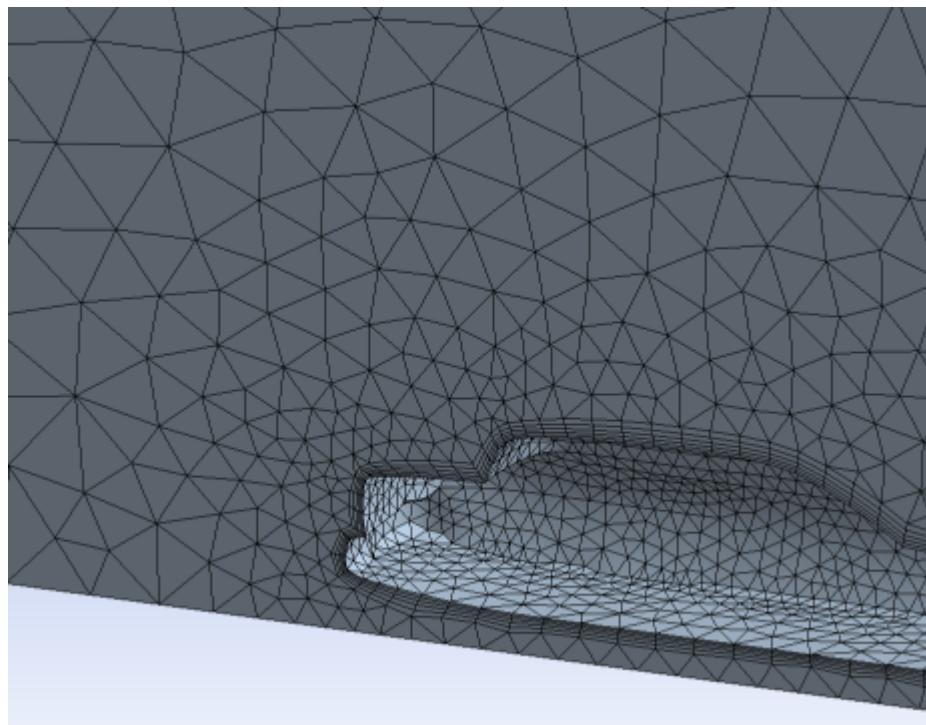
### Advancing front

The mesher uses advancing front as its primary algorithm, but switches to the Delaunay algorithm if problems occur.

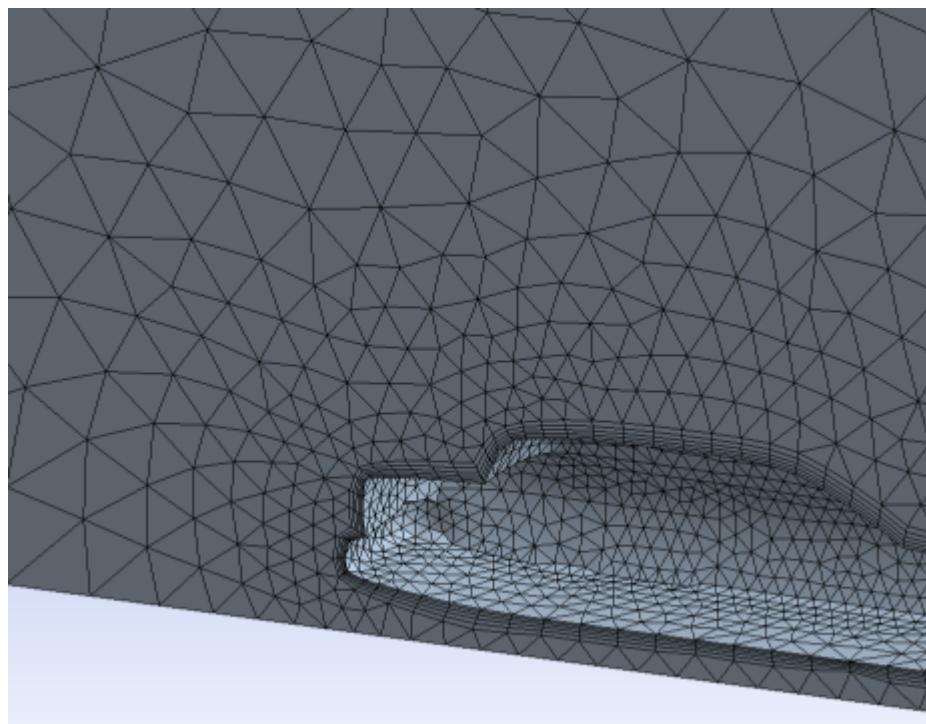
**Note:** If the mesher switches to Delaunay, the edge mesh from the advancing front algorithm may still be used and in some rare cases may lead to meshing failures. Changing your setting to **Automatically determined** may result in a successful mesh because the initial edge mesh may be better.

The figures below illustrate the difference between the **Automatically determined** and **Advancing front** options.

**Figure 3.3.9.6.3. Patch Conforming Triangular Surface Mesher = Automatically Determined**



**Figure 3.3.9.6.4. Patch Conforming Triangular Surface Mesher = Advancing Front**



# Loop Removal Tolerance

The **Loop removal tolerance** control sets the tolerance for loop removal. If you specify a value greater than **0**, loop removal is enabled and any loop with a radius less than or equal to the specified value is meshed over. A value of **0**, which is the default, means loop removal is disabled.

**Note:** The **Loop removal tolerance** option appears only if sheets are present in the model.

# Mesh Based Defeaturing

AIM defeatures small features and imperfections in the geometry (“dirty” geometry) automatically according to criteria you specify. When **Mesh based defeaturing** is enabled, features smaller than or equal to the **Defeaturing tolerance** value are removed automatically. Choose one of the following:

<b>Automatically determined</b>	This is the default. The defeaturing tolerance is calculated as described below.
<b>User defined</b>	You must specify a <b>Defeaturing tolerance</b> value greater than <b>0</b> before you can generate the mesh.
<b>Off</b>	No automatic defeaturing occurs.

**Note:** If your model contains a feature that you definitely want to capture in the mesh, make sure that the **Defeaturing tolerance** value is smaller than the size of that feature.

For the **Adaptive** size function method, the default **Defeaturing tolerance** is based on the mesher (for example, swept vs. patch conforming tetrahedron).

For all size function methods other than **Adaptive**, the default **Defeaturing tolerance** is set to 50% of the value of **Minimum size/Proximity minimum size** (whichever is smaller).

Remember these points:

- If you let the **Defeaturing tolerance** default and you modify the mesh size by applying local sizing controls (for example, local face or edge sizing), the tolerance may be modified automatically. In such cases, a warning message is issued to notify you that the global defeaturing tolerance has been modified. To prevent this behavior, manually specify a value for **Defeaturing tolerance**.
- If you let the **Defeaturing tolerance** default and you modify the **Minimum size**, a check is performed to determine whether the current **Defeaturing tolerance**  $\geq$  the product of factor \* new\_min\_size, where factor is 0.5. If yes, then the **Defeaturing tolerance** is recalculated. If no, then the **Defeaturing tolerance** remains unchanged.
- When specifying a value for **Defeaturing tolerance**, do not specify a value greater than the value of **Minimum size**.
- If you add a hard size that is smaller than the **Minimum size**, you must further reduce the **Defeaturing tolerance** with respect to the specified hard size (essentially, the hard size becomes the new **Minimum size**).

# Use All Processors

When set to **Yes** (the default), all available processors will be used for parallel part meshing. Using multiple processors enhances meshing performance. When set to **No**, the **Processor limit** option appears, where you can specify a number of processors to use. The **Use all processors** option is applicable to shared memory parallel (SMP) meshing (not distributed memory parallel). For related information, refer to [Parallel Part Meshing Best Practices](#) on page 149.

# Processor Limit

Available only when **Use all processors** is set to **No**. Specify a value from 1 to 256 (the default).

## 3.3.9.7. Troubleshooting Part-based Meshing Problems

This information provides tips and strategies for avoiding and handling problems that may occur during part-based meshing.

## Understanding Part-based Meshing and Mesh State

For part-based meshing, the mesh state is an aggregate state of all bodies and parts in the configuration. Making certain changes causes the mesh state of the entire configuration to become underdefined and require a remesh. Changes that affect mesh state include:

- Changes to a mesh setting, such as **Midside nodes**
- Addition or deletion of a mesh control
- Changes to mesh control properties
- Changes to topology scoped to a mesh control, including suppressing or unsuppressing topology
- Creation or deletion of a Selection Set
- Deletion, suppression, or unsuppression of a body or part that is a member of a Selection Set
- Refreshing geometry

### 3.3.10. Local Mesh Sizing

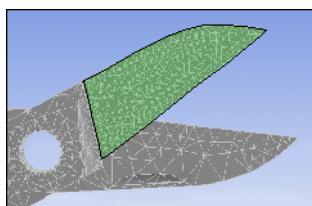
Local mesh sizing controls define sizing that is to be applied at a specified **Location**. You use local mesh sizing to control

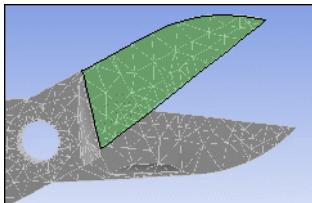
- the element size for a selected **body**, **face**, or **edge**.
- the number of divisions along an edge.
- the minimum mesh sizing used for a selected body, face, or edge. This setting overrides the default global sizing.

You can define multiple mesh sizing controls for topologies of different dimensions. For example, you may define a body sizing control scoped to one body, a second body sizing control scoped to another body, a face sizing control scoped to two faces, and an edge sizing control scoped to four edges. The geometry available for selection is filtered appropriately (that is, you can only scope a body sizing control to bodies, you can only scope a face sizing control to faces, and so on).

The following figures illustrate a sizing control applied to a face. For the first figure, the element size was set to 0.5 on one face; for the second figure, the default element size was retained.

**Figure 3.3.10.1. Element Size = 0.5**



**Figure 3.3.10.2. Element Size = Default**

## Usage Notes

- When multiple sizing controls are scoped to the same entity, the smallest size that is specified for that entity is respected. For example:
  - Sizing Control 1 specifies a size of 10 mm on Body A, Body B, and Body C.
  - Sizing Control 2 specifies a size of 5 mm on Body B, Body D, and Body E.
 In this case, bodies A and C are assigned a size of 10 mm, and bodies B, D, and E are assigned a size of 5 mm.
- An exception to this rule involves edge sizing controls. In cases where one edge sizing control specifies **Element size** for an edge and another specifies **Edge divisions** for the same edge, the specified **Edge divisions** value is respected.
- If multiple sizing controls are scoped to the same entity and the specified element size is the same for all of them, the last control that was created is respected. If a sizing control is placed on an edge and then another is placed on a face or body that contains that edge, the edge sizing takes precedence.
- For all size function methods other than **Adaptive**, any local size applied to an entity is also applied to all lower topology entities.
- Faces adjacent to a face that has a scoped size control applied to it respect the source as part of the size function. Meshes on the adjacent faces will transition smoothly to the size on the scoped face. When size controls that have differing sizes are on adjacent entities, the adjacent topology receives the smallest size.
- When applying sizes to edges, if possible, the meshing algorithm places the requested number of divisions on the specified edge. Otherwise, the algorithm adjusts the number to allow a successful mesh generation.
- Regardless of the value for the sizing control you set, other factors such as edge and face curvature and the proximity of small features may override the effect of the sizing control.
- **Local minimum size** is not respected for **proximity size functions**. It also might not be respected by sweepable bodies due to the interval assignment used to generate structured meshes.

## 3.3.10.1. Defining Sizing on a Body

**Volume Creation > Objects > Size Controls > Add > Body Sizing**

**Mesh > Objects > Size Controls > Add > Body Sizing**

To define mesh sizing on a body, you must create a body sizing control by performing these steps. For descriptions of the options mentioned below, refer to [Local Mesh Sizing Control Options](#) on page 173.

**Note:** When you define an **Extracted Volume** interactively, the **Body Sizing** option is unavailable from the **Volume Creation** panel. In addition, any **Body Sizing** control that is previously defined will have no effect on the mesh generated for a volume extraction. For example, if you define a **Body Sizing** control for a **Geometry Simplification**, you cannot reuse that control for an **Extracted Volume**; the control will have no effect on the **Extracted Volume** mesh. Similarly, a **Body Sizing** control in a journal file will have no effect on a volume extraction.

1. **Select the Location** (bodies) in your geometry for which you want to define the body sizing.

2. Specify the **Element size**.
3. Additionally, in the **Body Sizing** panel, you can make modifications to the following settings:

- a) Choose the  **Behavior**.
- b) Specify the  **Curvature normal angle**. **Curvature normal angle** appears only if **Behavior** is set to **Soft** and **Size function method** is set to either **Curvature and proximity** or **Curvature**.
- c) Specify the  **Growth rate**. **Growth rate** appears only if **Behavior** is set to **Soft** and **Size function method** is not **Adaptive**.
- d) Specify the  **Local minimum size**. **Local minimum size** appears only if **Behavior** is set to **Soft** and **Size function method** is not **Adaptive**.

### 3.3.10.2. Defining Sizing on a Face

**Volume Creation > Objects > Size Controls > Add > Face Sizing**

**Mesh > Objects > Size Controls > Add > Face Sizing**

To define mesh sizing on a face, you must create a face sizing control by performing these steps. For descriptions of the options mentioned below, refer to [Local Mesh Sizing Control Options](#) on page 173.

1. [Select the Location](#) (faces) in your geometry for which you want to define the face sizing.
2. Specify the **Element size**.
3. Additionally, in the **Face Sizing** panel, you can make modifications to the following settings:

- a) Choose the  **Behavior**.
- b) Specify the  **Curvature normal angle**. **Curvature normal angle** appears only if **Behavior** is set to **Soft** and **Size function method** is set to either **Curvature and proximity** or **Curvature**.
- c) Specify the  **Growth rate**. **Growth rate** appears only if **Behavior** is set to **Soft** and **Size function method** is not **Adaptive**.
- d) Specify the  **Local minimum size**. **Local minimum size** appears only if **Behavior** is set to **Soft** and **Size function method** is not **Adaptive**.

### 3.3.10.3. Defining Sizing on an Edge

**Volume Creation > Objects > Size Controls > Add > Edge Sizing**

**Mesh > Objects > Size Controls > Add > Edge Sizing**

To define mesh sizing on an edge, you must create an edge sizing control by performing these steps. For descriptions of the options mentioned below, refer to [Local Mesh Sizing Control Options](#) on page 173.

1. [Select the Location](#) (edges) in your geometry for which you want to define the edge sizing.
2. Choose the **Edge sizing method** (**Element size** or **Edge divisions**).
3. Depending on the **Edge sizing method**, specify the **Element size** or the number of **Edge divisions**.
4. Additionally, in the **Edge Sizing** panel, you can make modifications to the following settings:

- a) Choose the  **Behavior**.

- b) Specify the **Curvature normal angle**. **Curvature normal angle** appears only if **Behavior** is set to **Soft** and **Size function method** is set to either **Curvature and proximity** or **Curvature**.
  - c) Specify the **Growth rate**. **Growth rate** appears only if **Behavior** is set to **Soft** and **Size function method** is not **Adaptive**.
  - d) Specify the **Local minimum size**. **Local minimum size** appears only if **Behavior** is set to **Soft** and **Size function method** is not **Adaptive**.
5. Additionally, for part-based meshing, continue with the remaining steps. Otherwise, you are done with the **Edge Sizing** panel:
- a) **Bias option** is always set to **Smooth transition**.
  - b) Choose a **Bias type**.
  - c) If you set **Bias type** to anything other than **Constant**, specify a **Bias growth rate** or accept the default of 1.2.

### 3.3.10.4. Local Mesh Sizing Control Options

Use these options to further define mesh sizing on [bodies](#), [faces](#), or [edges](#). The options that are available depend on the selected topology.

Option	Available for This Topology	Definition
<b>Location</b>	<ul style="list-style-type: none"> <li>• Body</li> <li>• Face</li> <li>• Edge</li> </ul>	Specify the bodies, faces, or edges to which the sizing is to be applied.
<b>Edge sizing method</b>	• Edge	Choose <b>Element size</b> (default) or <b>Edge divisions</b> . Choose <b>Edge divisions</b> if you want the mesh to be sized according to a discrete number of divisions along an edge.
<b>Element size</b>	<ul style="list-style-type: none"> <li>• Body</li> <li>• Face</li> <li>• Edge</li> </ul>	Enter a positive value, or enter 0 for the value to be determined automatically. Smaller values generate more divisions. Refer to <a href="#">Local Element Size and the Size Function</a> on page 174.
<b>Edge divisions</b>	• Edge	Specify the number of divisions that should occur along the edge.
<b>(Optional) Behavior</b>	<ul style="list-style-type: none"> <li>• Body</li> <li>• Face</li> <li>• Edge</li> </ul>	Choose <b>Soft</b> (default) or <b>Hard</b> . The <b>Hard</b> setting places more constraints on the mesher. Refer to <a href="#">Behavior of Soft and Hard Settings</a> on page 175.
<b>(Optional) Curvature normal angle</b>	<ul style="list-style-type: none"> <li>• Body</li> <li>• Face</li> <li>• Edge</li> </ul>	This is the maximum allowable angle that one element edge is allowed to span. Enter a positive value in degrees or radians to override the global setting. If you enter 0, the global setting is used. Appears only if <b>Behavior</b> is <b>Soft</b> and <b>Size function method</b> is set to either <b>Curvature and proximity</b> or <b>Curvature</b> .
<b>(Optional) Growth rate</b>	<ul style="list-style-type: none"> <li>• Body</li> <li>• Face</li> </ul>	This represents the increase in element edge length with each succeeding layer of elements. For example, a growth rate of 1.2 results in a 20%

Option	Available for This Topology	Definition
	• Edge	increase in element edge length with each succeeding layer of elements. Enter a value from 1 to 5 or accept the default. If you enter 0, the global setting is used. This local <b>Growth rate</b> is similar to the global <b>Growth rate</b> . However, the value you specify for <b>Growth rate</b> on a local sizing control must always be smaller than or equal to the specified global <b>Growth rate</b> . Specifying a growth rate for a face or body affects the growth on the face or on the boundary of the body, and its effect continues outside of the selected entity as well. Appears only if <b>Behavior</b> is <b>Soft</b> and <b>Size function method</b> is not <b>Adaptive</b> .
 (Optional) <b>Local minimum size</b>	• Body • Face • Edge	Use <b>Local minimum size</b> to override the global <b>Minimum size</b> on local entities. If you enter 0, the mesher uses either the global <b>Minimum size</b> or the local <b>Element size</b> , whichever is smaller. This setting is useful where you want a different local mesh control than the global setting. For example, you can set a larger global minimum/maximum size, then use a smaller <b>Local minimum size</b> to obtain a finer mesh on some of the smaller parts. Appears only if <b>Behavior</b> is <b>Soft</b> and <b>Size function method</b> is not <b>Adaptive</b> . Hard sizes ( <b>Behavior = Hard</b> ) essentially mean an edge has a fixed size function. Thus, in cases where an edge is hard, the <b>Local minimum size</b> is hidden because its use does not make sense.  <b>Note:</b> <b>Local minimum size</b> may not be respected in swept meshes due to the interval assignment used to generate structured meshes.
 (Optional) <b>Bias option</b>	• Edge	<b>Bias option</b> is always set to <b>Smooth transition</b> . When <b>Bias type</b> is set to the default setting of <b>Constant</b> , the value of <b>Smooth transition</b> is 1.2. If the <b>Bias type</b> is anything other than <b>Constant</b> , the prescribed growth rate = <b>Bias growth rate</b> <sup>(1/n-1)</sup> where n is the number of <b>Edge divisions</b> .
 (Optional) <b>Bias type</b>	• Edge	Use <b>Bias type</b> to adjust the spacing ratio of nodes on an edge. <b>Bias type</b> is useful for any engineering problem where nodes need to be clustered on an edge or group of edges, or if there is a need to bias the mapped mesh of a face towards a specific direction with respect to the edges of the face. Choose <b>Constant</b> (the default), <b>Right</b> , <b>Left</b> , <b>Center In</b> , or <b>Center Out</b> .
 (Optional) <b>Bias growth rate</b>	• Edge	The ratio of the largest edge to the smallest edge. If you set <b>Bias type</b> to anything other than <b>Constant</b> , specify a <b>Bias growth rate</b> or accept the default of 1.2.

### 3.3.10.4.1. Local Element Size and the Size Function

For all **size function methods** other than **Adaptive**, the behavior of the local **Element size** is as follows:

- For an edge, **Element size** is the maximum size on the edge. It overrides **Maximum face size**.
- For a face, **Element size** is the maximum size on the face, which is then propagated to the edges. The value of **Element size** overrides **Maximum face size**.
- For a body, **Element size** is not only the maximum size on the body, but also the maximum sizes on the faces and edges of the body. If the body size setting is **Hard**, the value of **Element size** overrides **Maximum face size** and **Maximum size**. For a **Soft** body size, if **Element size** is smaller than **Maximum face size** or **Maximum size**, it overrides **Maximum face size** or **Maximum size** as applicable; otherwise, **Element**

**size** is equal to **Maximum face size** or **Maximum size** (that is, the smallest size is the maximum size on the body).

### 3.3.10.4.2. Behavior of Soft and Hard Settings

The **Soft** and **Hard** settings for **Behavior**, combined with [size function settings](#), affect mesh generation.

## All Size Function Methods

- Choosing the **Hard** option for edges means that the size or number of divisions is fixed on the edge and cannot be changed by the meshing algorithm. The likelihood of a mesh failure increases in this scenario.
- Choosing the **Soft** option for edges, faces, and bodies means the size control will be affected by proximity, curvature, and local remeshing during the meshing process.
- With respect to interval assignment, choosing **Soft** on a face or body means the edges are only seeded with this information; no additional constraints are placed on the mesher. A soft constraint in interval assignment places a higher weight in the interval solver to meet that number of divisions on the edge. When hard size constraints are placed on faces or bodies that are swept, the edges of those faces/bodies receive soft edge sizes to guide the interval solver.

## All Size Function Methods Other than Adaptive

- If a soft size is specified on an entity, the specified **Growth rate** is used.
- If a hard size is specified on an entity, the **Growth rate** is not respected; the mesher will try to maintain a uniform size on the entity. If two entities with hard sizes of different values are adjacent to one another, no growth occurs and there will be an abrupt size change between them. However, growth will occur on all entities without hard size constraints that are adjacent to a hard size entity, even if the hard size applied is smaller than **Minimum size/Proximity minimum size**.
- Avoid applying local hard size controls on faces and bodies if the specified local size falls between the values of the size function's minimum size (**Minimum size**) and maximum size (**Maximum face size** or **Maximum size**, as applicable) options. In such cases use soft size controls instead, and the soft controls will act like a maximum size.
- If an entity has a local soft size defined on it, the local control is used in place of the global **Maximum size**. However, the global **Minimum size** is still used. If instead the entity has a local hard size defined on it, the global **Minimum size** is ignored and the whole entity would be meshed at the size of the local control.
- The smallest size that the last size control specifies for the entity overrides all other size controls. In this way, the smallest size will be the maximum size on the entity. This is true unless a hard size control is used. When size controls are applied to entities of differing dimensions (for example, a face and an edge), a hard size control will override all others and the sizing will propagate from the hard size outward.

## Adaptive Size Function Method

Choosing **Hard** on a face or body means curvature pre-refinement is not performed on the edges of the face or body, but an edge can be split by the mesher.

## Additional Usage Notes

- If multiple sizing controls are scoped to the same entity and the specified element size is the same for all of them, the last sizing control that was created is respected.
- For a slender body, specifying hard local mesh sizing on the body might result in a mesh containing elements that are much larger than the specified size. To work around this, either use soft local mesh sizing on the body instead of hard sizing, or apply hard local mesh sizing to the faces of the body.

### 3.3.11. Volume Definitions

Volume definitions guide the mesher in the creation of a new engineering topology.

From your volume definitions, AIM creates objects that it uses in the **Volume Creation** process. Specifically, you can use volume definitions to:

- Define a flow region for meshing in cases where a model does not contain a body to represent the flow region that you want to mesh (**Extracted Volume**). Objects used for this purpose are composed of solid bodies, surface bodies (that is, capping surfaces), and a point. When you scope the **Bodies bounding flow region** for an **Extracted Volume**, the capping faces that are associated with the selected bodies are collected automatically for meshing. You do not need to select them manually.
- Identify solid bodies that should be united for meshing (**Geometry Simplification**). The bodies to unite must have the same material property and have no gaps between them. The mesher treats the united bodies as one part.

Faces in Selection Sets, as well as those to which physics conditions, boundary layer controls, and so on have been applied, are preserved in the resultant topology. Refer to [Protected Topologies in a Mesh](#) on page 183.

#### 3.3.11.1. Defining a Volume - Extracted Volume

**Volume Creation > Objects > Volume Definitions > Add > Extracted Volume**

To extract a volume for a **Volume Creation** task, you must define the volume by performing these steps. For descriptions of the options mentioned below, refer to [Extracted Volume Properties](#) on page 176. Also see [Extracted Volume Workflow](#) on page 106.

1. For **Bodies bounding flow region**, select the bodies that bound the volume to be extracted.
2. For **Point inside flow region**, select or create a point to identify the wetted region that will make up the flow volume.
3. For  **Feature resolution**, specify the sampling size to be used by the wrapper.

#### 3.3.11.2. Defining a Volume - Geometry Simplification

**Volume Creation > Objects > Volume Definitions > Add > Geometry Simplification**

To simplify geometry for a **Volume Creation** task, you must define a volume by performing these steps. For descriptions of the options mentioned below, refer to [Geometry Simplification Properties](#) on page 177. Also see [Geometry Simplification Workflow](#) on page 107.

1. For **Bodies to simplify**, select the solid bodies that you want to unite.
2. If more than one body is selected for **Bodies to simplify** and the selected bodies are in different parts, enable or disable  **Merge bodies**.
3. For  **Feature resolution**, specify the sampling size to be used by the wrapper.

#### 3.3.11.3. Extracted Volume Properties

The properties described here define how a volume definition that is being used to extract a volume is processed when the mesh for a **Volume Creation** task is generated.

Option	Definition
<b>Bodies bounding flow region</b>	<p>Scope the <b>Bodies bounding flow region</b> to the solid bodies that bound the volume to be extracted.</p> <p><b>Note:</b> The capping faces that are associated with the selected bodies are collected automatically for meshing. You do not need to select them.</p> <p>There are several scenarios involving <b>Extracted Volume</b> in which you can scope the same topological entity to multiple volume definitions, but a software check may determine the setup is invalid and require you to make adjustments. These scenarios include:</p> <ul style="list-style-type: none"> <li>• You may scope an internal volume definition (<b>Extracted Volume</b>) and an external volume definition (<b>Geometry Simplification</b>) to the same topological entity within a single <b>Volume Creation</b> task.</li> <li>• You may scope two volume definitions that are being used for <b>Extracted Volume</b> to the same topological entity within a single <b>Volume Creation</b> task as long as the volume definitions specify <i>different points</i>. Do not define multiple volumes that are scoped to the same topological entity and the <i>same point</i>.</li> <li>• You may scope two volume definitions (regardless of whether they are being used for <b>Extracted Volume</b> or <b>Geometry Simplification</b>) to the same topological entity in different (that is, parallel) <b>Volume Creation</b> tasks that are connected to the same import task.</li> </ul> <p>When you generate the mesh for the <b>Volume Creation</b> task in these cases, the software checks your setup to determine whether it is valid and reports any issues it finds.</p>
<b>Point inside flow region</b>	Select a <b>Point inside flow region</b> to identify the wetted region that will make up the flow volume.
 <b>(Optional) Feature resolution</b>	Specify the sampling size to be used by the wrapper. A value greater than 1 indicates that the sampling size is coarser while a value less than 1 indicates that the sampling size is finer than the sizes for remeshing. All Cartesian meshes are scaled by this factor. For example, if you set <b>Feature resolution</b> to 0.5, this factor reduces the mesh sizes throughout the domain by half. If you set <b>Feature resolution</b> to 2.0, the mesh sizes are doubled, which is useful if you want the mesher to ignore some features. Specify a value from 0.5 to 2.0; the default is 0.75.

### 3.3.11.4. Geometry Simplification Properties

The properties described here define how a volume definition that is being used to simplify geometry is processed when the mesh for a **Volume Creation** task is generated.

Option	Definition
<b>Bodies to simplify</b>	<p>For <b>Bodies to simplify</b>, select the solid bodies that you want to unite.</p> <p>There are several scenarios involving <b>Geometry Simplification</b> in which you can scope the same topological entity to multiple volume definitions, but a software check may determine the setup is invalid and require you to make adjustments. These scenarios include:</p> <ul style="list-style-type: none"> <li>• You may scope an internal volume definition (<b>Extracted Volume</b>) and an external volume definition (<b>Geometry Simplification</b>) to the same topological entity within a single <b>Volume Creation</b> task.</li> <li>• You may scope two volume definitions (regardless of whether they are being used for <b>Extracted Volume</b> or <b>Geometry Simplification</b>) to the same topological entity in</li> </ul>

Option	Definition
	<p>different (that is, parallel) <b>Volume Creation</b> tasks that are connected to the same import task.</p> <p>When you generate the mesh for the <b>Volume Creation</b> task in these cases, the software checks your setup to determine whether it is valid and reports any issues it finds.</p>
 (Optional) <b>Merge bodies</b>	<p>If more than one body is selected for <b>Bodies to simplify</b> and the selected bodies are in different parts, choose one of these options:</p> <ul style="list-style-type: none"> <li>To simplify the geometry by unifying multiple solid surfaces into one and ignoring any interior voids and faces, enable <b>Merge bodies</b>.</li> <li>To create a conformal, well-connected surface mesh for each of the selected bodies, disable <b>Merge bodies</b>.</li> </ul> <p>If the bodies selected for <b>Bodies to simplify</b> are members of the same multibody part, all selected bodies are merged regardless of the <b>Merge bodies</b> setting.</p>
 (Optional) <b>Feature resolution</b>	<p>Specify the sampling size to be used by the wrapper. A value greater than 1 indicates that the sampling size is coarser while a value less than 1 indicates that the sampling size is finer than the sizes for remeshing. All Cartesian meshes are scaled by this factor. For example, if you set <b>Feature resolution</b> to 0.5, this factor reduces the mesh sizes throughout the domain by half. If you set <b>Feature resolution</b> to 2.0, the mesh sizes are doubled, which is useful if you want the mesher to ignore some features. Specify a value from 0.5 to 2.0; the default is 0.75.</p>

### 3.3.12. Boundary Layers

Boundary layers are useful for improving the mesh for shear flow in fluid flow simulations or high stress concentrations in structural simulations. Boundary layer controls are available for **Meshing** tasks only. When defining these tasks, you can specify common **Boundary Layer Settings**, which will be applicable to all of the boundaries scoped to any local boundary layer controls defined for the task. To apply boundary layers to specific faces, you can [add additional boundary layer controls](#) as required.

**Note:** If you set up your simulation by launching the **Fluid Flow**, **Fluid-Structure Interaction**, or **Fluid-Solid Heat Transfer** template, or the **Thermal** template with **Fluid flow** physics selected, AIM adds a boundary layer control to the **Meshing** task automatically.

## 3.3.12.1. Defining Boundary Layers

### Mesh > Objects > Mesh Controls > Add > Boundary Layer

To identify and define boundary layers, you apply a boundary layer control to a single face or multiple faces.

**Note:** If you set up your simulation by launching the **Fluid Flow**, **Fluid-Structure Interaction**, or **Fluid-Solid Heat Transfer** template, or the **Thermal** template with **Fluid flow** physics selected, AIM adds a boundary layer control to the **Meshing** task automatically. Navigate to the control to specify its settings. You can also define additional boundary layer controls.

1. When **Automatically defined** is disabled on the **Boundary Layer** panel, use the **Location** field to select the desired faces manually, or enable **Automatically defined** to apply the control to all faces except those that are used in Selection Sets.
2. Specify the **Maximum layers**.

The **Maximum layers** control is applicable only when boundary layers are defined by **Smooth transition**, **First layer thickness**, **First aspect ratio**, or **Last aspect ratio**.

3. Choose how the boundary layers are to be defined.

#### 4. Specify the **First layer height**.

The **First layer height** control is applicable only when boundary layers are defined by **First layer thickness** or **Last aspect ratio**.

#### 5. Specify the **First aspect ratio**.

The **First aspect ratio** control is applicable only when boundary layers are defined by **First aspect ratio**.

#### 6. Specify the **Transition ratio**.

The **Transition ratio** control is applicable only when boundary layers are defined by **Smooth transition**.

#### 7. Specify the **Number of layers**.

The **Number of layers** control is applicable only when boundary layers are defined by **Total thickness**.

#### 8. Specify the **Growth rate**.

The **Growth rate** control is applicable only when boundary layers are defined by **Smooth transition**, **First layer thickness**, **Total thickness**, or **First aspect ratio**.

#### 9. Specify the **Aspect ratio (base/height)**.

The **Aspect ratio (base/height)** control is applicable only when boundary layers are defined by **Last aspect ratio**.

#### 10. Specify the **Maximum thickness**.

The **Maximum thickness** control is applicable only when boundary layers are defined by **Total thickness**.

For detailed descriptions of the local boundary layer controls, refer to [Local Boundary Layer Control Options](#) on page 179.

## 3.3.12.2. Local Boundary Layer Control Options

For a **Meshing** task, use these options to define layers for the specific boundaries (faces) that are scoped to a [boundary layer control](#).

### Automatically Defined

To allow AIM to determine boundary layer location automatically, enable **Automatically defined**. AIM associates the control to the DefaultBoundaryLayer() function, which returns all faces except those that are used in Selection Sets. This setting may not be appropriate for your simulation.

### Location

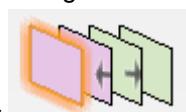
To manually specify the faces to which the boundary layer control is to be applied, disable **Automatically defined** and use **Location** to select the desired faces.

#### Location and Boundary Layer Growth in Part-based Meshing

When a selected face is shared topology, layers are generated differently depending on which face picking tool you use:

- To generate layers in both directions, use **Face selection**  only. Layers grow into both bodies that share the face.

- To generate layers in one direction only, use the disambiguate face sides feature of the topology selector to select the body on which you want the layers to grow. The disambiguate face sides feature is available

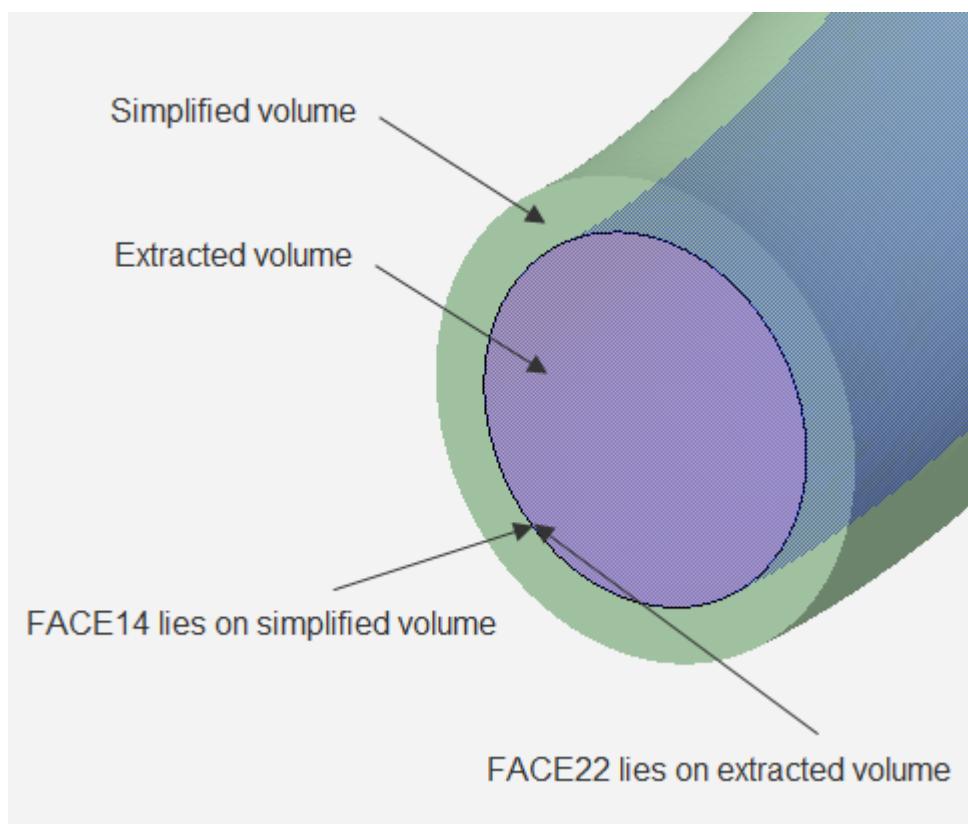


when you see arrows in the topology selector . Layers grow into the body adjacent to the disambiguated face side only.

If a selected face is not shared, the disambiguate face sides feature is unavailable and layers are generated in one direction only.

### Location and Boundary Layer Growth in Flow Volume Meshing

The figure below shows a model of fluid flow in a pipe. Geometry simplification was used to create a volume for the solid pipe, and volume extraction was used to create a volume for the fluid flowing inside it. In each volume, there is a distinct face in the overlapping region; it is not a shared face. If you select FACE14 as the **Location**, boundary layers are generated into the simplified volume only. If you select FACE22, boundary layers are generated into the extracted volume only. To generate boundary layers in both directions, define a Selection Set on the face during the **Volume Creation** task, and select that Selection Set as the **Location** of the boundary layer control in the **Meshing** task.



## Define by

The **Define by** option determines how the heights of the boundary layers are defined. Choose one of the following:

- Smooth Transition** – This is the default. The **Smooth transition** option uses the local tetrahedral element size to compute each local initial height and total height so that the rate of volume change is smooth. Each triangle that is in the boundary layer will have an initial height that is computed with respect to its area,

averaged at the nodes. This means that for a uniform mesh, the initial heights will be roughly the same, while for a varying mesh, the initial heights will vary.

The computations used for prism growth are as follows:

- The following value is computed at each node on the prism base:

$$\text{Height of last prism (H)} = \text{Transition ratio} * \text{average\_edge\_length}$$

- The height of the first layer ( $h$ ) is computed using the following formula, where  $g = \text{Growth rate}$  and  $n = \text{Number of layers}$ :

$$H = h * (g ^ (n-1))$$

Increasing the value of **Growth rate** reduces the total height of the layer. The total height approaches an asymptotic value with respect to the number of layers.

For details about the additional controls that appear when **Smooth transition** is selected, refer to the descriptions of the **Transition ratio**, **Maximum layers**, and **Growth rate** controls.

- **Total Thickness** – The **Total thickness** option creates constant layers using the values of the **Number of layers** and **Growth rate** controls to obtain a total thickness as defined by the value of the **Maximum thickness** control. Unlike the **Smooth transition** option, the **Total thickness** option means the thicknesses of the first layer and each following layer are constant.

For details about the additional controls that appear when **Total thickness** is selected, refer to the descriptions of the **Number of layers**, **Growth rate**, and **Maximum thickness** controls.

- **First Layer Thickness** – The **First layer thickness** option creates constant layers using the values of the **First layer height**, **Maximum layers**, and **Growth rate** controls to generate the layered mesh. Unlike the **Smooth transition** option, the **First layer thickness** option means the thicknesses of the first layer and each following layer are constant.

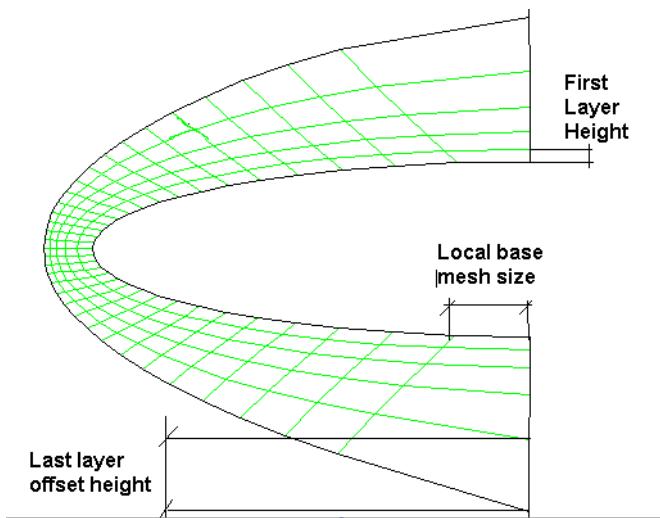
For details about the additional controls that appear when **First layer thickness** is selected, refer to the descriptions of the **First layer height**, **Maximum layers**, and **Growth rate** controls.

- **First Aspect Ratio** – The **First aspect ratio** option creates layers using the values of the **First aspect ratio**, **Maximum layers**, and **Growth rate** controls to generate the mesh.

For details about the additional controls that appear when **First aspect ratio** is selected, refer to the descriptions of the **First aspect ratio**, **Maximum layers**, and **Growth rate** controls.

- **Last Aspect Ratio** – The **Last aspect ratio** option creates layers using the values of the **First layer height**, **Maximum layers**, and **Aspect ratio (base/height)** controls to generate the mesh.

The figure below illustrates this option. Using the **First layer height** control, you can specify the height of the first layer. Local base mesh size is used to find the offset height for the last layer. For example, if you specify 20 for **Aspect ratio (base/height)**, the offset height of the last layer will be 0.2 times the local base mesh size. Local growth rate is used to calculate the other intermediate offset heights exponentially.



For details about the additional controls that appear when **Last aspect ratio** is selected, refer to the descriptions of the **First layer height**, **Maximum layers**, and **Aspect ratio (base/height)** controls.

## First Layer Height

The **First layer height** control determines the height of the first layer. This first layer consists of a single layer of prism elements that is formed against the faces of the boundary. You must enter a value for this control, and it must be greater than 0.

The **First layer height** control is applicable only when boundary layers are defined by **First layer thickness** or **Last aspect ratio**.

## First Aspect Ratio

By choosing the **First aspect ratio** option to define boundary layers, you can control the heights of the layers by defining the aspect ratio of the layer thickness that is extruded from the first layer. The *aspect ratio* is defined as the ratio of the local layer base size to the next layer height. Use the **First aspect ratio** control to specify the first aspect ratio to be used. Enter a value greater than 0. The default is 5.

The **First aspect ratio** control is applicable only when boundary layers are defined by **First aspect ratio**.

## Transition Ratio

In basic terms, the **Transition ratio** control determines the rate at which adjacent elements grow. It is the volume-based size change between the last layer of elements in the boundary layer and the first elements in the tetrahedron region. The value of **Transition ratio** is an ideal value and should produce accurate size change for layers from a planar boundary. However, be aware that areas of strong curvature will introduce an inaccuracy into the size change. If a proximity-based size function is being used, elements in proximity and elements with prism aspect ratios meeting the value defined by the **Maximum height over base** control will ignore this transition ratio.

The **Transition ratio** control is applicable only when boundary layers are defined by **Smooth transition**. Value must be 0 or greater. The default is **0.272**.

## Number of Layers

The **Number of layers** control determines the actual number of boundary layers in the mesh, except in places where layers are removed locally for reasons of improving mesh quality (for example, in areas where layers would otherwise collide with each other). Value must be 1 or greater. The default is **3**.

The **Number of layers** control is applicable only when boundary layers are defined by **Total thickness**.

## Maximum Layers

The **Maximum layers** control determines the maximum number of layers to be created in the mesh. Value must be 1 or greater. The default is **3**.

The **Maximum layers** control is applicable only when boundary layers are defined by **Smooth transition**, **First layer thickness**, **First aspect ratio**, or **Last aspect ratio**.

## Growth Rate

The **Growth rate** control determines the relative thickness of adjacent layers. As you move away from the face to which the boundary layer control is applied, each successive layer is approximately one growth rate factor thicker than the previous one. Valid values are from 1.0 to 5.0. The default is **1.2**.

The **Growth rate** control is applicable only when boundary layers are defined by **Smooth transition**, **First layer thickness**, **Total thickness**, or **First aspect ratio**.

## Aspect Ratio (Base/Height)

By choosing to define the boundary layers by the **Last aspect ratio**, you can control the heights of the boundary layers by defining the aspect ratio of the layers that are extruded from the base layer. The *aspect ratio* is defined as the ratio of the local base layer size to the boundary layer height. Use the **Aspect ratio (base/height)** control to specify the aspect ratio to be used. Value must be 0.5 or greater. The default is **3**.

The **Aspect ratio (base/height)** control is applicable only when boundary layers are defined by **Last aspect ratio**.

## Maximum Thickness

The **Maximum thickness** control determines the desired thickness of the boundary layer. You must enter a value for this control, and it must be greater than 0.

The **Maximum thickness** control is applicable only when boundary layers are defined by **Total thickness**.

### 3.3.13. Protected Topologies in a Mesh

When a mesh is generated, protected topologies are identified and “protected” by the mesher. Protected topologies are created for any topologies that are scoped to physics conditions, loads, interfaces, boundary layer controls, Selection Sets, and so on, at the time of meshing, so it is important to define them prior to mesh generation. The mesher respects the boundaries of the protected topologies and does not cross them. A **Topology Protections** list, which appears for **Volume Creation** tasks only, is populated with all protected topologies associated with the current task and any connected **Meshing** tasks.

**Note:** Selection Sets are the only type of protected topology that you can add to **Volume Creation** tasks. Refer to [Fluid Flow Meshing Best Practices](#) on page 110 for an example of protected topology and volume creation.

## 3.3.13.1. Creating a Selection Set to Preserve Topologies in the Mesh

**Volume Creation > Auxiliary Definitions > Selection Sets > Add > Selection Set**

You can add a Selection Set directly to a **Volume Creation** task to create a named grouping of entities that will be **protected** by the mesher.

For an example of protected topology and volume creation, refer to [Fluid Flow Meshing Best Practices](#) on page 110.

1. Rename the **Selection Set** panel with a meaningful title (for example, `Bolt faces`).
2. For **Location**, select the entities that you want to group.

## 3.3.13.2. Protected Topologies and Mesh State

Updates to previous connected tasks and changes to **protected topologies** may affect the mesh state, which in turn affects updates of the **Topology Protections** list:

- When you update any previous connected task, the mesh state becomes underdefined. If the update involved changes to protected topologies, the **Topology Protections** list is updated automatically and the task requires a remesh.
- Changes to protected topologies in the current task cause the mesh state of the task to become underdefined. The **Topology Protections** list is updated automatically and the task requires a remesh.
- Changes to protected topologies in any next connected task do not affect the mesh state of the task. As long as the mesh is up-to-date, the **Topology Protections** list is not updated. Once the mesh state becomes underdefined, the list is updated automatically and the task requires a remesh.

## 3.3.14. Element Shape

The **Element Shape** determines whether the mesher generates a tetrahedral or hexahedral mesh for a selected body. If you do not specify element shape, the mesher attempts to mesh sweepable bodies with hexahedrons and provides a tetrahedral mesh on bodies that are not sweepable or if the quality of the hexahedral mesh is poor.

In some cases, the best mesh is a hexahedral mesh, while in others it is a tetrahedral mesh. The model being simulated and the meshing constraints often dictate the best element shape to use. A hybrid mesh that contains hexahedrons in anisotropic or critical areas and tetrahedrons in other areas is often ideal.

## 3.3.14.1. Defining the Element Shape of a Mesh

**Mesh > Objects > Mesh Controls > Add > Element Shape**

The **Element Shape** determines whether the mesher generates a tetrahedral or hexahedral mesh for the selected body.

For more details about these settings, refer to [Element Shape Settings](#) on page 185.

1. Select the **Location** (body) in your geometry for which you want to define the element shape.
2. Choose a **Shape**.
  - **Automatic** - This is the default. The mesher attempts to mesh sweepable bodies with hexahedrons and provides a tetrahedral mesh on bodies that are not sweepable or if the quality of the hexahedral mesh is poor.
  - **Tetrahedrons** - The mesher generates a tetrahedral mesh.

- **Hexahedrons** - The mesher generates a hexahedral mesh.

The mesh includes prism elements if boundary layers are generated.

### 3. Choose how to handle **Midside nodes**.

- **Use engineering intent** - This is the default. The mesher determines whether to create quadratic or linear elements for the body based on the value of the **Engineering intent** control.
- **Dropped** - No elements in the body have midside nodes. Overrides the setting of the **Engineering intent** control.
- **Kept** - All elements in the body have midside nodes. Overrides the setting of the **Engineering intent** control.

### 4. If you chose **Hexahedrons** and you want to select **Source face(s)** manually, select the source faces in the graphics view.

If **Source face(s)** is left blank, AIM determines the source faces automatically.

If the model contains multiple bodies and mesh generation fails on some bodies, the mesher returns as much mesh as it can. In such cases, the graphics view shows the mesh for the bodies where mesh generation was successful but shows no mesh for any failed bodies.

## 3.3.14.2. Element Shape Settings

Use these settings to specify **Element Shape** on a body.

### Location

Select the body in your geometry for which you want to specify the element shape.

### Shape

Choose one of the following:

Option	Description	Considerations
<b>Automatic</b>	The mesher attempts to mesh sweepable bodies with hexahedrons and provides a tetrahedral mesh on bodies that are not sweepable or if the quality of the hexahedral mesh is poor.	<ul style="list-style-type: none"> <li>• Default; nothing to set up</li> </ul>
<b>Tetrahedrons</b>	The mesher generates a tetrahedral mesh.	<ul style="list-style-type: none"> <li>• Preferred when geometry is not suitable for sweeping</li> <li>• Easier to mesh more complex geometry</li> <li>• Mesh quality is often easier to achieve</li> <li>• Better mesh transitioning</li> </ul>
<b>Hexahedrons</b>	The mesher generates a hexahedral mesh.	<ul style="list-style-type: none"> <li>• Preferred when geometry is suitable for sweeping</li> </ul>

Option	Description	Considerations
		<ul style="list-style-type: none"> <li>• Fewer elements</li> <li>• Faster solution time</li> <li>• Better accuracy</li> <li>• Mesh transitioning can be problematic</li> </ul>

**Note:** The mesh includes prism elements if boundary layers are generated.

## Midside Nodes

The **Element Shape** feature includes support for mixed order meshing. When you specify an element shape on a body in a multibody part, you can set the **Midside nodes** option to **Kept** for some bodies (resulting in higher order elements) and to **Dropped** for other bodies (resulting in lower order elements).

The order in which bodies are meshed is determined internally. Midside node handling on the body that is meshed first affects elements at the interface between bodies. This means that if you specify **Kept** on one body and **Dropped** on the adjacent body and the higher order body is meshed first, one layer of higher order elements is generated at the interface, with mixed order elements transitioning away from the interface on the second body.

Choose one of the following:

- |                               |  |
|-------------------------------|--|
| <b>Use engineering intent</b> | This is the default. The mesher determines whether to create quadratic or linear elements for the body based on the value of the <a href="#">Engineering intent</a> control. |
| <b>Dropped</b>                | No elements in the body have midside nodes. Overrides the setting of the <a href="#">Engineering intent</a> control.   |
| <b>Kept</b>                   | All elements in the body have midside nodes. Overrides the setting of the <a href="#">Engineering intent</a> control.  |

### 3.3.14.2.1. Optional Element Shape Settings

 These settings are available for specifying **Element Shape** on a body if you turn Filtering off.

## Source Face(s)

If you set **Shape** to **Hexahedrons** and you want to scope **Source face(s)** manually, select the source faces in the graphics view.

**Note:** If **Source face(s)** is left blank, AIM determines the source faces automatically.

### 3.3.15. Evaluating Mesh Quality

Mesh diagnostics help you evaluate mesh quality. For [Volume Creation](#) tasks and for [Meshing](#) tasks used to mesh the output of [Volume Creation](#) tasks, you can review mesh diagnostics for the entire model. For [Physics](#) tasks used with [automatic physics-aware meshing](#) and for [Meshing](#) tasks used for [part-based meshing](#), you can review mesh diagnostics for the entire model, an individual part, or an individual body.

Mesh diagnostics include the number of nodes in the mesh; number of elements in the mesh; and the minimum, maximum, average, and standard deviation values for these metric types:

- [Element quality](#)
- [Aspect ratio calculation for triangles](#)

- Aspect ratio calculation for quadrilaterals
- Jacobian ratio
- Warping factor
- Parallel deviation
- Maximum corner angle
- Skewness
- Orthogonal quality

### 3.3.15.1. Viewing Mesh Diagnostics

**Volume Creation > Output > Metrics > Add > Mesh Diagnostics**

**Mesh > Output > Metrics > Add > Mesh Diagnostics**

**Physics > Output > Metrics > Add > Mesh Diagnostics**

By default, the numbers of nodes and elements for the entire model are reported on the **Mesh Diagnostics** panel after you generate the mesh. You can view mesh metric information also. Follow these steps.

1. For **Physics** tasks used with [automatic physics-aware meshing](#) and for **Meshing** tasks used for [part-based meshing](#), you may scope the **Location** on the **Mesh Diagnostics** panel to an individual body or part whose diagnostics you want to view. If you are working in a **Volume Creation** task or a **Meshing** task that was used to mesh the output of a **Volume Creation** task, you cannot scope a **Location**.
2. On the **Mesh Diagnostics** panel, choose the metric of interest from the **Mesh metric** drop-down menu.

The available metrics depend on which type of task you are working in.

The minimum, maximum, average, and standard deviation values are reported for the selected metric type. If you select a different metric type (or change the scope for a **Meshing** task), the data is updated automatically.

### 3.3.15.2. Element Quality

The **Element quality** option provides a composite quality metric that ranges between 0 and 1. This metric is based on the ratio of the volume to the sum of the square of the edge lengths for 2D quad/tri elements, or the square root of the cube of the sum of the square of the edge lengths for 3D elements. A value of 1 indicates a perfect cube or square while a value of 0 indicates that the element has a zero or negative volume.

This can also be expressed as follows:

- For two-dimensional quad/tri elements:

$$\text{Quality} = C \left( \frac{\text{area}}{\sum (\text{EdgeLength})^2} \right)$$

**Note:** If [Shape checking](#) is set to [Aggressive mechanical](#) and the [Jacobian ratio](#) is less than zero, an error occurs.

- For three-dimensional brick elements:

$$\text{Quality} = C \left[ \frac{\text{volume}}{\sqrt{\sum (\text{Edge length})^2}} \right]^3$$

For each type of element, the following table lists the value of C:

Element	Value of C
Triangle	6.92820323

Element	Value of C
Quadrangle	4.0
Tetrahedron	124.70765802
Hexagon	41.56921938
Wedge	62.35382905
Pyramid	96

### 3.3.15.3. Aspect Ratio Calculation for Triangles

The aspect ratio for a triangle is computed in the following manner, using only the corner nodes of the element:

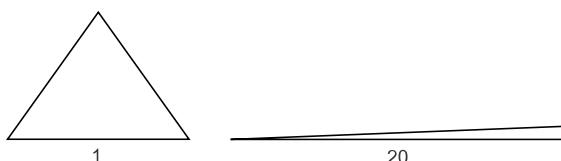
**Figure 3.3.15.3.1. Triangle Aspect Ratio Calculation**



1. A line is constructed from one node of the element to the midpoint of the opposite edge, and another through the midpoints of the other two edges. In general, these lines are not perpendicular to each other or to any of the element edges.
2. Rectangles are constructed centered about each of these two lines, with edges passing through the element edge midpoints and the triangle apex.
3. These constructions are repeated using each of the other two corners as the apex.
4. The aspect ratio of the triangle is the ratio of the longer side to the shorter side of whichever of the six rectangles is most stretched, divided by the square root of 3.

The best possible aspect ratio, for an equilateral triangle, is 1. A triangle having an aspect ratio of 20 is shown in the figure below.

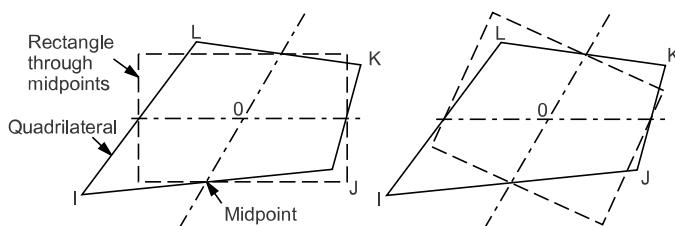
**Figure 3.3.15.3.2. Aspect Ratio for Triangles**



### 3.3.15.4. Aspect Ratio Calculation for Quadrilaterals

The aspect ratio for a quadrilateral is computed by the following steps, using only the corner nodes of the element:

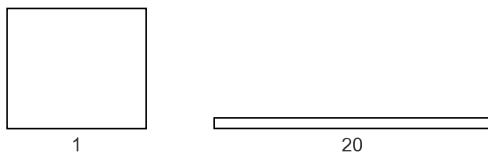
**Figure 3.3.15.4.1. Quadrilateral Aspect Ratio Calculation**



1. If the element is not flat, the nodes are projected onto a plane passing through the average of the corner locations and perpendicular to the average of the corner normals. The remaining steps are performed on these projected locations.
2. Two lines are constructed that bisect the opposing pairs of element edges and which meet at the element center. In general, these lines are not perpendicular to each other or to any of the element edges.
3. Rectangles are constructed centered about each of the two lines, with edges passing through the element edge midpoints. The aspect ratio of the quadrilateral is the ratio of a longer side to a shorter side of whichever rectangle is most stretched.

The best possible aspect ratio, for a square, is 1. A quadrilateral having an aspect ratio of 20 is shown in the figure below.

**Figure 3.3.15.4.2. Aspect Ratio for Quadrilaterals**



### 3.3.15.5. Jacobian Ratio

Jacobian ratio is computed and tested for all elements except triangles and tetrahedra that (a) are linear (have no midside nodes) or (b) have perfectly centered midside nodes. A high ratio indicates that the mapping between element space and real space is becoming computationally unreliable.

### Jacobian Ratio Calculation

An element's Jacobian ratio is computed by the following steps, using the full set of nodes for the element:

1. At each sampling location listed in the table below, the determinant of the Jacobian matrix is computed and called  $R_J$ .  
 $R_J$  at a given point represents the magnitude of the mapping function between element natural coordinates and real space. In an ideally-shaped element,  $R_J$  is relatively constant over the element, and does not change sign.

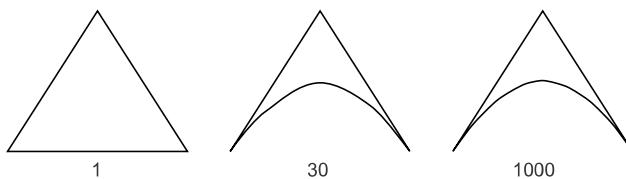
Element Shape	$R_J$ Sampling Locations
10-node tetrahedra	corner nodes
5-node or 13-node pyramids	base corner nodes and near apex node (apex $R_J$ factored so that a pyramid having all edges the same length will produce a Jacobian ratio of 1)
8-node quadrilaterals	corner nodes and centroid
20-node bricks	all nodes and centroid
all other elements	corner nodes

2. The Jacobian ratio of the element is the ratio of the maximum to the minimum sampled value of  $R_J$ . If the maximum and minimum have opposite signs, the Jacobian ratio is arbitrarily assigned to be -100 (and the element is clearly unacceptable).

3. If the element is a midside-node tetrahedron, an additional  $R_j$  is computed for a fictitious straight-sided tetrahedron connected to the four corner nodes. If that  $R_j$  differs in sign from any nodal  $R_j$  (an extremely rare occurrence), the Jacobian ratio is arbitrarily assigned to be -100.
4. If the element is a line element having a midside node, the Jacobian matrix is not square (because the mapping is from one natural coordinate to 2D or 3D space) and has no determinant. For this case, a vector calculation is used to compute a number which behaves like a Jacobian ratio. This calculation has the effect of limiting the arc spanned by a single element to about 106 degrees.

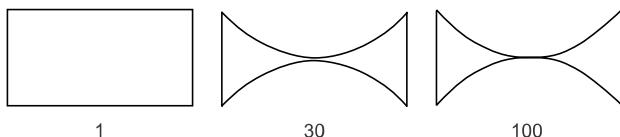
A triangle or tetrahedron has a Jacobian ratio of 1 if each midside node, if any, is positioned at the average of the corresponding corner node locations. This is true no matter how otherwise distorted the element may be. Hence, this calculation is skipped entirely for such elements. Moving a midside node away from the edge midpoint position will increase the Jacobian ratio. Eventually, even very slight further movement will break the element. Refer to the figure below. We describe this as “breaking” the element because it suddenly changes from acceptable to unacceptable (that is, “broken”).

**Figure 3.3.15.5.1. Jacobian Ratios for Triangles**



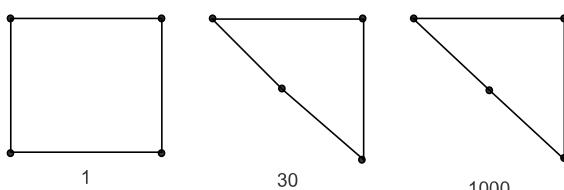
Any rectangle or rectangular parallelepiped having no midside nodes, or having midside nodes at the midpoints of its edges, has a Jacobian ratio of 1. Moving midside nodes toward or away from each other can increase the Jacobian ratio. Eventually, even very slight further movement will break the element. Refer to the figure below.

**Figure 3.3.15.5.2. Jacobian Ratios for Rectangles**



A quadrilateral or brick has a Jacobian ratio of 1 if (a) its opposing faces are all parallel to each other, and (b) each midside node, if any, is positioned at the average of the corresponding corner node locations. As a corner node moves near the center, the Jacobian ratio climbs. Eventually, any further movement will break the element. Refer to the figure below.

**Figure 3.3.15.5.3. Jacobian Ratios for Quadrilaterals**



### 3.3.15.6. Warping Factor

Warping factor is computed and tested for some quadrilateral shell elements, and the quadrilateral faces of bricks, wedges, and pyramids. A high factor may indicate a condition the underlying element formulation cannot handle well, or may simply hint at a mesh generation flaw.

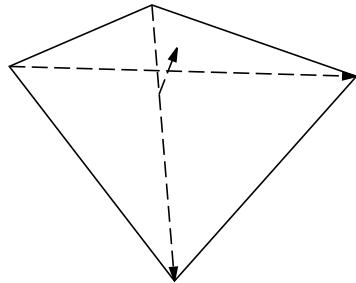
## Warping Factor Calculation for Quadrilateral Shell Elements

A quadrilateral element's warping factor is computed from its corner node positions and other available data by the following steps:

**Note:** When computing the warping factor for a quadrilateral shell element, AIM assumes 0 thickness for the shell.

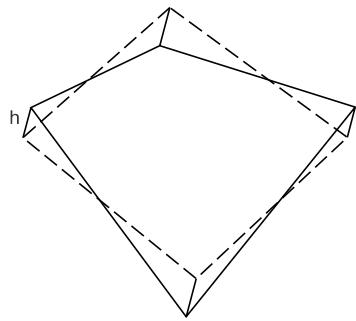
1. An average element normal is computed as the vector (cross) product of the two diagonals.

**Figure 3.3.15.6.1. Shell Average Normal Calculation**



2. The projected area of the element is computed on a plane through the average normal (the dotted outline in the figure below).
3. The difference in height of the ends of an element edge is computed, parallel to the average normal. In the figure below, this distance is  $2h$ . Because of the way the average normal is constructed,  $h$  is the same at all four corners. For a flat quadrilateral, the distance is zero.

**Figure 3.3.15.6.2. Shell Element Projected onto a Plane**



4. The “area warping factor” ( for the element is computed as the edge height difference divided by the square root of the projected area.

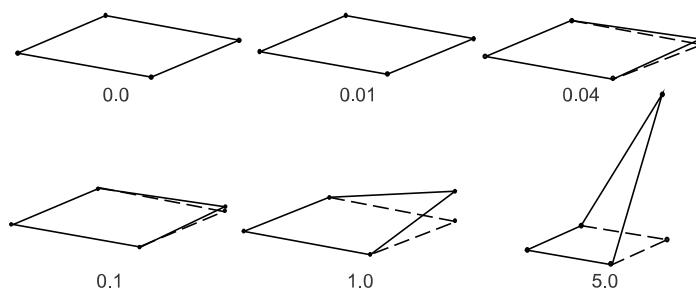
5. For all shells except those in the “membrane stiffness only” group, if the thickness is available, the “thickness warping factor” is computed as the edge height difference divided by the average element thickness. This could be substantially higher than the area warping factor computed in 4 (above).
6. The warping factor tested against warning and error limits (and reported in warning and error messages) is the larger of the area factor and, if available, the thickness factor.
7. The best possible quadrilateral warping factor, for a flat quadrilateral, is zero.

The figure below shows a “warped” element plotted on top of a flat one. Only the right-hand node of the upper element is moved. The element is a unit square, with a real constant thickness of 0.1.

When the upper element is warped by a factor of 0.01, it cannot be visibly distinguished from the underlying flat one.

When the upper element is warped by a factor of 0.04, it just begins to visibly separate from the flat one.

**Figure 3.3.15.6.3. Quadrilateral Shell Having Warping Factor**



Warping of 0.1 is visible given the flat reference, but seems trivial; however, it is well beyond the error limit for a membrane shell. Warping of 1.0 is visually unappealing. This is the error limit for most shells.

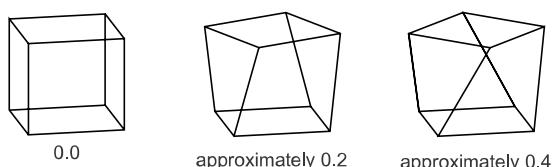
Warping beyond 1.0 would appear to be obviously unacceptable; however, SHELL181 permits even this much distortion. Furthermore, the warping factor calculation seems to peak at about 7.0. Moving the node further off the original plane, even by much larger distances than shown here, does not further increase the warping factor for this geometry. Users are cautioned that manually increasing the error limit beyond its default of 5.0 for these elements could mean no real limit on element distortion.

## Warping Factor Calculation for 3D Solid Elements

The warping factor for a 3D solid element face is computed as though the four nodes make up a quadrilateral shell element with no real constant thickness available, using the square root of the projected area of the face as described in 4 (above).

The warping factor for the element is the largest of the warping factors computed for the six quadrilateral faces of a brick, three quadrilateral faces of a wedge, or one quadrilateral face of a pyramid. Any brick element having all flat faces has a warping factor of zero. Refer to the figure below.

**Figure 3.3.15.6.4. Warping Factor for Bricks**



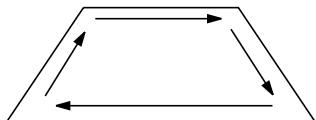
Twisting the top face of a unit cube by 22.5 degrees and 45 degrees relative to the base produces warping factors of about 0.2 and 0.4, respectively.

### 3.3.15.7. Parallel Deviation

Parallel deviation is computed using the following steps:

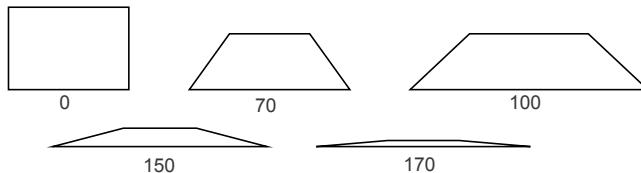
1. Ignoring midside nodes, unit vectors are constructed in 3D space along each element edge, adjusted for consistent direction, as demonstrated in the figure below.

**Figure 3.3.15.7.1. Parallel Deviation Unit Vectors**



2. For each pair of opposite edges, the dot product of the unit vectors is computed, then the angle (in degrees) whose cosine is that dot product is computed. The parallel deviation is the larger of these two angles. (In the illustration above, the dot product of the two horizontal unit vectors is 1, and  $\text{acos}(1) = 0$  degrees. The dot product of the two vertical vectors is 0.342, and  $\text{acos}(0.342) = 70$  degrees. Therefore, this element's parallel deviation is 70 degrees.)
3. The best possible deviation, for a flat rectangle, is 0 degrees. The figure below shows quadrilaterals having deviations of 0 degrees, 70 degrees, 100 degrees, 150 degrees, and 170 degrees.

**Figure 3.3.15.7.2. Parallel Deviations for Quadrilaterals**

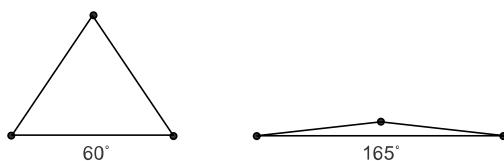


### 3.3.15.8. Maximum Corner Angle

The maximum corner angle is computed as follows.

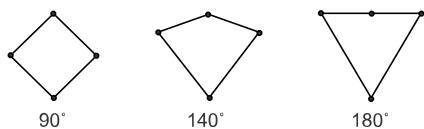
The maximum angle between adjacent edges is computed using corner node positions in 3D space. (Midside nodes, if any, are ignored.) The best possible triangle maximum angle, for an equilateral triangle, is 60 degrees. The figure below shows triangles having maximum corner angles of 60 degrees and 165 degrees.

**Figure 3.3.15.8.1. Maximum Corner Angles for Triangles**



The best possible quadrilateral maximum angle, for a flat rectangle, is 90 degrees. The figure below shows quadrilaterals having maximum corner angles of 90 degrees, 140 degrees, and 180 degrees.

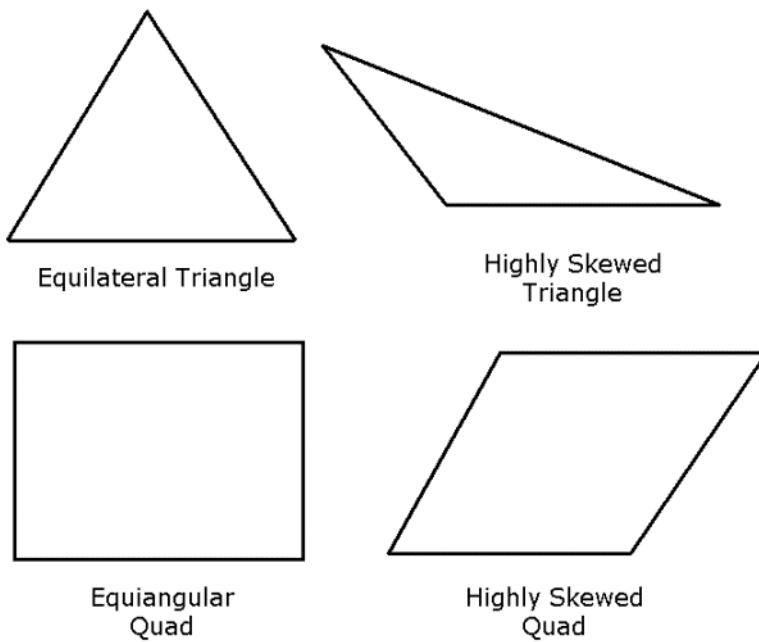
**Figure 3.3.15.8.2. Maximum Corner Angles for Quadrilaterals**



### 3.3.15.9. Skewness

Skewness is one of the primary quality measures for a mesh. Skewness determines how close to ideal (that is, equilateral or equiangular) a face or cell is.

**Figure 3.3.15.9.1. Ideal and Skewed Triangles and Quadrilaterals**



The following table lists the range of skewness values and the corresponding cell quality.

Value of Skewness	Cell Quality
1	degenerate
0.9 — < 1	bad (sliver)
0.75 — 0.9	poor
0.5 — 0.75	fair
0.25 — 0.5	good
> 0 — 0.25	excellent
0	equilateral

According to the definition of skewness, a value of 0 indicates an equilateral cell (best) and a value of 1 indicates a completely degenerate cell (worst). Degenerate cells (slivers) are characterized by nodes that are nearly coplanar (collinear in 2D).

Highly skewed faces and cells are unacceptable because the equations being solved assume that the cells are relatively equilateral/equiangular.

Two methods for measuring skewness are:

- Based on the equilateral volume (applies only to triangles and tetrahedra).
- Based on the deviation from a normalized equilateral angle. This method applies to all cell and face shapes, for example, pyramids and prisms.

## Equilateral-Volume-Based Skewness

In the equilateral volume deviation method, skewness is defined as

$$\text{Skewness} = \frac{\text{Optimal Cell Size} - \text{Cell Size}}{\text{Optimal Cell Size}}$$

where, the optimal cell size is the size of an equilateral cell with the same circumradius.

Quality meshes have a skewness value of approximately 0.1 for 2D and 0.4 for 3D. The table above provides a general guide to the relationship between cell skewness and quality.

In 2D, all cells should be good or better. The presence of cells that are fair or worse indicates poor boundary node placement. You should try to improve your boundary mesh as much as possible, because the quality of the overall mesh can be no better than that of the boundary mesh.

In 3D, most cells should be good or better, but a small percentage will generally be in the fair range and there are usually even a few poor cells.

**Note:** The Equilateral-Volume-Based Skewness quality metric applies to any mesh element that includes a triangular face. For triangular and tetrahedral elements, all faces of which are strictly triangular, the Equilateral-Volume-Based Skewness metric applies directly. For wedge or pyramidal elements, which include combinations of triangular and quadrilateral faces, the software computes both Equilateral-Volume-Based Skewness metrics (for the triangular faces) and Normalized Equiangular Skewness metrics (for the quadrilateral faces and 3D element, itself) and reports the maximum computed metric as the element skewness. As a result, Equilateral-Volume-Based Skewness metrics reported for meshes that contain wedge and/or pyramidal elements may include skewness values attributable to Normalized Equiangular Skewness computations.

## Normalized Equiangular Skewness

In the normalized angle deviation method, skewness is defined (in general) as

$$\max \left[ \frac{\theta_{\max} - \theta_e}{180 - \theta_e}, \frac{\theta_e - \theta_{\min}}{\theta_e} \right]$$

where

- $\theta_{\max}$  = largest angle in the face or cell
- $\theta_{\min}$  = smallest angle in the face or cell
- $\theta_e$  = angle for an equiangular face/cell (for example, 60 for a triangle, 90 for a square)

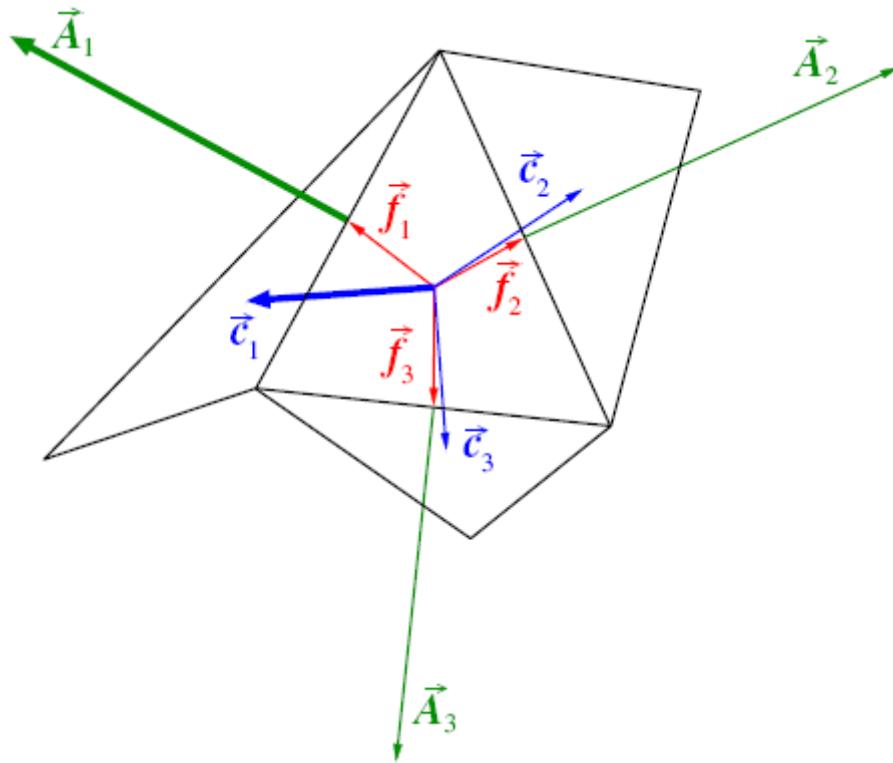
For a pyramid, the cell skewness will be the maximum skewness computed for any face. An ideal pyramid (skewness = 0) is one in which the four triangular faces are equilateral (and equiangular) and the quadrilateral base face is a square. The guidelines in the table above apply to the normalized equiangular skewness as well.

### 3.3.15.10. Orthogonal Quality

The range for orthogonal quality is 0-1, where a value of 0 is worst and a value of 1 is best.

The orthogonal quality for cells is computed using the face normal vector,  $\vec{A}_i$  for each face; the vector from the cell centroid to the centroid of each of the adjacent cells,  $\vec{c}_i$ ; and the vector from the cell centroid to each of the faces,  $\vec{f}_i$ . The figure below illustrates the vectors used to determine the orthogonal quality for a cell.

**Figure 3.3.15.10.1. Vectors Used to Compute Orthogonal Quality for a Cell**



For each face, the cosines of the angle between  $\vec{A}_i$  and  $\vec{c}_i$ , and between  $\vec{A}_i$  and  $\vec{f}_i$ , are calculated. The smallest calculated cosine value is the orthogonality of the cell. Finally, **Orthogonal Quality** depends on cell type:

- For tetrahedral, prism, and pyramid cells, the **Orthogonal Quality** is the minimum of the orthogonality and (1 - cell skewness).
- For hexahedral and polyhedral cells, the **Orthogonal Quality** is the same as the orthogonality.

**Note:**

- When the cell is located on the boundary, the vector  $\vec{c}_i$  across the boundary face is ignored during the quality computation.
- When the cell is separated from the adjacent cell by an internal wall (for example, a baffle), the vector  $\vec{c}_i$  across the internal boundary face is ignored during the quality computation.
- When the adjacent cells share a parent-child relationship, the vector  $\vec{f}_i$  is the vector from the cell centroid to the centroid of the child face while the vector  $\vec{c}_i$  is the vector from the cell centroid to the centroid of the adjacent child cell sharing the child face.

Orthogonal quality in AIM is equivalent to Inverse Orthogonal Quality in ANSYS Fluent Meshing, except that the scale is reversed:

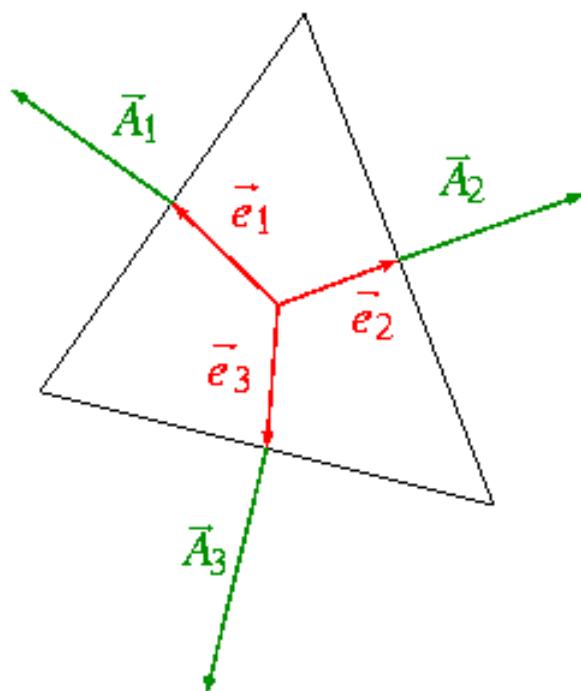
Inverse Orthogonal Quality = 1 - Orthogonal Quality

The orthogonal quality values may not correspond exactly with the inverse orthogonal quality values in ANSYS Fluent because the computation depends on boundary conditions on internal surfaces (*WALL* vs. *INTERIOR/FAN/RADIATOR/POROUS-JUMP*). ANSYS Fluent may return different results which reflect the modified mesh topology on which fluid flow simulations are performed.

For more information about Inverse Orthogonal Quality, see .

In a similar way, orthogonal quality for faces is computed as the smallest cosine of the angle between the edge normal vector,  $\vec{A}_i$ , for each edge and the vector from the face centroid to the centroid of each edge,  $\vec{e}_i$ . The figure below illustrates the vectors used to determine the orthogonal quality for a face.

**Figure 3.3.15.10.2. Vectors Used to Compute Orthogonal Quality for a Face**



### 3.3.15.11. Size Change

Size change is the ratio of the area (or volume) of a face (or cell) in the geometry to the area (or volume) of each neighboring face (or cell). This ratio is calculated for every face (or cell) in the domain.

# Chapter 4: Physics

---

Define your physics and solution settings of your simulation, including setting:

- [Physics Regions](#)
  - [Material Assignments](#)
  - [Physics Options](#)
  - [Physics Conditions](#)
  - [Initial Conditions](#)
  - [Interface Conditions](#)
  - [Solver Options](#)
- 

## 4.1. Adjusting the Calculation Type Settings

### **Physics > Settings > Calculation type**

In the Physics task, the **Calculation Type** property enables you to determine how the calculations are performed in your simulation. All physics types allow for a steady state, or **Steady/Static**, calculation by default.

For structural simulations, you can model vibration characteristics using the **Modal** setting. You can also select **Optimization** to compute an optimal structural design for a selected region of your model with specific design goals and constraints, or select **Eigenvalue buckling** to predict the buckling strength of your model.

For thermal and fluid flow simulations, you can model time-varying thermal and flow behavior using the **Time-dependent** setting.

For the magnetics simulation, you can model designs in which sinusoidal time-varying, alternating currents flowing in conductors produce a time-varying magnetic field that induces Eddy currents by using the **Frequency response** settings.

### 4.1.1. Frequency Response Calculation Type Settings

**Frequency Response** settings are available for Physics Regions with the Physics type set to Electromagnetics and the Physics Calculation type set to Frequency response.

#### **Physics > Settings > Frequency Response**

Frequency response calculation type settings allow you to:

- [Specify a single frequency for which to solve.](#)
- [Specify a range of frequencies for which to solve.](#)
- [Specify a tabular list of frequencies for which to solve.](#)

## 4.1.1.1. Specifying a Single Frequency for Frequency Response Calculations

A single value **Frequency** setting is available for Physics Regions with the Physics type set to Electromagnetics, the Physics Calculation type set to Frequency response, and the Frequency specification set to Single value.

For frequency response simulations, you can specify a single frequency to solve for as follows:

1. In the Workflow view, click **Physics** to open the Physics task.
2. In the Physics data panel, select **Frequency response** as the **Calculation type**.
3. Select **Single value** as the **Frequency specification**.
4. Enter the **Frequency** value or expression to solve for. This value is also used for calculating the adaptive meshing solution.

## 4.1.1.2. Specifying a Range of Frequencies for Frequency Response Calculations

Frequency **Range** settings are available for Physics Regions with the Physics type set to Electromagnetics, the Physics Calculation type set to Frequency response, and the Frequency specification set to Range.

You can define a range of frequencies (the sweep range) for which to solve a frequency response simulation as follows:

1. In the Workflow view, click **Physics** to open the Physics task.
2. In the Physics data panel, select **Frequency response** as the **Calculation type**.
3. Select **Range** as the **Frequency specification**.  
The total number of values in the sweep range must be less than 300.
4. Specify a value or expression for the starting frequency of the sweep range.  
**Start frequency** must be greater than 0.
5. Specify a value or expression for the ending frequency of the sweep range.  
**End frequency** must be greater than the Start frequency.
6. For the frequency **Distribution** setting:

If you want to...	Then choose...
Generate frequency values of equal step size within the sweep range	<b>Linear</b> , and specify a value or expression for the <b>Frequency step size</b> for values between the start and end frequencies.
Generate frequency values logarithmically spaced within the sweep range	<b>Logarithmic</b> , and specify a value or expression for the number of <b>Frequencies per decade</b> between the start and end frequencies.

7. For **Frequency for adaptive meshing**, you can accept the default **Program controlled** to use the maximum frequency in the sweep range for computing the adaptive meshing solution; or select **User defined**, which enables you to specify any **Frequency** in the sweep range.

## 4.1.1.3. Specifying a Tabular List of Frequencies for Frequency Response Calculations

Frequency Range settings are available for Physics Regions with the Physics type set to Electromagnetics, the Physics Calculation type set to Frequency response, and the Frequency specification set to Tabular.

You can define a tabular list of frequencies (the sweep range) for which to solve a frequency response simulation as follows:

1. In the Workflow view, click **Physics** to open the Physics task.
2. In the Physics data panel, select **Frequency response** as the **Calculation type**.
3. Select **Tabular** as the **Frequency specification**.

The total number of values in the tabular list that constitutes the sweep range must be less than 300.

4. In the **Frequencies** table, enter the desired frequency values that you want the solver to sweep.

Regardless of the order in which the frequencies are entered, the solver will sweep them from lowest to highest value.

5. For **Frequency for adaptive meshing**, you can accept the default **Program controlled** to use the maximum frequency in the tabular sweep range for computing the adaptive meshing solution; or select **User defined**, which enables you to specify any **Frequency** in the sweep range.
- 

## 4.2. Fidelity Capture

If you are using automatic physics-aware meshing but need more fidelity control than what the **Solution fidelity** slider provides, you can change the size function method that AIM is using to calculate mesh sizes. You can then modify the size function properties so that AIM can provide a more appropriate size distribution for your mesh.

- In general, size function properties work similarly regardless of whether you are using automatic physics-aware meshing or manual meshing. However, if you are using automatic physics-aware meshing, AIM may apply additional controls that override your specified property settings.
- You cannot parameterize size function properties when you are using them with automatic physics-aware meshing. If you want to create design points to perform what-if studies, use [manual meshing](#) instead.

### 4.2.1. Controlling the Mesh by Using Fidelity Capture

#### Physics > Settings > Global Fidelity Capture

Using Global Fidelity Capture, you can select the most appropriate size function method for your simulation and modify the corresponding size function properties to obtain additional fidelity control.

1. Click  **Show all properties** in the Physics panel.
  2. Under **Settings**, expand **Global Fidelity Capture** and select the **Size function method** that is most appropriate for your simulation.  
The default property values for the selected size function method are shown.
  3. Disable **Use predefined settings** to modify the size function properties.  
The available size function properties differ depending on whether you set **Size function method** to [curvature](#), [proximity](#), [curvature and proximity](#), or [fixed](#).
- 

## 4.3. Physics Regions

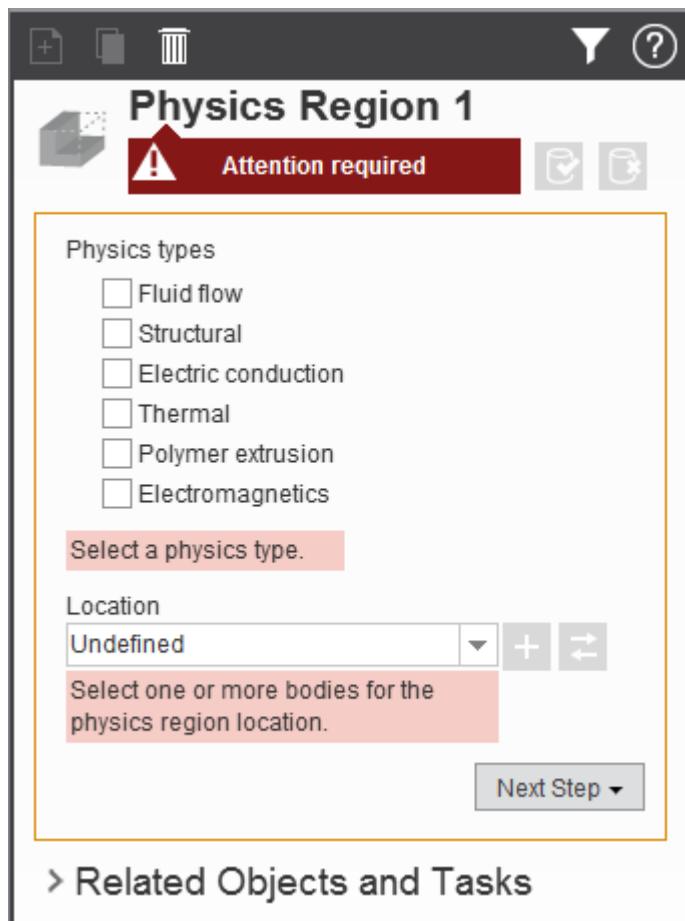
Physics regions are distinct locations in your simulation domain for which you define the types of physics that will be solved. If you begin your simulation by choosing a template, you define the default physics for the simulation; however, you can adjust those settings by using the **Physics > Physics Regions** panel.

In the Physics task's **Physics Region** panel, you can remove or add the following physics types in a physics region:

- [Fluid Flow Physics](#)
- [Structural Physics](#)
- [Thermal Physics](#)
- [Electromagnetics Physics](#)
- [Electric Conduction Physics](#)

- Electrostatic Physics
- Polymer Extrusion Physics
- Polymer Blow Molding Physics

Note that not all physics types can be combined in a given physics region.

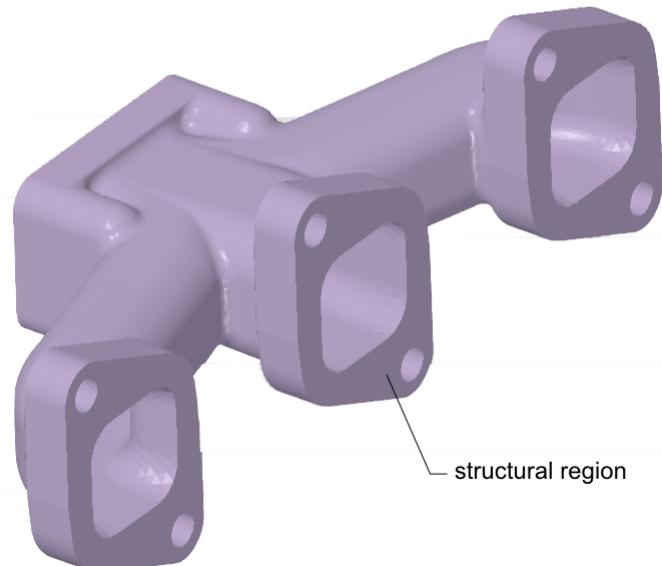


**Note:** The image above shows the maximum possible physics types; when you create a simulation by using a template, only mutually consistent options are presented.

**Figure 4.3.1. Example of a Fluid Flow Physics Region**



**Figure 4.3.2. Example of a Structural Physics Region**



When defining physics regions, note that:

- The physics characteristics you define, and sometimes the order in which you define them, control the solver used in the simulation. The solver used affects the time required to solve the simulation and the results returned.

The simulation type determines the number of physics regions you can define:

- Fluid flow and polymer extrusion simulations can have [multiple physics regions](#).
- Structural simulations can have only one physics region.

- The type of physics region determines the physics you can define. For example:
  - In a fluid flow simulation, you can assign only one material per physics region.
  - In a structural simulation, you can assign unique materials to individual bodies of a physics region.
- You can [change the physics region's type](#), depending on your simulation type and requirements.
- Elsewhere in the Physics task, you will define characteristics such as materials, boundaries, and solver operations for each physics region.
- By this point in the simulation, any parts of the geometry that need to be suppressed should have been suppressed.

### 4.3.1. Fluid Flow Physics Regions

A fluid flow physics region is composed of one or more volume bodies representing a space around and between solid objects through which fluid may flow; that space is filled with a fluid material. The bodies of a single fluid flow region must be of the same material, but a fluid flow simulation can have multiple regions.

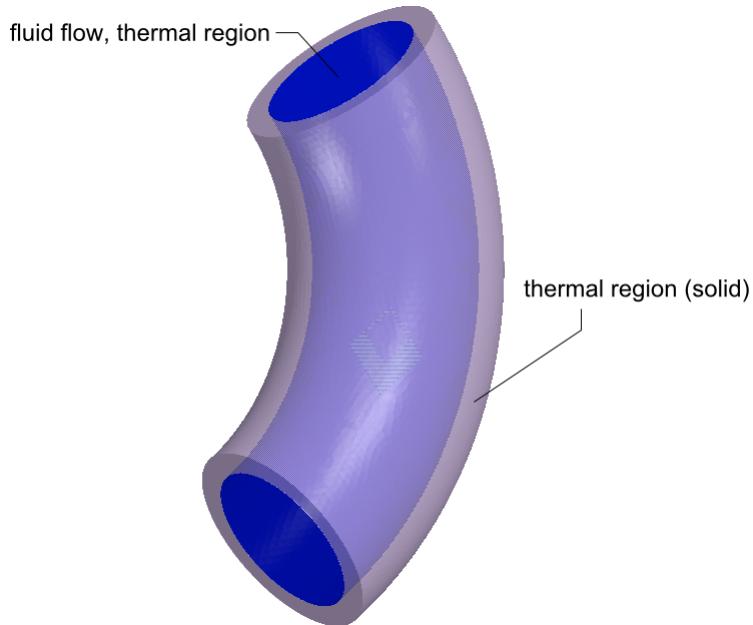
Depending on the problem you are trying to simulate, you may need to represent your model using distinct physics regions within the same Physics task. For fluid flow simulations, you can add a physics region to your Physics task, and associate the new physics region with other objects in your Study.

Fluid flow regions can be of two types:

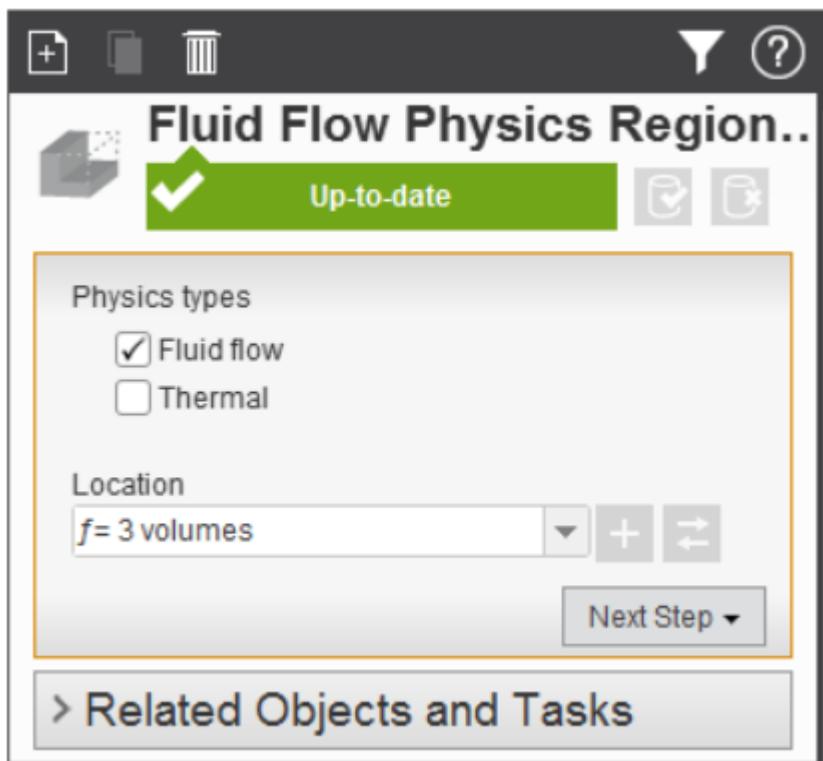
- For a simulation that represents internal flow, motion of the fluid material is constrained by boundary conditions to the region by walls representing the bounding solid bodies, and may enter or exit the region through inlets, outlets, and openings that represent conditions outside of the modeled region.
- For a simulation that represents external flow, the fluid material surrounds the body.

When combined with thermal physics, the solution includes both fluid flow and heat transfer and includes the thermal energy in the fluid flow simulation.

**Figure 4.3.1.1. Example of Multiple Physics Regions**



You can add or remove thermal physics to an existing fluid flow simulation in the **Physics > Physics Regions > Fluid Flow Physics Region** panel:

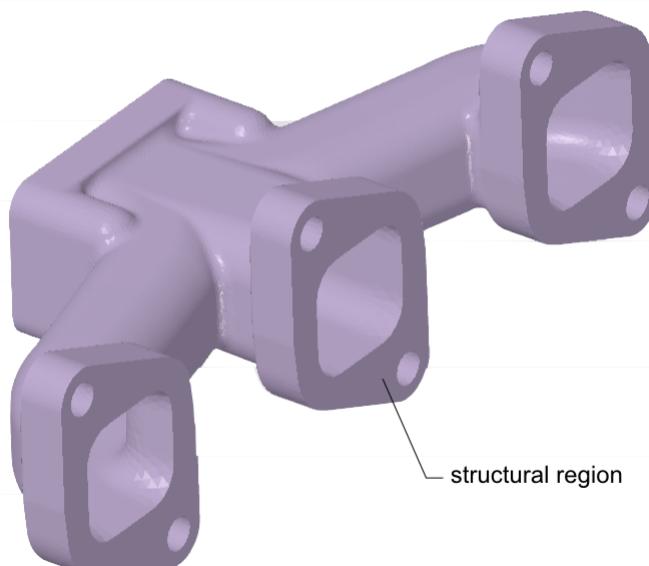


If you have a fluid flow simulation that has multiple parts that exhibit fundamentally different physics from the other regions in the model, you can define those parts to be different physics regions.

#### 4.3.2. Structural Physics Regions

Structural physics regions are made up of one or more bodies representing solid objects. Each body defines the undeformed, stress-free state of a fixed solid mass. Materials must be assigned to the bodies of a structural physics region.

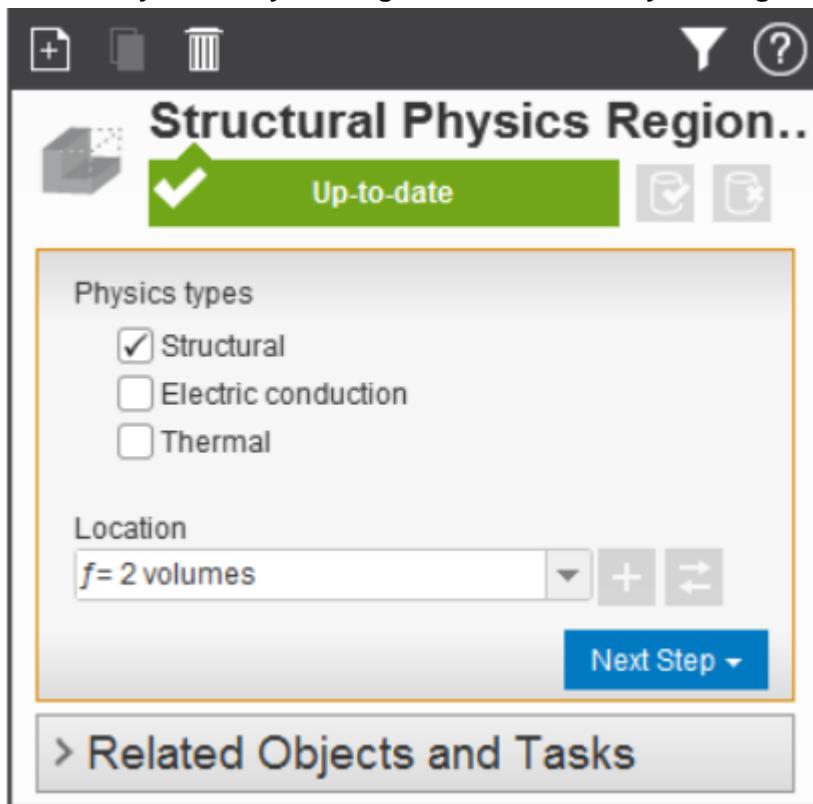
**Figure 4.3.2.1. Example of a Structural Physics Region**



The structural physics options model the deformation (strain) and forces (stress) of a body subjected to external displacements and forces.

- When combined with thermal physics, the solution includes both structural mechanics and heat transfer, and includes the thermal strains in the structural mechanics simulation.
- When combined with electric conduction physics, the solution includes both structural mechanics and electric conduction independently.
- When combined with thermal and electric conduction physics, the solution includes structural, thermal, and electric conduction degrees of freedom such as translations and temperatures. This coupled field solution models situations where electric effects create thermal effects, and, in turn, thermal effects cause structural strains and deformations.

You can add or remove thermal physics, electric conduction physics, or both in an existing structural simulation from the **Physics > Physics Regions > Structural Physics Region** panel:



#### 4.3.3. Electromagnetics Physics Region

Electromagnetic physics regions have the physics type set to **Electromagnetics**. These regions are made up of one or more bodies. These bodies may be all solids, or may include solids and an air field or vacuum. Unique materials can be assigned to the individual bodies.

The electromagnetic physics type cannot be combined with other physics types. A Physics task with an electromagnetic physics region cannot include other physics regions.

Electromagnetic physics [conditions](#) and [options](#) enable you to compute either static magnetic fields arising from DC currents flowing in conductive bodies (as well as from other sources such as permanent magnets and external magnetic fields); or—for frequency response designs—time-varying magnetic fields and induced Eddy currents arising from sinusoidal time-varying currents flowing in conductors.

Solutions also include current distribution, and magnetic flux density. Magnetic force, torque, inductance, impedance, and heat loss can also be computed.

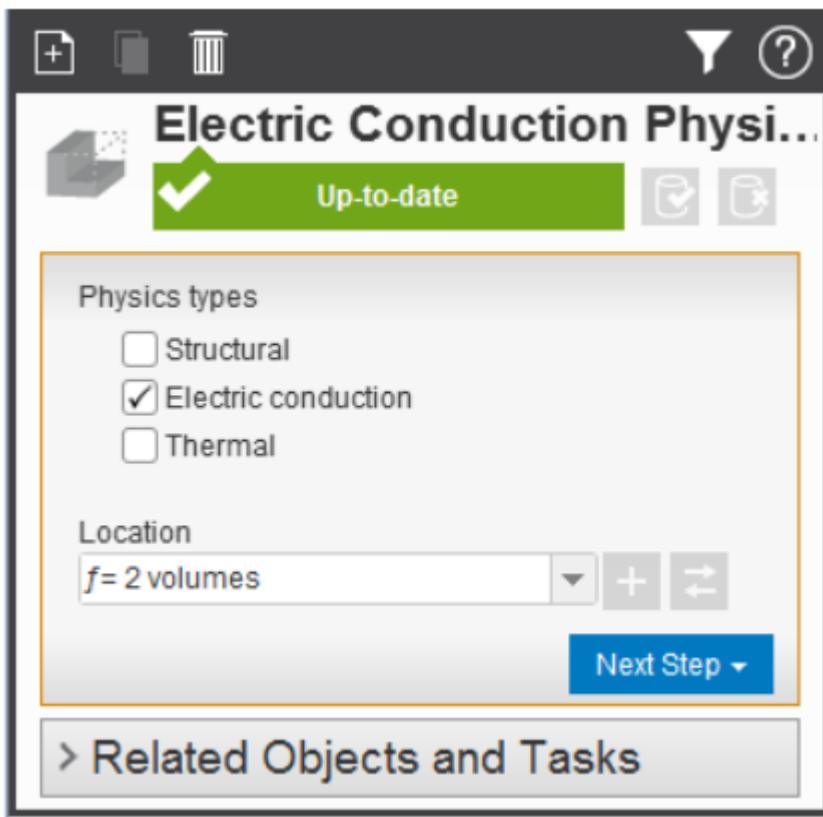
#### 4.3.4. Electric Conduction Physics Regions

Electric conduction physics regions are made up of one or more bodies representing solid objects. Materials must be assigned to bodies of an electric conduction physics region. The electric conduction physics options enable you to determine the electric potential in a conducting body created by the external application of voltage or current loads.

**Note:** The electric conduction physics region is also assigned when you specify you want to model a pure electric simulation in the **Electromagnetic** template, and then choose to determine the electric potential.

- When combined with thermal physics, the solution includes both electric conduction and thermal fields. This coupled-field capability models joule heating for resistive materials and contact electric conductance as well as Seebeck, Peltier, and Thomson effects for thermoelectricity.
- When combined with thermal physics and structural physics, the solution includes structural, thermal, and electric conduction degrees of freedom such as translations and temperatures. This coupled field solution models situations where electric effects create thermal effects, and, in turn, thermal effects cause structural strains and deformations.

If you have created a simulation using the Electromagnetic template, you can use the **Physics > Physics Regions > Electric Conduction Physics Region** panel to add structural and thermal results to the analysis.



#### 4.3.5. Electrostatic Physics Regions

Electrostatic physics regions are made up of one or more bodies representing solid objects. Materials must be assigned to bodies of an electrostatic physics region. The electrostatic physics options enable you to apply a charge or voltage to simulate an electrostatic charge.

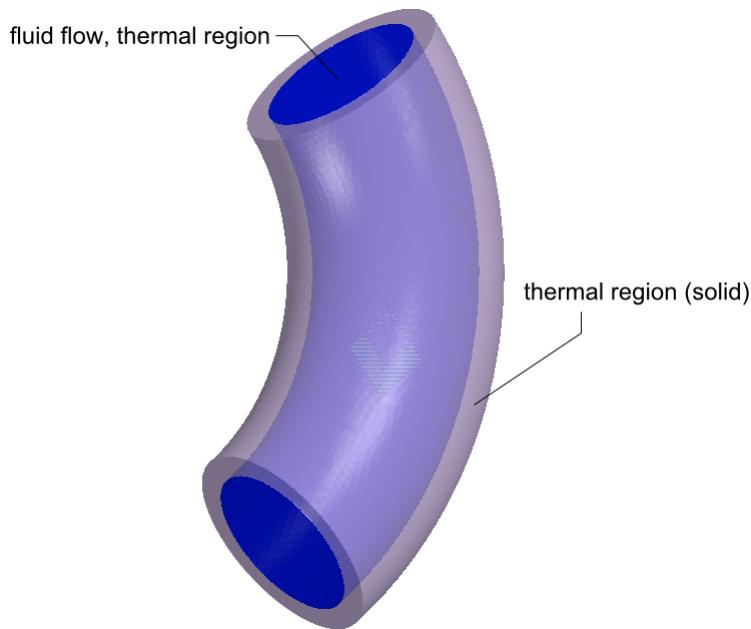
**Note:** The electrostatic physics region is also assigned when you specify you want to model a pure electric simulation in the **Electromagnetic** template, and then choose to apply a charge or voltage.

Electrostatic physics currently do not support multi-physics or physics coupling analyses.

#### 4.3.6. Thermal Physics Regions

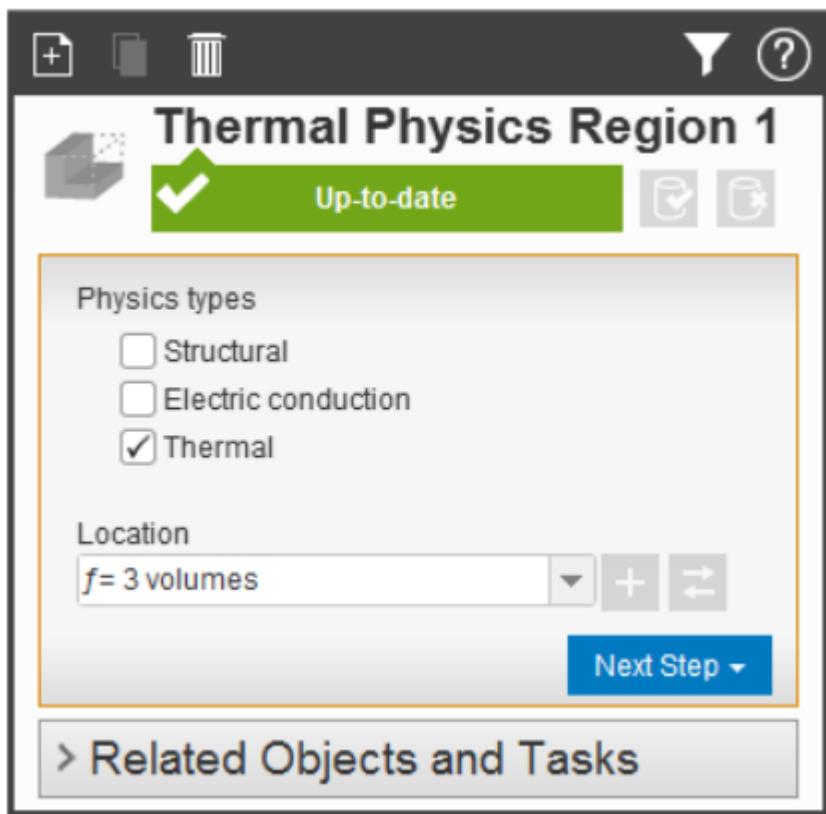
Thermal physics regions are made up of one or more bodies representing fluids or solid objects. The thermal physics options enable the transport of heat energy throughout a body.

**Figure 4.3.6.1. Example of Thermal Physics Regions**



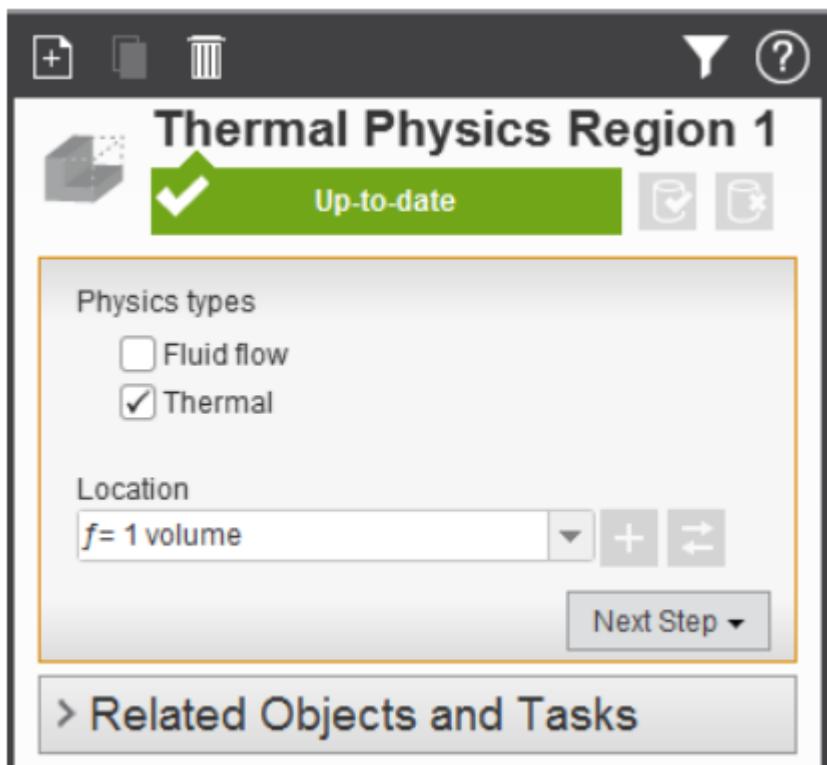
- When applied exclusively to a physics region, the thermal physics options model the temperature distribution within an undeformed solid body or stagnant fluid subject to internal heat sources and external heat sources and temperature conditions.
- When combined with fluid flow physics, the solution includes both heat transfer and fluid flow, and includes the thermal energy in the fluid flow simulation.
- When combined with structural physics, the solution includes both heat transfer and structural mechanics, and includes the thermal strains in the structural mechanics simulation.

You can add or remove thermal physics to an existing fluid flow simulation in the **Physics > Physics Regions > Fluid Flow Physics Region** panel:



When you create a simulation using the Thermal template, AIM uses the structural solver and the only other physics types you can add are **Structural** and **Electric conduction**.

**Note:** If you start with a Fluid Flow template and enable the thermal effects option, AIM uses the fluids solver; this solver is retained even if you deselect **Fluid flow** as a physics type on the **Physics Region** page:

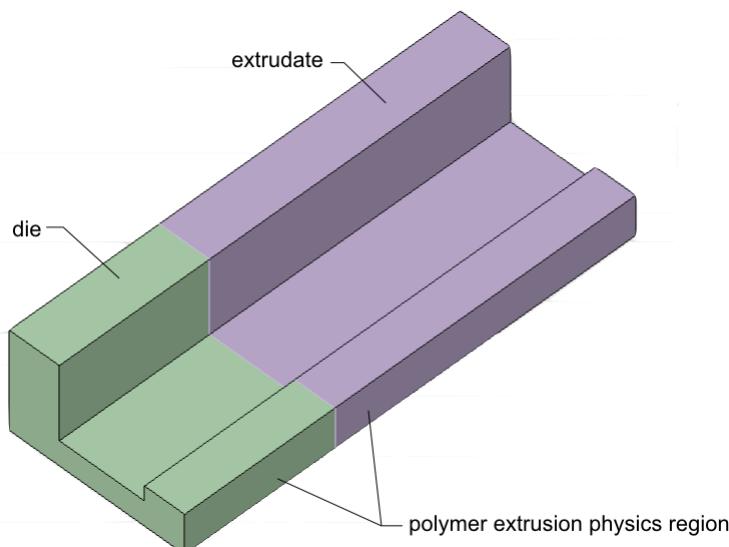


#### 4.3.7. Polymer Extrusion Physics Regions

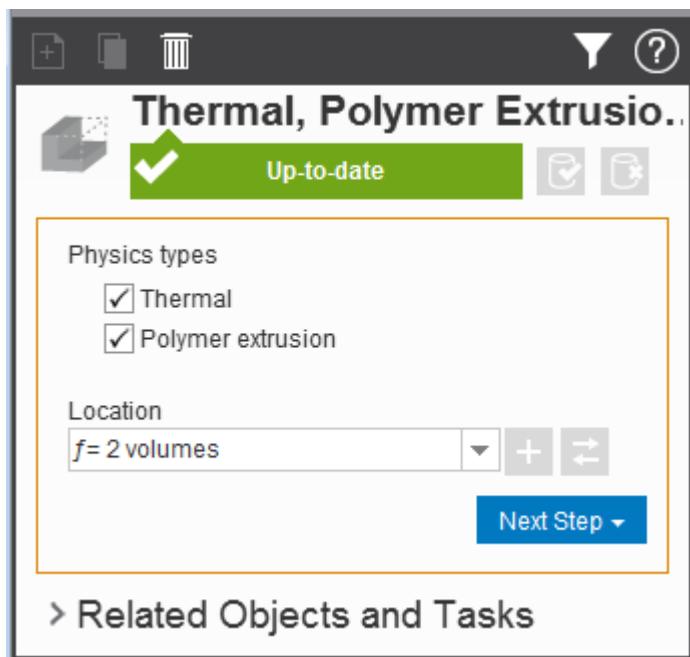
A polymer extrusion physics region is composed of volume bodies representing a space that is filled with a molten polymer material and solid die objects through which the polymer may flow. The motion of the polymer material is constrained to the region by walls representing the bounding solid bodies; the outside conditions are modeled by the inlets through which the material enters and the outlets through which the material exits the region. The extrudate may itself be a deforming body, and may contain an exit, and a free surface.

All bodies of a single polymer extrusion region must have the same material assignment.

**Figure 4.3.7.1. Example of a Polymer Extrusion Physics Region**



When combined with thermal physics, the solution includes both polymer extrusion and heat transfer, and includes the thermal energy in the polymer extrusion simulation.

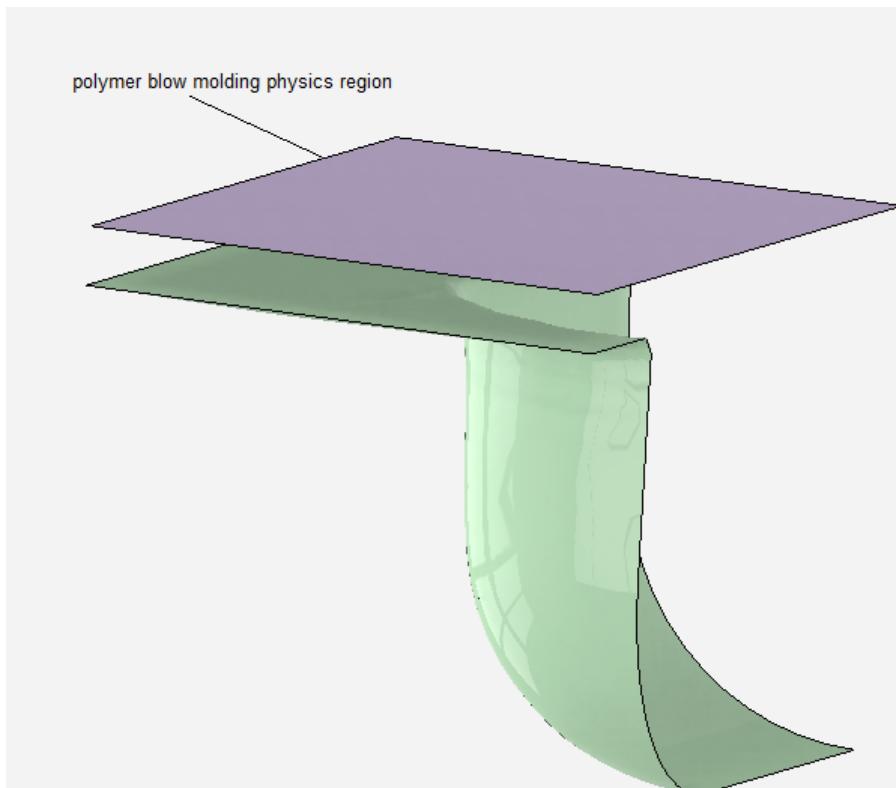


#### 4.3.8. Polymer Blow Molding Physics Regions

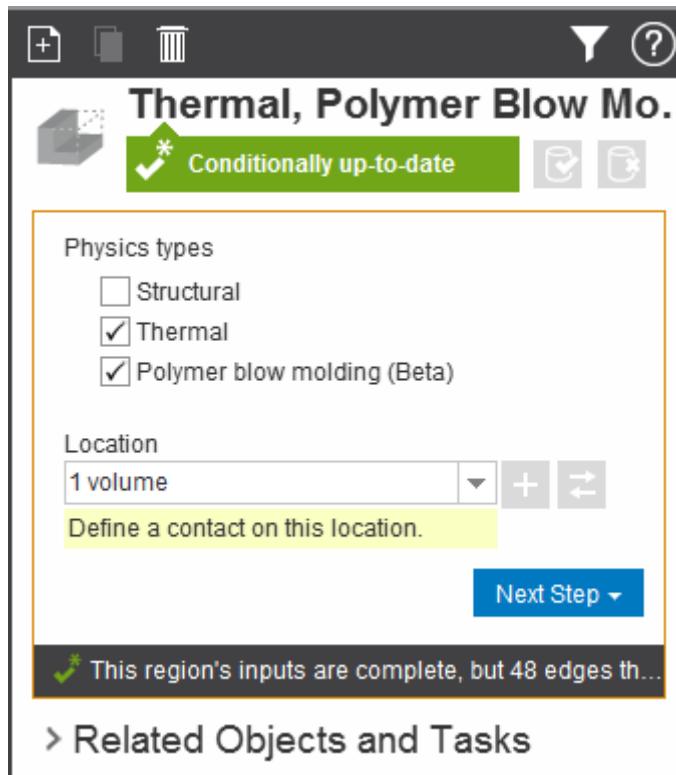
A polymer blow molding physics region is composed of one or several surface bodies, representing the polymer sheet, which can be made up of multiple layers (layer thicknesses are defined in the initial conditions). The mold is a non-deforming rigid body and is defined as a structural physics region. The mold's motion can be fixed or imposed by a velocity or force.

A layer can cover the whole polymer blow molding physics region or only some of its surface bodies, each layer has a material assignment. If a plunger is used in the process, it will have a separate material assignment.

**Figure 4.3.8.1. Example of Blow Molding Physics Regions for Blister Packaging**



When combined with thermal physics, the solution includes both polymer blow molding and heat transfer, and includes the thermal energy in the polymer blow molding simulation.



### 4.3.9. Changing a Region's Physics Type

The Physics task can be updated and a solution achieved only when the region's physics type and all of the corresponding physics objects are compatible. A region's physics type determines which physics objects are compatible with the Physics task. If you change the region's physics type, and if you have objects that no longer match your region's new physics type, then you will have to manually remove the incompatible physics objects from the Physics task.

To change a region's physics type:

1. In the Physics Region panel, make a new selection based on the available options.

For instance, when using a conjugate heat transfer template, you might want to select **Electric conduction** and deselect **Structural**.

2. Click **Apply**.

3. Return to the Physics panel and review the current settings.

For instance, using the example above, you will want to remove any Support, Displacement, Pressure, Force, Inertia Load, and Moment objects.

Use the following table as a reference for what objects apply to each region's physics type:

Table 4.3.9.1. Summary of Objects and Compatible Physics Region and Calculation Types

Object	Physics Region Type								Calculation Type				
	Fluid flow	Structural	Electric conduction	Elastic	Thermal	Polymer extrusion	Electromagnetics	Structural	Modal	Eigenvalue Buckling	Transient	Frequency Response	
Body Temperature		*	*	*				*		*			
Bolt Tension		*						*		*			
Displacement		*						*	*	*			
Force		*						*		*			
Inertia Load		*						*		*			
Moment		*						*		*			
Pressure		*						*		*			
Support		*						*	*	*			
Current			*					*		*			
Electric field			*	*				*		*			
Voltage			*	*				*		*			
Charge				*				*		*			
Conductivity					*			*		*	*		
Heat Flux					*			*		*	*		
Heat Flow					*			*		*	*		

Object	Physics Region Type						Calculation Type				
Heat Generation				*			*		*	*	
Radiation				*			*		*	*	
Temperature				*			*		*	*	
Far Field	*						*		*		
Heat Source	*			*			*		*		
Inlet	*				*		*		*		
Momentum Source	*						*		*		
Opening	*						*		*		
Outlet	*				*		*		*		
Symmetry	*				*		*		*		
Wall	*				*		*		*		
Region Interface	*						*		*		
Physics Coupling Interface		*		*		*	*		*	*	*
Contacts		*	*	*	*		*	*	*	*	
Joints		*					*	*	*		
Die Deformation					*		*		*		
Extrude Deformation					*		*		*		
Extrude Exit					*		*		*		
Free Surface					*		*		*		
Force Computation						*	*		*		
Torque Computation						*	*		*		
Current						*	*		*		*
Insulating						*	*		*		*
Terminal						*	*		*		*

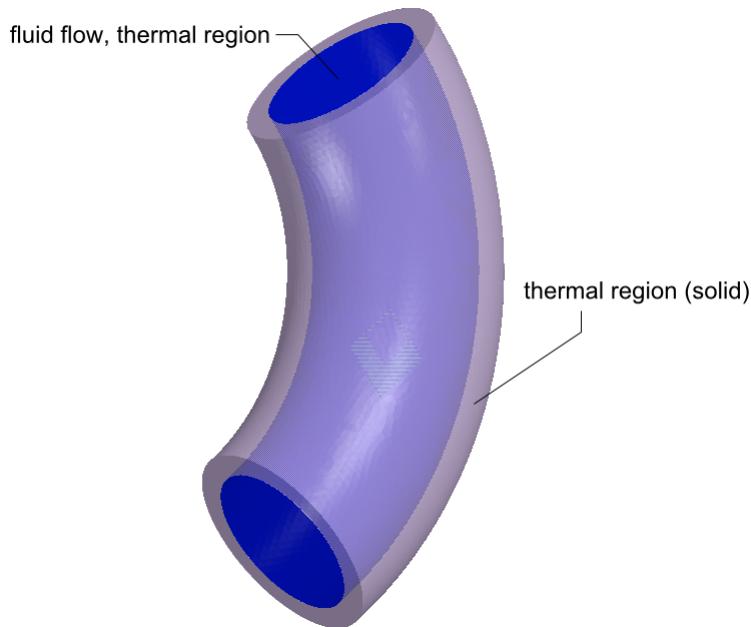
For conjugate heat transfer simulations, **Heat Source** and **Insulated** are also available as solid thermal condition objects.

#### 4.3.10. Defining a New Physics Region

##### Physics > Physics Regions > Add > Physics Region

A fluid flow or polymer extrusion simulation that has multiple parts that exhibit fundamentally different physics from the other regions in the model.

**Figure 4.3.10.2. Example of Multiple Physics Regions**



For fluid flow or polymer extrusion simulations, you can define parts of your model to have different physics:

1. In the Workflow view, click **Physics** to open the Physics task.
2. In the Physics data panel, select **Physics Region > Add > Physics Region**.
3. On the **Physics Region** panel, set the physics types; for the **Location**, specify the parts of the model that make up the new physics region.
4. Use the **Next Step** button to associate the new physics region with other objects in your Study, such as material assignments, physics conditions, and solution settings that are now specific to the new physics region.

When you create an object within a task, you are associating that object with that specific task. In addition, some objects are associated with other objects rather than tasks. This is particularly important for fluid flow simulations, where the content of an object is heavily dependent on the physical effects being modeled in that physics region.

Objects such as **Physics Options**, **Numerical Controls**, **Initial Conditions**, and **Physics Conditions** are all associated with a physics region (and listed under **Related Objects and Tasks**). So, a simulation with different physics regions can have different physics options, numerical controls, physics conditions, etc. There also may be occasions when these associated objects are shared between physics regions.

Simulations with multiple physics regions (such as conjugate heat transfer in a fluid flow or polymer extrusion simulation) require [physics region interface conditions](#) to address any discontinuities that may exist between the physics regions.

For polymer extrusion simulations, there are also some [additional considerations](#) regarding region connectivity and extrudate/die deformation.

## 4.3.10.1. Region Connectivity and Polymer Extrusion

For polymer extrusion simulations, some additional considerations should be noted:

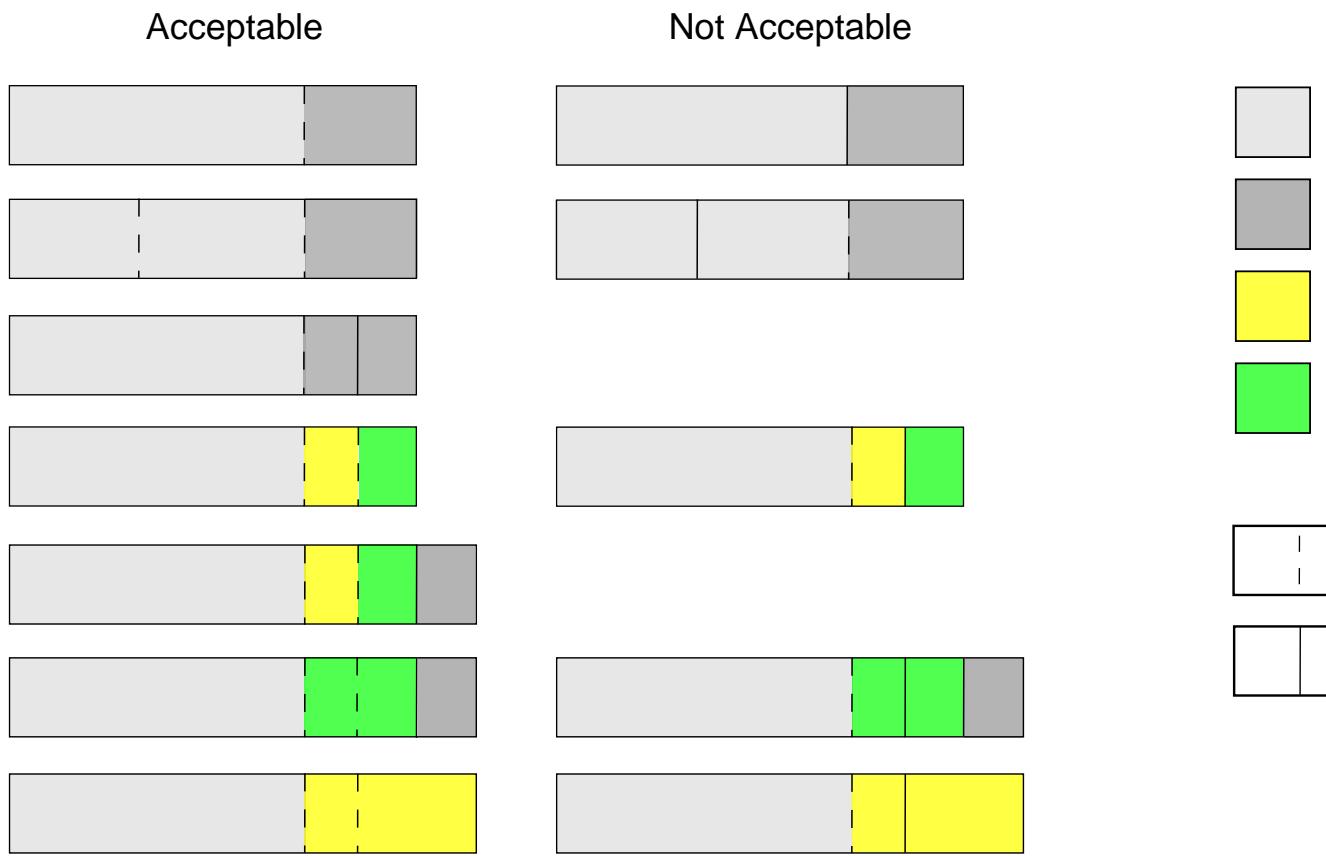
- For flow within a die:
  - Bodies may be connected or not, however, connected bodies always provide better results.
  - Region interfaces should only touch walls and/or symmetry conditions.
  - Faces that define the location of the inlets and outlets must be connected.
- For extrudate deformation:
  - The whole extrudate must be made up of connected bodies.
  - The extrudate cannot be adjacent to a region interface.
  - Having disconnected bodies inside the die is acceptable, however, all bodies in the die and adjacent to the extrudate must be connected.
  - While the extrudate and the polymer within the die adjacent to the extrudate must be a single, connected physics region, they cannot be a single body. Therefore, they must be made into separate, connected bodies.
  - Faces that define the location of inlets, the free surface, or the extrudate exit must be connected.
- For die deformation:
  - All the bodies involved in the die and extrudate deformation must be connected.
  - Faces that define the location of inlets, the free surface, or the extrudate exit must be connected.

**Note:** A multi-body part is connected when the mesh matches and is conformal where the two bodies meet. Otherwise the multi-body part is considered disconnected, or non-conformal.

If you need to connect bodies, then you will need a multi-body part with connected bodies, and you will need to use an external CAD system (for example, SpaceClaim Direct Modeler) to edit the geometry accordingly. Similarly, if you need to connect faces, then you will need to edit the geometry accordingly.

**Note:** If you are not using the polymer extrusion template, you must import the multi-body part geometry using a Data Import task (since multi-body parts are not supported by Geometry Modeling tasks).

The following figure illustrates the acceptable use of connected bodies for polymer extrusions:



## 4.4. Material Assignments

The default materials are structural steel for solids, air for fluids, and a generic polymer for extrusions. If you need more materials or different materials for your simulation, you may need to:

- **Assign materials** to your model from the supplied materials library or from an [imported materials library](#).
- **Define new properties** for existing materials
- **Create materials** and [define their material properties](#)
- **Export materials** for use in other simulations

### [Assigning Materials and Defining Material Properties](#)

### 4.4.1. Assigning Materials

#### Physics > Material Assignment

To assign a material to one or more bodies:

1. **Select a region in your simulation** to which you want to assign a material.
2. Select **Add > Physics Modeling > Material Assignment**.
3. For the **Material**, select one of the following options:

- A favorite material (  )
- A recently used material (  )
- A material that already exists in the Study (  )

If your desired material does not appear in the list, you can:

- Search the available libraries
- [Add a library](#) (  )
- [Create a material](#)

4. If necessary, you can modify or add [material properties](#).
5. Complete your material assignment. The next step depends on the type of physics:

	<b>Set</b>	<b>Description</b>
If this material is a solid in a structural physics simulation	<b>Zero Thermal Strain Temperature</b>	The <b>Zero Thermal Strain Temperature</b> is used to calculate the amount of thermal expansion or contraction in a material.
If this material is a solid in a structural physics simulation	<b>Plasticity model</b>	Select a plasticity model, if applicable.
If this material is a solid in a structural physics simulation	<b>Elasticity model</b>	Select an <a href="#">elasticity model</a> , if applicable. For Orthotropic Elasticity, you must additionally <a href="#">define the material orientation</a> .
If modeling spherical particles (such as droplets or bubbles) dispersed in the continuous fluid phase	<b>Material distribution</b>	To model <a href="#">particles in fluid flow</a> , such as water droplets in the atmosphere, select <b>Continuous</b> for air, therefore representing the continuous phase and select <b>Particle</b> for your water material, which represents your second discrete phase.
If this material is a fluid in a fluid flow physics simulation	<b>Material model</b>	For properties that do not vary as a function of temperature or pressure, select <b>Constant properties</b> . For <a href="#">compressible flows</a> where the density will vary as a function of temperature, select <b>Ideal gas</b> .

	Set	Description
If this material is part of a polymer extrusion physics simulation	<b>Material model</b>	<p>For polymer flows that exhibit constant viscosity, or to obtain a quick initial evaluation of the flow within a die, select <b>Newtonian fluid</b>. A Newtonian fluid has a constant viscosity that leads to a large deformation of the extrudate because there is a significant velocity rearrangement after the die lip.</p> <p>For polymer flows exhibiting shear-thinning, select <b>Generalized Newtonian fluid</b> for a more realistic simulation. When using this option, several <b>viscosity laws</b> are available to apply to your material. Generalized Newtonian fluids have a viscosity that most often decreases with the shear-rate (shear-thinning). The shear-thinning leads to an important velocity gradient close to the walls and a low gradient in the center of the channel. This leads to a smaller velocity rearrangement after the die lip and therefore a smaller deformation of the extrudate.</p> <p>For polymer flows where the extrudate shape is important, select <b>Simplified viscoelastic fluid</b> for a better prediction of extrudate swelling.</p>

#### 4.4.1.1. Zero-Thermal-Strain Reference Temperature

The Zero-Thermal-Strain Reference Temperature is used to calculate the amount of thermal expansion or contraction in a material. It is the temperature at which the structural member has zero strain. As the member heats up or cools down over the course of the simulation, it will expand or contract and if fixed will cause stress in the member.

An application to demonstrate this feature's use is the shrink-fitting of a bushing into its housing. In this process a bushing may be cooled to -50C before being inserted into its housing, then allowed to return to ambient temperature with the corresponding thermal expansion achieving an interference fit. To model this case, you would need to have two Material Assignments: one for the bushing with a **Zero-Thermal-Strain Reference Temperature** of -50C and one for the housing with a **Zero-Thermal-Strain Reference Temperature** at ambient conditions.

#### 4.4.1.2. Ideal Gas Law

**Compressible flows** are described by the standard continuity and momentum equations and the treatment of density as detailed below.

The compressible form of the gas law (the ideal gas law) is written in the following form:

$$\rho = \frac{(P_{abs})}{(R / M_w T)}$$

where  $P_{abs}$  is the absolute pressure,  $R$  is the universal gas constant,  $M_w$  is the molecular weight, and  $T$  is the local temperature.

## 4.4.1.3. Defining the Material Orientation for Orthotropic Elasticity

You can define orientation on the **Material Assignment** panel under **Orientation** using a single reference frame (default) or a surface and edge guide. The surface and edge guides define an orthogonal reference frame at the centroid of every element. The closest normal at the surface becomes the Z-Axis of the reference frame, while the X-axis lies in the plane created by the normal and closest tangent at the edge guide. Therefore the material orientation changes at each element in the model based on the guide (therefore it is effectively representing multiple reference frames).

To define the material orientation as a reference frame:

1. Select **Define by > Reference frame**.
2. From **Relative to**, select or define a reference frame.

To define the material orientation using geometry:

1. Select **Define by > Geometry**.
2. Select a face for the **Surface guide**.
3. Select an edge for the **Edge guide**.
4. Optionally, enter a value for **Maximum number of display points**, which controls the number of points on which you want to display the material orientation in the graphics window.

When defining the material orientation with a surface and edge guide, every element could potentially have a different orientation depending on the selected surface guide and edge guide. However, displaying the orientation on all elements may affect performance. By default, AIM displays the orientations for 200 points per material assignment, but you can enter your own value to view more or fewer orientations.

Note that this only impacts the display and has no effect on the solve or results.

## 4.4.2. Creating Materials

### Physics > Material Assignment

To create a new material:

1. Right click in the graphics window to select **Add > Physics Modeling > Material Assignment**.
2. Enter the name for the new material in the **Material** field, and then select **Create material\_name**.
3. In the newly created material panel:
  - a) Enter a **Description** for your material.
  - b) Set the **Default state** of your material to **Solid**, **Liquid**, or **Gas**.
  - c) For **Solid properties**, **Liquid properties**, and/or **Gas properties**, select **Add** and choose a new property from favorites, recent, or search items. You can also use the search field for quicker access to a particular property.

Alternatively, you can use an existing material as a template, then change the material name and selected properties as required.

## 4.4.3. Exporting Materials

### Physics > Material Assignment

You can export a material for use in another project. Exporting materials is only supported for HDF5 (.h5) files. Follow the procedure below:

1. In the **Material Assignment** panel, click the link to the material.



2. Click the library icon ( ) to show the source information of the material which includes the library name, material name, and library location.
3. Click **Export this material to a library**.
4. Aside from exporting the material to an already loaded library, you have two options:

If you want to ...	then choose ...
Export the material to an existing library not loaded inside AIM.	<b>Existing Library</b>
Export the material to a new library.	<b>New Library</b>

The default location is your Engineering Data Library, which stores the material in your local application data folder.

5. Click **OK**.

#### 4.4.4. Copy Material Assignments from Other Simulations

When you create a new Physics Solution task from a Geometry task that already has a single Physics Solution task connected to it, any material assignments from the previous Physics Solution task are copied to the new one automatically.

You can copy one or more material assignments from any Physics Solution task that shares the same Geometry connection.

Right-click and select **Add > Physics Modeling > Copy from**, and then choose the Physics Solution task and material assignment you'd like to copy from.

#### 4.4.5. Adding a Library

##### Physics > Material Assignment

To add a material library to your Study:

1. Click **Add Library** ( ).
2. Choose a valid material library. The following file formats are supported:
  - MatML 3.1 schema (.xml)
  - HDF5 (.h5)

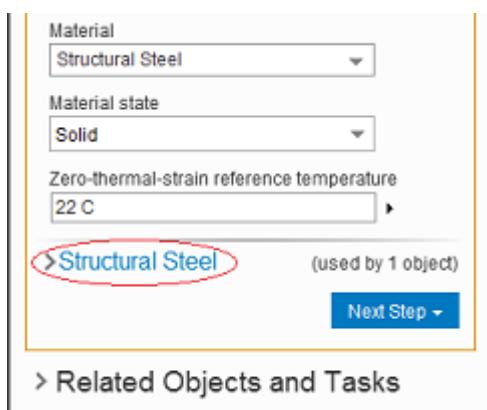
You can now use the search function ( ) to find the newly imported materials and [assign them to your physics region](#) as needed.

#### 4.4.6. Defining Material Properties

AIM provides a collection of material properties that can be used in fluid flow, thermal, structural, polymer extrusion, electric conduction, or electromagnetics simulations. Material properties appropriate for your project are dependent on the type of simulation performed. The material properties will be displayed on a material or can be added to the material (see [Creating Materials](#) on page 219 for details). The properties for a material can be defined in a solid, liquid, or gas state. You can set properties for a material in multiple states if, for example, you would like to use the liquid state in one material assignment and the gas state in another.

To define the material properties:

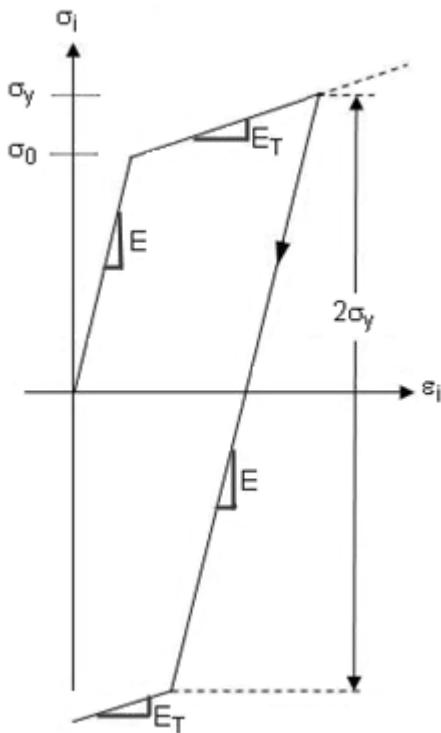
1. After assigning a material to a region, select the material name to view the material.



2. Modify the material properties as desired. Some material properties can be defined using [tabular data](#).
3. For **Solid properties**, **Liquid properties**, and/or **Gas properties**, select **Add** and choose a new property from favorites, recent, or search items. You can also use the search field for quicker access to a particular property.

#### 4.4.6.1. Bilinear Isotropic Hardening Model

The bilinear isotropic hardening plasticity material model, often used in large strain analyses, is described by a bilinear effective stress versus effective strain curve. The initial slope of the curve is equivalent to the Young's modulus of the material. Beyond the specified **Yield strength value** ( $\sigma_y$ ), plastic strain develops and stress-vs.-total-strain continues along a line with slope defined by the **Tangent Modulus** ( $E_t$ ). The tangent modulus cannot be less than zero or greater than the elastic modulus.



You can define yield strength and tangent modulus as constants or expressions.

## 4.4.6.2. Density

The density of a material is the mass per unit volume. You can define Density ( $\rho$ ) as a constant or as an [expression](#). The units are entered as [mass length $^{-3}$ ], for example "8750 [kg m $^{-3}$ ]".

## 4.4.6.3. Elasticity

AIM support three elasticity models:

- Isotropic Elasticity
- Orthotropic Elasticity
- Neo-Hookean Hyperelasticity

## Isotropic Elasticity

Isotropic elasticity is used to model a material behavior where the stress versus strain relationship is linear and the loading is kept within the elastic range. The molecular structure of the material is homogeneous so that a load in any direction will exhibit the same stress versus strain relationship. The model follows Hooke's law and so assumes that the stretch or compression remains in the elastic range of the material and the model will return to its original shape. Hooke's law requires two terms to be input for computation, these inputs can be chosen with the Derive from field. Each of the inputs can be defined as a constant or an [expression](#).

- Young's Modulus (E), base units are entered as [force length $^{-2}$ ], for example "7.1E10 [N m $^{-2}$ ]"
- Poisson's Ratio (v), base unit is unitless, for example "0.33"
- Bulk Modulus (K), base units are entered as [force length $^{-2}$ ], for example "6.96E10 [N m $^{-2}$ ]"
- Shear Modulus (G), base units are entered as [force length $^{-2}$ ], for example "2.67E10 [N m $^{-2}$ ]"

## Orthotropic Elasticity

Orthotropic elasticity is used to model a material behavior where the stress-strain relationship differs along three mutually orthogonal planes of elastic symmetry. Orthotropic elasticity models are a subset of anisotropic elasticity models, because their properties change when measured from different directions. One example of a material that would utilize orthotropic elasticity is wood.

The elastic properties (Young's Modulus [E], Poisson's Ratio [v], and Shear Modulus [G]) are therefore specified in three orthogonal directions.

- Young's Modulus (E), base units are entered as [force length $^{-2}$ ], for example "7.1E10 [N m $^{-2}$ ]"
- Poisson's Ratio (v), base unit is unitless, for example "0.33"
- Shear Modulus (G), base units are entered as [force length $^{-2}$ ], for example "2.67E10 [N m $^{-2}$ ]"

You can define the orientation of the orthotropic elastic material by selecting a single reference frame or defining a surface and edge guide. When you define the orientation using a reference frame, all elements are assigned the same reference frame. With a surface and edge guide, the material orientation changes at each element in the model based on the guide (therefore it is effectively representing multiple reference frames). For more information, see [Defining the Material Orientation for Orthotropic Elasticity](#) on page 219.

## Neo-Hookean Hyperelasticity

The Neo-Hookean hyperelasticity material model represents the simplest form of strain energy potential and has an applicable strain range of 20-30%. It can be used for predicting the nonlinear stress-strain behavior of materials undergoing large deformations.

The Neo-Hookean hyperelasticity material model requires two terms to be input for computation. Each of the inputs can be defined as a constant or an [expression](#). They include:

- Initial Shear Modulus, base units are entered as [force length<sup>-2</sup>]
- Bulk Modulus, base units are entered as [force length<sup>-2</sup>]

Note that the initial bulk modulus is 2/incompressibility factor.

## 4.4.6.4. Isotropic Resistivity

Isotropic resistivity is used to model how strongly a material opposes the flow of electric current in all directions. A low resistivity indicates a material that readily allows the movement of electrical charge. You can define isotropic resistivity as a constant, as an [expression](#), or in the form of tabular data. For details on how tabular data can be defined, see [Defining Tabular Data](#) on page 84 for information. The units are entered as [electrical resistance length], for example "1.65x10<sup>-8</sup>[ohm meter]."

## 4.4.6.5. Isotropic Seebeck Coefficient

The isotropic Seebeck coefficient of a material is a measure of the magnitude of an induced voltage in response to a change in temperature across that material. This value is independent of the direction of the applied temperature gradient.

The isotropic Seebeck coefficient can be defined as a constant or an [expression](#), or in the form of tabular data. For details on how tabular data can be defined, see [Defining Tabular Data](#) on page 84 for information. The base units are entered as [voltage temperature<sup>-1</sup>], for example "6.5 [V C<sup>-1</sup>]."

## 4.4.6.6. Molar Mass

Molar mass is the average mass of a substance per mole of molecules. It can be defined as a constant or as an [expression](#).

The base units are entered as [mass mole<sup>-1</sup>], for example "28.965 [kg kmol<sup>-1</sup>]."

## 4.4.6.7. Specific Heat Capacity

The specific heat capacity of a material is the heat required to raise the unit mass of a substance by one degree of temperature.

The specific heat at constant pressure ( $c_p$ ) can be defined as a constant or as an [expression](#). The base units are entered as [energy mass<sup>-1</sup> temperature<sup>-1</sup>], for example "875 [J kg<sup>-1</sup> C<sup>-1</sup>]."

## 4.4.6.8. Thermal Conductivity

The thermal conductivity of a material is the ability of the material to conduct heat. Thermal conductivity is assumed to be isotropic in behavior which conducts heat equally in all directions.

The thermal conductivity (k) can be defined as a constant, as an [expression](#), or in the form of tabular data. For details on how tabular data can be defined, see [Defining Tabular Data](#) on page 84 for information. The base units are entered as [power length<sup>-1</sup> temperature<sup>-1</sup>], for example "148.6[W m<sup>-1</sup> C<sup>-1</sup>]."

## 4.4.6.9. Thermal Expansion

The thermal expansion of a material is the volumetric response of the material to a change in temperature. Thermal expansion is assumed to be isotropic in behavior and so expands or contracts equally in all directions.

The thermal expansion ( $\alpha$ ) can be defined as a constant or an [expression](#). The base units are entered as [temperature^-1], for example "2.3e-05 [C^-1]".

## 4.4.6.10. Viscosity

The viscosity is the dynamic viscosity of a material and is the resistance to flow. The dynamic viscosity must be defined when modeling fluid flow.

The viscosity ( $\mu$ ) can be defined as a constant or an [expression](#). The base units are entered as [mass length^-1 time^-1], for example "1.1376E-03 [kg m^-1 s^-1]".

### 4.4.6.10.1. Non-Newtonian Fluids

Some fluids are non-Newtonian; they do not obey the simple linear relationship between shear stress and shear strain rate. Many practical fluids fall into this class, and their behavior is generally well understood and described using various mathematical models.

You can specify the viscosity of non-Newtonian fluids as a function of strain rate using an [expression](#). A common non-Newtonian model is the Power Law:

$$\mu = K(\lambda \dot{\gamma})^{n-1}$$

where K is the Consistency Index,  $\lambda$  is a time constant,  $H(T)$  is the Strain Rate (computed by the flow solver), and n is the Power Law Index. The following expression uses the Power Law model for a shear-thinning ( $n < 1$ ) fluid having  $K=10$ . [kg/m s],  $\lambda=1$  [s] and  $n=0.4$ :

```
10. [kg m^-1 s^-1]*pow(StrainRate * 1[s] + 1e-10, -0.6)
```

More [complex non-Newtonian models](#), such as Carreau-Yasuda and Bingham, can be implemented in a similar manner. Temperature can also be included in the expression if viscosity has a significant dependence on the fluid temperature.

## 4.4.6.11. Viscosity Models for Polymer Extrusion Simulations

Polymer extrusion simulations in AIM can utilize the following types of viscosity models:

- [Generalized Newtonian Viscosity Models](#) on page 224
- [Temperature-Dependent Viscosity Models](#) on page 234
- [Simplified Viscoelastic Fluid Model](#) on page 238

### 4.4.6.11.1. Generalized Newtonian Viscosity Models

For polymer extrusion simulations, AIM can be used to model generalized Newtonian flows. The generalized Newtonian category of flow includes both Newtonian flows and inelastic non-Newtonian flows. AIM has a number of modeling options available for simulations of these types of flows.

Several shear rate dependent viscosity laws are available for generalized Newtonian flows.

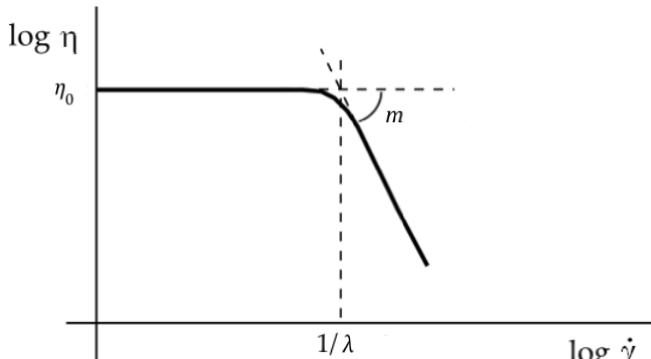
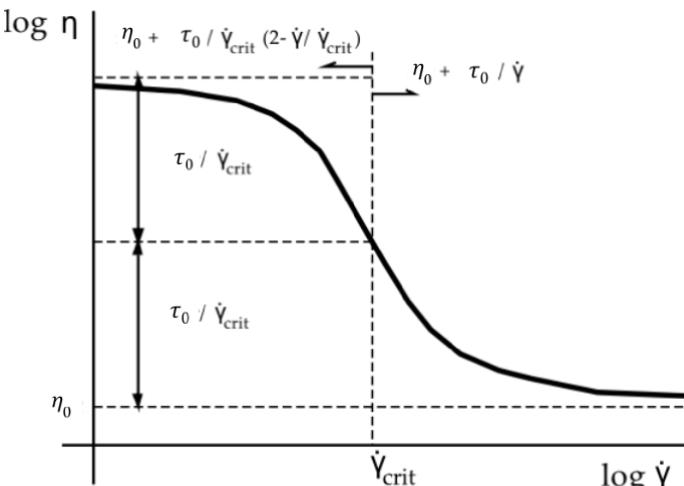
Once you have chosen a generalized Newtonian viscosity model, you will need to [set up the viscosity model](#) and assign various properties.

Table 4.4.6.11.1.1. Selecting a Generalized Newtonian Viscosity Model

If you want to...	... then select	More information
Choose a viscosity model that allows you to control the transition from the Newtonian plateau to the power-law region	Carreau-Yasuda ...	<p><b>Carreau-Yasuda</b></p> <p>The Carreau-Yasuda law for viscosity is</p> $\eta = \eta_{\infty} + (\eta_0 - \eta_{\infty}) [ 1 + (\lambda \dot{\gamma})^a ]^{\frac{n-1}{a}}$ <p>where</p> <ul style="list-style-type: none"> <li><math>\eta_0</math> = <b>Zero shear viscosity.</b></li> <li><math>\eta_{\infty}</math> = <b>Infinite shear viscosity.</b></li> <li><math>\lambda</math> = <b>Time constant.</b></li> <li><math>a</math> = <b>Plateau index.</b></li> <li><math>n</math> = <b>Power law index.</b></li> </ul>

If you want to...	... then select	More information
Describe the low-shear-rate behavior of the viscosity, especially at the plateau zone of the viscosity curve and allows you to define a plateau zone at high shear-rates.	Bird-Carreau ...	<p><b>Bird-Carreau</b></p> <p>The Bird-Carreau law for viscosity is</p> $\eta = \eta_\infty + (\eta_0 - \eta_\infty) (1 + \lambda^2 \dot{\gamma}^2)^{\frac{n-1}{2}}$ <p>where</p> <ul style="list-style-type: none"> <li><math>\eta_\infty</math> = <b>Infinite shear viscosity</b>.</li> <li><math>\eta_0</math> = <b>Zero shear viscosity</b>.</li> <li><math>\lambda</math> = <b>Time constant</b>.</li> <li><math>n</math> = <b>Power law index</b>.</li> </ul>

If you want to...	... then select	More information
Consider a model that is very similar to the Bingham law but uses an analytic expression that may be easier to calculate, leading to a more stable solution.	<b>Modified Bingham</b> ...	<p>The modified Bingham law for viscosity is:</p> $\eta = \eta_0 + \tau_0 \left( \frac{1 - \exp(-m\dot{\gamma})}{\dot{\gamma}} \right)$ <p>where</p> <ul style="list-style-type: none"> <li><math>m</math> = Plastic viscosity.</li> <li><math>\tau_0</math> = Yield stress threshold.</li> <li><math>\dot{\gamma}_c</math> = Critical shear rate.</li> </ul>
Describe the low-shear-rate behavior of the viscosity, especially at the plateau zone of the viscosity curve.	<b>Cross law</b>	<p>The Cross law for viscosity is</p> $\eta = \frac{\eta_0}{1 + (\lambda\dot{\gamma})^m}$ <p>where</p> <ul style="list-style-type: none"> <li><math>\eta_0</math> = Zero shear viscosity.</li> <li><math>\lambda</math> = Time constant.</li> <li><math>m</math> = Cross law index.</li> </ul>

If you want to...	... then select	More information
Consider a special case of the Carreau-Yasuda viscosity law, where the exponent $\alpha$ has a value of 1.	<b>Modified Cross law</b> ...	 <p>A modified Cross law for viscosity is also available:</p> $\eta = \frac{\eta_0}{(1 + \lambda \dot{\gamma})^m}$
Describe materials such as concrete, mud, dough, and toothpaste, for which a constant viscosity after a critical shear stress is a reasonable assumption, typically at rather low shear rates.	<b>Bingham</b>	 <p>The Bingham law for viscosity is</p> $\eta = \begin{cases} \eta_0 + \frac{\tau_0}{\dot{\gamma}}, & \dot{\gamma} \geq \dot{\gamma}_c \\ \eta_0 + \tau_0 \frac{(2 - \dot{\gamma})}{\dot{\gamma}_c}, & \dot{\gamma} < \dot{\gamma}_c \end{cases}$ <p>where</p> <ul style="list-style-type: none"> <li><math>\tau_0</math> = Yield stress threshold.</li> <li><math>\dot{\gamma}_c</math> = Critical shear rate.</li> </ul> <p>By default, <math>\tau_0</math>, <math>\dot{\gamma}_c</math>, and <math>\eta_0</math> are 1.</p>

If you want to...	... then select	More information
Describe materials such as concrete, mud, dough, and toothpaste, for which a power-law viscosity after a critical shear stress is a reasonable assumption.	Herschel-Bulkey ...	<p><b>Herschel-Bulkey</b></p> <p>The Herschel-Bulkey law for viscosity is</p> $\eta = \begin{cases} \frac{\tau_0}{\dot{\gamma}} + K \left( \frac{\dot{\gamma}}{\dot{\gamma}_c} \right)^{n-1}, & \dot{\gamma} > \dot{\gamma}_c \\ \frac{\tau_0(2-\dot{\gamma})}{\dot{\gamma}_c} + K \left[ (2-n) + (n-1) \frac{\dot{\gamma}}{\dot{\gamma}_c} \right], & \dot{\gamma} \leq \dot{\gamma}_c \end{cases}$ <p>where</p> <ul style="list-style-type: none"> <li><math>\tau_0</math> = <b>Yield stress threshold</b>.</li> <li><math>\dot{\gamma}_c</math> = <b>Critical shear rate</b>.</li> <li><math>K</math> = <b>Consistency factor</b>.</li> <li><math>n</math> = <b>Power law index</b>.</li> </ul> <p>The Herschel-Bulkey law exhibits a shear-thinning behavior, in addition to the transition behavior between a flow and no-flow regime.</p>

If you want to...	... then select	More information
Consider a model that is very similar to the Herschel-Bulkley law but uses an analytic expression that may be easier to calculate, leading to a more stable solution.	<b>Modified Herschel-Bulkley</b> ...	<p><b>Modified Herschel-Bulkley</b></p> <p>A modified Herschel-Bulkley law is also available:</p> $\eta = \tau_0 \left( \frac{1 - \exp\left(\frac{-3\dot{\gamma}}{\dot{\gamma}_c}\right)}{\dot{\gamma}} \right) + K \left( \frac{\dot{\gamma}}{\dot{\gamma}_c} \right)^{n-1}$ <p>The integer value 3 that appears in the argument of the exponential term has been selected so that the standard and modified Herschel-Bulkley laws exhibit the same behavior above the critical shear rate, <math>\dot{\gamma}_c</math>.</p>

If you want to...	... then select	More information
Describe the viscous behavior of polymeric materials, such as polyethylene, with shear rates ranging over 2 to 3 decades, not allowing for the behavior at low shear rates.	<b>Power law</b> ...	<p>The power law for viscosity is</p> $\eta = K(\lambda \dot{\gamma})^{n-1}$ <p>where</p> <ul style="list-style-type: none"> <li><b>K</b> = Consistency factor.</li> <li><b>λ</b> = Time constant.</li> <li><b>n</b> = Power law index.</li> </ul>

#### 4.4.6.11.1.1. Setting Up a Generalized Newtonian Viscosity Model

Simulations involving viscous flows, such as polymer extrusion simulations, often require the use of a generalized Newtonian viscosity model. Assign viscosity models as a property of the particular material you want to simulate.

To define the parameters of the Generalized Newtonian viscosity law:

1. In the material panel (for example, the **Generic Polymer** panel, for **Liquid Properties**), click **Add**.
2. In the search field, type **Generalized** to display a list of generalized Newtonian viscosity models.
3. **Select a generalized Newtonian viscosity model**.
4. For the selected model, specify values for the model properties.

Table 4.4.6.11.1.2. Generalized Newtonian Viscosity Models and Their Properties

Model	Property	Details
Carreau-Yasuda	<b>Zero shear viscosity</b>	The value of the viscosity as the shear rate approaches zero.
	<b>Infinite shear viscosity</b>	The value of the viscosity as the shear rate approaches infinity.
	<b>Time constant</b>	Natural time, or the inverse of the shear rate at which the fluid changes from Newtonian to power-law behavior.
	<b>Power law index</b>	An index that controls the rate at which the viscosity decreases with increasing shear-rate. A value of 1 corresponds to a Newtonian fluid (no shear thinning), a low value corresponds to a viscosity that decreases rapidly (important shear thinning). A value smaller than 0 would lead to unstable flow as the shear stresses decreases with increasing shear-rate.
	<b>Plateau index</b>	Controls the transition from the Newtonian plateau to the power-law region. A low value (less than 1) lengthens the transition, and a high value (greater than 1) results in an abrupt transition.
Bird-Carreau	<b>Zero shear viscosity</b>	The value of the viscosity as the shear rate approaches zero.
	<b>Infinite shear viscosity</b>	The value of the viscosity as the shear rate approaches infinity.
	<b>Time constant</b>	Natural time, or the inverse of the shear rate at which the fluid changes from Newtonian to power-law behavior.
	<b>Power law index</b>	An index that controls the rate at which the viscosity decreases with increasing shear-rate. A value of 1 corresponds to a Newtonian fluid (no shear thinning), a low value corresponds to a viscosity that decreases rapidly (important shear thinning). A value smaller than 0 would lead to unstable flow as the shear stresses decreases with increasing shear-rate.
Modified Bingham	<b>Plastic viscosity</b>	The viscosity when shear-rate approaches infinity.
	<b>Yield stress threshold</b>	At zero shear-rate, the viscosity is equivalent to the <b>Plastic viscosity</b> added to the ratio of three times the <b>Yield stress threshold</b> and the <b>Critical shear rate</b> .
	<b>Critical shear rate</b>	The shear-rate corresponding to the center of the transition between the log shear-rate plateau and the high shear-rate plateau.

Model	Property	Details
Cross law	<b>Zero shear viscosity</b>	The value of the viscosity as the shear rate approaches zero.
	<b>Time constant</b>	Natural time, or the inverse of the shear rate at which the fluid changes from Newtonian to power-law behavior.
	<b>Cross law index</b>	An index that controls the rate at which the viscosity decreases with increasing shear-rate. A value of 0 corresponds to a Newtonian fluid (no shear thinning), a value of 1 corresponds to a viscosity that decreases rapidly (important shear thinning). A value greater than 1 would lead to unstable flow as the shear stresses decreases with increasing shear-rate.
Modified Cross law	<b>Zero shear viscosity</b>	The value of the viscosity as the shear rate approaches zero.
	<b>Time constant</b>	Natural time, or the inverse of the shear rate at which the fluid changes from Newtonian to power-law behavior.
	<b>Cross law index</b>	An index that controls the rate at which the viscosity decreases with increasing shear-rate. A value of 0 corresponds to a Newtonian fluid (no shear thinning), a value of 1 corresponds to a viscosity that decreases rapidly (important shear thinning). A value greater than 1 would lead to unstable flow as the shear stresses decreases with increasing shear-rate.
Bingham law	<b>Plastic viscosity</b>	The viscosity when shear-rate approaches infinity.
	<b>Yield stress threshold</b>	The viscosity at very low shear-rates is equal to the viscosity at high shear-rates plus twice the ratio of the <b>Yield stress threshold</b> and the <b>Critical shear rate</b> .
	<b>Critical shear rate</b>	The shear-rate beyond which Bingham's constitutive equation is applied. At this precise shear-rate, the viscosity is equal to the viscosity at high shear-rates plus the ratio of the <b>Yield stress threshold</b> and the <b>Critical shear rate</b>
Herschel-Bulkey law	<b>Yield stress threshold</b>	Adding the <b>Consistency factor</b> to the <b>Yield stress threshold</b> provides the zero shear-rate viscosity. The <b>Yield stress threshold</b> also reduces shear-thinning.
	<b>Critical shear rate</b>	The shear-rate corresponding to the center of the transition between the log shear-rate plateau and the high shear-rate plateau.
	<b>Consistency factor</b>	Adding the <b>Consistency factor</b> to the <b>Yield stress threshold</b> provides the zero shear-rate viscosity.
	<b>Power law index</b>	An index that controls the rate at which the viscosity decreases with increasing shear-rate. A value of 1 corresponds to a Newtonian fluid (no shear thinning), a low value corresponds to a viscosity that decreases rapidly (important shear thinning). A value smaller than 0 would lead to unstable flow as the shear stresses decreases with increasing shear-rate.

Model	Property	Details
Modified Herschel-Bulkey	<b>Yield stress threshold</b>	At zero shear-rate, the viscosity is equivalent to the <b>Plastic viscosity</b> added to the ratio of three times the <b>Yield stress threshold</b> and the <b>Critical shear rate</b> .
	<b>Critical shear rate</b>	The shear-rate corresponding to the center of the transition between the log shear-rate plateau and the high shear-rate plateau.
	<b>Consistency factor</b>	Adding the <b>Consistency factor</b> to the <b>Yield stress threshold</b> provides the zero shear-rate viscosity.
	<b>Power law index</b>	An index that controls the rate at which the viscosity decreases with increasing shear-rate. A value of 1 corresponds to a Newtonian fluid (no shear thinning), a low value corresponds to a viscosity that decreases rapidly (important shear thinning). A value smaller than 0 would lead to unstable flow as the shear stresses decreases with increasing shear-rate.
Power law	<b>Consistency factor</b>	The viscosity at shear-rate equal to the inverse of the <b>Time constant</b> .
	<b>Time constant</b>	The inverse of the shear-rate at which the viscosity is equal the <b>Consistency factor</b> .
	<b>Power law index</b>	An index that controls the rate at which the viscosity decreases with increasing shear-rate. A value of 1 corresponds to a Newtonian fluid (no shear thinning), a low value corresponds to a viscosity that decreases rapidly (important shear thinning). A value smaller than 0 would lead to unstable flow as the shear stresses decreases with increasing shear-rate.

## 4.4.6.11.2. Temperature-Dependent Viscosity Models

If the flow is nonisothermal, the temperature dependence of the viscosity must be taken into account along with the shear-rate dependence. The viscosity law can be factorized as follows:

$$\eta = H(T) \eta_0(\dot{\gamma})$$

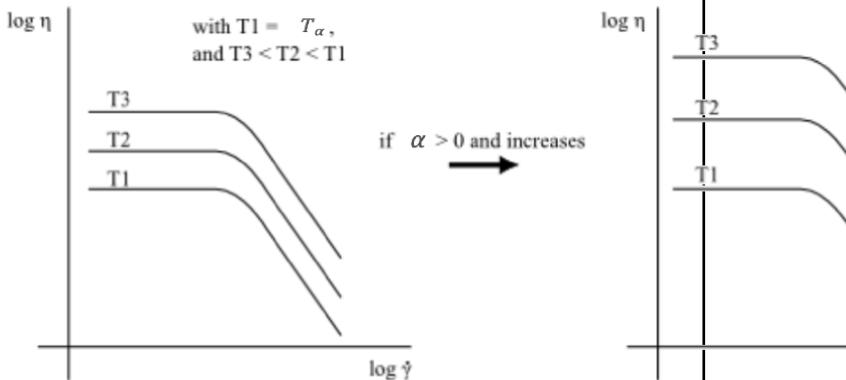
where  $H(T)$  is the Arrhenius law (or one of the other available laws) and  $\eta_0(\dot{\gamma})$  is the viscosity law at some reference temperature  $T_\alpha$  (as computed by one of the shear-rate-dependent laws).

Several temperature-dependent viscosity laws are available for generalized Newtonian flows.

Once you have chosen a temperature-dependent viscosity model, you will need to [set up the viscosity model](#) and assign various properties.

Table 4.4.6.11.2.1. Selecting a Temperature-dependent Viscosity Model

If you want to...	... then select	More information
Account for standard temperature dependence based on the assumption that the fluid flow obeys the Arrhenius equation for molecular kinetics	Arrhenius ...	<p>The Arrhenius law is given as</p> $H(T) = \exp \left[ \alpha \left( \frac{1}{T - T_0} - \frac{1}{T_\alpha - T_0} \right) \right]$ <p>where</p> <p><math>\alpha</math> = the ratio of the activation energy to the thermodynamic constant</p> <p><math>T_\alpha</math> = a reference temperature for which <math>H(T) = 1</math></p> <p>The temperature shift <math>T_0</math> is automatically set according to the selected units so, if you use Kelvin as the temperature unit, <math>T_0 = 0</math>, and if you use Celsius, <math>T_0 = -273.15</math>. <math>T_0</math> corresponds to the lowest temperature that is thermodynamically acceptable, therefore <math>T</math> and <math>T_\alpha</math> are absolute temperatures. They can also be defined relative to a non-absolute temperature scale, in which case <math>T_0</math> corresponds to the absolute zero temperature in the current temperature scale.</p> <p>The graphs correspond to a <b>Shift function</b> set to <b>Vertical shift</b>, where the <math>\eta = \eta(\dot{\gamma})H(T)</math>. If the <b>Shift function</b> is set to <b>Vertical &amp; horizontal shifts</b>, then the <math>\eta = \eta(H(T)\dot{\gamma})H(T)</math>.</p>

If you want to...	... then select	More information
Consider a model that is a close approximation to the Arrhenius model when the temperature difference is small.	<b>Arrhenius Approximate</b> ...	<p>The approximate Arrhenius law is written as follows:</p> $H(T) = \exp [ -\alpha(T - T_\alpha) ]$ <p>The behavior with this formulation is similar to that described by the Arrhenius law in the neighborhood of <math>T_\alpha</math>. This formulation is valid as long as the temperature difference <math>T - T_\alpha</math> is not too large.</p>  <p>The graphs correspond to a <b>Shift function</b> set to <b>Vertical shift</b>, where <math>\eta = \eta(\dot{\gamma})H(T)</math>. If the <b>Shift function</b> is set to <b>Vertical &amp; horizontal shifts</b>, then <math>\eta = \eta(H(T)\dot{\gamma})H(T)</math>.</p>
Consider a model mainly used for glass-based simulations.	<b>Fulcher</b>	<p>Another definition for <math>H(T)</math> comes from the Fulcher law:</p> $H(T) = 10^{-f_1 + \frac{f_2}{T-f_3}}$ <p>where <math>f_1</math>, <math>f_2</math>, and <math>f_3</math> are the Fulcher constants.</p>
Consider a model that fits experimental data better than the Arrhenius law for a wide range of temperatures, especially close to the glass transition temperature.	<b>Williams-Landel-Ferry</b>	<p>The Williams-Landel-Ferry (WLF) model is given as:</p> $\ln(H(T)) = \frac{c_1(T_r - T_a)}{c_2 + T_r - T_a} - \frac{c_1(T - T_a)}{c_2 + T - T_a}$ <p>where <math>c_1</math> and <math>c_2</math> are the WLF constants, and <math>T_r</math> and <math>T_a</math> are reference temperatures.</p> <p>If the <b>Shift function</b> is set to <b>Vertical shift</b>, then <math>\eta = \eta(\dot{\gamma})H(T)</math>. If the <b>Shift function</b> is set to <b>Vertical &amp; horizontal shifts</b>, then <math>\eta = \eta(H(T)\dot{\gamma})H(T)</math>.</p>

## 4.4.6.11.2.1. Setting Up a Temperature-Dependent Newtonian Viscosity Model

Simulations involving viscous thermal flows, such as polymer extrusion simulations, often require the use of a temperature-dependent viscosity model. Assign viscosity models as a property of the particular material you want to simulate.

To define the properties of the temperature-dependent viscosity law:

1. In the material panel (for example, the **Generic Polymer** panel, for **Liquid Properties**), click **Add**.
2. In the search field, type **Temperature** to display a list of temperature-dependent viscosity models.
3. **Select a temperature-dependent viscosity model**.

Valid options are:

- Arrhenius
- Arrhenius Approximate
- Fulcher
- Williams-Landel-Ferry

4. For the selected model, specify values for the model properties.

Table 4.4.6.11.2.1.2. Temperature-Dependent Viscosity Models and Their Properties

Model	Property	Details
<b>Arrhenius</b>	<b>Shift function</b>	Dictates how the viscosity will change with temperature: a vertical shift means that the viscosity curve is simply shifted downwards in a viscosity/shear-rate diagram when temperature increases, a vertical+horizontal shift means that the viscosity curve is shifted downwards and to the right.
	<b>Activation energy</b>	The ratio of the activation energy to the thermodynamic constant.
	<b>Reference temperature</b>	The reference temperature at which the Arrhenius law is equivalent to 1.
<b>Arrhenius Approximate</b>	<b>Shift function</b>	Dictates how the viscosity will change with temperature: a vertical shift means that the viscosity curve is simply shifted downwards in a viscosity/shear-rate diagram when temperature increases, a vertical+horizontal shift means that the viscosity curve is shifted downwards and to the right.
	<b>Activation energy</b>	The ratio of the activation energy to the thermodynamic constant.
	<b>Reference temperature</b>	The reference temperature at which the Arrhenius law is equivalent to 1.
<b>Fulcher</b>	<b>Constant, F1</b>	One of the Fulcher constants.
	<b>Constant, F2</b>	One of the Fulcher constants.
	<b>Constant, F3</b>	One of the Fulcher constants.

Model	Property	Details
Williams-Landel-Ferry	<b>Shift function</b>	Dictates how the viscosity will change with temperature: a vertical shift means that the viscosity curve is simply shifted downwards in a viscosity/shear-rate diagram when temperature increases, a vertical+horizontal shift means that the viscosity curve is shifted downwards and to the right.
	<b>Constant, C1</b>	One of the Williams-Landel-Ferry constants.
	<b>Constant, C2</b>	One of the Williams-Landel-Ferry constants.
	<b>Reference temperature, Ta</b>	A reference temperature.
	<b>Reference temperature difference, Tr-Ta</b>	The difference between two reference temperatures.

### 4.4.6.11.3. Simplified Viscoelastic Fluid Model

When you would like to better predict extrudate swelling, or when you would like a qualitative prediction of the extrudate free surface, use the **Simplified viscoelastic fluid** option when assigning a **Material model** for a generic polymer material in polymer extrusion simulations.

Enhanced extrudate swelling in polymer extrusion flow is primarily due to the first normal stress difference. Since this is typically a viscoelastic property, the simplified viscoelastic model is an extension of existing Newtonian fluid models, in which a normal stress difference has been incorporated into the constitutive behavior. In simple shear flow along the first axis and with a shear rate  $\dot{\gamma}$ , the total extra-stress tensor  $\mathbf{T}$  is given by:

$$\mathbf{T} = \begin{pmatrix} \psi\mu(\dot{\chi})\dot{\chi} & \eta(\dot{\gamma})\dot{\gamma} & \cdot \\ \eta(\dot{\gamma})\dot{\gamma} & \ddots & \cdot \\ \cdot & \cdot & \ddots \end{pmatrix}$$

where,

$\eta(\dot{\gamma})\dot{\gamma}$  = the shear stress component, which involves the shear rate dependent viscosity  $\eta(\dot{\gamma})$ .

$\eta(\dot{\gamma})$  = the shear viscosity

$\psi\mu(\dot{\chi})\dot{\chi}$  = the first normal stress

$\mu(\dot{\chi})$  = the normal viscosity

$\psi$  = the weighting coefficient

$\lambda(\dot{\gamma})$  = the relaxation time function

$\dot{\chi}$  = the pseudo shear rate that obeys a transport equation involving the relaxation time  $\lambda(\dot{\gamma})$ :

$\lambda(\dot{\gamma}) \frac{D\dot{\chi}}{Dt} + \dot{\chi} = \dot{\gamma}$ . The equation is such that the solution  $\dot{\chi} = \dot{\gamma}$  is recovered in simple steady shear flow.

The first normal viscosity  $\mu(\dot{\chi})$  is described by means of functions similar to those available for the shear viscosity  $\eta(\dot{\gamma})$ , where  $\dot{\gamma}$  is replaced by  $\dot{\chi}$ . In order to facilitate the setup of a flow simulation involving the simplified viscoelastic model, identical dependences for  $\mu(\dot{\chi})$  and  $\eta(\dot{\gamma})$  should be considered. However, it is important to note that different functions can be selected for the shear and normal viscosities.

You can specify these properties when you [set up the Simplified Viscoelastic fluid model](#).

## 4.4.6.11.3.1. Setting Up a Simplified Viscoelastic Fluid

Polymer extrusion simulation allows you to study the effects of swelling of the polymer extrudate.

To better predict swelling of the extrudate in a polymer extrusion simulation, for the **Material model**, you should use the [Simplified viscoelastic fluid](#).

- Specify a **Generalized Newtonian viscosity model** for the shear viscosity,  $\eta(\dot{\gamma})$ . Typically, standard viscosity data (shear viscosity vs. shear rate) should be used for identifying the shear viscosity function. In most situations, shear thinning is experimentally observed, and algebraic relationships such as the power law, the Bird-Carreau law, or the Cross law will be good candidates. However, you should consider a law that exhibits a zero-shear plateau if you expect regions in which there is no deformation in the flow domain.

Valid options are:

- Power law
- Bird-Carreau
- Carreau-Yasuda
- Cross law
- Modified Cross law

- Specify a **First normal viscosity**,  $\mu(\dot{\chi})$ . The first normal viscosity is described by means of functions similar to those available for the shear viscosity  $\eta(\dot{\gamma})$ , where  $\dot{\gamma}$  is replaced by  $\dot{\chi}$ .
- Specify a **Relaxation time function**,  $\lambda(\dot{\gamma})$ . The relaxation time function controls the development of the extrudate diameter along the jet, and may also have an influence on the developed extrudate geometric attributes.
- Specify a **Weighting coefficient**,  $\psi$ . Typically, the weighting coefficient controls the swelling intensity vs. flow rate. A series of calculations should be performed to examine the development of extrudate vs. flow rate. A comparison with experimental data on swelling should enable the selection of an appropriate numerical value for the weighting coefficient using the **Constant law** field.
- For non-isothermal cases, you must also specify an appropriate [temperature-dependent viscosity model](#).
- In the **Material Assignment** panel, for the **Material model**, select [Simplified viscoelastic fluid](#).

### 4.4.6.11.3.1.1. Setting First Normal Viscosity Properties

For polymer extrusion flows employing a [simplified viscoelastic fluid model](#), you need to describe a first normal viscosity law.

To select a **First Normal Viscosity** law, and assign its properties:

- In the material panel (for example, the **Generic Polymer** panel, for **Liquid Properties**), click **Add**.
- In the search field, enter **First normal viscosity** to see a list of available options.
- Select a first normal viscosity model.

Valid options are:

- Bird-Carreau
- Power law
- Cross law
- Modified Cross law
- Carreau-Yasuda
- Constant

4. Assign property values for the selected normal viscosity.

Table 4.4.6.11.3.1.1.1. First Normal Viscosity Model Options and Their Properties

Viscosity Model	Property	Details
Bird-Carreau Law	<b>Zero first normal viscosity</b>	The normal viscosity at zero pseudo shear rate strain-rate.
	<b>Infinite first normal viscosity</b>	The normal viscosity when the pseudo shear rate strain-rate approaches infinity.
	<b>Time constant</b>	Natural time, or the inverse of the pseudo shear rate strain-rate at which the normal viscosity starts to decrease with increasing simplified viscoelastic variable strain-rate.
	<b>Power law index</b>	An index that controls the pseudo shear rate strain-rate at which the normal viscosity decreases with increasing pseudo shear rate strain-rate. A value of 1 corresponds to a constant normal viscosity, a low value corresponds to a normal viscosity that decreases rapidly.
Power Law	<b>Consistency factor</b>	The normal viscosity at pseudo shear rate strain-rate equal to the inverse of the <b>Time constant</b> .
	<b>Time constant</b>	The inverse of the pseudo shear rate strain-rate at which the normal viscosity is equal the <b>Consistency factor</b> .
	<b>Power law index</b>	An index that controls the pseudo shear rate strain-rate at which the normal viscosity decreases with increasing pseudo shear rate strain-rate. A value of 1 corresponds to a constant normal viscosity, a low value corresponds to a normal viscosity that decreases rapidly.
Cross Law	<b>Zero first normal viscosity</b>	The normal viscosity at zero pseudo shear rate strain-rate.
	<b>Time constant</b>	Natural time, or the inverse of the pseudo shear rate strain-rate at which the normal viscosity starts to decrease with increasing simplified viscoelastic variable strain-rate.
	<b>Cross law index</b>	An index that controls the pseudo shear rate strain-rate at which the normal viscosity decreases with increasing pseudo shear rate strain-rate. A value of 1 corresponds to a constant normal viscosity, a low value corresponds to a normal viscosity that decreases rapidly.
Modified Cross Law	<b>Zero first normal viscosity</b>	The normal viscosity at zero pseudo shear rate strain-rate.
	<b>Time constant</b>	Natural time, or the inverse of the pseudo shear rate strain-rate at which the normal viscosity starts to decrease with increasing simplified viscoelastic variable strain-rate.
	<b>Cross law index</b>	An index that controls the pseudo shear rate strain-rate at which the normal viscosity decreases with increasing pseudo shear rate strain-rate. A value of 1 corresponds to a constant normal viscosity, a low value corresponds to a normal viscosity that decreases rapidly.

Viscosity Model	Property	Details
Carreau-Yasuda	<b>Zero first normal viscosity</b>	The normal viscosity at zero pseudo shear rate strain-rate.
	<b>Infinite first normal viscosity factor</b>	The normal viscosity when the pseudo shear rate strain-rate approaches infinity.
	<b>Time constant</b>	Natural time, or the inverse of the pseudo shear rate strain-rate at which the normal viscosity starts to decrease with increasing simplified viscoelastic variable strain-rate.
	<b>Power law index</b>	An index that controls the pseudo shear rate strain-rate at which the normal viscosity decreases with increasing pseudo shear rate strain-rate. A value of 1 corresponds to a constant normal viscosity, a low value corresponds to a normal viscosity that decreases rapidly.
	<b>Plateau Index</b>	An index that controls the transition from the plateau to the power-law region. A low value (less than 1) lengthens the transition, and a high value (greater than 1) results in an abrupt transition.
Constant	<b>First normal viscosity</b>	The normal viscosity at any pseudo shear rate strain-rate.

#### 4.4.6.11.3.1.2. Setting Relaxation Time Function Properties

For polymer extrusion flows employing a [simplified viscoelastic fluid model](#), you need to describe a relaxation time function.

To select a **Relaxation Time Function** law, and assign its properties:

1. In the material panel (for example, the **Generic Polymer** panel, for **Liquid Properties**), click **Add**.
2. In the search field, enter **Relaxation time function** to see a list of available options.
3. Select a relaxation time function.

Valid options are:

- Bird-Carreau
- Power law
- Constant law

4. Assign property values for the selected relaxation time function.

Table 4.4.6.11.3.1.2.2. Relaxation Time Function Viscosity Model Options and Their Properties

Viscosity Model	Property	Details
Bird-Carreau Law	<b>Zero relaxation time</b>	The relaxation time at zero shear-rate.
	<b>Time constant</b>	Natural time, or the inverse of the shear-rate at which the relaxation time starts to decrease with an increasing shear-rate.
	<b>Power law index</b>	An index that controls the rate at which the relaxation time decreases with increasing shear-rate. A value of 1 corresponds to a constant relaxation time, a low value corresponds to a relaxation time that decreases rapidly.

Viscosity Model	Property	Details
Power law	Consistency factor	The relaxation time at shear-rate equal to the inverse of the <b>Time constant</b> .
	Time constant	The inverse of the shear-rate at which the relaxation time is equal the <b>Consistency factor</b> .
	Power law index	An index that controls the rate at which the relaxation time decreases with increasing shear-rate. A value of 1 corresponds to a constant relaxation time, a low value corresponds to a relaxation time that decreases rapidly.
Constant law	Relaxation time	Controls the development of the extrudate swelling rather than its intensity. Experimental data based on swelling can be used to fine-tune the values for this property.

#### 4.4.6.11.3.1.3. Setting Weighting Coefficient Properties

For polymer extrusion flows employing a [simplified viscoelastic fluid model](#), you need to describe a weighting coefficient.

To select a **Weighting Coefficient** law, and assign its properties:

1. In the material panel (for example, the **Generic Polymer** panel, for **Liquid Properties**), click **Add**.
2. In the search field, enter **Weighting coefficient** to see the **Weighting Coefficient - Constant law** option.
3. Assign a value to the **Weighting coefficient**. This field controls the swelling intensity versus the flow rate. Experimental data based on swelling can be used to fine-tune the values for this property.

## 4.4.6.12. Viscosity Models for Polymer Blow Molding Simulations

Polymer blow molding simulations in AIM can utilize the following types of viscosity models:

- Newtonian fluid
- [Generalized Newtonian fluid](#)
- [Integral Viscoelastic Fluid Model](#)

### 4.4.6.12.1. Setting Up an Integral Viscoelastic Fluid Model

An important aspect of the behavior of a polymer in blow molding is the strain hardening, which results in the reinforcement of the material under deformation. The integral viscoelastic model is especially suited for modeling strain hardening and obtaining accurate blow molding results. It exhibits both viscous and elastic properties.

In the **Material Assignment** panel, select **Integral viscoelastic fluid** for the **Material model**.

To define the parameters of the integral viscoelastic fluid model:

1. In the material panel (for example, the **Generic Polymer** panel, for **Liquid Properties**), click **Add**.
2. In the search field, enter **Integral** to display the relaxation time spectrum and additional viscosity parameters.

## 4.4.6.13. Fatigue Material Properties

Fatigue Life material properties encompass both the set of properties required for stress life and strain life analysis.

### Stress Life Material Properties

The properties supported for stress life include:

<a href="#">Alternating Stress</a> on page 243	Define the S-N curve data (alternating stress vs. number of cycles to failure) to perform the most accurate stress life analysis.
<a href="#">Tensile Ultimate Strength</a> on page 243	In the absence of S-N curve data, you can specify an ultimate strength value. Then AIM internally constructs a simplified S-N curve using just the ultimate strength.  Also required if you want to use Goodman or Gerber for your Mean stress theory.
<a href="#">Tensile Yield Strength</a> on page 244	Required if you want to use Soderberg for your Mean stress theory.

### Strain Life Material Properties

Six material properties are required for [strain life analysis](#). Four define the strain life curve:

- Strength Coefficient
- Strength Exponent
- Ductility Coefficient
- Ductility Exponent

And two define the cyclic stress-strain curve:

- Cyclic Strength Coefficient
- Cyclic Strain Hardening Exponent

#### 4.4.6.13.1. Alternating Stress

Alternating Stress is defined as the amount of stress that deviates from the mean. It is also called stress amplitude.

Under this property, you can define the S-N curve as [tabular data](#), where S is the Alternating Stress and N is the Fatigue Life (the number of cycles to failure). The unit of Alternating Stress is defined as [\[stress\]](#), for example [\[Pa\]](#). The unit of Fatigue Life is in Cycles.

In addition, you can specify the type of interpolation method for the S-N curve. This interpolation method is used in both the chart display and the fatigue solver.

#### 4.4.6.13.2. Tensile Ultimate Strength

Tensile ultimate strength is defined as the highest point on the Stress-Strain curve obtained from a tensile test.

You can define Tensile ultimate strength ( $S_{ut}$ ) as a constant or as an [expression](#). The units are entered as [\[stress\]](#). For example "460000000 [Pa]."

### 4.4.6.13.3. Tensile Yield Strength

Tensile yield strength is defined as the point on the Stress-Strain curve where the material starts to deform plastically.

You can define Tensile yield strength ( $S_{yt}$ ) as a constant or as an [expression](#). The units are entered as [stress], for example "250000000 [Pa]".

### 4.4.6.13.4. Strain Life Parameters

While Stress life fatigue is based on empirical S-N curves, Strain life fatigue is based upon the Strain Life Relation Equation where the Strain Life Parameters are values for a particular material that best fit the equation to measured results.

The Strain Life Relation requires a total of six parameters to define the strain-life material properties; four strain-life parameter properties and the two cyclic stress-strain parameters.

The Strain Life Relation equation is shown below:

$$\frac{\Delta\varepsilon}{2} = \frac{\sigma'_f}{E} (2N_f)^b + \varepsilon'_f (2N_f)^c$$

The two cyclic stress-strain parameters are part of the equation below:

$$\Delta\varepsilon = \frac{\Delta\sigma}{E} + 2 \left( \frac{\Delta\sigma}{2K'} \right)^{\frac{1}{n'}}$$

Where:

$\frac{\Delta\varepsilon}{2}$  is the Total Strain Amplitude

$\Delta\sigma$  is 2x Stress Amplitude

$E$  is the Modulus of Elasticity

$N_f$  is the Number of Cycles to Failure

$2N_f$  is the Number of Reversals to Failure

And the parameters required for a Strain Life analysis are:

$\sigma'_f$  is the Fatigue Strength Coefficient

$b$  is the Fatigue Strength Exponent (Basquin's Exponent)

$\varepsilon'_f$  is the Fatigue Ductility Coefficient

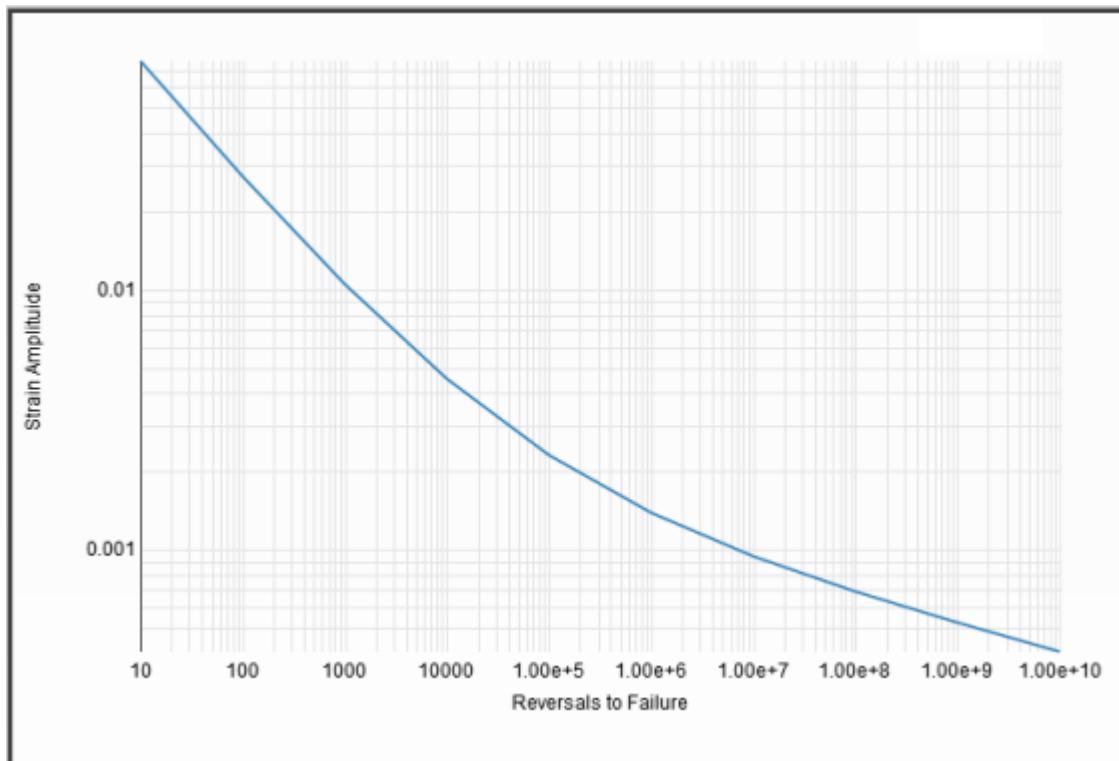
$c$  is the Fatigue Ductility Exponent

$K'$  is the Cyclic Strength Coefficient  $n'$  is the Cyclic Strain Hardening Exponent

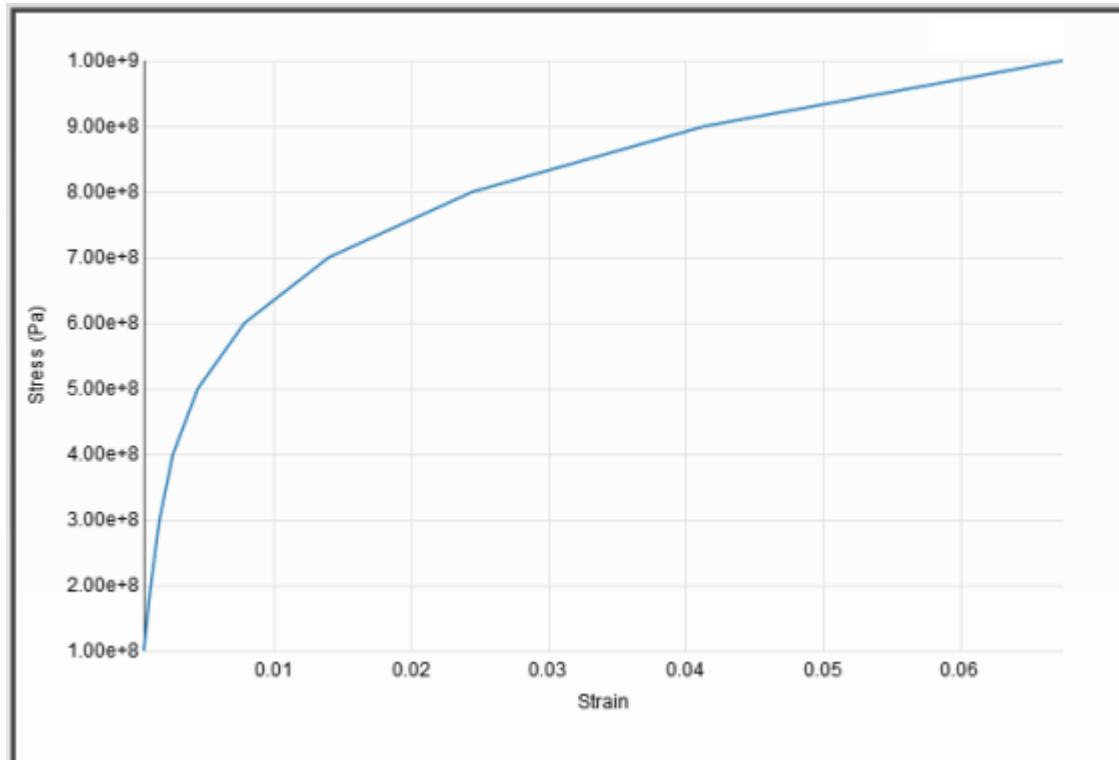
The total Strain amplitude

$$\frac{\Delta\varepsilon}{2}$$

can be plotted as shown below in the Strain-life curve:



Similarly the cyclic stress strain equation can be plotted as shown below in the Cyclic Stress-Strain curve:



## 4.4.6.14. Electromagnetic Material Properties

In addition to the standard structural material properties, the following electromagnetics properties are also available for materials used in electromagnetics simulations.

- Isotropic Relative Permittivity
- Isotropic Relative Permeability
- B-H Curve (Nonlinear relative permeability)
- Isotropic Electrical Conductivity
- Isotropic Magnetic Loss Tangent
- Magnetic Coercivity
- Core Loss Power Ferrite

### 4.4.6.14.1. Isotropic Magnetic Loss Tangent

To represent a magnetic material such as ferrite that dissipates the power of a high-frequency magnetic field, enter a magnetic loss tangent property value for the material. The smaller the loss tangent, the less lossy the material.

Isotropic magnetic loss tangent  $\tan\delta_m$  can be parameterized, defined as a constant, defined by an [expression](#) or in the form of tabular data (valid only for Frequency response simulations) where magnetic loss tangent is tabulated as a function of frequency. For details on how tabular data can be defined, refer to [Defining Tabular Data](#) on page 84. The base values are unitless.

### 4.4.6.14.2. Isotropic Relative Permittivity

The permittivity of a material relates the electric flux density to the electric field. Relative permittivity  $\epsilon_r$  is the ratio of the permittivity  $\epsilon$  of a material to the permittivity  $\epsilon_0$  of free space. The relative permittivity of free space (vacuum) is 1.

Isotropic relative permittivity  $\epsilon_r$  can be parameterized, defined as a constant, defined by an [expression](#), or in the form of tabular data. Relative permittivity can be tabulated as a function of either frequency (valid only for frequency response designs) or temperature. For details on how tabular data can be defined, refer to [Defining Tabular Data](#) on page 84. The base values are unitless.

Refer to [Temperature-Dependent Electromagnetics Materials](#) for details on how this property can be used for designs that include temperature-dependent materials.

### 4.4.6.14.3. Isotropic Relative Permeability

The permeability of a material relates the magnetic flux density to the magnetic field. Isotropic relative permeability  $\mu_r$  is the ratio of the permeability  $\mu$  of a material to the permeability of free space  $\mu_0$ . The relative permeability of free space (vacuum) is 1.

Isotropic relative permeability  $\mu_r$  can be parameterized, defined as a constant, defined by an [expression](#), or in the form of tabular data. Relative permeability can be tabulated as a function of either frequency (valid only for frequency response designs) or temperature. For details on how tabular data can be defined, refer to [Defining Tabular Data](#) on page 84. The base values are unitless.

Refer to [Temperature-Dependent Electromagnetics Materials](#) for details on how this property can be used for designs that include temperature-dependent materials.

The relative permeability is a dimensionless quantity, defined as follows:

$$\mu = \mu_r \mu_0$$

where  $\mu$  is the permeability,  $\mu_r$  is the relative permeability, and  $\mu_0$  is the permeability of free space.

Some materials exhibit a permeability that includes both a real and imaginary component:

$$\mu = \mu' \left( 1 - j \frac{\mu''}{\mu'} \right)$$

where  $\mu'$  is the real portion of  $\mu$  and  $\mu''/\mu'$  is the [magnetic loss tangent](#).

## 4.4.6.14.4. Isotropic Electrical Conductivity

Isotropic electrical conductivity is the measure of a material's ability to conduct an electric current. A high conductivity indicates a material that readily allows the movement of electrical charge.

Isotropic electrical conductivity  $\sigma$  can be parameterized, defined by an [expression](#), or in the form of tabular data. Electrical conductivity can be tabulated as a function of either frequency (valid only for frequency response designs) or temperature. For details on how tabular data can be defined, refer to [Defining Tabular Data](#) on page 84. The base units are siemens per meter.

Refer to [Temperature-Dependent Electromagnetics Materials](#) for details on how this property can be used for designs that include temperature-dependent materials.

## 4.4.6.14.5. Magnetic Coercivity

Magnetic coercivity is a measure of the ability of a ferromagnetic material to withstand an external magnetic field without becoming demagnetized. For a ferromagnetic material the coercivity is the intensity of the applied magnetic field required to reduce the magnetization of that material to zero after the magnetization of the material has been driven to saturation. Thus Magnetic Coercivity measures the resistance of a ferromagnetic material to becoming demagnetized. Typically, the magnetic coercivity of a magnetic material is determined by measurement of the magnetic hysteresis loop. Specifically, the magnetic coercivity is a horizontal intercept of the hysteresis loop.

Magnetic Coercivity can be used together with either the [Isotropic relative permeability property](#), or the [B-H Curve property](#):

- If used with the [isotropic relative permeability](#) property, the **Magnitude** of the Magnetic Coercivity is editable. Enter a non-zero value for the magnitude, and then choose a coordinate system for expressing the coercivity vector from the **Magnetization direction** drop-down list.
  - Cartesian (defined by the X, Y, and Z axes)
  - Cylindrical (defined by the R, Phi, and Z axes)
  - Spherical (defined by the Rho, Theta, and Phi axes)

For example, the following coercivity and relative permeability settings are for the SmCo 24 material.

**Magnetic Coercivity** 

Magnitude  
 

Magnetization direction  
 

X  
 

Y  
 

Z  
 

---

Isotropic magnetic loss tangent,  $\tan \delta_m$   
 

---

Isotropic relative permeability,  $\mu_r$   
 

- If used with the B-H curve property, the **Magnitude** of the Magnetic Coercivity is not editable. It is obtained from the B-H curve. Choose a coordinate system type for expressing the coercivity vector from the **Magnetization direction** drop-down list.

Select **No Magnetization** from the **Magnetization direction** drop-down list if an external magnetic field has no effect.

Magnetic coercivity **Magnitude** can be parameterized, or defined by an [expression](#).

## 4.4.6.14.6. Core Loss Power Ferrite

Core losses are generated by the changing magnetic flux field within a power ferrite material. The power loss equation values needed to calculate **Core Loss Power Ferrite** are: the core loss coefficient  $C_m$ , Exponent X for the frequency (f) term of the core loss equation, and Exponent Y for the magnetic flux (B) term of the core loss equation.

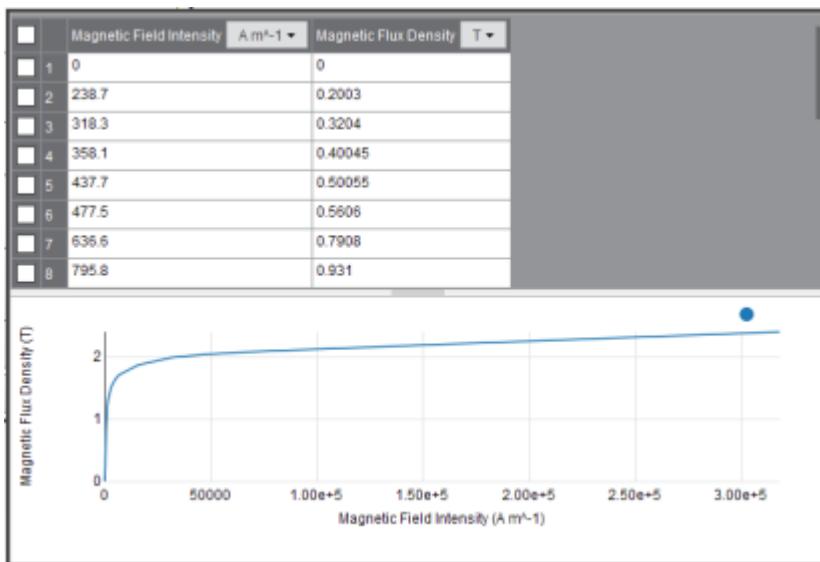
$$P_v = C_m f^X B_m^Y$$

Typically, values for  $C_m$ , Exponent X and Exponent Y are obtained from a manufacturer's data sheet.

These values can be parameterized, or defined by an [expression](#).

## 4.4.6.14.7. B-H Curve (Nonlinear relative permeability)

In nonlinear materials, when a material has a permeability that varies with the flux density, a B-H curve is used to describe the material's nonlinear behavior. The *B* refers to the Magnetic Flux Density measured in Teslas. The *H* refers to the Magnetic Field Intensity, measured in Amperes per meter.



Nonlinear relative permeability is not a constant, and can be defined by an [expression](#), or in the form of tabular data. For details on how tabular data can be defined, refer to [Defining Tabular Data](#) on page 84.

#### 4.4.6.14.8. Temperature-Dependent Electromagnetics Materials

Temperature-dependent materials have one or more material properties that vary with temperature. The following electromagnetics material properties can be temperature dependent: [Isotropic Electrical Conductivity](#), [Isotropic Relative Permeability](#), and [Isotropic Relative Permittivity](#). The use of temperature-dependent materials is required for magnetics designs that include [thermal to magnetics coupling](#).

**Temperature** is an independent variable whose values are used by the solver to determine the values of temperature-dependent properties used when calculating solutions. Temperature values come from one of the following three sources:

- Thermal feedback - if a temperature-dependent material is used in a [thermal to magnetics coupling](#) design.
- [Body temperature](#)
- [Ambient temperature](#) (default: 22C)

Thermal feedback temperatures, if present, override both ambient temperature and any body temperatures that may be assigned. Similarly, body temperature assignments override ambient temperature.

Temperature-dependent electromagnetic material properties can be defined in any of the following ways:

- A simple expression of Temperature, such as:  $100 * \text{Temperature}/1[\text{K}]$ .
- A temperature table of property values. (For details on how tabular data can be defined, refer to [Defining Tabular Data](#) on page 84.)
- For frequency response designs only, you can define a Frequency table, with Temperature as a scaling function. (For details on how tabular data can be defined, refer to [Defining Tabular Data](#) on page 84.)

## ✓ Solid Properties

Add ▾

Isotropic relative permeability,  $\mu_r$

1



Isotropic relative permittivity,  $\epsilon_r$

table(Frequency)\*Temperature/1[K]



- For frequency response designs only, you can define both a Frequency table and a Temperature table, as well specify a reference **Frequency for temperature-dependent values**.

Isotropic electrical conductivity,  $\sigma$

table1(Frequency)\*

(table2(Temperature)/table1(60 [Hz]))



### ▼ Inputs for isotropic electrical conductivity tables

Frequency dependence

table1(Frequency)



Temperature dependence

table2(Temperature)



Frequency for temperature-dependent values

60 Hz



If the specified reference frequency does *not* exist in the frequency table, but is within the range of frequencies in the table, interpolation is used on the frequency table to determine the property value at the reference frequency. If the specified reference frequency is less than the lowest frequency in the table, the value at the lowest frequency in the table is used. If the reference frequency is greater than the highest frequency, the value at the highest frequency is used.

**Note:** The solver applies its own internal temperature and frequency values of 1C and 1GHz, respectively, to evaluate the material property for conduction path calculations and pre-validation. During solve, active Frequency and/or Temperature values are used to evaluate the material property values; interpolation/extrapolation are used as needed.

**Note:** If a temperature-dependent material also has frequency dependency defined, the material cannot be used in Static design calculations. Remove the frequency table to correct this condition.

Isotropic electrical conductivity,  $\sigma$

table1(Frequency)\*

(table2(Temperature)/table1(60 [Hz]))



Frequency is not a valid field variable  
in this calculation type.

## 4.4.7. Customizing the Appearance of Materials

### Physics > Material Assignment

You can make a material appear more realistic when the graphics viewer is in **Enhanced display** mode by applying color, transparency, a finish, and texture.

**Note:** These changes will apply wherever this material is assigned in this study. If you want the same material with a different appearance elsewhere, [duplicate the material](#) first and assign the different versions of the material as appropriate.

To modify the appearance of a material:

1. Click the **Choose display mode** icon and select **Enhanced display**.



2. In the **Physics > Material Assignment** panel, select the material you want to modify.
3. In the **material\_name** panel, under **Appearance Properties** you can modify any or all the categories listed in the table that follows.

Table 4.4.7.1. Material Appearance Properties

Option	Description
<b>Color</b>	Adjust the levels of red, green, and blue until you achieve the desired color.
<b>Opacity</b>	Choose from 0 (transparent) to 1 (opaque).
<b>Metallic finish</b>	Choose from 0 (plastic) to 1 (metallic) finish for materials with high opacity, such as solids.
<b>Roughness</b>	Choose from 0 (polished) to 1 (rough). Roughness is easiest to see when <b>Metallic finish</b> and <b>Opacity</b> have high values.
<b>Transmittance</b>	Choose from 0 (little light passes through) to 1 (all light passes through) for materials with low opacity to determine how much light (radiant energy) is transmitted through the material.

**Note:** If you are [creating a new material](#), (or you discarded a material's **Appearance Properties**) and want to include the option to modify your material's appearance:

- a. Select **Add** for the **Solid properties**, **Liquid properties**, and/or **Gas properties**.
- b. Enter **Appearance Properties** in the quick search field and select this option.

For any default material that already exists in the Study, the appearance properties are preset for optimum visualization. Any newly created materials will not have any preset appearance properties.

## 4.5. Physics Options

### Physics > Physics Options

The Physics options that AIM provides are based on the type of simulation:

- For [fluid flow physics options](#), you have flow, turbulence, and in some cases energy modeling options.
- For [structural physics options](#), you can set solid and beam modeling and topology optimization options.

- For [polymer physics options](#), you have polymer inertia and gravity options.
- For [electromagnetic physics options](#), you have inductance matrix as well as force and torque calculation options.
- For [topology optimization options](#), you can modify your design goal and set the removal strategy in the case of multiple materials in the model.

## 4.5.1. Fluid Flow Physics Options

### Physics > Physics Options

The fluid flow physics options that AIM provides are grouped as:

- [Flow model options](#), which enable you to set the operating conditions and flow viscosity for your simulation, as well as model buoyancy-driven flows.
- [Turbulence model options](#).

## 4.5.1.1. Flow Model Options

The **Flow Model** physics options enable you to:

- model [flow viscosity](#) (available when you click the **Show all properties** icon - [set the operating conditions](#) for your simulation.
- model [buoyancy-driven flows](#).

### 4.5.1.1.1. Viscous vs. Inviscid Flows

A **Viscous** analysis involves a solution of the full Navier-Stokes equations whereas an **Inviscid** analysis ignores all viscous effects. While all wall bounded flows include viscous effects at the wall (shear due to the no-slip condition), it can be useful to ignore these effects in some cases and perform an **Inviscid** analysis to quickly obtain approximate solutions. You can choose which type of analysis to perform using the

**Stress-strain model** option, which is available when you click the **Show all properties** icon .

When the ratio of inertial forces to viscous forces, the Reynolds Number, is large, viscous effects may play a smaller role. One example of this situation is an aerodynamic analysis of a high-speed projectile. In a case like this, where the pressure forces on the body dominate the viscous forces, you can use an **Inviscid** analysis to get a quick estimate of the primary forces acting on the body. After the body shape has been modified to maximize the lift forces and minimize the profile drag forces, you can perform a **Viscous** analysis to include the effects of the fluid viscosity and turbulent viscosity on the lift and drag forces.

### 4.5.1.1.2. Setting Operating Conditions

#### Physics > Physics Options > Flow Model > Operating Conditions

1. Expand the **Operating Pressure** options.
2. Optionally modify the default value.

The **Operating Pressure** should approximate the mean absolute pressure of the flow region.

3. Select the **Reference pressure location** (This property is available when the **Show all properties** icon  is active.)
  - If you know the absolute pressure at a specific location in the flow region, choose **Cartesian coordinates**. For this option, you need to specify the **X**, **Y**, and **Z** coordinates, respectively of the **Reference pressure location**.
  - If you don't know the absolute pressure at a specific location, select **Automatic**.

## 4.5.1.1.2.1. Operating Pressure

All calculations use gauge pressure, instead of absolute pressure, to avoid significant round-off errors. The operating pressure relates gauge pressure to absolute pressure by:

$$P_{abs} = P_{op} + P_{gauge}$$

When the material density definition does not depend on pressure, the solution is independent of the operating pressure. In this case, the solution depends only on the relative pressures in the analysis.

## 4.5.1.1.2.2. Reference Pressure Location



This property is available when the **Show all properties** icon is active.

For incompressible flows that do not involve any pressure boundaries, the pressure level is defined by holding pressure constant at the **Reference Pressure Location**. This location will have an absolute pressure equal to the **Operating Pressure** you specify, and therefore a gauge pressure of zero. If pressure boundaries are involved, the adjustment is not needed and the **Reference Pressure Location** is ignored.

The reference pressure location is, by default, the cell center at or closest to (0,0,0). There may be cases in which you might want to move the **Reference Pressure Location**, perhaps locating it at a point where the absolute static pressure is known (for example, if you are planning to compare your results with experimental data).

## 4.5.1.1.3. Buoyancy-Driven Flows

Fluid flow for which gravity has an important effect is called buoyant flow. Because the force of gravity is proportional to fluid density, buoyant flow can arise whenever the fluid density varies for any reason.

The most common buoyant flow situation is natural (or free) convection, where the fluid density varies with temperature. An important dimensionless number for natural convection flows is the Grashof number

$$Gr = g\beta\Delta TL^3 / \nu^2$$

where  $\mathcal{G}$  is the magnitude of the gravity vector,  $\Delta T$  is the temperature range,  $L$  is the length scale,  $\nu$  is the kinematic viscosity, and  $\beta$  is the thermal expansion coefficient

$$\beta = -\frac{1}{\rho} \left( \frac{\partial \rho}{\partial T} \right)_P$$

The Grashof number represents the ratio of buoyancy force to viscous force, and therefore plays the same role as the Reynolds number ( $Re$ ) for forced convection flows. A related dimensionless number is the Rayleigh number

$$Ra = Gr Pr$$

where  $Pr$  is the Prandtl number. A natural convection flow is typically laminar if  $Ra < 10^8$  and turbulent if  $Ra > 10^{10}$ .

Mixed convection flow refers to flow in which both natural and forced convection processes are important.

The relative importance of natural and forced convection effects is characterized by the ratio  $Gr / Re^2$ . If this ratio is significantly less than unity, natural convection effects can safely be neglected.

## 4.5.1.1.3.1. Solving Buoyancy-Driven Flows

The procedure for including buoyancy forces in the simulation of mixed or natural convection flows is described below.

To ensure heat transfer is included in the calculation, make sure you have enabled **Thermal effects** in the Fluid Flow template, or selected the **Physics type** to be **Thermal** in the **Physics Region** panel.

In the **Physics Options** panel:

- Under **Buoyancy**, select **On** to enable buoyancy modeling.
- For the **Gravity definition**, select **Create new** and enter the X, Y, and Z components of the gravitational acceleration in the **Gravity Definition** panel. You can enter them as constant values or use an expression that evaluates to a constant value.

To modify a previously created gravity definition, expand **Related Objects > Behaviors** and click the **Gravity Definition** link to open the panel where you can make changes to the X, Y, and Z components.

- Select the **Buoyancy model** and specify the **model details**. Note that the **Boussinesq model** will be applied to flows of constant density and the **Density difference** model will be applied to flows with varying density.
- The **Buoyancy reference location** is used in the calculation of the absolute pressure, as explained in [Buoyancy and Pressure](#) on page 254. The default **Automatic** location is (0,0,0), but the **User defined** option allows you to change this.
- If you are modeling turbulence, under **Turbulence Model > RANS Model** specify how you would like **Buoyancy effects** to play a role in the [turbulence production terms](#).
- Define the **boundary conditions**. The boundary pressures that you specify at pressure inlet and outlet boundaries are the modified pressures as explained in [Buoyancy and Pressure](#) on page 254.

### 4.5.1.1.3.1.1. Buoyancy Modeling

Buoyancy leads to an additional source term in the momentum equation. Depending on the material properties, there are two models available for this source term:

- Density difference** model: This is available when the material assignment for the flow region has a variable density, and uses a source term

$$\mathbf{S} = (\rho - \rho_{op})\mathbf{g}$$

where **g** is the gravity vector. The **Operating density**  $\rho_{op}$  should be selected to be close to the average fluid density.

- Boussinesq** model: This model assumes that the fluid density variations can be neglected for all terms in the equation except for the buoyancy source term. It is available when the material assignment for the flow region has a constant density and the simulation includes thermal effects, and uses the Boussinesq approximation for the source term

$$\mathbf{S} = \rho\beta(T - T_{op})\mathbf{g}$$

The **Operating temperature**  $T_{op}$  should be selected to be close to the average fluid temperature.

The **Thermal expansion coefficient** depends on the material, and is typically a function of temperature. For ideal gases that are modeled with a constant density approximation, select the thermal expansivity as  $1/T_{op}$ . In other situations, the expansion coefficient value should be obtained from a reference.

### 4.5.1.1.3.1.2. Buoyancy and Pressure

It is important to note that the solver does not directly compute the absolute pressure  $P_{abs}$ , but rather a modified pressure  $P$  which excludes both the operating pressure  $P_{op}$  and the hydrostatic pressure due to the buoyancy reference conditions:

$$P = P_{abs} - P_{op} - \rho_0 \mathbf{g}(\mathbf{r} - \mathbf{r}_{ref})$$

where  $\rho_0$  is either the operating density (if using the Density Difference model) or the constant fluid density (if using the Boussinesq model),  $\mathbf{r}$  is the local position coordinate, and  $\mathbf{r}_{ref}$  is the buoyancy reference location.

Any gauge pressure inputs (boundary conditions and initial conditions), as well as the pressure field displayed during post-processing, refer to this modified pressure field. Most notably, this means that the gauge pressure which is specified at a vertical pressure boundary should change if the operating temperature/density change. It is for this reason that choosing the operating temperature/density to be close to the average values is beneficial - it typically allows you to choose this specified pressure as a uniform value rather than a position-dependent expression.

Note also that the solver includes the hydrostatic pressure in the absolute pressure field. This field is available for post-processing and also used to compute any property which depends on pressure (such as fluid density for a compressible fluid).

## 4.5.1.2. Turbulence Modeling Options

A vast majority of industrial flows are turbulent. While conditions at boundaries may be steady, shear action near walls and other velocity profiles lead to transient pressure and velocity fluctuations transported by and affecting the mean path of the fluid. These "turbulent" fluctuations span a wide range of length and timescales. As a result, computational demands for exact solutions of industrial turbulent flows far exceed available computing resources.

Reynolds-Averaged Navier-Stokes (RANS) models provide an economic approach to solving these complex, turbulent flows by modeling turbulence as fluctuations within a steady, average velocity field. These models are not exact, but have been tuned to provide reasonable accuracy on appropriately scaled meshes.

The default [turbulence model](#) selection is appropriate for most cases and need not be modified, however, additional models and [options](#), [turbulence specification methods](#), and [boundary conditions](#) are available, depending on your simulation requirements. In the **Physics Options** panel, **Turbulence Model** is on by default and AIM provides alternative selections of RANS models if needed. Alternatively, laminar flow can be modeled by turning the **Turbulence Model** off.

### 4.5.1.2.1. Reynolds Averaged Navier-Stokes (RANS) Turbulence Models

RANS models solve additional transport equations for turbulence and introduce an eddy viscosity (also known as turbulent viscosity) to the simulation to mimic the effect of turbulence, a modeling approach that is suitable for many engineering applications.

# RANS Models

Option	Details
<b>k-<math>\omega</math> SST</b>	<p>A model that allows more accurate resolution of boundary layer behavior as the near wall resolution is refined. This model is useful also for free-stream flows, in particular, flows with adverse pressure gradients, flows around airfoils, etc. This model is appropriate for most industrial flows, and is therefore the default model. Where the near wall mesh resolution is coarse, the SST model assumes the boundary layer is fully developed. This is not always the case, especially under adverse pressure gradients when flows decelerate. Under these conditions, the SST model allows you to refine the near wall mesh and capture boundary layer instabilities which lead to separation. If this occurs, then you should switch to the k-<math>\omega</math> BSL model.</p> <p>k-<math>\omega</math> models are typically better in predicting flow separation, and this is one reason why the k-<math>\omega</math> SST model is among the most widely used models for aerodynamic flows.</p>
<b>k-<math>\epsilon</math> standard</b>	<p>A general industrial model that has historically been used heavily in fluid flow simulations. The wall treatment for this model assumes a fully developed boundary layer, which is not often valid. The <math>\omega</math>-based models offer a more consistent wall treatment.</p>
<b>k-<math>\omega</math> BSL</b>	<p>A general industrial model that offers a reasonable compromise of accuracy and robustness and is generally conservative on separation prediction.</p>
 <b>k-<math>\epsilon</math> realizable</b>	<p>A general industrial model that is a fair compromise between accuracy and robustness for many industrial applications. The k-<math>\omega</math>-based SST and BSL models are a better alternative and offer a more consistent wall treatment.</p> <p>Other, newer models, such as the k-<math>\omega</math> SST model, offer various improvements over the historic realizable k-<math>\epsilon</math> model and are normally favored. Should be used in combination with the <math>y+</math> insensitive two layer wall treatment. For cases where the flow separates under adverse pressure gradients from smooth surfaces (airfoils, etc.), k-<math>\epsilon</math> models are generally not recommended as they can lead to overly optimistic design evaluations. This is one reason why k-<math>\epsilon</math> models are not widely used in external aerodynamics.</p> <p>For cases where the flow separates under adverse pressure gradients from smooth surfaces (airfoils, etc.), k-<math>\epsilon</math> models are generally not recommended as they can lead to overly optimistic design evaluations.</p>
 <b>Spalart Allmaras</b>	<p>A computationally efficient one equation model designed for aerodynamics/aerospace applications.</p>

## 4.5.1.2.2. Turbulence Model Options

Additional sub-options are available for each [turbulence model](#).

# Boundary Layer Transition Settings

Specify whether to model the laminar/turbulent transitional region of the boundary layer by selecting the **Intermittency re theta** option. This option also enables the **Kato launder** production setting option. Applicable only for k- $\omega$  models.

# Curvature Correction for Swirling Flows

To model flows with strong swirl, either enable **Swirling flow** in the fluid flow related [templates](#) or enable **Curvature correction** in the **Physics Options** panel. Curvature correction sensitizes the standard eddy-viscosity models to the effects of streamline curvature and system rotation by modifying the turbulence

 **Curvature correction coefficient** allows you to influence the strength of the curvature correction if needed for a specific flow. The default value is 1, however, you can enter values between 0 and 2 to control the strength of the curvature correction.

# Buoyancy Effects Settings

Specify whether to include buoyancy effects on the turbulent dissipation. Available for any of the k- $\epsilon$  or k- $\omega$  models when buoyancy is enabled.

Option	Details
<b>Production only</b>	Include buoyancy effects <i>only</i> on the generation of turbulent kinetic energy.
<b>Full</b>	Also include buoyancy effects on turbulent dissipation.

# Near Wall Treatment Settings

Specify options that allow you to control the sensitivity near the wall boundary, since turbulent flows are significantly affected by the presence of walls and boundary layers.



These options are available when the **Show all properties** icon is active.

Option	Details
<b>RANS Model &gt; Near wall treatment</b>	<p><b>y+ insensitive:</b> Default near-wall treatment that uses a single wall law for the entire <math>y+</math>-range. Recommended when using any of the k-<math>\omega</math> models or the Spalart Allmaras model.</p> <p><b>y+ insensitive Menter Lechner:</b> Near-wall treatment that uses a single wall law for the entire <math>y+</math>-range. Recommended when using any of the k-<math>\epsilon</math> models.</p> <p><b>y+ insensitive two layer:</b> Blend of algebraic mixing length and epsilon equation near wall. y-insensitive. Limited robustness. Available for any of the k-<math>\epsilon</math> models.</p>

Option	Details
<b>RANS Model &gt; Near wall treatment &gt; Scalable wall function</b>	Wall functions that avoid the deterioration of the standard wall functions under mesh refinement. Prevents resolution of the viscous sublayer. Available for any of the k- $\epsilon$ models.
<b>RANS Model &gt; Near wall treatment &gt; Standard wall function</b>	No longer recommended due to grid sensitivity and robustness problems under grid refinement. Available for any of the k- $\epsilon$ models.

## Turbulence Production Settings

Specify options to control the production of turbulent kinetic energy for two-equation turbulence models.



These options are available when the **Show all properties** icon is active.

Option	Details
<b>Production settings &gt; Production</b>	<p>Useful in avoiding the build-up of turbulent kinetic energy in stagnation areas, and limit the production term in the standard two-equation turbulence models.</p> <p><b>Standard</b> does not affect the shear layer performance of the model, but avoids the stagnation point buildup in aerodynamic simulations. This is a production term based on the strain-rate square, and can lead to over-production of turbulence kinetic energy in stagnation regions.</p> <p><b>Production Limiter C pk lim</b> limits the production to a factor times dissipation. The factor has a default value of 10. The production limiter is activated by default for many models. It is less intrusive than Kato Launder and is therefore recommended for all two-equation models.</p> <p><b>Kato launder</b> accounts for very high levels of shear strain rate in the stagnation areas when the flow field is nearly irrotational, with a very small vorticity rate. This is a production term based on the product of the strain-rate and the vorticity. This avoids over-production of turbulent kinetic energy in stagnation regions.</p> <p>In addition to the <b>Production</b> settings, there are also the following coefficients to consider:</p> <p><b>C dok mu</b>: Coefficient that avoids model singularities for the ratio of epsilon and k. The default value should not be changed. Values should be very small.</p> <p><b>C pk mu</b>: Coefficient that helps the model recover from zero or low values of eddy-viscosity levels. The default value should not be changed. Values should be very small.</p>

# Model Constant Settings

Several turbulence models in AIM employ model constant settings meant for advanced numerical modeling. Changing the default values is not recommended.



These options are available when the **Show all properties** icon is active.

## Thermal Considerations for Turbulent Flows

For thermal fluid flow simulations, there may be viscous and thermal considerations to take into account for turbulent flows.



These options are available when the **Show all properties** icon is active.

Option	Details
<b>Energy Model &gt; Turbulence Energy Interaction &gt; Near wall treatment thermal effects</b>	<p>Account for near-wall viscous and thermal effects.</p> <p>In addition to enabling the <b>Near wall treatment thermal effects</b> settings, there are also the following coefficients to consider:</p> <ul style="list-style-type: none"> <li><b>Turbulent prandtl number:</b> The ratio of the momentum eddy diffusivity and the heat transfer eddy diffusivity.</li> <li><b>Wall prandtl number:</b> The ratio of the momentum eddy diffusivity and the heat transfer eddy diffusivity at the wall.</li> </ul>
<b>Energy Model &gt; Turbulence Energy Interaction &gt; Energy Near Wall Treatment</b>	<p>Account for smooth walls in the turbulent boundary layer.</p> <p><b>P function:</b> A function using the Jayatilleke formulation in the modeling of the conductive, convective, and viscous heating derivation for the law-of-the-wall.</p>

### 4.5.2. Structural Physics Options

#### Physics > Physics Options

The structural physics options include solid and global (mesh) modeling options. The [Solid Modeling Options](#) enable you to set the physics characteristics of your solid elements, while the [Global Modeling Options](#) enable you to specify whether meshes create midside nodes.

If you are running a [Topology Optimization](#) simulation, you can also [set Optimization Options](#).

### 4.5.2.1. Setting Solid Physics Options

#### Physics > Physics Options

To set the solid modeling options:

Under **Solid Modeling Options**, for **Brick integration type**, select an option for how your 3D solid brick elements are integrated. The default is **Program Controlled**, which allows ANSYS AIM to select the best integration method for your simulation, or you can explicitly choose from the following:

- **Reduced**: The reduced integration mode helps to prevent the undesirable locking of the volumetric mesh locking in nearly incompressible cases. However, an undesired hourglass mode might propagate in the model if there are not at least two layers of elements in each direction.
- **Full**: The full integration method does not cause hourglass mode, but can cause volumetric locking in nearly incompressible cases. This method is used primarily for linear analyses, or when the model has only one layer of elements in each direction.

## 4.5.2.2. Global Modeling Options

### Physics > Physics Options > Global Modeling Options

To define your global modeling options, specify whether meshes contain midside nodes. By default, AIM models elements based on the engineering intent, but you can also specify that midside nodes are **Dropped** (modeled as linear elements) or **Kept** (modeled as quadratic elements).

### 4.5.3. Electromagnetic Physics Options

#### Physics > Physics Options

For electromagnetic physics simulations, the **Physics Options** panel enables you to:

- Compute the model's [inductance matrix](#) for static magnetics designs.
- Compute the model's [impedance matrix](#) for frequency response designs.
- Specify the [ambient temperature](#) to be applied to the model.
- Adjust [conductor threshold settings](#).
- Calculate forces acting on specific bodies by adding [force computation](#) or [torque computation](#).
- Control [Eddy effect](#) and [Displacement current computations](#).
- Control [Core loss computations](#).

### 4.5.3.1. Compute Inductance

**Inductance** is available for Physics Regions with the physics type set to Electromagnetics and the Physics Calculation type set to Static.

#### Physics > Physics Options > Physics Options

Compute inductance to compute the model's inductance matrix, which represents the magnetic flux linkage between current loops; the resulting computations are available in the [Matrix Results](#). Selecting this option will increase solution time.

To compute inductance, in the **Physics Options** data panel, check **Compute inductance**.

### 4.5.3.2. Compute Impedance

#### Physics > Physics Options > Physics Options

**Impedance** is available for Physics Regions with the physics type set to Electromagnetics and the Physics Calculation type set to Frequency response.

Compute impedance to compute the impedance matrix, available in the [Matrix Results](#). Selecting this option will increase solution time.

To compute impedance, in the **Physics Options** data panel, check **Compute impedance**.

### 4.5.3.3. Setting Ambient Temperature

**Ambient temperature** is available for Physics Regions with the physics type set to Electromagnetics.

#### Physics > Physics Options > Ambient Temperature

The ambient temperature value is applied to all solid bodies to which no explicit **Body Temperature** condition has been applied. The effect of temperature on bodies composed of temperature-dependent material in the design is calculated by the solver for simulation results. For example, the **electrical conductivity** of a material (the ability of a material to conduct electric current) typically decreases as temperature increases.

To set the ambient temperature, in the **Physics Options** data panel, enter the desired value or expression in the **Ambient temperature** field. The default value is 22 degrees C. The value must be greater than absolute zero.

### 4.5.3.4. Conductor Threshold Settings

#### Physics > Physics Options > Physics Options

Conductor threshold settings tell the magnetics solvers how to deal with conductors and insulators. Materials with conductivity values above the  **Perfect conductor threshold** are treated as having infinite conductivity and surface current only. Materials with conductivity values below the  **Conductor threshold** are treated as insulators with no conductivity and no current carrying capability. Materials with conductivity values that fall between the two thresholds are treated as normal conductors that can carry current throughout the volume of the material.

To set the threshold options:

- For **Perfect conductor threshold**, the default setting is **Program controlled**, which instructs the solver to use the default conductor threshold conductivity value (1e+030 Siemens per meter); or you can select **User defined** to display the **Perfect conductor threshold value** property where you can specify a value or expression.
- For **Conductor threshold**, the default setting is **Program controlled**, which instructs the solver to use the conductor threshold conductivity value (1 Siemen per meter); or you can select **User defined** to display the **Conductor threshold value** property where you can specify a value or expression. The default value is 1 Siemen per meter.

### 4.5.3.5. Controlling Eddy Effect and Displacement Current Computations

**Eddy Effect Computation** is available for Physics Regions with the Physics type set to Electromagnetics and the Physics Calculation type set to Frequency response.

#### Physics > Physics Options > Eddy Effect Computation

Eddy Effect Computation allows you to control both Eddy effect and Displacement current computations for the conductors in an electromagnetics frequency response design. Eddy effect refers to current flow (eddy currents) induced by a time-varying magnetic field in conductors; while displacement current refers to currents due to a time-varying electric field. The effects of these currents on the solution tend to be more significant at higher frequencies.

To control Eddy effect and displacement current computations:

1. Select an option for how **Eddy effect computation** is applied. The default is **Program controlled**, which automatically selects all conductors for the computation, or you can explicitly choose from the following:

- **User defined**, which enables you to select the conductors you want to include.
- Note:** **Eddy effect computation** must only include conductors.
- **Off**, which instructs the solver not to perform Eddy current calculations.
2. Select an option for how **Displacement current computation** is applied. The default is **Program controlled**, which automatically selects all conductors for the computation, or you can explicitly choose from the following:
    - **User defined**, which enables you to select the objects you want to include.

**Note:**

    - Conductors selected for **Displacement current computation** must also be selected for **Eddy effect computation**.
    - You can choose to apply **Displacement current computation** alone for non-conductors you select.
    - **Off**, which instructs the solver not to perform displacement current calculations.

## 4.5.3.6. Controlling Core Loss Computation

**Core loss computation** is available for Physics Regions with the Physics type set to Electromagnetics and the Physics Calculation type set to Frequency response.

### Physics > Physics Options > Core Loss Computation

When a solid material used to form the core of devices such as inductors and transformers is subjected to a changing magnetic field, such as that resulting from a sinusoidal current flowing through the device, some of the power that would otherwise be transferred through the device is dissipated in the core as heat. Core loss computations are used to calculate these losses for electromagnetics frequency response designs.

To control core loss computation:

1. Select an option for how **Core loss computation** is applied. The default is **Program controlled**, which automatically selects all solid objects in the design whose material properties include core loss, or you can explicitly choose from the following:
  - **User defined**, which enables you to select the objects you want to include in the core loss computation.

**Note:** The objects you select must have core loss material properties defined.

  - **Off**, which instructs the solver not to perform core loss calculations.

## 4.5.3.7. Setting Torque Computation

**Torque Computation** is available for Physics Regions with the physics type set to Electromagnetics.

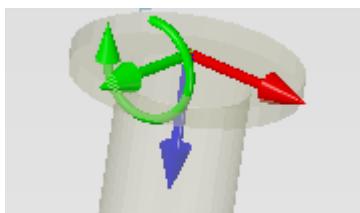
### Physics > Physics Options > Torque Computation

Add a **Torque Computation** physics option to calculate the torque on the assigned object. Depending on the use case, multiple magnetic force conditions can be assigned.

To add **Torque Computation**:

1. For the location, select bodies on which to calculate the torque.
2. For the **Torque type**, select **Virtual** or **Lorentz**. **Virtual** and **Lorentz** torque types both represent the average torque on a body (or group of bodies). Select **Virtual** if the **isotropic relative permeability** of the body's material is greater than 1.
3. Select the appropriate coordinate system **Reference frame** for calculating the torque from the drop-down list. **Global Reference Frame** is the default. Changing the reference frame requires resolving the design.

- Select the **Axis** of rotation for the torque calculation, and specify the **Axis direction**, which determines the direction of rotation around the axis of rotation. Changing the **Axis direction** requires resolving the design. A representation of the selected reference frame at the selected location is displayed. A circular arrow indicates the direction of rotation for the computed torque around the selected axis.



### 4.5.3.8. Setting Force Computation

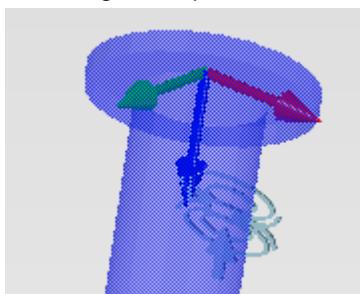
**Force Computation** is available for Physics Regions with the physics type set to Electromagnetics.

#### Physics > Physics Options > Force Computation

Add a **Force Computation** physics option to calculate the magnetic force on the assigned object. Depending on the use case, multiple magnetic force conditions can be assigned.

To add **Force Computation**:

- For the location, select bodies on which to calculate the magnetic force.
- For the **Force type**, select **Virtual** or **Lorentz**. **Virtual** and **Lorentz** force types both represent the average magnetic force on a body (or group of bodies). Select **Virtual** if the **isotropic relative permeability** of the body's material is greater than 1.
- Select the appropriate coordinate system **Reference frame** for calculating the force from the drop-down list. **Global Reference Frame** is the default. Changing the reference frame does not require resolving the design. A representation of the selected reference frame at the selected location is displayed.



### 4.5.4. Optimization Options

The optimization options enable you to modify your design goal and set the removal strategy in the case of multiple materials in the model.

Optimization options are available under **Physics Options**.

### 4.5.4.1. Specifying Optimization Options

To specify optimization options:

- Select the location.**

The default is the same as the location of the physics region

- Select one of the Design goals:**

- Select **Maximize strength** to maximize the stiffness and minimize compliance. This option reduces volume or mass but maintains the maximum stiffness possible for the remaining structure.
  - Select **Minimize response to free vibration** to maximize frequency. Then, specify a mode number (from 1 to 6) for the frequency to be optimized in **Mode to base frequency on** and specify the number of natural frequencies to solve for in **Total modes to extract**.
3. Select **Limit material removal** to maintain at least a percentage of the mass or volume of the model, then specify the maximum percentage to be removed under **Maximum removal**.
  4. Select the **Removal strategy for multiple materials**. You can minimize for product mass or volume.
  5. When minimizing the response to free vibration, select **Limit natural frequency** if you want to constrain a particular mode to a specified frequency range. Then:
    - a) Specify the mode number at which the frequency is to be constrained in **Mode number to limit**.
    - b) Specify the **Minimum frequency** for the selected mode number.
    - c) Specify the **Maximum frequency** for the selected mode number.
6.  Optionally, select **Preserve bonded contact locations** to ensure bonded contact locations are respected during optimization.  
If left unchecked, the optimizer might remove the contact in between bodies and work through it as if they were single body.

#### 4.5.5. Polymer Physics Options

##### Physics > Physics Options

AIM enables you to [set the physics characteristics](#) of your polymer extrusion or blow molding, such as the effects of inertia and gravity.

### 4.5.5.1. Setting Polymer Physics Options

#### Extrusion/Blow molding > Physics Options

To set the polymer extrusion or blow molding modeling options:

1. [Select the location](#).
2. Select from the following options:
  - To consider the effects of inertia for polymer extrusion, select **Include inertia effects**. For blow molding simulations, inertia effects are always included.
  - If the effects of gravity are important to your simulation, select **Include gravity effects**. For the **Gravity** definition, select **Create New** and enter the X, Y, and Z components of the gravitational acceleration in the **Gravity Definition** panel.
  - If the effects of thermal viscous dissipation are important to your simulation, select **Include viscous dissipation effects**.

## 4.6. Physics Conditions

Depending on the type of physics involved ([fluid flow](#), [polymer extrusion](#), [polymer blow molding](#), [structural](#), [steady-state thermal](#), [electric conduction](#), [electrostatic](#), or [electromagnetic](#), or a [topology optimization](#)), AIM provides several means to control the solution of the physics simulation.

AIM allows you to apply physical conditions to your model and specify them in your simulation.

## 4.6.1. Fluid Flow Conditions

Fluid flow conditions are used to define the problem to be solved based on their shape and location, which is provided by the geometry, as well as the conditions described at the boundaries. Volumetric heat and momentum sources provide a mechanism to account for the effect of features which are not otherwise resolved by the geometry and/or physics.

Boundary conditions should represent sensible engineering assumptions of the true conditions and therefore should be considered carefully when defining the analysis.

You can define your problem by explicitly setting the following boundary types:

Table 4.6.1.1. Boundary Types

Boundary Type	Description
Far Field	An open boundary used to model free-stream compressible flow at infinity.
Inlets	An open boundary where fluid is entering the physics region.
Outlets	An open boundary where fluid is exiting the physics region.
Openings	An open boundary where fluid may enter and/or exit the physics region.
Walls	A closed boundary across which fluid cannot flow into or out of the physics region.
Symmetry	A closed boundary at a plane of symmetry across which fluid is not expected to flow.

You can add the following volumetric sources to define the flow:

Source Type	Description
Heat	Additional volumetric sources in the thermal energy equation.
Momentum	A force per unit volume acting on the fluid(s). It is used to model directional losses in porous regions or momentum increases such as fans.

### 4.6.1.1. Far Field

Far field boundary conditions are used to model free-stream compressible flow at infinity. The actual boundary values are computed based on the far field specifications and local flow conditions. This boundary condition is applicable only when the density is calculated using the [ideal gas law](#).

To effectively approximate true infinite-extent conditions, you must place the far field boundary far enough from the object of interest. For example, in lifting airfoil calculations, it is not uncommon for the far field boundary to be a circle with a radius of 20 chord lengths.

#### 4.6.1.1.1. Specifying Pressure Far Fields

**Physics > Fluid Flow Conditions > Far Field**

Available only when the fluid's **Material Assignment > Material model** is set to **Ideal gas**.

The far field boundary condition models free-stream compressible flow at infinity using the Mach number, static pressure, flow direction, and temperature.

To specify the far field boundary conditions:

1. Select the location for which you want to specify the pressure far field.
2. Enter the **Gauge static pressure**. This is the pressure that is maintained at a fixed specified value over the far field boundary.
3. Enter the **Mach number**. The Mach number allows you to characterize the flow regime as subsonic, mixed (transonic), or supersonic.
4. Define the **Direction** of flow in **Cartesian** or **Cylindrical** coordinates. See [Coordinate Systems](#) on page 77 for more information.
5. Under **Energy specification**, specify the **temperature** at the far field boundary.
6. Additionally, if you want to use a reference frame other than **Global Reference Frame**, in the **Far Field** panel click the **Reference frame** dropdown arrow and then choose a reference frame, or click **Create New** to define a reference frame. For more information, see [Understanding Reference Frames](#) on page 73.
7. In the **Far Field** panel you can modify turbulence levels at the far field boundary by selecting **Turbulence specification** and then selecting a method. See [Turbulence Specification Methods](#) on page 286.

## 4.6.1.2. Heat Source

Volumetric sources provide a mechanism to account for the effect of features which are not otherwise resolved by the geometry and/or physics. For example, a heat source might be used to account for heat generated by a circuit board, resistance heater, or fire, without explicitly including their geometrical and physical details. For details on how to set up heat sources, see [Specifying Heat Sources](#) on page 266.

### 4.6.1.2.1. Specifying Heat Sources

#### Physics > Fluid Flow Conditions > Heat Source

To add a heat source:

1. For the **location**, select one or more bodies in your physics region where you would like to include the heat source.
2. If you want to use a local **Reference frame** which differs from the **Global Reference Frame**, [select or define the reference frame](#).
3. Specify the heat source in one of two ways:
  - **Source per unit volume** can be input as either a constant or a position-dependent expression. Position-dependent coordinates are with respect to the local reference frame.
  - **Total source** is the total heat source.

**Note:** A negative heat source represents a heat sink.

## 4.6.1.3. Inlets

Inlets define conditions where flow is expected to enter the solution domain.

To determine the type of inlet you should apply depends on the available flow information.

Upstream conditions from the inlet	Flow specification	Additional information
A plenum or device delivering flow.	<a href="#">Pressure inlet</a>	Solves the velocity profile and mass flow. Gauge total pressure is the absolute total pressure minus the operating pressure.
A positive displacement device delivering a certain mass or volume flow rate.	<a href="#">Mass flow inlet</a>	Solves for pressure. Velocity is derived based on local density. The <a href="#">expression</a> cannot vary spatially. Local direction can be specified.
	<a href="#">Velocity inlet</a>	Solves for pressure. Mass flow is derived based on the local density. Local direction or velocity components can be specified.

## 4.6.1.3.1. Specifying Velocity Inlets

### Physics > Fluid Flow Conditions > Inlet

To specify velocity inlets:

1. [Select the location](#) in your geometry for which you want to specify the inlet boundary condition.
2. For **Flow specification**, select **Velocity**.
3. For the **Magnitude**, enter the velocity at which fluid is flowing into the domain.
4. Define the **Direction** of fluid flow. By default the direction is **Normal to boundary**, however, you can specify the flow in **Cartesian** or **Cylindrical** coordinates. See [Understanding Reference Frames](#) on page 73 for more information.
5. If your simulation includes a thermal analysis, make sure to specify the inlet **Temperature**.
6. Additionally, in the **Inlet** panel, you can make modifications to the following settings:

If you want to...	Then choose...	For more information:
Use a reference frame other than <b>Global Reference Frame</b>	<a href="#">Reference frame</a> and <a href="#">select or define the reference frame</a> .	See <a href="#">Understanding Reference Frames</a> on page 73.
Express the velocity by specifying its directional components	<a href="#">Define by</a> and select <b>Directional components</b> . Specify the <b>Coordinate type</b> to be <b>Cartesian</b> or <b>Cylindrical</b> .	See <a href="#">Understanding Reference Frames</a> on page 73.
Modify turbulence levels at your inlet	<a href="#">Turbulence specification</a> and select a method.	See <a href="#">Turbulence Specification Methods</a> on page 286.
Specify total temperature (also known as the stagnation temperature) rather than the static temperature	<a href="#">Total temperature</a> under <b>Energy specification</b>	The total temperature represents the temperature of the fluid in a plenum at rest upstream. The local static temperature will be lower, depending on the velocity of the fluid.

## 4.6.1.3.2. Specifying Pressure Inlets

### Physics > Fluid Flow Conditions > Inlet

To specify pressure inlets:

1. **Select the location** in your geometry for which you want to specify the inlet boundary condition.
2. For **Flow specification**, select **Pressure**.
3. Enter the **Gauge total pressure**, which represents the pressure obtained by bringing the flow isentropically to rest (eg, upstream plenum pressure). It is measured relative to the operating pressure (i.e.  $P_{T_{gauge}} = P_{T_{absolute}} - P_{T_{operating}}$ ).
4. Define the **Direction** of flow. By default the direction is **Normal to boundary**, however, you can specify the flow in **Cartesian**, **Cylindrical**, or **Cylindrical with tangential velocity** coordinates. For cylindrical components, the radial velocity component points radially outward from the Z axis, the axial velocity component is in the direction of the Z axis, and the tangential component is in the positive circumferential direction; all three component magnitudes scale based on mass flow. If the Tangential Velocity is specified, the tangential velocity component is given directly and only the radial and axial components scale with mass flow. See [Understanding Reference Frames](#) on page 73 for more information.
5. If your simulation includes a thermal analysis, make sure to specify the inlet **Total temperature**. Also known as the stagnation temperature, it represents the temperature of the fluid in a plenum at rest upstream.
6. Additionally, in the **Inlet** panel, you can make modifications to the following settings:

If you want to...	Then choose...	For more information:
Use a reference frame other than <b>Global Reference Frame</b>	<b>Reference frame</b> and <a href="#">select or define the reference frame</a> .	See <a href="#">Understanding Reference Frames</a> on page 73.
Account for reverse flow	<b>Yes</b> under <b>Allow reverse flow</b> . Select  <b>Total pressure</b> for the <b>Backflow pressure specification</b> .	If the specified inlet pressure is less than the pressure at the neighboring cell, it cannot push fluid in. In such cases, the static pressure is held constant at the same value as the specified total pressure, allowing the fluid to flow out the inlet. If the specified pressure can overcome the back pressure, inflow will be restored.
Modify turbulence levels at your inlet	<b>Turbulence specification</b> and select a method.	See <a href="#">Turbulence Specification Methods</a> on page 286.

## 4.6.1.3.3. Specifying Mass Flow Inlets

### Physics > Fluid Flow Conditions > Inlet

To specify mass flow inlets:

1. **Select the location** in your geometry for which you want to specify the inlet boundary condition.
2. For **Flow specification**, select **Mass flow**.
3. Enter the **Mass flow rate**. This is where fluid is delivered by a positive displacement device upstream and is capable of overcoming the back pressure of the system being modeled.
4. Define the **Direction** of flow. By default the direction is **Normal to boundary**, however, you can specify the flow in **Cartesian**, **Cylindrical**, or **Cylindrical with tangential velocity** coordinates. For cylindrical

components, the radial velocity component points radially outward from the Z axis, the axial velocity component is in the direction of the Z axis, and the tangential component is in the positive circumferential direction; all three component magnitudes scale based on mass flow. If the Tangential Velocity is specified, the tangential velocity component is given directly and only the radial and axial components scale with mass flow. See [Understanding Reference Frames](#) on page 73 for more information.

5. If your simulation includes a thermal analysis, make sure to specify the inlet **Temperature**.
6. Additionally, in the **Inlet** panel, you can make modifications to the following settings:

If you want to...	Then choose...	For more information:
Use a reference frame other than <b>Global Reference Frame</b>	<b>Reference frame</b> and <a href="#">select or define the reference frame</a> .	See <a href="#">Understanding Reference Frames</a> on page 73.
Specify the tangential velocity	<b>Direction</b> to be <b>Cylindrical with Tangential Velocity</b> .	See <a href="#">Understanding Reference Frames</a> on page 73. If the Tangential Velocity is specified, the tangential velocity component is given directly and only the radial and axial components scale with mass flow.
Modify turbulence levels at your inlet	<b>Turbulence specification</b> and select a method.	See <a href="#">Turbulence Specification Methods</a> on page 286.
Specify total temperature (also known as the stagnation temperature) rather than the static temperature	<b>Total temperature under Energy specification</b>	The total temperature represents the temperature of the fluid in a plenum at rest upstream. The local static temperature will be lower, depending on the velocity of the fluid.

## 4.6.1.4. Momentum Source

Volumetric sources provide a mechanism to account for the effect of features which are not otherwise resolved by the geometry and/or physics. For example, a momentum source might be used to account for the effect of a [fan](#) or screen without explicitly modeling its geometry.

### 4.6.1.4.1. Specifying Momentum Sources

#### Physics > Fluid Flow Conditions > Momentum Source

To add a momentum source:

1. For the [location](#), select one or more bodies in your physics region where you would like to include the momentum source.
2. If you want to use a local **Reference frame** which differs from the **Global Reference Frame**, [select or define the reference frame](#).
3. Specify the momentum source components in either Cartesian or cylindrical coordinates, with respect to the selected reference frame. The source can be input as either a constant or a position-dependent expression. Position-dependent coordinates are with respect to the local reference frame.
  - **Cartesian**: enter the **X**, **Y**, and **Z** components of the momentum source.
  - **Cylindrical**: enter the **p**, **φ**, and **Z** (radial, angular, and axial) components of the momentum source. The axial direction is aligned with the z-axis of the local reference frame.

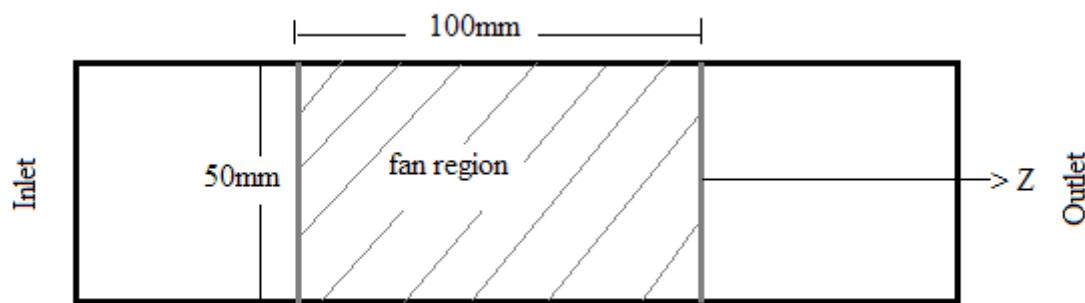
**Note:** A negative momentum source represents a momentum sink. If you know that the momentum source will lead to a pressure rise or drop  $\Delta P$  of a particular magnitude as the flow passes through the body length L, you would choose the momentum source in the relevant direction to be  $\Delta P/L$ .

## 4.6.1.4.2. Fan Modeling Using Momentum Source Terms

Many applications such as ventilation and cooling systems involve internal forced recirculation of air using blowers or fans. To model such systems, you will need to calculate the momentum source value from either a known mass flow rate pushed by the fan, or a pressure rise across the fan.

In general, fans can be represented as a cylindrical region. The fan region should be treated as a separate region from the rest of the domain to enable the application of a momentum source explicitly for the fan.

Consider the example shown in the schematic below, where the fan region is represented by a cylinder of 50mm in diameter and a length of 100mm placed inside a larger containment. The two methods for calculating the momentum source are described below.



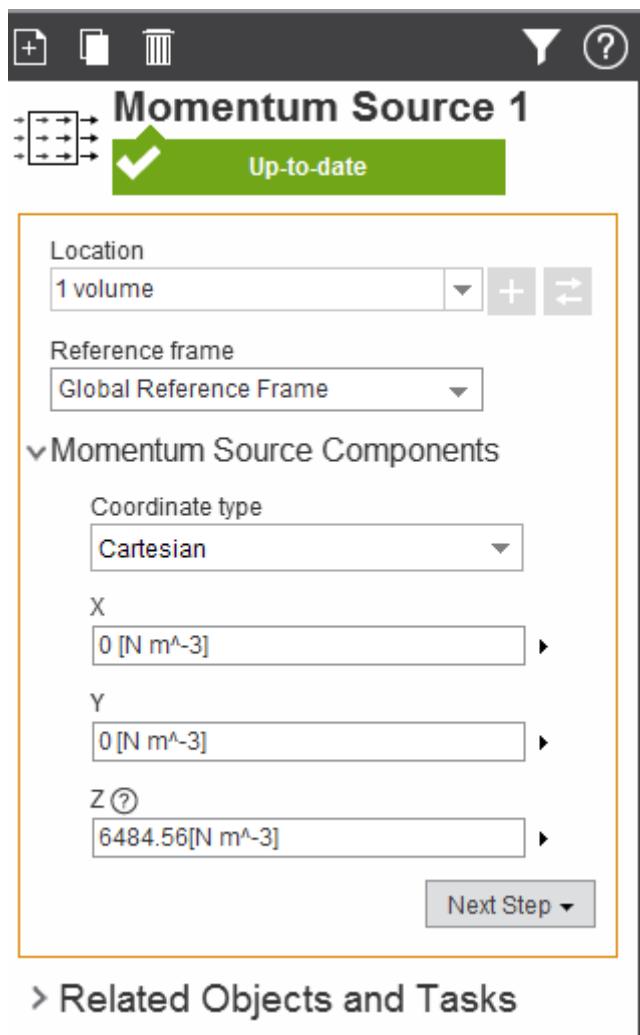
### Method 1: Mass flow rate pushed by the fan is known

If the mass flow rate or velocity imparted by the fan is known, the momentum source can be calculated as

$$S_{mom} [N/m^3] = \frac{\text{Mass flow rate } [kg/s] \times \text{Average velocity } [m/s]}{\text{Volume of the region } [m^3]}$$

The mass flow rate of the air pushed by the fan is known to be 0.05kg/s in the +Z direction. The average velocity of flow through the cylindrical region, considering an air density of  $1 \text{ kg/m}^3$ , is obtained as  $v_z = \dot{m}/\rho A = 25.46 \text{ m/s}$  where  $\dot{m}$  is the mass flow rate,  $\rho$  is the density of air, and  $A$  is the cross-section area of the cylinder. The volume of the fan region is  $\pi D^2 L / 4$ , and the momentum source in the Z direction  $S_{mom,Z}$  is calculated as  $6484.56 \text{ N/m}^3$ .

The calculated momentum source term can then be used in the setup as shown below:



## › Related Objects and Tasks

### Method 2: Pressure rise across the fan is known

If the pressure rise across the fan is known and assuming a constant flow cross-sectional area, then the momentum source can be calculated as

$$S_{mom}[N/m^3] = \frac{\text{Pressure rise}[N/m^2]}{\text{Length of the region}[m]}$$

## 4.6.1.5. Openings

An opening is used where the flow may enter and/or exit through a boundary.

To determine the type of opening you should apply depends on the available flow information.

Upstream conditions from the inlet	Flow specification	Additional information
Velocity profile and boundary shape result in flow in and out of the boundary in different areas.	Velocity opening	Solves for pressure. Mass flow is derived based on the local density. Local direction or velocity components can be specified.

Upstream conditions from the inlet	Flow specification	Additional information
An open window where flow is entering and/or exiting.	Pressure opening: Gauge entrainment pressure (Constant value)	The specified pressure will be applied over the entire face. The velocity is a function of the solution. The direction is derived from neighboring cells.
	Pressure opening: Gauge entrainment pressure (Varying spatially)	The specified pressure is applied by evaluating the expression at each position.

## 4.6.1.5.1. Specifying Velocity Openings

### Physics > Fluid Flow Conditions > Opening

To specify velocity openings:

1. Select the location in your geometry for which you want to specify the opening boundary condition.
2. For **Flow specification**, select **Velocity**.
3. For the **Magnitude**, enter the velocity at which fluid is flowing through the domain.
4. Define the **Direction** of fluid flow. You can specify the flow in **Cartesian** or **Cylindrical** coordinates. See [Understanding Reference Frames](#) on page 73 for more information.
5. If your simulation includes a thermal analysis, make sure to specify the **Temperature** at the opening.
6. Additionally, in the **Opening** panel, you can make modifications to the following settings:

If you want to...	Then choose...	For more information:
Use a reference frame other than <b>Global Reference Frame</b>	<b>Reference frame</b> and select or define the reference frame.	See <a href="#">Understanding Reference Frames</a> on page 73.
Express the velocity by specifying a magnitude and defining the direction of flow	<b>Define by</b> and select <b>Magnitude and direction</b> . Specify <b>Direction</b> to be <b>Cartesian</b> or <b>Cylindrical</b> .	See <a href="#">Understanding Reference Frames</a> on page 73.
Express the velocity by specifying its directional components	<b>Define by</b> and select <b>Directional components</b> . Specify the <b>Coordinate type</b> to be <b>Cartesian</b> or <b>Cylindrical</b> .	See <a href="#">Understanding Reference Frames</a> on page 73.
Modify turbulence levels at the opening	<b>Turbulence specification</b> and select a method.	See <a href="#">Turbulence Specification Methods</a> on page 286.
Specify total temperature (also known as the stagnation temperature) rather than the static temperature	<b>Total temperature under Energy specification</b> .	The total temperature represents the temperature of the fluid in a plenum at rest upstream. The local static temperature will be lower, depending on the velocity of the fluid.

## 4.6.1.5.2. Specifying Pressure Openings

### Physics > Fluid Flow Conditions > Opening

To specify pressure openings:

1. Select the location in your geometry for which you want to specify the opening boundary condition.
2. For **Flow specification**, select **Pressure**.
3. Enter the **Gauge entrainment pressure**. When used in external flow where the boundary is open to the air and the velocity is nearly tangent, you will apply a gauge atmospheric pressure.
4. If your simulation includes a thermal analysis, make sure to specify the opening **Entrainment temperature**. When used in external flow where the boundary is open to the air and the velocity is nearly tangent, you will apply an atmospheric temperature.
5. Additionally, in the **Opening** panel, you can make modifications to the following settings:

If you want to...	Then choose...	For more information:
Use a reference frame other than <b>Global Reference Frame</b>	<b>Reference frame</b> and select or define the reference frame.	See <a href="#">Understanding Reference Frames</a> on page 73.
Modify turbulence levels at the opening	<b>Turbulence specification</b> and select a method.	See <a href="#">Turbulence Specification Methods</a> on page 286.

## 4.6.1.6. Outlets

Outlets are boundary conditions that permit flow to exit the solution domain.

To determine the type of outlet you should apply depends on the available flow information.

Downstream conditions from the <b>Flow specification outlet</b>	Additional information
Fluid is exiting the domain into a large volume where the static pressure is known.	<b>Pressure outlet</b> : Static pressure (Constant value) A constant static pressure will be applied over the entire face. The velocity and mass flow are functions of the solution.
The pressure profile is known downstream of the outlet.	<b>Pressure outlet</b> : Static pressure (Varying spatially) An <b>expression</b> can be provided as a function of position to describe how the pressure varies.
The real geometry extends beyond the outlet location, but is not modeled.	<b>Pressure outlet</b> : Static pressure (Pressure averaging) The solver allows the pressure to vary naturally over the face, while maintaining a constant area averaged pressure.
If the velocity profile is known at the outlet or a target velocity profile is desired (usually used in conjunction with pressure inlet)	<b>Velocity outlet</b> The velocity is constrained over the face according to the <b>expression</b> provided. Pressure at the outlet will be a function of the solution, however the upstream velocity and pressure profiles will be strongly influenced by the downstream velocity.

Downstream conditions from the <b>Flow specification outlet</b>	Additional information
<p>When the mass flow through the location is known or expected and at least one pressure boundary has been specified.</p>	<p><b>Mass flow outlet</b></p> <p>The Mass flow update is defined using:</p> <p>Scale mass flow: the pressure and velocity are allowed to vary naturally, while maintaining a net mass flow.</p> <p>Uniform flux: forces a constant mass flux over the face, the velocity will also be constant over the face.</p>

## 4.6.1.6.1. Specifying Velocity Outlets

### Physics > Fluid Flow Conditions > Outlet

To specify velocity outlets:

1. **Select the location** in your geometry for which you want to specify the outlet boundary condition.
2. For **Flow specification**, select **Velocity**.
3. For the **Magnitude**, enter the velocity at which fluid is flowing out of the domain.
4. Define the **Direction** of fluid flow. By default the direction is **Normal to boundary**, however, you can specify the flow in **Cartesian** or **Cylindrical** coordinates. See [Understanding Reference Frames](#) on page 73 for more information.
5. Additionally, in the **Outlet** panel, you can make modifications to the following settings:

If you want to...	Then choose...	For more information:
Use a reference frame other than <b>Global Reference Frame</b>	<b>Reference frame</b> and select or define the reference frame.	See <a href="#">Understanding Reference Frames</a> on page 73.
Express the velocity by specifying its directional components	<b>Define by</b> and select <b>Directional components</b> . Specify the <b>Coordinate type</b> to be <b>Cartesian</b> or <b>Cylindrical</b> .	See <a href="#">Understanding Reference Frames</a> on page 73.

## 4.6.1.6.2. Specifying Pressure Outlets

### Physics > Fluid Flow Conditions > Outlet

To specify pressure outlets:

1. **Select the location** in your geometry for which you want to specify the outlet boundary condition.
2. For **Flow specification**, select **Pressure**.
3. Enter the **Gauge static pressure**. This is the pressure that is maintained at a fixed specified value over the outlet boundary.
4. Additionally, in the **Outlet** panel, you can make modifications to the following settings:

If you want to...	Then choose...	For more information:
Use a reference frame other than <b>Global Reference Frame</b>	<b>Reference frame</b> and select or define the reference frame.	See <a href="#">Understanding Reference Frames</a> on page 73.
Allow the pressure at the boundary to vary radially consistent with swirling flow, as in turbomachinery or swirling flow in a pipe	<b>Radial equilibrium</b> and enter the <b>Gauge static pressure at minimum radius</b> . Under <b>Number of Bands</b> , a default of 40 is applied as the maximum number. Selecting <b>Automatic</b> produces a solver generated number of circumferential bands.	The specified gauge pressure applies only to the position of minimum cylindrical radius at the boundary (about the z axis of the boundary reference frame).  The averaged static pressure is constrained within radial circumferential bands defined by either a local rotation axis or the domain rotation axis. The band-averaged pressure satisfies radial equilibrium between the radial pressure gradient and the centrifugal force calculated using the band-averaged density and circumferential velocity.
Allow the pressure to vary naturally over the face, while maintaining a constant area averaged pressure	<b>Pressure averaging</b> and select <b>Average</b> . Apply a <b>Pressure profile blend</b> .	A blend factor of 0 gives the natural pressure profile (zero shear). A value of 1 results in a constant pressure applied over the face. If the solution is unstable (due to reverse flow), a value 0.05 will generally suffice.
Allow backflow at the outlet	<b>Yes</b> under <b>Allow reverse flow</b> . Select  <b>Total Pressure</b> for the <b>Backflow pressure specification</b> . Specify the <b>Backflow direction</b> as being <b>From neighboring cell</b> , <b>Normal to boundary</b> , <b>Cartesian</b> , or <b>Cylindrical</b> .	If the pressure at a localized area at the outlet is less than the specified outlet pressure, fluid will enter through the boundary face at this location. By default, artificial walls will be placed at these locations to prevent reverse flow. If reverse flow is allowed, a pressure will be applied at these locations.  Applying <b>Total pressure</b> (default) will allow the static pressure to drop as the inflow velocity increases, maintaining a constant total pressure. The <b>Entrainment pressure</b> applies a similar condition, however, only the normal component of the inflow velocity is taken into consideration.  If reverse flow is permitted, a portion of the outlet is effectively acting as an <a href="#">inlet</a> and additional boundary condition information may be required, such as the temperature or <a href="#">turbulence</a> .
Specify total temperature (also known as the stagnation temperature) rather than the static temperature	<b>Total temperature</b> under <b>Energy specification</b>	The total temperature represents the temperature of the fluid in a plenum at rest upstream. The local static temperature will be lower, depending on the velocity of the fluid.

### 4.6.1.6.3. Specifying Mass Flow Outlets

#### Physics > Fluid Flow Conditions > Outlet

To specify mass flow outlets:

1. Select the location in your geometry for which you want to specify the outlet boundary condition.
2. For Flow specification, select Mass flow.
3. Enter the Mass flow rate if you know the mass flow rate through your system.
4. Additionally, in the Outlet panel, you can make modifications to the following settings:

If you want to...	Then choose...	For more information:
Use a reference frame other than <b>Global Reference Frame</b>	<b>Reference frame</b> and select or define the reference frame.	See <a href="#">Understanding Reference Frames</a> on page 73.
Define the mass flow distribution	<b>Mass flow update</b> and select <b>Scale mass flows</b> or <b>Uniform flux</b> .	<b>Scale mass flow</b> : the pressure and velocity are allowed to vary naturally, while maintaining a net mass flow. <b>Uniform flux</b> : forces a constant mass flux over the face.

## 4.6.1.7. Particle Injections

[Particle injections](#) define the initial conditions of the individual particles. The initial position, velocity, size, and temperature of the particle are the physical properties of the discrete phase that are used to initiate the trajectory and heat transfer calculation.

Defining how a particle behaves when it reaches a physical boundary will determine the fate of the trajectory at that boundary. Particles experience elastic collisions at symmetry boundaries and may be reflected or escape through all other boundaries.

### 4.6.1.7.1. Specifying Particle Injections

#### Physics > Fluid Flow Conditions > Particle Injection

To define a **Particle Injection** at any of the boundaries:

1. For the **Location**, select a surface in your geometry where you would like to apply particle injection.
2. Select the **Particle** you would like to inject, or click **Create New** to create and inject a new particle that is not available in the list.
3. Enter the **Mass flow rate** of the injection.
4. Enter or select a **Diameter specification** model.

Option	Description
<b>Constant diameter</b>	Enter a constant specified particle diameter for all particles.
<b>Uniform distribution</b>	A uniform probability distribution between the <b>Minimum diameter</b> and <b>Maximum diameter</b> .

Option	Description
<b>Normal distribution</b>	Uses a normal distribution of particle diameters centered about a specified <b>Mean diameter</b> . The width of the normal distribution is determined by the specified <b>Diameter standard deviation</b> . The <b>Minimum diameter</b> and <b>Maximum diameter</b> are used to clip the normal distribution. Specifying the minimum/maximum diameters are optional. If you do not set those values, the minimum particle size injected will be 0.1 $\mu\text{m}$ .
<b>Rosin Rammler distribution</b>	Requires a <b>Mean diameter</b> and <b>Rosin Rammler power</b> . You can control the <b>Minimum diameter</b> and <b>Maximum diameter</b> . See <a href="#">Rosin Rammler Distribution</a> on page 277 for more information.

5. Specify the **Velocity** of the particle. By default, **Magnitude and direction** is specified. The normal speed, cartesian velocity components, and local velocity can be set. **Use local velocity** causes the particles to be injected at the local velocity of the continuous phase. It can be used at an inflow boundary that is specified using a mass flow or pressure option for the continuous phase.
6. Define the injection **Distribution** to be:
  - **Random** to produce random injections over the entire boundary. Specify the **Number of particle streams** injected at a boundary condition or particle injection region.
 

**Note:** When a single particle injection region is located on multiple non-adjacent faces, the random distribution can lead to all tracks starting from one face. To ensure a better distribution, use separate particle injections.
  - **Per element face** to inject particles at the center of each element face. Enter the **Particle streams per element face** to specify how many particles should be released from each face on the boundary.
7. For a simulation that includes thermal conditions, specify the **Temperature** of the injected particles.

### 4.6.1.7.1.1. Rosin Rammler Distribution

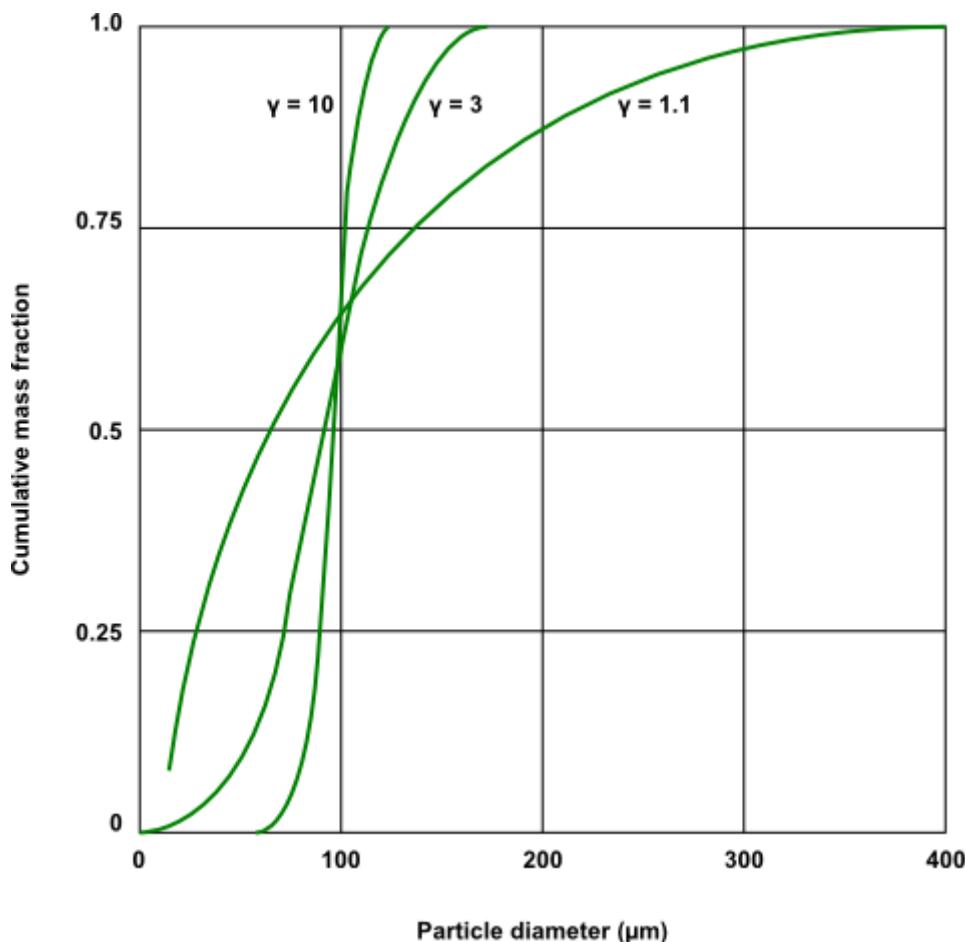
A Rosin Rammler distribution can be used to determine the distribution of the mass flow among particle sizes. It was developed for pulverized solid fuels but may also be applicable to certain sprays.

The mass fraction,  $R$ , above a given particle diameter,  $d$ , is calculated from:

$$R = \exp[-(d/d_e)^\gamma]$$

where  $d_e$  is a measure of the fineness and is equal to the diameter at which  $R$  is  $1/e$  or 0.368. The **Rosin Rammler power**,  $\gamma$ , is a measure of dispersion of particle sizes, a lower value indicating a wider dispersion. A typical value of  $\gamma$  for pulverized fuels is 1 to 1.3 and for sprays is 1.5 to 3.0. For a nearly monodisperse size distribution,  $\gamma$  may have a value of 10 to 20. Given  $d_e$  and  $\gamma$ , it is possible to calculate the particle size distribution. Examples of Rosin Rammler distributions for  $d_e = 10^{-4}$  and various values of  $\gamma$  are shown in the figure below.

**Figure 4.6.1.7.1.1.1. Particle Size Distributions for Various Rosin Rammler Powers**



## 4.6.1.7.2. Particle Behavior at Boundaries

### Physics > Fluid Flow Conditions > Particle Behavior

Defining how a particle reacts when it reaches a boundary will determine the fate of the trajectory at that boundary. You can define the particle behavior in any of the fluid flow boundary conditions, except for symmetry, in one of the following ways:

- **Reflection:** The particle may be reflected via an elastic or inelastic collision.
- **Escape:** The particle may escape through the boundary. The particle is removed from the calculation at the point where it encounters the boundary.

**Note:** At symmetry boundary conditions, the particle will undergo perfectly elastic collisions. Particle interaction with the interface between a fluid, thermal and a solid region cannot be specified.

#### Particle Reflection:

1. At, for example, a wall boundary the particle will likely be reflected. If the particle loses momentum upon reflection, you can model this behavior by modifying the default settings under **Momentum Change**. The table below provides information about each of the settings:

Option	Description
<b>Smooth/Rough wall</b>	Select the surface to be a <b>Smooth wall</b> or <b>Rough wall</b> . A rough wall can capture more realistic behavior of dispersed flows in confined geometries, such as cyclones and pipe systems.

Option	Description
 <b>Normal restitution coefficient</b>	Specifies the fraction of particle momentum normal to the wall retained by the particle following a reflection.
 <b>Parallel restitution coefficient</b>	Specifies the fraction of particle momentum parallel to the wall retained by the particle following a reflection.
 <b>Average roughness height</b>	Specifies the mean value for the roughness height. The default values of the wall roughness parameters are adequate for most situations. The roughness parameters may need to be increased for very large particles to see an effect on the trajectories.
 <b>Std dev of roughness height</b>	Specifies the standard deviation of the roughness height. The default values of the wall roughness parameters are adequate for most situations. The roughness parameters may need to be increased for very large particles to see an effect on the trajectories.
 <b>Average roughness width</b>	Specifies the mean value for the roughness width. The default values of the wall roughness parameters are adequate for most situations. The roughness parameters may need to be increased for very large particles to see an effect on the trajectories.

2. If after reflection the particle is trapped, you can model this behavior by enabling **Trapping** and modifying the default settings. Note that the particle will be 'trapped' if it hits the boundary when the **Minimum/Maximum particle velocity** and **Minimum/Maximum impact angle** are within the specified ranges. The mass and energy of the particle is not transferred to the continuous phase as the particle 'sticks' to the wall. **Trapping** will have an effect on the calculated particle force on that boundary since a particle that sticks to a wall will impart a different amount of momentum than one that reflects.
3. If after reflection material is removed from a wall surface due to micromechanical deformation or cracking of the wall's surface, then **Erosion** can be applied to model this behavior.
  - If erosion is due to abrasion, select **Abrasive** as the erosion model. The default model parameters (accessible by clicking  to show all properties) are based on the [Oka Erosion Model](#) on page 279 and have been tuned for sand erosion of carbon steel, but may need adjustment for other systems.
  - If erosion is due to other degradation processes, select **User specified** and enter the **Erosion rate**. This is the specified rate multiplied by the mass flow rate of the particles. The specified rate can be defined using expressions.

### 4.6.1.7.2.1. Oka Erosion Model

In the erosion model by Oka, the erosion rate  $E$  is determined as:

$$E = E_{90} \left( \frac{V}{V_{ref}} \right)^{k_2} \left( \frac{d}{d_{ref}} \right)^{k_3} f(\gamma)$$

where:

$E_{90}$  = reference erosion ratio at 90° impact angle

$V$  = particle impact velocity

$V_{ref}$  = reference velocity

$d$  = particle diameter

$d_{ref}$  = particle reference diameter

$k_2, k_3$  = velocity and diameter exponents, respectively

$f(\gamma)$  = function of impact angle

The impact angle dependence of the erosion rate is given by:

$$f(\gamma) = (\sin\gamma)^{n_1} (1 + H_v(1 - \sin\gamma))^{n_2}$$

where

$\gamma$  = wall impact angle (rad)

$H_v$  = wall material Vickers hardness (GPa)

$n_1, n_2$  = angle function constants

Table 4.6.1.7.2.1.1. Example of the Oka Erosion Model Constants for Sand Erosion of Carbon Steel

Constant	Sand - Steel Value
$E_{90}$	$6.154 \times 10^{-4}$
$H_v$	1.8 [GPa]
$n_1, n_2$	0.8 and 1.3, respectively
$k_2, k_3$	2.35 and 0.19, respectively
$d_{ref}$	326 [ $\mu\text{m}$ ]
$V_{ref}$	104 [m/s]

*Y.I. Oka, T. Yoshida, "Practical Estimation of Erosion Damage Caused by Solid Particle Impact, Part 2: Mechanical Properties of Materials Directly Associated With Erosion Damage", Wear, 2005, pp.102-109*

## 4.6.1.8. Porous Media

Porous media can be used to model a wide variety of applications, such as fluid flow through packed beds, filter papers, perforated plates, flow distributors, and tube banks. The porous medium is modeled as a momentum loss, the magnitude of which involves resistance coefficients.

### 4.6.1.8.1. Specifying Porous Media

Physics > Fluid Flow Conditions > Porous Medium

To set up a porous medium:

1. Select a body or bodies for the **Location** in your geometry for which you want to specify porous conditions.
2. If you want to use a reference frame other than **Global Reference Frame**, click the **Reference frame** drop-down arrow and then choose a reference frame, or click **Create New** to define a reference frame.
3. Select **Isotropic** if the porous loss coefficients are the same in all directions, and **Orthotropic** if they are direction-dependent. Porous media such as packed beds are typically isotropic, while filters and perforated plates are often orthotropic because the porous medium has a direction-dependent orientation.
4. For **Resistance velocity type** select whether the resistance coefficients multiply the **Superficial** or **Physical** velocity. By default, **Superficial** is selected and is more commonly used, since experimental data is usually obtained using flow sensors which are outside the porous medium. Choose **Physical** velocity if the coefficients were instead correlated using physical velocity. In this case, you must also specify the **Volume porosity**, which is the local ratio of fluid volume to physical volume, with 1 representing a fully open medium. (Note that Superficial velocity = Volume porosity \* Physical velocity.)
5. Under **Porous Loss**, set the resistance coefficients which control the momentum loss.

In general, the loss consists of two parts: a linear term, which models the resistance due to viscous losses; and a quadratic term, which models the resistance due to inertial losses. The loss is expressed as:

$$\mathbf{S} = -\mu C_1 \mathbf{V} - C_2 \frac{\rho}{2} |\mathbf{V}| \mathbf{V}$$

where

**S** is the momentum source which accounts for the porous loss

**C**<sub>1</sub> is the user-specified **Viscous resistance coefficient**

**C**<sub>2</sub> is the user-specified **Inertial resistance coefficient**

$\mu$  is the fluid viscosity

$\rho$  is the fluid density

**V** is the fluid velocity vector.

For low-Reynolds number flows, the viscous loss typically dominates and the **Inertial resistance coefficient** can often be set to 0. For turbulent flows, the inertial loss typically dominates and the **Viscous resistance coefficient** can often be set to 0.

The values you select for the **Viscous resistance coefficient** and **Inertial resistance coefficient** typically depend on the application and what data you know. For instance:

- If you are modeling flow through a porous medium where Darcy's law applies, set the **Viscous resistance coefficient** to  $1/K_{perm}$ , where  $K_{perm}$  is the permeability. (If the viscous resistance is large compared to other terms in the momentum equation, the governing equations will automatically recover Darcy's law.)
- If you are modeling flow in a packed bed, the Ergun equation is very similar to the momentum loss expression. The resistance coefficients can be derived from the Ergun equation parameters.
- The **Inertial resistance coefficient** can be interpreted as a loss coefficient per unit length. So if you are modeling a device such as a perforated plate, filter, or tube bank with a known loss coefficient  $kg/m^3$  and thickness  $t$ , you can select  $v_z = \dot{m}/\rho A$ .
- If you have experimental tabulated data of pressure drop as a function of velocity for a porous medium of thickness  $t$ , you can perform a curve fit of the data in the following form to estimate the resistance coefficients:

$$\Delta P / \Delta x = -(\mu C_1 \mathbf{V} + C_2 \frac{\rho}{2} |\mathbf{V}| \mathbf{V})$$

For an orthotropic porous medium, the resistance coefficients are set independently in each direction, with respect to the selected Reference frame. In addition, you may specify the coefficients in either Cartesian or cylindrical components.

**Note:** The solver uses a superficial formulation of the governing equations, which means that the porosity is modeled as a momentum loss only. The velocity magnitude therefore does not change as the fluid enters/exits the porous medium.

## 4.6.1.8.2. Derivation of Porous Coefficients Based on Experimental Pressure and Velocity Data

Experimental data, that is available in the form of pressure drop against velocity through the porous component, can be extrapolated to determine the coefficients for the porous media. To effect a pressure drop across a porous medium, the coefficients of the porous media are determined in the manner described below.

If the experimental data over the thickness of the media,  $\Delta x$ , is:

Velocity (m/s)	Pressure Drop (Pa)
20.0	197.8
50.0	948.1
80.0	2102.5
110.0	3832.9

Then a curve can be plotted through these points, in the form  $\Delta P / \Delta x = Av^2 + Bv$ , in this case yielding  $\Delta P / \Delta x = 0.27394v^2 + 4.68816v$ .

## Viscous Resistance Coefficient, $C_1$

From this curve the viscous resistance factor,  $C_1$ , can be determined by the following:

$$C_1 = B / \mu$$

Where B is from the fitted curve, and  $\mu$  is the fluid viscosity. In this example,  $B = 4.68816$  and if  $\mu = 1.7894 \times 10^{-5}$ , then  $C_1 = 261996 \text{ m}^2$ .

## Inertial Resistance Coefficient, $C_2$

From this curve the inertial resistance factor,  $C_2$ , can be determined by the following:

$$C_2 = 2A / \rho$$

Where A is from the fitted curve, and  $\rho$  is the density of the fluid. In this example,  $A = 0.27394$  and if  $\rho = 1.225 \text{ kg/m}^3$ , this leads to  $C_2 = 0.2236 \text{ m}$ .

## 4.6.1.9. Symmetry

The symmetry boundary condition imposes constraints that mirrors the expected pattern of the flow or thermal solution on either side of it. At a symmetry boundary, all gradients normal to the face are zero. Thus, the normal velocity component at the boundary is set to zero:

$$U_n = 0$$

and the scalar variable gradients normal to the boundary are also set to zero:

$$\frac{\partial \varphi}{\partial n} = 0$$

## 4.6.1.9.1. Specifying Symmetry

### Physics > Fluid Flow Conditions > Symmetry

To specify symmetry boundary conditions:

Select the location in your geometry for which you want to specify **Symmetry** conditions. Note that the location of a valid symmetry condition should be planar.

## 4.6.1.10. Walls

Wall boundary conditions assigned to surfaces of a fluid region prevent flow through those surfaces.

An expression of `DefaultWalls()` can be used to define the location of a boundary condition. This resolves to all the bounding faces of a region that do not have an explicit boundary condition or region interface applied. Creating a new wall boundary condition without a location selected will automatically default to the `DefaultWalls()` expression.

The conditions that best describe the forces applied to a wall are summarized below.

Condition at the wall	Option	Additional information
The fluid sticks to the wall. If the wall is moving, the fluid moves with the same velocity as the wall.	No Slip	Specify the <b>Wall velocity</b> to be stationary or moving. The motion is specified relative to the boundary's reference frame.
A frictionless wall or streamline.	Free Slip	Components of velocity normal to free slip wall are zero.  <b>Note:</b> For symmetry or free stream conditions, consider using a symmetry or an opening boundary condition.
The shear stress is known along a surface but velocity is unknown.	Specified Shear	Components of the shear stress may be specified as values or an expression. This assumes the fluid is moving at the wall and the velocity is a result of the solution.

## 4.6.1.10.1. Specifying No Slip Walls

### Physics > Fluid Flow Conditions > Wall

To specify no slip walls:

1. Select the location in your geometry for which you want to specify the wall boundary condition.
2. For **Flow specification**, select **No slip**.
3. For **Wall velocity**, select **Stationary** or **Moving**.

If you want to...	Then choose...
Model a non-stationary wall	<b>Wall velocity</b> and select <b>Moving</b> .
Express the wall velocity by specifying a magnitude and defining the direction	The <b>Moving Wall</b> option of <b>Magnitude and direction</b> . Enter the speed of the moving wall and the direction in <b>Cartesian</b> coordinates.
Express the wall velocity by specifying its directional components	The <b>Moving Wall</b> option of <b>Components</b> . Specify the coordinates to be <b>Cartesian</b> or <b>Cylindrical</b> .
Model a rotating wall	The <b>Moving Wall</b> option of <b>Rotating</b> and enter the <b>Angular speed</b> of the rotating wall.

- For **Energy specification**, if your simulation includes a thermal analysis, select one of the [energy specification options](#) that best describes the system.
- Additionally, in the **Wall** panel, if you want to use a **Reference frame** other than **Global Reference Frame**, [select or define the reference frame](#).

## 4.6.1.10.2. Specifying Free Slip Walls

**Physics > Fluid Flow Conditions > Wall**

To specify free slip walls:

- [Select the location](#) in your geometry for which you want to specify the wall boundary condition.
- For **Flow specification**, select **Free slip** for frictionless walls.
- For **Energy specification**, if your simulation includes a thermal analysis, select one of the [energy specification options](#) that best describes the system.
- Additionally, in the **Wall** panel, if you want to use a **Reference frame** other than **Global Reference Frame**, [select or define the reference frame](#).

## 4.6.1.10.3. Specifying Shear Walls

**Physics > Fluid Flow Conditions > Wall**

Click  to show all properties.

To specify shear conditions at a wall:

- [Select the location](#) in your geometry for which you want to specify the wall boundary condition.
- For **Flow specification**, select **Specified shear** if it is known along a surface but velocity is unknown. In the **Wall** panel, under **Shear stress components**, specify the X, Y, and Z components as constant values or expressions.
- For **Energy specification**, if your simulation includes a thermal analysis, select one of the [energy specification options](#) that best describes the system.
- Additionally, in the **Wall** panel, if you want to use a **Reference frame** other than **Global Reference Frame**, [select or define the reference frame](#).

## 4.6.1.10.4. Specifying Wall Roughness

**Physics > Fluid Flow Conditions > Wall**

Fluid flow over rough surfaces are encountered in diverse situations such as flow over the surfaces of airplanes, ships, turbomachinery, heat exchangers, piping systems, and atmospheric boundary layers over terrain of varying roughness. Wall roughness affects drag (resistance) and heat and mass transfer on the walls.

To specify wall roughness conditions:

1. Select the location in your geometry for which you want to specify the wall boundary condition.
2. For **Flow specification**, select an appropriate slip condition for the wall.
3. For **Energy specification**, if your simulation includes a thermal analysis, select one of the [energy specification options](#) that best describes the system.
4. For **Turbulence specification**, if your simulation needs to account for wall roughness effects in wall-bounded turbulent flows (when using k- $\omega$  turbulence models only), specify wall roughness parameters. For **Wall roughness**, select **Rough**, and set a value for the **Roughness height** and the **Roughness constant**. **Wall roughness** affects drag (resistance) and heat and mass transfer on the walls. A roughness height of zero is equivalent to a smooth wall.
5. Additionally, in the **Wall** panel, if you want to use a **Reference frame** other than **Global Reference Frame**, [select or define the reference frame](#).

#### 4.6.1.10.4.1. Turbulence Specification for Walls

Table 4.6.1.10.4.1.1. Wall Roughness Effects in Turbulent Wall-Bounded Flows

Roughness Height Indicator	Roughness Constant Guidance
For a uniform sand-grain roughness, you only need to specify the height of the sand-grain.	Adjust the <b>Roughness Constant</b> when the roughness you want to model departs much from uniform sand-grain. For uniform sand-grain surfaces, a value of 0.5 is reasonable.
For a non-uniform sand-grain, the mean diameter is a more meaningful roughness height.	For non-uniform sand-grains, ribs, and wire-mesh roughness a <b>Roughness Constant</b> between 0.5 and 1.0 can be used to describe the non-uniformity of the wall roughness.
For all other types of roughness, you should use an equivalent sand-grain roughness height.	Non-constant <b>Roughness Height</b> and <b>Roughness Constant</b> can be specified using <a href="#">expressions</a> .

#### 4.6.1.10.5. Energy Specification for Walls

Table 4.6.1.10.5.1. Energy Specifications for Internal Wall (touching the fluid) and External Wall (opposite side of the fluid)

Energy specification	Description	Heat flux at the wall $q_w$ calculated from the boundary condition
<b>Insulated</b>	The wall is perfectly insulated (adiabatic).	$q_w = 0$
<b>Heat flux</b>	The heat flux (energy transfer rate $q_w = q_{specified}$ per unit area [W/m^2 or equivalent]) is specified. Entering a value of 0 is equivalent to a perfectly insulated boundary.	

Energy specification	Description	Heat flux at the wall $q_w$ calculated from the boundary condition
<b>Heat flow</b>	The heat flow (energy transfer rate $q_w = Q_{specified} / A$ [W or equivalent]) is specified.	$Q_{specified}$ is the specified heat flow $A$ is the area
<b>Temperature</b>	The wall temperature is specified.	The solver discretizes $q_w$ based on the local temperature gradient adjacent to the wall.
<b>Convection</b>	Heat transfer through the wall occurs through external convection. The heat transfer coefficient and external convection temperature are specified.	$q_w = h(T_{convection} - T_w)$ $h$ is the specified heat transfer coefficient
<b>Radiation</b>	Heat transfer through the wall occurs through external radiation exchange with a blackbody having a specified <b>Radiation temperature</b> .	$T_{convection}$ is the specified convection temperature $T_w$ is the wall temperature as computed by the solver $q_w = \varepsilon\sigma(T_r^4 - T_w^4)$ $\varepsilon$ is the wall emissivity $\sigma$ is the Stefan-Boltzmann constant
<b>Convection and Radiation</b>	Heat transfer through the wall boundary occurs through both external convection and radiation.	$q_w = h(T_{convection} - T_w) + \varepsilon\sigma(T_r^4 - T_w^4)$ $T_r$ is the specified radiation temperature

## 4.6.1.11. Turbulence Specification Methods

Table 4.6.1.11.1. Turbulence Specification Methods

Option	Description
<b>Low intensity and viscosity ratio</b>	For mildly turbulent flow <ul style="list-style-type: none"> <li>• Intensity is 1%</li> <li>• Eddy Viscosity Ratio is 1</li> </ul>
<b>Medium intensity and viscosity ratio</b>	For moderately turbulent flow <ul style="list-style-type: none"> <li>• Intensity is 5%</li> <li>• Eddy Viscosity Ratio is 10</li> </ul>

Option	Description
<b>High intensity and viscosity ratio</b>	For highly turbulent flow <ul style="list-style-type: none"> <li>Intensity is 10%</li> <li>Eddy Viscosity Ratio is 100</li> </ul>
 <b>Intensity and viscosity ratio</b>	For turbulent flow that does not necessarily fall into one of the low, medium, or high range for the intensity and Eddy viscosity ratio. Values that best describe turbulent flow should be specified.
 <b>Transport variables</b>	
 <b>Intensity and length scale</b>	
 <b>Intensity and hydraulic diameter</b>	

## 4.6.2. Structural Loads and Constraints

Loads and constraints for structural physics include:

- **Body temperature loads** (for thermal strain)
- **Bolt pretension**
- Zero and nonzero displacements
- Externally applied forces, pressures, and moments
- Steady-state inertial forces
- **Supports**
- For a **random vibration modal analysis**, acceleration PSD (either in acceleration<sup>2</sup> units or in G<sup>2</sup> units), velocity PSD, or displacement PSD

**Note:** If a condition is no longer required for your structural simulation, but may be needed in the future, you can **suppress it** rather than delete it entirely.

### 4.6.2.1. Body Temperature

You can use a Body Temperature load to apply a temperature over the selected geometry. Body Temperature loads are used to calculate temperature-dependent material properties and thermal strains (if Thermal physics is not activated).

#### 4.6.2.1.1. Setting a Body Temperature

**Physics > Structural Conditions > Body Temperature**

To set a Body Temperature load:

1. Select the location for this structural condition.

You can apply a temperature to one or more bodies.

2. For **Temperature**, enter a constant or expression directly into the field; any expression must resolve to a constant.

## 4.6.2.2. Bolt Pretension

The bolt pretension load applies a pretension load to a cylindrical face, to a single body, or to multiple bodies -- typically to model a bolt under pretension. For example, you can use the bolt pretension to model a bolt which is holding together two bodies by applying a known value of axial load or adjustment to the length of the bolt (to simulate the tightening of a nut or bolt).

If you have multiple bolts with the same load value and axial direction, you can either simulate each of the bolts with the same preload value, or you can distribute the applied load value over all the selected bolts.

Bolt pretension must be defined in conjunction with [multi-step analysis](#) to ensure that, during the simulation, the bolt pretension load is applied and then locked, typically before other conditions are applied. To simulate sequencing of bolt tightening, bolt pretension can be suppressed before being active, or to simulate the removal of a bolt, it can be suppressed after it has been locked. The following table describes how the state set in a multi-step analysis is implemented in bolt pretension.

State	Description
Factor	Applies an Axial force or Adjustment as a preload using a constant factor. This factor defaults to 1.
Locked	Fixes all displacements. You can set this state for any step except the first step.
Suppressed	<p>Leaves the Bolt Pretension load set as open, so that the load has no effect on the applied step. This option effectively suppresses the load for the step.</p> <p>Note that in order to avoid convergence issues resulting from under-constrained conditions, a small load (0.01 % of the maximum value of load across all solution steps for the current bolt pretension) is applied.</p> <p>You can set this state for any step.</p>

### 4.6.2.2.1. Defining a Bolt Pretension

#### Physics > Boundary Conditions > Bolt Pretension

To set up a bolt pretension load:

1. Select the location for this load.

You can apply a bolt pretension load to a cylindrical face or one or more bodies. When applied to a body, you must specify the bolt axial direction. When applied to a cylinder face, the cylinder axis becomes the bolt axis direction.

2. When the location is set to multiple volumes, you can choose how to model the resulting bolt(s) by setting **Model as**.

- If you want to create only one bolt out of the selected volumes and distribute the supplied load value, select **One bolt for all locations**.
- If you want to create a separate bolt from each selected volume and apply the supplied load value to each of them, select **One bolt per location**.

**Note:** This option is available when the location is set to multiple volumes.

3. For **Define by**, select if you want to define your bolt pretension as an **Axial force** or an **Adjustment**. Then enter your **Axial force** load or **Length of adjustment**.
4. For **Axial direction based on geometry selection** (applicable to volumes), select a single face or edge to define the axial direction of the bolt.
5.  Additionally, you can make modifications to the following settings in the **Bolt Pretension** panel:

If you want to...	Then choose...	More information
Define the bolt axial direction based on a reference frame	<b>Axial direction definition &gt; Reference frame</b>	By default, AIM defines the position of the virtual cut plane on the centroid of the body. However, you can also select a reference frame. Then AIM defines the position of the cut plane based on frame origin. If the cut plane is defined by a referenced frame, the cut plane must pass through the bolt. You can change the axis to define the cut plane using <b>Axial direction</b> and move any user defined reference frames into position
Specify the pretension tolerance	<b>Pretension tolerance specification &gt; User Defined</b>	The <b>Pretension tolerance</b> value denotes the tolerance up to which nodes below the plane where the bolt is virtually cut (located at the centroid of the body, surface, or reference frame) are considered to be above the surface -- in effect, shifting the plane downwards by the specified distance.

### 4.6.2.3. Displacement

A displacement constraint specifies that a portion of the model displaces relative to its original location. You can also use this constraint to prescribe rotations to a solid model via an originating point in space.

You have several options for defining displacement:

- By specifying translation and rotation components in a reference frame.
- By specifying translation by direction which is either defined by magnitude and direction or normal to a boundary.
- By specifying rotation magnitude about an axis.

You also have the option of defining displacement via an originating point in space. You can use this remote displacement constraint as an alternative to building a rigid link and applying a constraint to it.

#### 4.6.2.3.1. Defining Translation and Rotation by Directional Components

##### Physics > Structural Conditions > Displacement

To specify translation and rotation components:

1. **Select the location** for this structural condition.

You can apply this type of displacement to bodies, vertices, edges, or faces of a 3D geometry.

**Note:** When applying via an arbitrary point in space, you cannot select a body as the location.

2. From **Displacement specification**, select **Translation and Rotation**.
3. To apply this condition via an arbitrary point in space, select **Apply remotely from originating point**. You can then define the originating point on a body, face, edge, or vertex. If you select a body, face, or edge as the originating location, AIM uses the centroid of the selected topology as the location. You can also define the origin as a **Point**, either by selecting a previously-defined point or by creating a new one.
4. Specify the directional components. Enter a constant or expression directly into the field; any expression must resolve to a constant.

For a Cartesian reference frame, specify translation on the relevant axis. For a cylindrical reference frame, specify:

- Rho: The distance from the Z axis
- Phi: The angular distance in the XY plane
- Z: The translation on the Z axis.

**Note:** Depending on the DOF values you define, AIM may set other DOF values to a value of "Free"; you will be unable to edit these values.

5. Additionally, you can [select or define a frame of reference](#) under **Reference frame** if you want to use a reference frame other than **Global reference frame**.

### 4.6.2.3.2. Defining Translation Normal to a Face

#### Physics > Structural Conditions > Displacement

To set a translation applied in the normal direction to a face:

1. [Select the location](#) for this structural condition.  
You can apply this constraint to faces.
2. For **Displacement specification**, select **Translation by direction**.
3. For **Define by**, select **Normal to boundary**.
4. For **Displacement**, enter a constant into the field.  
You can also enter an expression that resolves to a constant.

### 4.6.2.3.3. Defining a Translation by Magnitude and Direction

#### Physics > Structural Conditions > Displacement

To specify a translation on a vector using magnitude and direction:

1. [Select the location](#) for this structural condition.  
You can apply this type of displacement to bodies, vertices, edges, or faces of a 3D geometry.
2. For **Displacement specification**, select **Translation by direction**.
3. To apply this condition via an arbitrary point in space, select **Apply remotely from originating point**.

You can then define the originating point on a body, face, edge, or vertex. If you select a body, face, or edge as the originating location, AIM uses the centroid of the selected topology as the location.

You can also define the origin as a **Point**, either by selecting a previously-defined point or by creating a new one.

4. For **Define by**, select **Magnitude and direction**.
5. For **Magnitude**, enter a constant or expression directly into the field; any expression must resolve to a constant.
6. For **Direction based on geometry selection**, select a face or edge to determine the vector direction.

The direction geometry assignment defaults to the first location selected, but you can replace this location.

If you want the direction to be opposite the vector for the selected location, click **Reverse direction**.

## 4.6.2.3.4. Defining a Rotation by Magnitude and Direction

### Physics > Structural Conditions > Displacement

To specify a rotation on a vector using magnitude and direction:

1. Select the **location** for this structural condition.

You can apply this type of displacement to bodies, vertices, edges, or faces of a 3D geometry.

**Note:** When applying via an arbitrary point in space, you cannot select a body as the location.

2. For **Displacement specification**, select **Rotation about an axis**.
3. To apply this condition via an arbitrary point in space, select **Apply remotely from originating point**.

You can then define the originating point on a body, face, edge, or vertex. If you select a body, face, or edge as the originating location, AIM uses the centroid of the selected topology as the location.

You can also define the origin as a **Point**, either by selecting a previously-defined point or by creating a new one.

4. For **Magnitude**, enter a constant or expression directly into the field; any expression must resolve to a constant.
5. For **Axis orientation based on geometry selection**, select a face or edge to determine the vector direction.

The direction geometry assignment defaults to the first location selected, but you can replace this location.

If you want the direction to be opposite the vector for the selected location, click **Reverse direction**.

## 4.6.2.4. Distributed Mass

You can distribute additional mass across the faces of the flexible parts in your model using the distributed mass load. This load enables you to idealize the inertial effects from the bodies/entities that are evenly spread across the surfaces of your model; for example, mass contribution from paint, external equipment, a large number of small objects spaced evenly across the surfaces, etc.

## 4.6.2.4.1. Specifying a Distributed Mass

### Physics > Boundary Conditions > Distributed Mass

To define a distributed Mass:

1. Select one or more faces for this condition.
2. For **Mass type**, select whether you're defining the distributed mass as a **Total mass** or **Mass per unit area**.  
For mass per unit area, the solver multiplies the specified value by the area of the selected face(s). For total mass, the specified value is divided by the sum of the selected area.
3. Enter the **Mass value**.

## 4.6.2.5. Inertia Load

An Inertia load is experienced by a body or a collection of bodies when it is in an accelerated reference frame. A reference frame is "accelerated" if you specify a linear acceleration or an angular velocity or angular acceleration to the frame.

Inertia itself is not a load, but it causes loads (forces) to be developed in the bodies opposite in direction to the specified accelerations. The bodies must also have mass specified for the developed inertial loads to be non-zero.

Inertia loads are used to model gravity effects (weight) or rotational effects in spinning equipment.

### 4.6.2.5.1. Setting Inertia Load

#### Physics > Structural Conditions > Inertia Load

To set an Inertia Load:

1. [Select the location](#) for this structural condition.

An inertia load is initially applied by default to all bodies of a model that have mass. However, you can change the location to apply an inertia load to a specific body or bodies.

2. For **Inertia source**, select the type of inertia load.
3. Specify the magnitude on one or more directions of the coordinate system in the **X**, **Y**, and **Z** fields. Enter a constant or expression directly into the field; any expression must resolve to a constant. Note that the **Coordinate type** shown is based on the reference frame.
4. Additionally, you can [select or define a frame of reference](#) under **Reference frame** if you want to use a reference frame other than **Global reference frame**.

## 4.6.2.6. Force

You can apply the force load either by defining a magnitude and direction or by defining the components of that vector in a given coordinate system. This load is defined in units such as pounds or Newtons.

You can also apply a pressure in a specific direction by defining a **Force per unit area**. And, when defining force per unit area with directional components, you can define a spatially varying load or a bearing load.

Finally, you can define a force intensity (a line load) to edges by selecting **Force per unit length**.

In addition to applying a force to geometry, you also have the option of defining the force as originating at an arbitrary point in space. You can use this remote force load as an alternative to building a rigid link and applying a boundary condition to it.

### 4.6.2.6.1. Defining Force by Directional Components

#### Physics > Structural Conditions > Force

To set a force load:

1. Select the location for this structural condition.

You can apply a force or force per unit area to edges or faces of 3D geometry, or to a single vertex. For a force per unit length, you can apply a force to edges.

2. If you want to apply the force from an arbitrary point in space:

a) Select **Apply remotely from originating point**.

b) For **Originating point location**, define the originating point on a body, face, edge, or vertex. If you select a body, face, or edge as the originating location, AIM uses the centroid of the selected topology as the location.

You can also define the origin as a **Point**, either by selecting a previously-defined point or by creating a new one.

c) For **Formulation**, select a behavior for the remote connection:

- Select **Deformable** if attached geometry from the remote location is free to deform.
- Select **Rigid** if attached geometry from the remote location can not deform (maintains the initial shape).
- Select **Coupled** if the attached geometry has the same DOF solution on its underlying nodes as the remote point location.

3. For **Specification**, select if you want to define a total force, or a force per unit area or unit length.

4. For **Define by**, select **Directional components**.

5. Specify the magnitude on one or more directions of the coordinate system in the **X**, **Y**, and **Z** fields. Enter a constant or expression directly into the field. Note that the **Coordinate type** shown is based on the reference frame.

To define a spatially varying load, specify the component values as functions of a single spatial variable (Position.x, Position.y, Position.z). These spatial variables will be defined in the same reference frame used to define components.

For example, you could specify the spatially varying load using the following expression:

```
-250 * (Position.z ** 3) * 1 [Pa m^-3]
```

When defining Force per unit area, you can specify a bearing load by using an if/else expression. For example:

```
-1000*Position.z*Position.z*1[Pa m^-2] if Position.z<0[m] else 0 [Pa]
```

View a chart by clicking  and then the **Show chart**  icon

6. Additionally, you can [select or define a frame of reference](#) under **Reference frame** if you want to use a reference frame other than **Global reference frame**.

## 4.6.2.6.2. Defining Force by Magnitude and Direction

### Physics > Structural Conditions > Force

To set a force load:

1. Select the location for this structural condition.

You can apply a force or force per unit area to edges or faces of 3D geometry, or to a single vertex. For a force per unit length, you can apply a force to edges.

2. If you want to apply the force from an arbitrary point in space:
  - a) Select **Apply remotely from originating point**.
  - b) For **Originating point location**, define the originating point on a body, face, edge, or vertex. If you select a body, face, or edge as the originating location, AIM uses the centroid of the selected topology as the location.

You can also define the origin as a **Point**, either by selecting a previously-defined point or by creating a new one.

  - c) For **Formulation**, select a behavior for the remote connection:
    - Select **Deformable** if attached geometry from the remote location is free to deform.
    - Select **Rigid** if attached geometry from the remote location can not deform (maintains the initial shape).
    - Select **Coupled** if the attached geometry has the same DOF solution on its underlying nodes as the remote point location.
3. For **Specification**, select if you want to define a total force, or a force per unit area or unit length.
4. For **Define by**, select **Magnitude and direction**.
5. For **Magnitude**, enter a constant or expression.
6. For **Direction based on geometry selection**, select a face or edge to determine the vector direction.

The direction geometry assignment defaults to the first location selected, but you can replace this location.

If you want the direction to be opposite the vector for the selected location, click **Reverse direction**.

## 4.6.2.7. Moment

This vector-based load distributes a moment "about" (the vector of) an axis across one or more flat or curved faces, or about one or more edges or vertices. Use the right-hand rule to determine the sense of the moment. This load has a moment or torque (Newton meter) unit.

You can apply the moment load either by defining a magnitude and direction or by defining the components of that vector in a given coordinate system.

### 4.6.2.7.1. Defining a Moment by Directional Components

#### Physics > Structural Conditions > Moment

To define a moment load by directional components:

1. Select the location for this structural condition.  
You can apply a moment to vertices, edges, or faces.
2. For **Define by**, select **Directional components**.
3. Specify the magnitude on one or more directions of the coordinate system in the **X**, **Y**, and **Z** fields. Enter a constant or expression directly into the field; any expression must resolve to a constant. Note that the **Coordinate type** shown is based on the reference frame.
4. Additionally, you can select or define a frame of reference under **Reference frame** if you want to use a reference frame other than **Global reference frame**.

## 4.6.2.7.2. Defining a Moment by Magnitude and Direction

### Physics > Structural Conditions > Moment

To define a moment load by magnitude and direction:

1. Select the location for this structural condition.

You can apply a moment to vertices, edges, or faces.

2. For Define by, select **Magnitude and direction**.

3. For **Magnitude**, enter a constant or expression directly into the field; any expression must resolve to a constant.

4. For **Direction based on geometry selection**, select a face or edge to determine the vector direction.

The direction geometry assignment defaults to the first location selected, but you can replace this location.

If you want the direction to be opposite the vector for the selected location, click **Reverse direction**.

## 4.6.2.8. Pressure

A pressure load applies a constant pressure on one or more flat or curved faces. A positive value for pressure acts into the face, tending to compress the solid body.

Pressure is a scalar load and can only be applied normal to a face. You can also define pressure as a spatially varying load.

**Note:** To apply a pressure in a specific direction, use **Force per unit area** as an alternative. For more information, see [Force](#) on page 292.

## 4.6.2.8.1. Setting a Pressure

### Physics > Structural Conditions > Pressure

To set a pressure load:

1. Select the location for this structural condition.

You can apply a pressure to 3D geometric faces.

2. For **Pressure**, enter the magnitude as a constant or expression directly into the field.

To define pressure as a spatially varying load, specify it as a function of a single spatial variable (Position.x, Position.y, Position.z). Once you have specified the pressure as a spatial function, select or define a frame of reference from **Reference frame**.

For example, you could specify the spatially varying load using the following expression:

```
-250 * (Position.z * Position.z * Position.z) * 1 [Pa m^-3]
```

You can view a chart by clicking ▶ and then the **Show chart**  icon

## 4.6.2.9. Power Spectral Density (PSD)

The base excitation for a random vibration analysis is applied in the form of Power Spectral Density (PSD). The PSD is a table of spectral values vs. frequency that captures the frequency content. The PSD captures the frequency and mean square amplitude content of the load's time history.

The base excitation could be an acceleration PSD (either in acceleration<sup>2</sup> units or in G<sup>2</sup> units), velocity PSD, or displacement PSD. In AIM, the base excitation, defined as a structural condition, is applied in the specified direction to all entities that have the selected Fixed Support. Other support points in a structure, such as Frictionless Support, are not excited by the PSD.

You can apply multiple uncorrelated PSDs -- useful if different, simultaneous excitations occur in different directions.

### 4.6.2.9.1. Specifying PSD Acceleration

#### Physics > Structural Conditions > PSD Acceleration

To define your base excitation as an acceleration PSD in acceleration<sup>2</sup> units:

1. Select the fixed support for the base excitation.  
To set the location to all supports, enter `AllFixedSupports()`.
2. For one or more of the directions in the global reference frame, define a table of spectral values vs. frequency that captures the frequency content.

### 4.6.2.9.2. Specifying PSD Displacement

#### Physics > Structural Conditions > PSD Displacement

To define your base excitation as a displacement PSD:

1. Select the fixed support for the base excitation.  
To set the location to all supports, enter `AllFixedSupports()`.
2. For one or more of the directions in the global reference frame, define a table of spectral values vs. frequency that captures the frequency content.

### 4.6.2.9.3. Specifying PSD G Acceleration

#### Physics > Structural Conditions > PSD G Acceleration

To define your base excitation as an acceleration PSD in G<sup>2</sup> units:

1. Select the fixed support for the base excitation.  
To set the location to all supports, enter `AllFixedSupports()`.
2. For one or more of the directions in the global reference frame, define a table of spectral values vs. frequency that captures the frequency content.

### 4.6.2.9.4. Specifying PSD Velocity

#### Physics > Structural Conditions > PSD Velocity

To define your base excitation as a velocity:

1. Select the fixed support for the base excitation.

To set the location to all supports, enter `AllFixedSupports()`.

- For one or more of the directions in the global reference frame, define a table of spectral values vs. frequency that captures the frequency content.

## 4.6.2.10. Point Mass

You can idealize the inertial effects from a body using a Point Mass. Applications include applying a force with an acceleration or any other inertial load; or adding inertial mass to a structure, which affects modal solutions.

### 4.6.2.10.1. Setting a Point Mass

#### Physics > Structural Conditions > Point mass

To set up a point mass:

- Select the location for this structural condition.

You can define a point mass anywhere in space. When you select a geometry location, the point mass will be created at the centroid of the selected location and the selected geometry is set as the **Connecting location**. You can also define the origin as a **Point**, either by selecting a previously-defined point or by creating a new one.

- For **Connecting location**, select the location by which the point mass is connected to the rest of the model.
- Specify the **Mass** as a value greater than 0.
- Define the **Mass moment of inertia** about each axis of the global reference frame.

- Optionally, you can select a **Formulation**. This option dictates the behavior of the attached geometry.
  - Deformable**: The geometry is free to deform. This general purpose option is used when applying loads and constraints such as a force or mass through "abstract" entities not explicitly represented as geometry.
  - Rigid**: The geometry will not deform (AIM maintains the initial shape). This option is useful when the "abstracted" object significantly stiffens the model at the attachment point. Note that thermal expansion effects cause artificially high stresses because the geometry cannot deform where the load is applied.
  - Coupled**: The geometry has the same degrees of freedom solution on its underlying nodes as the point location. This is useful when you want a portion of geometry to share the same degrees of freedom (such as UX) that may or may not be known.

## 4.6.2.11. Support

A support is a constraint that you create by fixing or freeing the available degrees of freedom at the location in the model where you apply it. Available degrees of freedom are automatically detected once the location is specified; they are fixed by default.

You have several options for defining a support. You can define:

- A **fixed support**, where all available degrees of freedom are automatically set to fixed.
- A **frictionless support**. In this case, ANSYS AIM fixes the direction that is perpendicular to the selected face or tangential to the selected edge. This direction can vary at every node on the selected edge or face.
- A **support you define**, where you can individually set each available DOF to fixed or free. The available DOFs will change according to selected location.

## 4.6.2.11.1. Setting a Fixed Support

### Physics > Structural Conditions > Support

A fixed support is one where all available degrees of freedom are automatically set to fixed.

To set a fixed support constraint:

1. [Select the location](#) for this structural condition.

You can apply a support directly to vertices, edges, bodies, or faces of 3D geometry.

2. For **Type**, select **Fixed**.

## 4.6.2.11.2. Setting a Frictionless Support

### Physics > Structural Conditions > Support

With a frictionless support, AIM fixes the direction that is perpendicular to the selected face or tangential to the selected edge. This direction can vary at every node on the selected edge or face.

To set a frictionless support:

1. [Select the location](#) for this structural condition.

You can apply a support directly to edges or faces of 3D geometry.

2. For **Type**, select **Frictionless**.

## 4.6.2.11.3. Defining a Custom Support

### Physics > Structural Conditions > Support

To manually define a support constraint:

1. [Select the location](#) for this structural condition.

You can apply a support directly to vertices, edges, bodies, or faces of 3D geometry.

2. For **Type**, select **User specified**.

3. For the **Degrees of freedom** fields, specify the translation and rotation for each as fixed or free.

For a Cartesian reference frame, specify translation on the relevant axis. For a cylindrical reference frame, specify:

- Rho: The distance from the Z axis
- Phi: The angular distance in the XY plane
- Z: The translation on the Z axis.

4. Additionally, you can [select or define a frame of reference](#) under **Reference frame** if you want to use a reference frame other than **Global reference frame**.

## 4.6.3. Thermal Loads and Constraints

Loads and constraints for thermal physics include:

- [Convection](#)
- [Convection and Radiation](#) (applicable to fluid-solid heat transfer simulations)
- [Heat Flow](#)
- [Heat Flux](#)

- Heat Generation
- Heat Source
- Insulated (applicable to fluid-solid heat transfer simulations)
- Radiation
- Temperature

## 4.6.3.1. Convection

The Convection load causes convective heat transfer to occur through one or more flat or curved faces (in contact with a fluid).

The bulk fluid temperature is measured at a distance from the face outside of the thermal boundary layer. The face temperature refers to the temperature at the face of the simulation model.

## Convective Heat Transfer

Convection is related to heat flux by use of Newton's law of cooling:

$$q/A = h(t_s - t_f)$$

where:

- $q/A$  is heat flux out of the face (calculated within the application)
- $h$  is the heat transfer coefficient (you provide)
- $t_s$  is the temperature on the face (calculated by AIM)
- $t_f$  is the bulk fluid temperature that you specify

When the fluid temperature exceeds face temperature, energy flows into a part. When the face temperature exceeds the fluid temperature, a part loses energy.

If you select multiple faces when defining convection, the same bulk fluid temperature and heat transfer coefficient are applied to all selected faces.

### 4.6.3.1.1. Setting Convection

#### Physics > Solid Thermal Conditions > Convection

To apply Convection:

1. Select the location for this solid-thermal condition.

You can apply convection to faces of 3D geometry. If you select multiple faces when defining convection, the same bulk fluid temperature and heat transfer coefficient are applied to all selected faces.

2. For **Heat transfer coefficient**, enter a constant value greater than 0 for a steady-state analysis. For a time-dependent simulation, enter as a constant value greater than 0, as a function of temperature, or as a table of temperatures. You cannot use a table if you are performing a fluid-solid heat transfer simulation.

The heat transfer coefficient (also called the film coefficient or unit thermal conductance) is based on the composition of the fluid in contact with the face, the geometry of the face, and the hydrodynamics of the fluid flow past the face. Refer to heat transfer handbooks or other references to obtain appropriate values for film coefficient.

3. For **Convection temperature**, enter the temperature of the surrounding fluid. For a steady-state analysis, enter the value as a constant. For a time-dependent analysis, enter the temperature of the surrounding fluid as a constant, as a function of time, or as a table of time. You cannot use a table if you are performing a fluid-solid heat transfer simulation.

- For fluid-solid heat transfer fluid flow simulations, you can [select or define a frame of reference](#) under **Reference frame** if you want to use a reference frame other than **Global reference frame**.

## 4.6.3.2. Convection and Radiation

This thermal condition applies only to fluid-solid heat transfer simulations, where it combines both [convection](#) and [radiation](#).

The heat flux for the combined convection and radiation thermal condition is defined as

$$q = h(t_s - t_f) + q_{rad}$$

### 4.6.3.2.1. Setting Convection and Radiation

#### Physics > Solid Thermal Conditions > Convection and Radiation

This thermal condition combines both convection and radiation for fluid-solid heat transfer simulations. For simulations comprising only solid-thermal regions within the physics task, you can overlap Convection and Radiation conditions.

To apply Convection and Radiation:

- [Select the location](#) for this solid-thermal condition.

If you select multiple faces when defining convection, the same bulk fluid temperature and heat transfer coefficient are applied to all selected faces.

- Specify the **Heat transfer coefficient** (also called the film coefficient or unit thermal conductance) as a positive constant value.
- Specify the **Convection temperature** as a constant.
- Specify the **Emissivity** as the ratio of the radiation emitted by a surface to the radiation emitted by a black body at the same temperature.
- Enter the **Radiation temperature**, or the temperature of the surrounding space.
- [Select or define a frame of reference](#) under **Reference frame** if you want to use a reference frame other than **Global reference frame**.

## 4.6.3.3. Heat Flow

Heat flow simulates the transmission across flat or curved surfaces of a specified amount of heat energy per unit time. As a result, heat flow adds energy to or from a body over time.

### 4.6.3.3.1. Setting Heat Flow

#### Physics > Solid Thermal Conditions > Heat Flow

To apply Heat Flow:

- [Select the location](#) for this solid-thermal condition.

In a 3D simulation, you can apply a heat flow to a face. If you select multiple faces when defining the heat flow rate, the magnitude is apportioned across all selected faces.

- For **Heat flow**, enter a value to represent heat energy per time. For a steady-state analysis, enter a constant or expression directly into the field; any expression must resolve to a constant. For a time-dependent analysis, enter a constant value greater than 0, a function of time, or a table of heat flow

vs. time. You cannot use a table if you are performing a fluid-solid heat transfer simulation. Positive heat flow adds energy to the body.

**Note:** For a time-dependent expression containing an `if ... else` condition, create a [Heat Flux](#) condition instead.

3. For fluid-solid heat transfer fluid flow simulations, you can [select or define a frame of reference](#) under **Reference frame** if you want to use a reference frame other than **Global reference frame**.

## 4.6.3.4. Heat Flux

Heat flux simulates the transmission across flat or curved surfaces of a specified amount of heat energy per unit area per unit time. As a result, heat flux adds energy to or from a body over time.

### 4.6.3.4.1. Setting Heat Flux

#### Physics > Solid Thermal Conditions > Heat Flux

To apply Heat Flux:

1. [Select the location](#) for this solid-thermal condition.

You can apply a heat flux to faces of 3D geometry. If you select multiple faces when defining the heat flow rate, the magnitude is applied to all selected faces.

2. For **Heat flux**, a value to represent heat energy per unit area per time. For a steady-state analysis, enter a constant or expression directly into the field; any expression must resolve to a constant. For a time-dependent analysis, enter a constant, a function of time, or a table of time. You cannot use a table if you are performing a fluid-solid heat transfer simulation. Positive heat flux adds energy to the body.
3. For fluid-solid heat transfer fluid flow simulations, you can [select or define a frame of reference](#) under **Reference frame** if you want to use a reference frame other than **Global reference frame**.

## 4.6.3.5. Heat Generation

This load applies a uniform generation rate internal to a body. A positive heat generation acts into a body, adding energy to it. The heat generation rate is defined as energy per unit time per unit volume. The input value for this heat generation is expected to be in watts. The input value is divided by the volume of the body to get the heat generation rate. This heat generation rate is then written in the input file for the solver.

If you select multiple bodies when defining the heat generation, the input value is divided by the total volume of the bodies to get the heat generation rate.

### 4.6.3.5.1. Setting Heat Generation

#### Physics > Solid Thermal Conditions > Heat Generation

Adding a heat generation to a solid-thermal region is applicable only to simulations comprising only solid-thermal regions within the physics task. For fluid-solid heat transfer simulations, see [Heat Source](#) on page 302.

To apply Heat Generation:

1. [Select the location](#) for this solid-thermal condition.

You can apply heat generation to bodies.

- For **Heat generation**, enter a value to represent heat energy per time. For a steady-state analysis, enter a constant or expression directly into the field; any expression must resolve to a constant. For a time-dependent analysis, enter a constant, a function of time, or a table of time. Positive heat generation adds energy to the body.

## 4.6.3.6. Heat Source

Volumetric sources provide a mechanism to account for the effect of features which are not otherwise resolved by the geometry and/or physics. For example, a heat source might be used to account for heat generated by a circuit board, resistance heater, or fire, without explicitly including their geometrical and physical details. For details on how to set up heat sources, see [Specifying Heat Sources](#) on page 266.

### 4.6.3.6.1. Specifying Heat Sources

#### Physics > Solid Thermal Conditions > Heat Source

Adding a heat source to a solid-thermal region is applicable only to fluid-solid heat transfer simulations. For simulations comprising only solid-thermal regions within the physics task, see [Heat Generation](#) on page 301.

To apply Heat Source:

- For the [location](#), select one or more bodies in your physics region where you would like to include the heat source.
- If you want to use a local **Reference frame** which differs from the **Global Reference Frame**, [select or define the reference frame](#).
- Specify the heat source in one of two ways:
  - Source per unit volume** can be input as either a constant or a position-dependent expression. Position-dependent coordinates are with respect to the local reference frame.
  - Total source** is the total heat source.

**Note:** A negative heat source represents a heat sink.

## 4.6.3.7. Insulated

An insulated (or adiabatic) boundary condition for a solid region is where the heat flux across the boundary is zero.

### 4.6.3.7.1. Setting an Insulated Solid Region

#### Physics > Solid Thermal Conditions > Insulated

Adding an insulated solid-thermal condition for a solid region is applicable only to fluid-solid heat transfer simulations. For simulations comprising only solid thermal regions within the physics task, the insulated condition is defined on all external faces of the solid region automatically and therefore no additional definition is required.

[Select the location](#) for this solid-thermal condition.

## 4.6.3.8. Radiation

This load simulates thermal radiation from a face of a 3D model to its surroundings at an ambient temperature. That is, the Form Factor is assumed to be 1.0. Surface-to-surface radiation is not supported.

If radiation is applied to the same location more than once, both radiation conditions independently contribute to the final solution (accumulation effect).

## 4.6.3.8.1. Setting Radiation

### Physics > Solid Thermal Conditions > Radiation

To apply Radiation:

1. [Select the location](#) for this solid-thermal condition.

You can apply radiation to one or more faces of a 3D model.

2. For **Emissivity**, enter the ratio of the radiation emitted by a surface to the radiation emitted by a black body at the same temperature.

Specify as a value or expression greater than 0 but less than or equal to 1.

3. For **Radiation temperature**, set the temperature of the surrounding space.
4. For fluid-solid heat transfer fluid flow simulations, you can [select or define a frame of reference](#) under **Reference frame** if you want to use a reference frame other than **Global reference frame**.

## 4.6.3.9. Temperature

The Temperature constraint simulates a uniform temperature over the selected geometry.

**Note:** If you apply multiple temperature constraints to the same area, the solver uses the last one applied.

## 4.6.3.9.1. Setting Temperature

### Physics > Solid Thermal Conditions > Temperature

To apply a Temperature constraint:

1. [Select the location](#) for this solid-thermal condition.

You can apply a temperature to vertices, edges, or faces of a 3D geometry.

2. For **Temperature**, enter the temperature value. For a steady-state analysis, enter a constant or an expression that resolves to a constant. For a time-dependent analysis, you can enter the temperature as a constant, a function of time, or a table of time. You cannot use a table if you are performing a fluid-solid heat transfer simulation.

3. For fluid-solid heat transfer fluid flow simulations, you can [select or define a frame of reference](#) under **Reference frame** if you want to use a reference frame other than **Global reference frame**.

## 4.6.4. Electromagnetics Conditions

Electromagnetic conditions model the effects of magnetic fields. These conditions are available for [physics regions](#) with the physics type set to **Electromagnetics**.

The conditions available are:

- [Terminal](#)
- [Current](#)
- [Insulating](#)
- [Body Temperature](#)

## 4.6.4.1. Current

The Current condition provides the excitation current that produces a magnetic field.

Current can be defined in one of three ways:

- With **Cross-section** setting, the current is directly assigned on an internal cross section of a conductor. You do not have to define any Terminals to use this setting.
- With the **Source and return** setting, the current flows from a source face to a return face through a conductor, creating a conduction path. These faces must coincide with the edges of an enclosure; otherwise, current will leak into the enclosure. You assign the source and return faces. You do not have to define any Terminals to use this setting.
- With the **Multiple terminals** setting, the current path is determined by one or more defined [Terminals](#). Before you create a Current condition with this setting, [add the needed Terminals to your simulation](#).

### 4.6.4.1.1. Setting Current using a Cross-Section

**Physics > Electromagnetic Conditions > Current**

The **Current** condition is available for Physics Regions with the physics type set to Electromagnetics.

With the **Cross-section** setting, the current is directly assigned on an internal cross section of a conductor. You do not have to define any Terminals to use this setting.

- For **Define path by**, select **Cross-section**.
- For **Cross-sectional face and body**, select the face or body for the internal cross section of the conductor.
- Specify the **Operating current** for the conductor.
- Note:** For branches connected in parallel, the operating current is equally divided among the branches. For example, if the number of branches is 2 and the specified operating current is 10A, the current flowing through each branch is 5A.
- Specify the **Number of conductors**. Typically, this is the number of times the conductor is wrapped around its core.
- When the number of conductors is greater than 1, specify a **Fill factor**.

The fill factor is the adjustment/correction ratio the solver applies to the conductivity when calculating the losses for stranded conductors.

Because the fill factor determines whether the loss values are accurate, we recommend that the value is obtained from actual experiments with similar settings.

- Select **Reverse polarity** to swap the direction of the current.
- Select **Connected in parallel** if the terminals are connected in parallel. Then, set the **Number of branches** in the circuit.
- For frequency response designs, specify the **Phase** angle of the operating current.
- Select **Show conduction path** to display the path of the current through the conductor.
- If the Physics task is valid, and conduction paths have not yet been calculated, you can **Generate Conduction Paths**. Otherwise, this button is disabled (greyed out).

### 4.6.4.1.2. Setting Current with a Source and Return

**Physics > Electromagnetic Conditions > Current**

The **Current** condition is available for Physics Regions with the physics type set to Electromagnetics.

With the **Source and return** setting, the current has a source and a return on the surfaces of a conductor (or a conduction path). You assign the source and return. You do not have to define any Terminals to use this setting.

1. For **Define path by**, select **Source and return**.
2. For **Source**, select a surface body or planar face as the source for the current.
3. For **Return**, select a surface body or planar face as the return for the current.
4. Specify the **Operating current** for the conductor.

**Note:** For branches connected in parallel, the operating current is equally divided among the branches. For example, if the number of branches is 2 and specified operating current is 10A, the current flowing through each branch is 5A.

5. Specify the **Number of conductors**. Typically, this is the number of times the conductor is wrapped around its core.
6. When the number of conductors is greater than 1, specify a **Fill factor**.

The fill factor is the adjustment/correction ratio the solver applies to the conductivity when calculating the losses for stranded conductors.

Because the fill factor determines whether the loss values are accurate, we recommend that the value is obtained from actual experiments with similar settings.

7. Select **Connected in parallel** if the terminals are connected in parallel. Then, set the **Number of branches** in the circuit.
8. For frequency response designs, specify the **Phase** angle of the operating current.
9. Select **Show conduction path** to display the path of the current through the conductor.
10. If the Physics task is valid, and conduction paths have not yet been calculated, you can **Generate Conduction Paths**. Otherwise, this button is disabled (greyed out).

### 4.6.4.1.3. Setting a Current with Multiple Terminals

#### Physics > Electromagnetic Conditions > Current

- The **Current** condition is available for Physics Regions with the physics type set to Electromagnetics.
- Magnetics simulations require that a **Current** condition using the **Multiple terminals** setting include at least one **Terminal**.

With the **Multiple terminals** setting, the current path is determined by one or more defined **Terminals**. Before you create a Current condition with this setting, [add the needed Terminals to your simulation](#).

To set a current condition with multiple terminals:

1. Select the terminal(s) to associate with the conductor. If needed, first [add terminals](#) to your simulation.
2. If two or more terminals are selected, check **Connected in parallel** if the terminals are connected in parallel. Then, set the **Number of branches** in the circuit.
3. Specify the **Operating current** for the conductor.

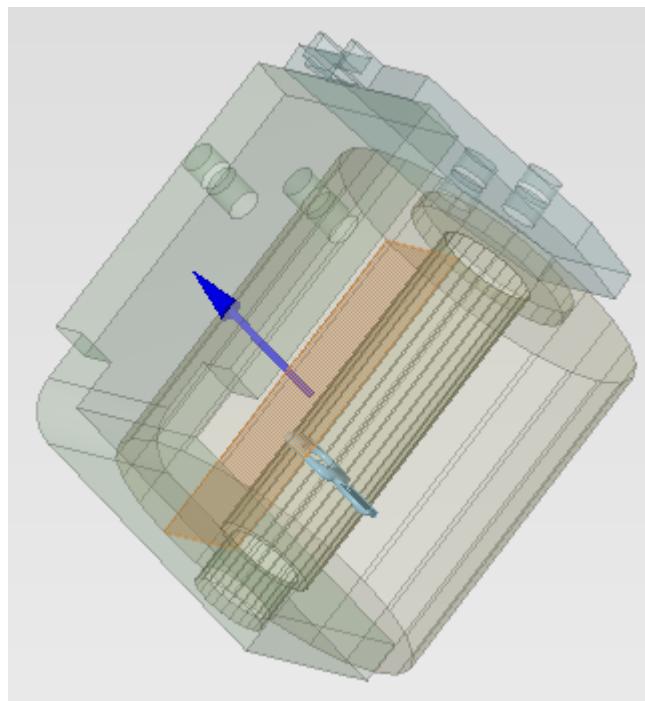
**Note:** For branches connected in parallel, the operating current is equally divided among the branches. For example, if the number of branches is 2 and the specified operating current is 10 amperes, the current flowing through each branch is 5 amperes.

4. For frequency response designs, specify the **Phase** angle of the operating current.
5. If the Physics task is valid, and conduction paths have not yet been calculated, you can **Generate Conduction Paths**. Otherwise, this button is disabled (greyed out).

## 4.6.4.2. Terminal

If the Current condition has been defined by **Multiple terminals**, then at least one Terminal electromagnetic condition is required.

A terminal specifies the total current in a conduction path. You can [assign a terminal](#) to a face of a solid body, or to a surface body that intersects a solid body. For terminals assigned to a face, a conduction path requires one "in" face terminal and one "out" face terminal. For terminals assigned to a surface body such as the one shown below, a conduction path requires one terminal. In stranded conductors the current density is constant across the cross section.



The arrow indicates the polarity, the direction of current flow, of the terminal.

- Enter the **Number of conductors** being modeled: 1 models solid conductors; a real number greater than 1 when modeling stranded conductors. For a typical current, this is the number of times the conductor is wrapped around its core.
- You can change the direction of current flow in the conduction path by enabling **Reverse polarity**.
- A conduction path must be calculated before it can be shown. If the design has not been solved, or the conduction path has not been calculated, **Show conduction path** is disabled (greyed out). If the Physics task is valid, and conduction paths have not yet been calculated, you can [Generate Conduction Paths](#).

### 4.6.4.2.1. Setting a Terminal

#### Physics > Electromagnetic Conditions > Terminal

The **Terminal** condition is available for Physics Regions with the physics type set to Electromagnetics.

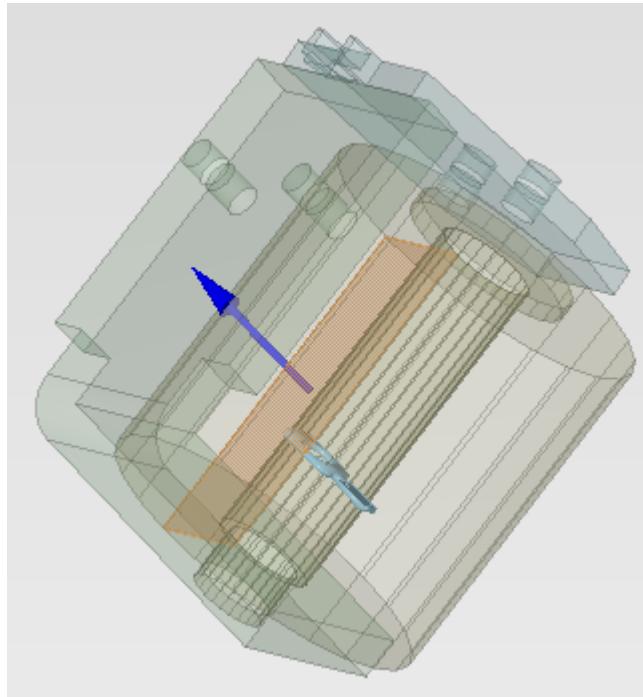
A terminal specifies the total current in a conduction path.

To set a terminal:

1. Select one or more faces or a surface body in the model for the location of the terminal.

You can assign a terminal to a face of a solid body, or to a surface body that intersects a solid body.

- For terminals assigned to a face, a conduction path requires one "in" face terminal and one "out" face terminal.
- For terminals assigned to a surface body such as the one in the image below, a conduction path requires one terminal.



In the image above, the arrow indicates the polarity (the direction of current flow) of the terminal condition.

**2.** Enter the **Number of conductors** being modeled.

- Enter 1 if you are modeling solid conductors.
- Enter a real number greater than 1 if you are modeling stranded conductors.

**3.** When the number of conductors is greater than 1, specify a **Fill factor**.

The fill factor is the adjustment/correction ratio the solver applies to the conductivity when calculating the losses for stranded conductors.

Because the fill factor determines whether the loss values are accurate, we recommend that the value is obtained from actual experiments with similar settings.

4. Checking **Reverse polarity** reverses the direction of current flow (indicated by the red arrow) in the coil current.
5. If the Physics task is valid, and conduction paths have not yet been calculated, you can **Generate Conduction Paths**. Otherwise, this button is disabled (greyed out).
6. Once the Physics task is solved, checking **Show conduction path** shows the conduction path of this terminal on the model. Otherwise, this check box is disabled (greyed out).

### 4.6.4.3. Insulating

An insulating boundary condition is used to model very thin sheets of perfectly insulating material between touching conductors. Current cannot cross an insulating boundary.

## 4.6.4.3.1. Setting an Insulating Boundary

The **Insulating** condition is available for Physics Regions with the physics type set to Electromagnetics.

### Physics > Electromagnetic Conditions > Insulating

An insulating boundary condition is used to model very thin sheets of perfectly insulating material between touching conductors. Current cannot cross an insulating boundary.

To set an insulating boundary, in the **Insulating** data panel, select faces or solid bodies in the model for the location of the insulating boundary.

## 4.6.4.4. Body Temperature

A **Body Temperature** condition can optionally be assigned to one or more solid bodies in a Magnetics design. Body temperature condition assignments override the [Ambient Temperature](#) setting on the Physics Options panel. The effect of temperature on bodies composed of temperature-dependent material in the design is calculated by the solver for simulation results. For example, the [electrical conductivity](#) of a material (the ability of a material to conduct electric current) typically decreases as temperature increases.

### 4.6.4.4.1. Setting a Body Temperature Condition

**Body Temperature** is available for Physics Regions with the physics type set to Electromagnetics.

### Physics > Electromagnetic Conditions > Body Temperature

A **Body Temperature** condition can optionally be assigned to one or more solid bodies in a Magnetics design. Body temperature condition assignments override the [Ambient Temperature](#) setting on the Physics Options panel.

**Note:** Only one body temperature condition can be assigned to any given location.

To set a body temperature:

1. In the **Body Temperature** data panel, select one or more solid bodies for the Location to which the body temperature will be applied.
2. Enter the desired value or expression in the **Temperature** field. The value must be greater than absolute zero.

## 4.6.5. Electric Conduction Loads and Constraints

Electric conduction conditions model the effects of electric fields. These conditions are available for [physics regions](#) with the physics type set to **Electric conduction**. In the Electromagnetics template, the physics type is set to **Electric conduction** when you select an Electric simulation and then apply a DC current.

The conditions available are:

- [Current](#)
- [Equipotential](#)
- [Body Temperature](#)
- [Voltage](#)

### 4.6.5.1. Current

A current load simulates the application of an electric current to the system.

You can also apply a current density, which applies the current per unit area.

## 4.6.5.1.1. Setting Current

### Physics > Electromagnetic Conditions > Current

To set a Current load:

1. [Select the location](#) for this condition.

You can apply current to edges, faces and vertices.

2. For **Current**, enter the current magnitude as a constant or expression; any expression must resolve to a constant.
3. For **Specification**, select how you want to apply the current. For a **Current Density**, AIM multiplies the current value by the mesh area of the selected geometry.

## 4.6.5.2. Equipotential

You can use the Equipotential constraint to apply a constant voltage condition to one or more equipotential surfaces of a model.

Any geometry assigned to an equipotential constraint cannot be used in any other load or constraint, including another Equipotential constraint, with the exception of a Voltage constraint.

### 4.6.5.2.1. Setting Equipotential

#### Physics > Electromagnetic Conditions > Equipotential

To set equipotential:

1. [Select the location](#) for this condition.

You can apply equipotential to one or more faces, edges, or vertices of 3D geometry.

**Note:** Any geometry assigned to an equipotential constraint cannot be used in any other load or constraint, including another Equipotential constraint, with the exception of a Voltage constraint.

## 4.6.5.3. Body Temperature

You can use a Body Temperature load to apply a temperature over the selected geometry. Body Temperature loads are used to calculate temperature-dependent material properties and thermal strains (if Thermal physics is not activated).

### 4.6.5.3.1. Setting a Body Temperature

#### Physics > Electromagnetic Conditions > Body Temperature

To set a body temperature load:

1. [Select the location](#) for this condition.

You can apply a temperature to one or more bodies.

2. For **Temperature**, enter a constant or expression directly into the field; any expression must resolve to a constant.

## 4.6.5.4. Voltage

A voltage constraint simulates the application of an electric potential to the system.

### 4.6.5.4.1. Setting Voltage

#### Physics > Electromagnetic Conditions > Voltage

To set a Voltage constraint:

1. **Select the location** for this condition.

You can apply voltage to faces, edges, bodies, or vertices of 3D geometry.

2. For **Voltage**, enter the voltage magnitude as a constant or expression; any expression must resolve to a constant.

## 4.6.6. Electrostatic Loads and Constraints

Electrostatic conditions model the effects of electrostatic charge. These conditions are available for **physics regions** with the physics type set to **Electrostatic**. In the Electromagnetics template, the physics type is set to **Electrostatic** when you set an Electric simulation and choose to simulate an electrostatic charge.

The conditions available are:

- Body Temperature
- Electric charge
- Equipotential
- Voltage

## 4.6.6.1. Body Temperature

You can use a Body Temperature load to apply a temperature over the selected geometry. Body Temperature loads are used to calculate temperature-dependent material properties and thermal strains (if Thermal physics is not activated).

### 4.6.6.1.1. Setting a Body Temperature

#### Physics > Electromagnetic Conditions > Body Temperature

To set a body temperature load:

1. **Select the location** for this condition.

You can apply a temperature to one or more bodies.

2. For **Temperature**, enter a constant or expression directly into the field; any expression must resolve to a constant.

## 4.6.6.2. Electric Charge

Electric charge enables you to specify the total charge (measured in Coulomb) as applied on a surface in a Electrostatic analysis. This charge will cause the body to experience a force in an electromagnetic field.

The SI unit of quantity of electric charge is the coulomb, which is defined as the quantity of charge that has passed through the cross section of an electrical conductor carrying one ampere within one second.

## 4.6.6.2.1. Setting an Electric Charge Condition

### Physics > Electromagnetic Conditions > Charge

To define a charge:

1. [Select the location](#) for this condition.

You can apply a charge to vertices, edges, or faces of a solid body.

2. Enter a value or expression for the quantity of electric charge. The SI unit of quantity of electric charge is the coulomb, which is defined as the quantity of charge that has passed through the cross section of an electrical conductor carrying one ampere within one second.

## 4.6.6.3. Equipotential

You can use the Equipotential constraint to apply a constant voltage condition to one or more equipotential surfaces of a model.

Any geometry assigned to an equipotential constraint cannot be used in any other load or constraint, including another Equipotential constraint, with the exception of a Voltage constraint.

### 4.6.6.3.1. Setting Equipotential

#### Physics > Electromagnetic Conditions > Equipotential

To set equipotential:

1. [Select the location](#) for this condition.

You can apply equipotential to one or more faces, edges, or vertices of 3D geometry.

**Note:** Any geometry assigned to an equipotential constraint cannot be used in any other load or constraint, including another Equipotential constraint, with the exception of a Voltage constraint.

## 4.6.6.4. Voltage

A voltage constraint simulates the application of an electric potential to the system.

### 4.6.6.4.1. Setting Voltage

#### Physics > Electromagnetic Conditions > Voltage

To set a Voltage constraint:

1. [Select the location](#) for this condition.

You can apply voltage to faces, edges, bodies, or vertices of 3D geometry.

2. For **Voltage**, enter the voltage magnitude as a constant or expression; any expression must resolve to a constant.

## 4.6.7. Optimization Conditions

Conditions available for optimization include:

1. [Manufacturing pull direction](#), where you specify the direction that the optimized product will be removed from the mold so as to ensure the integrity of the model.

2. **Size limit**, where you define the minimum member size of the supporting structures and maximum size of connected parts in the final design.
3. **Stress limit**, where you specify the maximum equivalent stress limit for the bodies specified in the Physics Options.
4. Symmetry, where you can enforce a design that is symmetric with respect to a user-defined plane. There are two types of symmetry, **reflective** and **rotational**.

## 4.6.7.1. Setting the Manufacturing Pull Direction

### Optimization Conditions > Stress Limit

The Manufacturing Pull Direction enables you to specify the direction that the optimized product will be removed from the mold so as to ensure the integrity of the model.

**Note:** You should use manufacturing pull direction in conjunction with size limits.

To set the pull direction:

1. Select or define a frame of reference under **Reference frame** if you want to use a reference frame other than the **Global reference frame**.
2. Select the **Axis** for the pull.
3. Select the **Direction** of the pull - **Along the axis**, **Opposite to axis**, or **Both directions**.

## 4.6.7.2. Specifying Size Limits

### Optimization Conditions > Size Limits

With Size Limits, you can define the minimum member size of the supporting structures and maximum size of connected parts in the final design.

You should specify a factor of two to three times the average mesh element size when trying to customize size limits. You can define only one set of size limits for each optimization analysis.

To set size limits:

1. Specify the **Minimum size**. Or choose **Minimum size specification** > **Program Controlled** to have the solver automatically set optimal default size limits.
2. Set the **Maximum size**. By default, AIM calculates the optimal maximum size, but you can also define one by selecting **Manual** and then specifying a **Maximum size**.

## 4.6.7.3. Setting the Stress Limit

### Optimization Conditions > Stress Limit

With Stress Limit, specify the maximum equivalent stress limit for the bodies specified in the Physics Region.

To specify a stress limit:

Enter a **Maximum stress**.

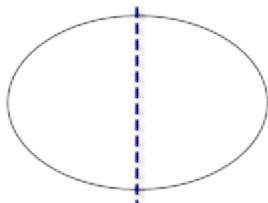
## 4.6.7.4. Specifying Reflective Symmetry

### Optimization Conditions > Reflective Symmetry

With symmetry constraints, you can enforce a design that is symmetric with respect to a user-defined plane.

Reflective symmetry is a type of symmetry in which the model is dimensionally reduced based on a mirror plane.

**Figure 4.6.7.4.1. Reflective Symmetry**



To set reflective symmetry:

Select or define a plane that divides the model into mirrored segments.

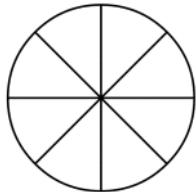
## 4.6.7.5. Specifying Rotational Symmetry

### Optimization Conditions > Rotational Symmetry

With symmetry constraints, you can enforce a design that is symmetric with respect to a user-defined plane.

Rotational symmetry, or cyclic symmetry, is a type of symmetry in which a model is symmetric about an axis, and therefore divided into identical segments. AIM can then reduce the model based on one sector, which is then applied to the others. Fan wheels, spur gears, and turbine blades are all examples of models that can benefit from rotational symmetry.

**Figure 4.6.7.5.2. Rotational Symmetry**



To set rotational symmetry:

1. For **Axial direction definition**, select how the axial direction is determined.
  - To define the axis through geometry, select **Geometry selection** and then, for **Axial direction based on geometry selection**, select a single face or edge to determine the axial direction for the cyclic symmetry.
  - To define the axis through a reference frame, select **Reference frame**. Then select or define a **Reference frame** and choose the axis for the **Axial direction**.
2. Specify the **Number of sectors**. For example, to divide the model into 8 identical segments, enter 8.

## 4.6.8. Polymer Extrusion Conditions

To properly set up a polymer extrusion simulation, you need to apply polymer flow-specific physics conditions at various boundaries (or bodies) within the geometry model. The physics conditions should represent sensible engineering assumptions of the true conditions for the problem being solved, and therefore should be considered carefully when defining the analysis.

When you use the polymer extrusion simulation **template**, several conditions are provided for you to get started, depending on your simulation goal.

Table 4.6.8.1. Polymer Extrusion Condition Types and Simulation Goals

Condition Type	Description	Simulation Goal		
		Analyze flow within die	Predict extrudate shape	Determine die lip shape
Extrudate Deformation	The deformation of the polymer extrudate.	*	*	*
Extrudate Exit	An open boundary where the extrudate is exiting the physics region.	*	*	*
Free Surface	A boundary that represents the outer surface of the polymer extrudate.	*	*	*
Die Deformation	The deformation of the die.			*
Inlet	An open boundary where fluid is entering the physics region.	*	*	*
Outlet	An open boundary where fluid is exiting the physics region.	*		
Wall	A closed boundary across which fluid cannot flow into or out of the physics region.	*	*	*
Symmetry	A closed boundary at a plane of symmetry across which fluid is not expected to flow.			

When setting up polymer flow conditions, you will need to make selections and provide entries that best describe your problem.

Setting	Description
Location	Assigns a fluid flow condition to a face of a physics region or to a selection set containing continuous faces that you have previously defined.
Flow	Defines the influence of the boundary on the flow (mass and momentum) equations.
Energy	Defines the influence of the boundary on the energy equations.

## 4.6.8.1. Inlet Conditions

Inlets define conditions where flow is expected to enter the solution domain.

To determine the type of inlet you should apply depends on the available flow information.

Flow specification	Additional information
Volume flow inlet	The inflow is calculated from a specified volumetric flow rate. The velocity profile can be controlled automatically, computed dynamically, or fully developed, in which case, a fully developed velocity profile is determined where it is assumed that all derivatives normal to the inlet are zero, except the pressure gradient, which is assumed to be constant.
Velocity inlet	The inflow is calculated from a specified normal velocity, tangential velocity is allowed or imposed to zero.
Mass flow inlet	The inflow is calculated from a specified mass flow rate. The velocity profile can be controlled automatically, computed dynamically, or fully developed, in which case, a fully developed velocity profile is determined where it is assumed that all derivatives normal to the inlet are zero, except the pressure gradient, which is assumed to be constant.

## 4.6.8.1.1. Specifying Mass Flow Inlets

### Extrusion > Fluid Flow Conditions > Inlet

To specify mass flow inlets for polymer extrusion simulations:

1. Select the location in your geometry for which you want to specify the inlet boundary condition.
2. For Incoming material, select a liquid material that passes through the inlet boundary. For co-extrusion simulations, you can only use Newtonian or generalized Newtonian fluid materials. In addition, you must define your multiple material assignments for the polymer extrusion physics region before you can specify the incoming material for your particular inlet fluid flow conditions.
3. For Flow specification, select Mass flow.
4. Define the Velocity profile, or keep the default settings.
5. Enter the Mass flow rate.
6. If your simulation includes a thermal analysis, make sure to specify the inlet Temperature.

## 4.6.8.1.2. Specifying Velocity Inlets

### Extrusion > Fluid Flow Conditions > Inlet

To specify velocity inlets for polymer extrusion simulations:

1. Select the location in your geometry for which you want to specify the inlet boundary condition.
2. For Incoming material, select a liquid material that passes through the inlet boundary. For co-extrusion simulations, you can only use Newtonian or generalized Newtonian fluid materials. In addition, you must define your multiple material assignments for the polymer extrusion physics region before you can specify the incoming material for your particular inlet fluid flow conditions.
3. For Flow specification, select Velocity.
4. For the Normal velocity, enter the velocity at which fluid is flowing into the domain.
5. Deselect Allow non-zero tangential velocity, if required.
6. If your simulation includes a thermal analysis, make sure to specify the inlet Temperature.

## 4.6.8.1.3. Specifying Volume Flow Inlets

### Extrusion > Fluid Flow Conditions > Inlet

To specify volume flow inlets for polymer extrusion simulations:

1. **Select the location** in your geometry for which you want to specify the inlet boundary condition.
2. For **Incoming material**, select a liquid material that passes through the inlet boundary. For co-extrusion simulations, you can only use Newtonian or generalized Newtonian fluid materials. In addition, you must define your multiple material assignments for the polymer extrusion physics region before you can specify the incoming material for your particular inlet fluid flow conditions.
3. For **Flow specification**, select **Volume flow**.
4. Define the **Velocity profile**, or keep the default settings.
5. Enter the **Volume flow rate**.
6. If your simulation includes a thermal analysis, make sure to specify the inlet **Temperature**.

## 4.6.8.2. Outlet Conditions

For polymer extrusion simulations, when the outlet represents an actual exit from which the fluid can freely flow, tangential velocities at the outlet should be allowed (such as a die exit). However, when the outlet is at the end of the computation domain, but not at the end of the channel in which the fluid flows, tangential velocities at outlets should not be allowed because the fluid is still confined after the outlet.

Flow specification	Additional information
Volume flow outlet	The outflow is calculated from a specified volumetric flow rate. Provide the flow rate in terms of the volume of material that flows through the outlet.
Pressure outlet	The pressure that you want to impose at the outlet will determine the level of pressure in the die.
Mass flow outlet	The outflow is calculated from a specified mass flow rate. Provide the flow rate in terms of the mass of material that flows through the outlet.

### 4.6.8.2.1. Specifying Pressure Outlet Conditions

#### Extrusion > Fluid Flow Conditions > Outlet

To specify pressure outlets:

1. **Select the location** in your geometry for which you want to specify the outlet boundary condition.
2. For **Flow specification**, select **Pressure**.
3. Enter the **Gauge static pressure**. This is the pressure that is maintained at a fixed specified value over the outlet boundary.

### 4.6.8.2.2. Specifying Mass Flow Outlet Conditions

#### Extrusion > Fluid Flow Conditions > Outlet

To specify mass flow outlets:

1. **Select the location** in your geometry for which you want to specify the outlet boundary condition.

2. For **Flow specification**, select **Mass flow**.
3. Enter the **Mass flow rate**.

### 4.6.8.2.3. Specifying Volume Flow Outlet Conditions

#### Extrusion > Fluid Flow Conditions > Outlet

To specify volume flow outlets:

1. [Select the location](#) in your geometry for which you want to specify the outlet boundary condition.
2. For **Flow specification**, select **Volume flow**.
3. Enter the **Volume flow rate**.

### 4.6.8.3. Walls for Polymer Extrusion Simulations

Wall boundaries [assigned](#) to surfaces of a fluid region prevent flow through those surfaces.

The conditions that best describe the forces applied to a wall are summarized below.

Condition at the wall	Option	Additional information
The fluid sticks to the wall. If the wall is moving, the fluid moves with the same velocity as the wall.	<a href="#">No Slip</a>	Specify the <b>Wall velocity</b> to be stationary or moving.
The fluid partially sticks to the wall.	<a href="#">Partial Slip</a>	Specify the <b>Wall velocity</b> to be stationary or moving. A slipping specification is required.
A frictionless wall.	<a href="#">Free Slip</a>	Components of velocity normal to free slip wall are zero. <b>Note:</b> For symmetry, consider using a symmetry condition.

### 4.6.8.3.1. Setting Wall Boundaries

#### Extrusion > Fluid Flow Conditions > Wall

To specify wall boundaries for polymer extrusion simulations:

1. [Select the location](#) (surface) in your geometry for which you want to specify the wall boundary. If the polymer extrusion simulation goal is set to **Analyze flow within a die**, then all walls should be selected. If the simulation goal is set to either **Predict extrudate shape** or **Determine die lip shape**, then only the die walls should be selected.
2. For **Flow specification**, specify slip conditions at the wall boundary. Options include [no slip](#), [partial slip](#), or [free slip](#).
3. For **Wall velocity**, specify whether the wall boundary is **Stationary** or **Moving**. For moving walls, you will need to specify the **X**, **Y**, and **Z** components of the wall translation motion.
4. For simulations where temperatures are important, under **Energy specification**, specify [the type of thermal conditions](#) at the wall.

## 4.6.8.3.2. Specifying No Slip Walls for Polymer Flows

### Extrusion > Fluid Flow Conditions > Wall

To specify no slip walls:

1. Select the location in your geometry for which you want to specify the wall boundary condition.
2. For Flow specification, select **No slip**.
3. For Wall velocity, select **Stationary** or **Moving**.

If you want to...	Then choose...
Model a non-stationary wall	<b>Wall velocity</b> and select <b>Moving</b> .
Express the wall velocity by specifying its directional components	Under <b>Translational Velocity</b> , specify the <b>X</b> , <b>Y</b> , and <b>Z</b> components of the wall translation motion.

4. For Energy specification, if your simulation includes a thermal analysis, select one of the [energy specification options](#) that best describes the system.

## 4.6.8.3.3. Specifying Partial Slip Walls for Polymer Flows

### Extrusion > Fluid Flow Conditions > Wall

To specify partial slip walls:

1. Select the location in your geometry for which you want to specify the wall boundary condition.
2. For Flow specification, select **Partial slip**, and under **Slip specification**:
  - Specify a **Velocity dependence** option, where a zero normal velocity component is imposed simultaneously using one of three relationships between the shear force and the tangential relative velocity. See [Table 21: Velocity-Dependent Slip Specification](#) on page 319 for details.
  - Specify a **Temperature dependence** option, when thermal considerations are important. See [Table 22: Temperature-Dependent Slip Specification](#) on page 321 for details.

Full slippage is assumed if the friction coefficient is zero.

3. For Wall velocity, select **Stationary** or **Moving**.

If you want to...	Then choose...
Model a non-stationary wall	<b>Wall velocity</b> and select <b>Moving</b> .
Express the wall velocity by specifying its directional components	Under <b>Translational Velocity</b> , specify the <b>X</b> , <b>Y</b> , and <b>Z</b> components of the wall translation motion.

4. For Energy specification, if your simulation includes a thermal analysis, select one of the [energy specification options](#) that best describes the system.

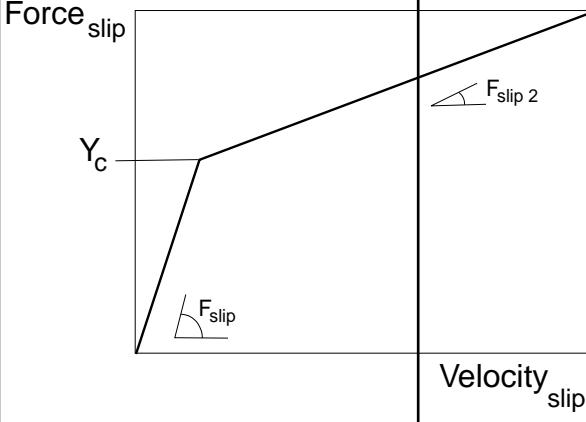
## 4.6.8.3.3.1. Velocity and Temperature Dependent Slip Specifications for Wall Conditions

When the slip condition is applied to non-moving parts, you can specify:

- The **Velocity dependence** field, where a zero normal velocity component is imposed simultaneously using one of three relationships between the shear force and the tangential relative velocity. See [Table 21: Velocity-Dependent Slip Specification](#) on page 319 for details.
- The **Temperature dependence** field, when thermal considerations are important. See [Table 22: Temperature-Dependent Slip Specification](#) on page 321 for details.

Table 4.6.8.3.3.1.1. Velocity-Dependent Slip Specification

If ...	... then ...	Description
You have a fluid that can support any shear stress or that the shear stress at the wall remains well under the maximum shear stress.  This is to say that you have a simple, proportional relationship between the slipping force and the slipping velocity (the default).	Select the <b>Navier Law</b> option. When this option is enabled, you need to specify the <b>Friction coefficient</b> for the slip condition.	This law is defined as: $f_s = -F_{slip}(\mathbf{v}_s - \mathbf{v}_{wall})$ where $\mathbf{v}_s$ is the tangential velocity of the fluid, $\mathbf{v}_{wall}$ is the tangential velocity of the wall, and $F_{slip}$ is the friction coefficient. $\mathbf{v}_{wall}$ is assumed to be zero, by default.  Note that full slip is obtained when $F_{slip} = 0$ . $\mathbf{v}_{wall}$ is zero if the <b>Wall velocity</b> is set to <b>Stationary</b> .

If ...	... then ...	Description
<p>You have a fluid that can support a given critical stress threshold and change its behavior above that stress threshold.</p> <p>Below the threshold, a specific coefficient determines the slipping force/velocity relationship, and above the threshold, another coefficient (usually smaller than the first) defines the force/velocity relationship.</p> <p>It is assumed that the shear stress at the wall can reach the threshold and that, above this threshold, the slipping force can still increase.</p> 	<p>Select the <b>Threshold Law</b> option. When this option is enabled, you need to specify the <b>First friction coefficient</b>, the <b>Second friction coefficient</b>, and the <b>Critical stress</b> for the slip condition. The critical stress is the critical force density at which the friction coefficient changes. When this stress is exceeded, the second friction coefficient is used.</p>	<p>This law is defined as:</p> $f_s = \begin{cases} -F_{\text{slip}}(\mathbf{v}_s - \mathbf{v}_{\text{wall}}), & \text{if } \mathbf{v}_s - \mathbf{v}_{\text{wall}} < y_c \\ -y_c - F_{\text{slip},2}(\mathbf{v}_s - \mathbf{v}_{\text{wall}} - \frac{y_c}{F_{\text{slip}}}), & \text{if } \mathbf{v}_s - \mathbf{v}_{\text{wall}} \geq y_c \end{cases}$ <p>where <math>F_{\text{slip}}</math> and <math>F_{\text{slip},2}</math> are two different slip coefficients, and <math>y_c</math> is the critical stress at which the slipping behavior changes.</p>

If ...	... then ...	Description
You have a fluid that can support a critical shear stress, but no more. Above the critical stress, the fluid deteriorates or molecules align with the flow, but the slipping force cannot increase.  It is assumed that the shear stress at the wall can approach the critical shear stress, but that the fluid is simply unable to sustain more than this limit.	Select the <b>Asymptotic Law</b> option. When this option is enabled, you need to specify the <b>Friction coefficient</b> and the <b>Scaling factor</b> for the slip condition.	This law is defined as: $f_s = -F_{slip} \left[ 1 - \exp \left( -\frac{v_s - v_{wall}}{v_c} \right) \right]$ where $v_c$ is a scaling factor with the dimensions of the velocity. $v_c$ affects the slope of the slip-velocity curve: a small value creates a sharp curve; a large value creates a broad curve.

Force<sub>slip</sub>

Velocity<sub>slip</sub>

Table 4.6.8.3.3.1.2. Temperature-Dependent Slip Specification

If ...	... then ...	Description
The slip conditions at the wall displays no temperature dependence (the slipping coefficient is constant, or $H(T) = 1$ ).	Select <b>None</b> .	No temperature dependence.
The temperature-dependent slip conditions at the wall obey the Arrhenius law.	Select the <b>Arrhenius Law</b> option.	The Arrhenius law is given as $H(T) = \exp \left[ \alpha \left( \frac{1}{T - T_0} - \frac{1}{T_\alpha - T_0} \right) \right]$ where $\alpha$ = the ratio of the activation energy to the thermodynamic constant $T_\alpha$ = a reference temperature for which $H(T) = 1$

If ...	... then ...	Description
The temperature-dependent slip conditions at the wall obey a first order approximation of the Arrhenius law.	Select the <b>Arrhenius Approximate Law</b> option.	<p>The approximate Arrhenius law is written as follows:</p> $H(T) = \exp [ -\alpha(T - T_\alpha) ]$ <p>The behavior with this formulation is similar to that described by the Arrhenius law in the neighborhood of <math>T_\alpha</math>. This formulation is valid as long as the temperature difference <math>T - T_\alpha</math> is not too large.</p>

## 4.6.8.3.4. Specifying Free Slip Walls for Polymer Flows

### Extrusion > Fluid Flow Conditions > Wall

To specify free slip walls:

1. Select the location in your geometry for which you want to specify the wall boundary condition.
2. For Flow specification, select **Free slip** for frictionless walls.
3. For Energy specification, if your simulation includes a thermal analysis, select one of the [energy specification options](#) that best describes the system.

## 4.6.8.4. Symmetry

A [symmetry](#) boundary imposes constraints that mirrors the expected pattern of the flow or thermal solution on either side of it. A symmetry boundary is equivalent to imposing zero values for the normal velocity and tangential force.

### 4.6.8.4.1. Setting Symmetry

#### Extrusion > Symmetry Conditions > Symmetry

To specify symmetry boundaries for polymer extrusion simulations:

Select the location in your geometry for which you want to specify **Symmetry**. Note that the location of a valid symmetry boundary should be planar.

## 4.6.8.5. Restrictors

Restrictors, in die extrusion applications, are solid structures placed in the polymer flow region to obtain optimum melt flow distribution. There are some limitations when using restrictors in AIM:

- Restrictors should not touch the inlet and die exit.
- Restrictors must be located in the die region (and not the extrudate).
- Restrictors are not permitted if the die design includes determining die lip shape.

To include restrictors in your polymer extrusion simulation, see [Setting Restrictors](#) on page 323.

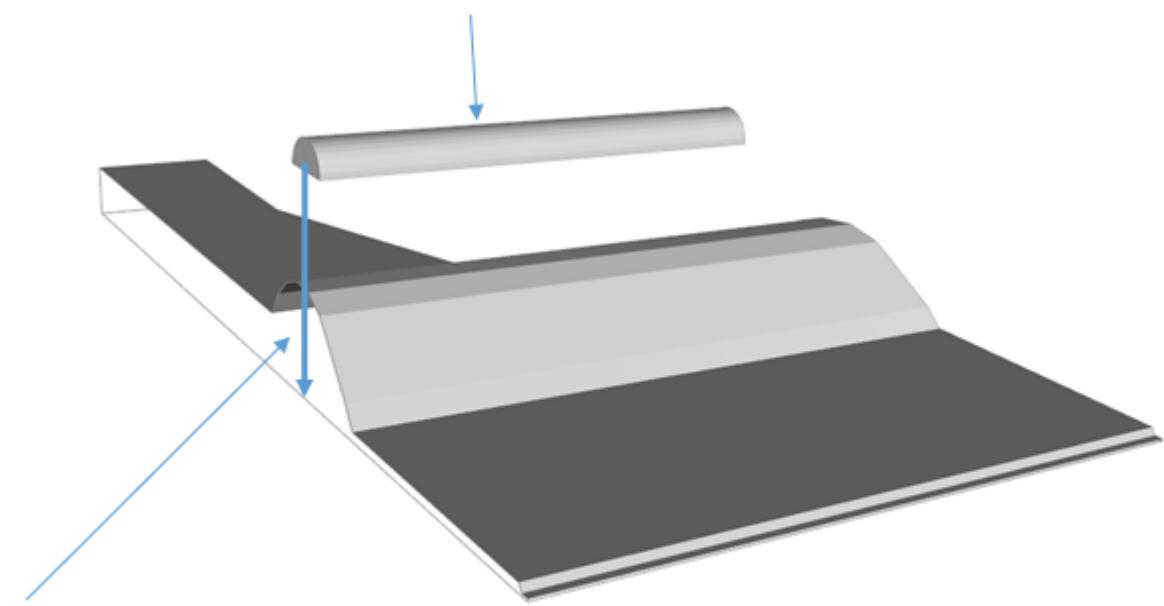
## 4.6.8.5.1. Setting Restrictors

### Extrusion > Fluid Flow Conditions > Restrictor

To specify restrictors for polymer extrusion simulations:

1. Select the location (body) where the restrictor is applied.
2. For the **Flow specification**, select **No slip** if the fluid sticks to the restrictor. Select **Partial slip** if the fluid partially sticks to the restrictor and enter the **Maximum slipping stress**. If the stress falls below this value, a full stick condition is assumed, otherwise a slip condition is modeled using this value.
3. Specify the X, Y, and Z components of the **Initial Displacement** of the restrictor. Different displacement values can be used for parametric studies to find the best location to produce the most uniform flow distribution at, for example, the die exit.

**Restrictor (located outside the die)**



**Initial displacement**  
to be applied so that the restrictor lays  
inside the flow domain

A maximum of 5 restrictors can be defined. AIM can detect whether a given velocity node is in the real flow domain or is overlapped by a restrictor. For example, in the Results task for **contours** and **isosurfaces**, in the **Variable** field, choose **Category** and select any one of **Inside 1** through **Inside 5** variables. A value of 1 means the node is inside the restrictor, while zero means we are in the fluid.

## 4.6.8.6. Extrudate Deformation Conditions

In polymer extrusion simulations, you can [specify the type of deformation to be applied to the extrudate](#). To cope with the displacement of free surfaces, a local mesh [deformation method](#) must be applied to the volume representing the extrudate. The purpose of a remeshing technique is to relocate internal nodes according to the displacement of boundary nodes. Remeshing techniques control mesh deformations in order to avoid unacceptable element shapes.

## 4.6.8.6.1. Setting Extrudate Deformations

### Extrusion > Fluid Flow Conditions > Extrudate Deformation

To specify extrudate deformations for polymer extrusion simulations:

1. **Select the location** (volume or body) in your geometry for which you want to specify the extrudate deformation.
2. For **Deformation method**, specify the **type of deformation** to be applied to the extrudate.

## 4.6.8.6.2. Extrudate Deformation Methods

Option	Details
<b>Program controlled</b>	Allow AIM to choose the most appropriate remeshing technique based on the specific nature of the simulation settings.
<b>Optimesh</b>	The Optimesh algorithm is based on a "minimum energy" rule and belongs to the family of elliptic remeshing rules and, as such, handles internal nodal displacements as variables. Optimesh is often more robust because the angular deformation of each individual element is under control of the remeshing rule.  For 3D extrusion problems, the domain is sliced in the direction of extrusion, generating a series of 2D meshes on which the Optimesh rule can be applied.
<b>Streamwise</b>	The Streamwise algorithm assumes a regularity of the domain, which can be seen as a series of mesh lines emitted from the die lip section.  Along these mesh lines, which can be on the boundary, on the interface between coextruded fluids, or inside the domain, the streamwise method aligns the mesh with the local velocity vector. The method is useful in situations where large isotropic deformations occur. A typical example is 3D fiber spinning with large drawdown ratios.
<b>Elastic</b>	This remeshing technique belongs to the family of elliptic remeshing schemes and has the advantage of complete topological generality, since no regular organization of the mesh is required.  In the interior of the domain, a nonlinear small displacement pseudo-elastic problem is solved in the deformed configuration and not on the original domain.  This technique should not be used when the extrudate is not maintained in the extrusion direction by planes of symmetry, slipping surfaces, or rigid insert(s).
<b>Elastic with torsion control</b>	This is a version of the Elastic method, but with a torsion factor that adds a rotation stiffness with respect to the initial configuration. This avoids deforming elements too much with respect to their initial shape.

## 4.6.8.7. Extrudate Exits

For polymer extrusion simulations where you want to predict the shape of the extrudate, or determine the die lip shape, you can also [specify the conditions of the extrudate](#) at the exit boundary.

### 4.6.8.7.1. Setting Extrudate Exits

#### Extrusion > Fluid Flow Conditions > Extrudate Exit

To specify extrudate exits for polymer extrusion simulations:

1. [Select the location](#) (surface) in your geometry for which you want to specify the extrudate exit.
2. For **Fixed edges at the extrudate exit**, specify whether to fix **All** or **None** of the edges of the extrudate exit in the frame of a die lip shape design. You must select **All** when determining the die lip leading to the desired extrudate shape. You should select **No** for predicting the extrudate shape exiting a given die geometry.
3. For **Flow specification**, specify the flow conditions at the exit. There may be **No force** acting at the exit, or there can be a **Take-up force**, a **Take-up velocity**, or a **Take-up force per unit area**, when the extrudate is pulled by the extrusion line.

## 4.6.8.8. Free Surface Boundaries

In an extrusion problem, where the shape of the extrudate is not known in advance, a [free surface](#) is used to represent the outer surface of the extrudate. A free-surface problem involves a boundary whose position is computed as part of the solution, since it is not known in advance.

### 4.6.8.8.1. Setting Free Surfaces

#### Extrusion > Fluid Flow Conditions > Free Surface

To specify free surfaces for polymer extrusion simulations:

1. [Select the location](#) (surface) in your geometry for which you want to specify the free surface.
2. For **Pressure**, specify a value for the pressure conditions on the free surface.
3. For simulations where temperatures are important, under **Energy specification**, specify the type of [thermal conditions](#) at the free surface.

## 4.6.8.9. Die Deformation Conditions

For polymer extrusion simulations that include a determination of the die lip shape, you can describe how the die will be deformed in the simulation using a die deformation condition.

The die deformation condition is [applied](#) to a specific volume or body in the simulation that corresponds to the die.

The die deformation condition applies localized remeshing to the die volume in addition to the extrudate, because the die shape is unknown beforehand and will be computed based on the given extrudate shape. Remeshing must be applied to the extrudate as well, because only the exit shape is known.

**Note:** You cannot model a thermal physics region if it is adjacent to a die deformation. See [Considerations for Region Interfaces and Polymer Extrusion](#) on page 376 for more information on this and other restrictions.

## 4.6.8.9.1. Setting Die Deformations

### Extrusion > Fluid Flow Conditions > Die Deformation

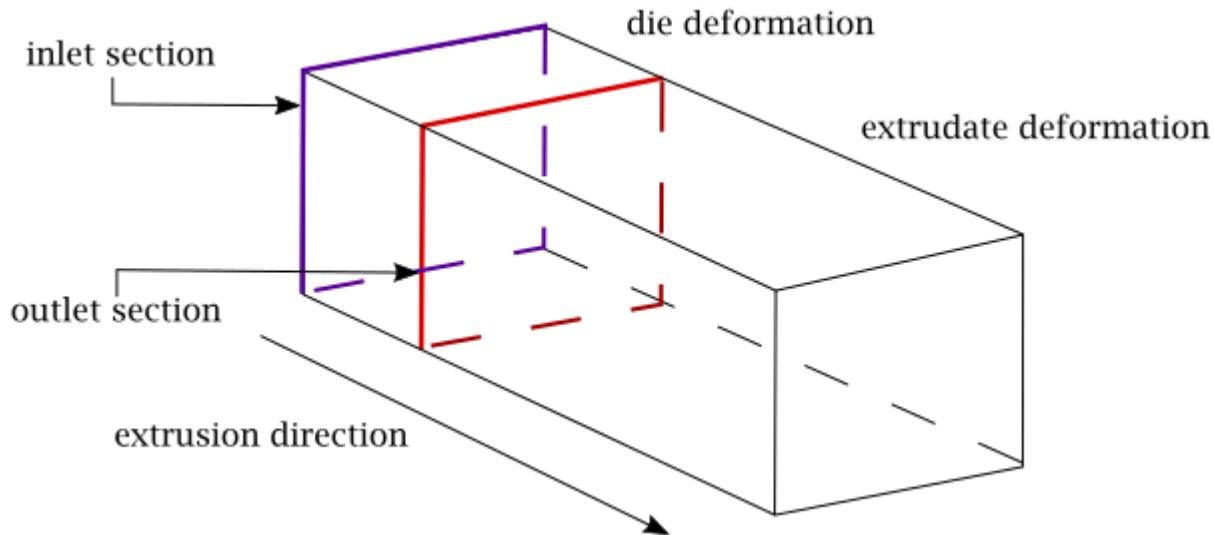
To specify die deformations for polymer extrusion simulations:

1. Select the location (volume or body) in your geometry for which you want to specify the die deformation.
2. For **Deformation method**, specify the type of deformation to be applied to the die. You can choose a **Constant** cross section, or a **Variable** cross section that varies linearly from the entry to the exit.
3. For the **Mesh type**, specify the type of mesh to be applied to the die. Select **Sliceable** when the mesh can be divided into a series of parallel slices that are perpendicular to the main flow direction, or select **Not sliceable** when it cannot, or select **Program controlled** to let AIM decide.

## 4.6.8.9.2. Considerations When Setting Die Deformation Conditions

There are occasions when a complex geometry and the required flow conditions create complications while setting up a die deformation (die lip design) extrusion simulation. For example, you may have bodies that are not going to be used in the problem setup for the die deformation.

Such complications come into play when considering a die deformation condition defined on one or more bodies with respect to surfaces upstream of the die deformation (also known as *inlet* section) and surfaces downstream of the die deformation (also known as *outlet* section).

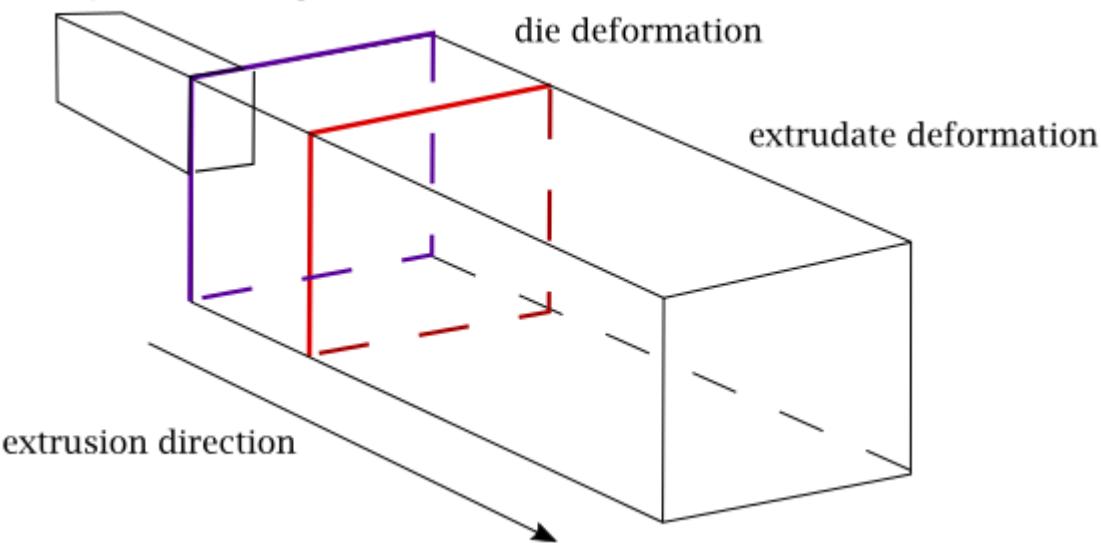


For such problematic scenarios, consider the following limitations:

- While you can have an inlet condition for the inlet section of a die deformation, or have an intersection with an upstream non-deformable body (that is used in the simulation), you cannot combine these conditions.

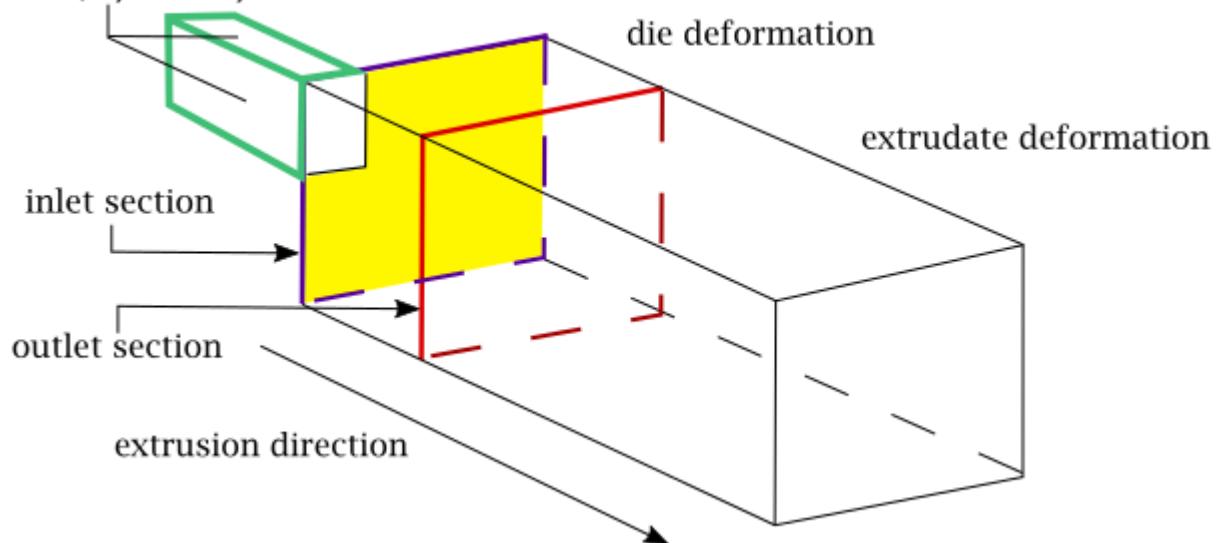
- You *cannot* have an unused body or a thermal region in contact with the die

### unused body/ thermal region



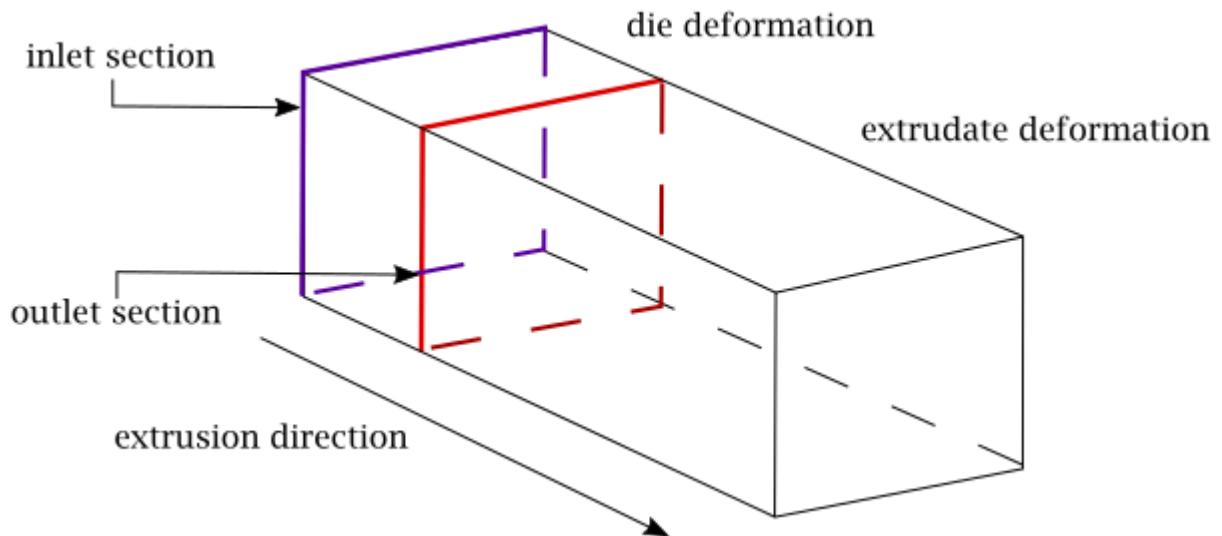
- You *cannot* have a wall condition or a symmetry condition partially or totally in the inlet section. That is, you cannot have a portion of the inlet section (in yellow) where a wall condition could be

### wall/symmetry condition

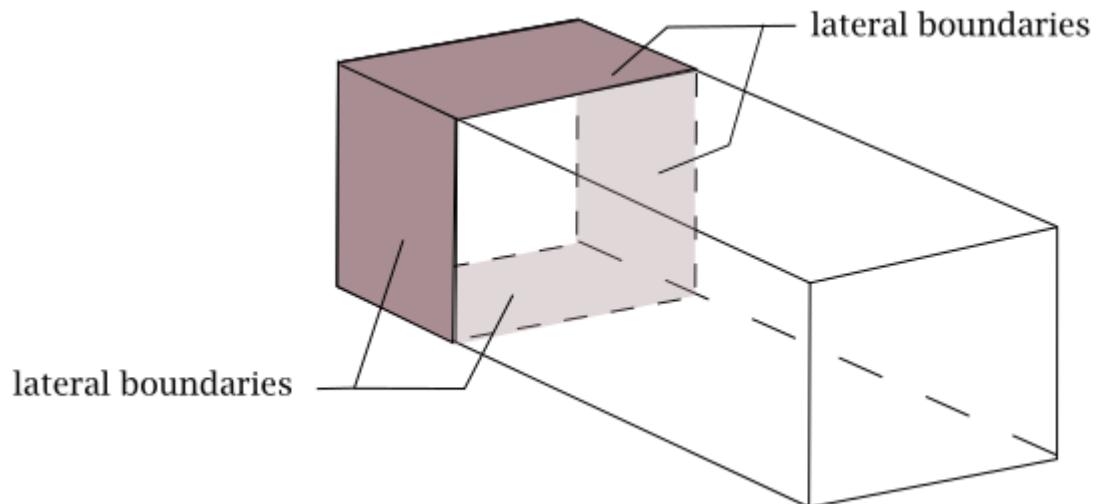


- The outlet section should contain only the faces at the intersection of the die deformation and the extrudate deformation objects.

- The inlet and the outlet sections must be roughly perpendicular to the extrusion



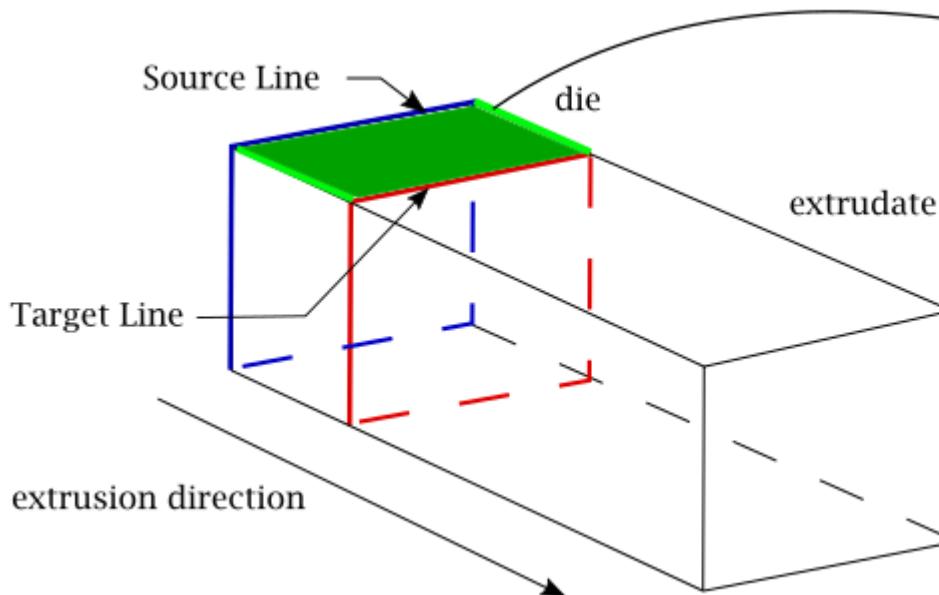
For the die deformation, any surface that is not in the inlet section or the outlet section, is referred to as a lateral



boundary.

To correctly set up the problem, you need to define enough lateral flow and/or symmetry conditions. The lateral conditions must be split into different boundary conditions, so that each condition is defined on a surface that has four bounding lines:

- One source line, intersecting the inlet section
- One target line, intersecting the outlet section
- Two generation lines (starting on the inlet section and ending on the outlet section)



Generation line 1

**Note:** You must define at least three lateral, non-overlapping boundary conditions.

The polymer extrusion solver implicitly assumes that the shape of the inlet and outlet sections are very similar (ideally, they should initially overlap) and the finite element faces of the lateral boundaries are perpendicular to the direction of extrusion. Also note that the polymer extrusion solver performs a check to see if the lateral faces have an angle greater than 60° with the direction of extrusion.

If the non-sliceable technique cannot be easily employed in an die deformation extrusion problem setup, due to a sufficiently complicated geometry, you can try to simplify the geometry appropriately.

If problems arise during the setup and solution of problematic scenarios, refer to the following table for relevant messages you might encounter, and the recommended actions you can perform.

Message	Recommended Action
The inlet and outlet sections must not come into contact with each other.	Make sure that the inlet section does not come into contact with the outlet section.
A lateral boundary is not in contact with either the inlet section or the outlet section.	Make sure the inlet section and the outlet section each are in contact with all lateral boundaries.
Several lateral boundaries are not in contact with either the inlet section or the outlet section.	
There are not enough lateral boundaries: define at least three lateral boundaries.	Make sure the inlet section and the outlet section each are in contact with all lateral boundaries.
A lateral boundary is in contact with more than two other lateral boundaries.	Make sure that each lateral boundary is in contact with two other lateral boundaries.
Several lateral boundaries are in contact with more than two other lateral boundaries.	
One lateral boundary is in contact with only a single other lateral boundary, or none at all.	Make sure that each lateral boundary is in contact with two other lateral boundaries.
Several lateral boundaries are in contact with only a single other lateral boundary, or none at all.	

Message	Recommended Action
The inlet and outlet sections are not parallel.	Make sure that the inlet section is more or less parallel with the outlet section.
One lateral boundary contains faces that are not properly oriented with respect to the direction of the extrusion.  Several lateral boundaries have faces that are not properly oriented with respect to the direction of the extrusion.	Make sure that lateral boundary faces are more or less parallel with the direction of the extrusion.

## 4.6.8.10. Energy Specification for Polymer Extrusion Flows

Energy specification	Description	Heat flux at the boundary $q_w$ calculated from the boundary condition
<b>Insulated</b>	The boundary is perfectly insulated $q_w = 0$ (adiabatic).	
<b>Heat flux</b>	The heat flux (energy transfer rate $q_w = q_{specified}$ per unit area [W/m^2 or equivalent]) is specified. Entering a value of 0 is equivalent to a perfectly insulated boundary.	
<b>Temperature</b>	The boundary temperature is specified.	The solver discretizes $q_w$ based on the local temperature gradient adjacent to the boundary.
<b>Convection</b>	Heat transfer through the boundary occurs through external convection. The heat transfer coefficient and external convection temperature are specified.	$q_w = h(T_{convection} - T_w)$  $h$ is the specified heat transfer coefficient  $T_{convection}$ is the specified convection temperature  $T_w$ is the boundary temperature as computed by the solver
<b>Heat Flux and Convection</b>	Heat transfer through the boundary occurs through both heat flux and external convection.	$q_w = q_{specified} + h(T_{convection} - T_w)$

## 4.6.9. Polymer Blow Molding Conditions

To properly set up a polymer blow molding simulation, you need to apply polymer flow-specific physics conditions at various boundaries (or bodies) within the geometry model. The physics conditions should represent sensible engineering assumptions of the true conditions for the problem being solved, and therefore should be considered carefully when defining the analysis.

When you use the polymer blow molding simulation template, the following fluid flow conditions are provided for you to get started:

- **Pressure** where you will apply an inflation pressure on the polymer sheet.
- **Polymer edge** where borders (lines) of the sheet or parison are fixed (clamped) or quite free to move.

A structural condition is provided to define the mold motion (considered as a rigid body with no deformation). You can specify the mold as:

- **Fixed**
- **Velocity Driven**
- **Force Driven**

## 4.6.9.1. Setting Pressure Conditions

### **Blow Molding > Fluid Flow Conditions > Pressure**

In a blow molding problem, a pressure condition is used to represent the blowing air that will inflate and stretch the polymer parison. Such problems involve a boundary whose position is computed as part of the solution, since it is not known in advance.

To specify the pressure for polymer blow molding simulations:

1. Select the location (surface or physics region) in your geometry for which you want to specify the pressure.
2. Specify the **Inflation pressure** acting along the normal to the parison. An arrow in the graphical window represents the inflation pressure. If the direction of the arrow is not correct, change the sign of the pressure.
3. For simulations where temperatures are important, under **Energy specification**, specify the type of thermal conditions at the free surface.

## 4.6.9.2. Setting Polymer Edge Conditions

### **Blow Molding > Fluid Flow Conditions > Polymer Edge**

In a blow molding problem, a polymer edge represents the border of the polymer sheet that will be held in place (fixed) or free to move.

To specify the polymer edge condition for a polymer blow molding simulation:

1. Select the location (edges) of the polymer that will be held in place or free to move.
2. For **Type**, specify whether that border will be **Fixed** (held in place) or **Free** to move.

## 4.6.9.3. Setting Fixed Mold Conditions

### **Blow Molding > Structural Conditions > Fixed Mold**

To define a fixed mold:

Select the body in the structural physics region of your geometry that is not in motion.

## 4.6.9.4. Setting Force Driven Mold Conditions

### **Blow Molding > Structural Conditions > Force Driven Mold**

For a mold that is in motion due to an applied force, specify the following:

1. For the **Location**, select the body in the structural physics region of your geometry that has an applied force.

2. For **Mold mass**, enter the mass of the mold. You must specify a constant value.
3. Enter the **Initial velocity** imposed on the mold.
4. Enter the **Translational force** acting on the mold. The force is aligned with the direction of displacement.
5. Specify the **Maximum displacement** due to the force on the mold. When the maximum displacement is achieved, the motion of the mold will stop. This value must be positive such that the motion is limited only in the direction of displacement.
6. Specify the X, Y, and Z directions for the displacement.

## 4.6.9.5. Setting Velocity Driven Mold Conditions

### **Blow Molding > Structural Conditions > Velocity Driven Mold**

For a mold that is in motion due to some imposed velocity, specify the following:

1. For the **Location**, select the body in the structural physics region of your geometry that has an imposed velocity.
2. Specify the X, Y, and Z components of the **Translational velocity**.

## 4.6.9.6. Setting Symmetry Conditions

### **Blow Molding > Symmetry Conditions > Symmetry**

A symmetry boundary imposes constraints that mirrors the expected pattern of the flow or thermal solution on either side of it. A symmetry boundary is equivalent to imposing zero values for the normal velocity and tangential force.

To specify symmetry boundaries for polymer blow molding simulations:

1. Select the **Location** in your geometry for which you want to specify Symmetry. Symmetry must be defined on edges bordering the polymer sheet.
2. Select a normal aligned with the X, Y, Z direction or other arbitrary direction. This normal in combination with the location of the symmetry condition fully defines the plane of symmetry. The plane of symmetry is displayed in the graphical window and allows you to check its definition.

## 4.7. Initial Conditions

Initial conditions consist of required information necessary to initialize a simulation.

For example, before performing a computational fluid dynamics (CFD) simulation, you must provide an initial guess for the solution. In order to obtain optimal results, solution initialization is required for:

- [fluid flow simulations](#)
- [polymer extrusion simulations](#)
- [polymer blow molding](#)

Because a time-dependent thermal analysis involves loads that are functions of time, you must first establish the [initial temperature distribution](#).

### 4.7.1. Initial Conditions for Fluid Flows

Before solving a steady-state fluid-flow simulation, an initial guess must be provided for the solution. The [initial conditions](#) will generally not affect the converged solution, but may affect the path to convergence. In some cases, a poor initial guess may lead to [divergence](#) during startup.

There may be situations where if the physics supports multiple solutions, the initial guess may affect which solution is obtained. For example, a converging-diverging nozzle may have purely supersonic or transonic flow with a shock; if the supersonic solution is desired, you should choose user-specified initial conditions which are supersonic throughout the geometry.

For time-dependent simulations, [initialization](#) provides the initial conditions for the simulation. If you are interested in how the transient flow field evolves from a known set of initial conditions, you must specify these initial conditions directly.

## 4.7.1.1. Initializing the Solution

### Physics > Initial Conditions

To initialize the solution, you can keep the default (**Automatic**) settings or you can specify the initial conditions for each of the equations. In most situations, the automatic initialization options behave well. However, there are situations where it may be useful to specify the initial flow.

To set the initial conditions:

**1. Set the Flow specification.**

- **Automatic** interpolates from boundary condition information to the interior by solving a Laplace equation for velocity and pressure.
  - To control how many iterations are used to solve the Laplace equation, click  to display and set the **Number of iterations**.
  - For situations where the automatic velocity initialization gives a good initial field for the velocity direction but not the magnitude, click  to display **Maintain constant velocity magnitude** and enable this setting. This replaces the velocity magnitude with the average of user-specified boundary values.
- **Static pressure and velocity** requires you to explicitly specify the initial **Gauge static pressure** and **Velocity** values.

**2. Initialize turbulence conditions.**

- **Automatic** initializes the turbulence field automatically using one of two methods, depending on the **Turbulence Specification**.  **Use averaged quantities** averages the user-specified turbulence boundary values; otherwise the specified **Turbulence intensity** and **Turbulence viscosity ratio** are used together with the initial velocity field.
- Specifying a [turbulence specification method](#) will give you the option to explicitly specify the initial turbulence values.

**3. Initialize thermal conditions.**

- **Automatic** initializes the temperature field by averaging user-specified energy boundary values.
- **Temperature** requires you to explicitly specify the initial temperature values.

## 4.7.2. Initial Conditions for Time-Dependent Fluid Flows

Depending on what information you are looking for in the time-dependent simulation, [initial conditions](#) may need to be specifically provided. For instance, if you would like to capture startup transient behavior from known initial conditions, you need to specify those initial conditions. On the other hand, if you are interested in longer-term transient behavior such as resolving a periodic-in-time solution, then the initial conditions are less important. By default, AIM assumes you are interested in the second type of simulation, and therefore applies **Automatic** initial conditions instead.

### 4.7.3. Initial Conditions for Time Dependent Thermal

A time-dependent thermal analysis involves loads that are functions of time. The first step in applying transient thermal loads is to establish initial temperature distribution at Time = 0.

The default initial condition for a time-dependent thermal analysis is a uniform temperature of 22°C or 71.6°F. You can change this to an appropriate value for your analysis. An example might be modeling the cooling of an object taken out of a furnace and plunged into water.

In the first iteration of a time-dependent thermal analysis, this initial temperature is used as the starting temperature value for the model except where temperatures are explicitly specified. In addition this temperature is also used to evaluate temperature-dependent material property values for the first iteration.

#### 4.7.3.1. Modifying the Initial Condition

##### **Physics > Initial Conditions**

A time-dependent thermal analysis involves loads that are functions of time. The first step in applying transient thermal loads is to establish the initial temperature distribution at Time = 0. The default initial condition for a time-dependent thermal analysis is a uniform temperature of 22°C or 71.6°F. You can change this to an appropriate value for your analysis.

1. [Select the location](#) for this initial temperature.  
You can apply an initial temperature to any volume.
2. Enter a value for the **Temperature**. You can enter a constant or an expression; any expression must resolve to a constant.

### 4.7.4. Initial Conditions for Polymer Extrusion Flows

Polymer extrusion flow simulations require you to [provide an initial temperature](#) for the flow field solution to aid in producing a final solution.

#### 4.7.4.1. Setting Initial Temperatures for Polymer Extrusion Flows

##### **Extrusion > Initial Conditions > Temperature**

To specify initial temperatures for polymer extrusion simulations:

1. [Select the location](#) (volume or body) in your geometry for which you want to specify the temperature.
2. Indicate the initial **Temperature** for the simulation.

### 4.7.5. Initial Conditions for Polymer Blow Molding

Polymer blow molding simulations require you to provide an [initial temperature](#) for the flow field solution and/or an [initial layer thickness](#).

#### 4.7.5.1. Setting Initial Temperatures for Polymer Blow Molding

##### **Blow Molding > Initial Conditions > Temperature**

To specify initial temperatures for polymer blow molding simulations:

1. Select the location (surface or body) in your geometry for which you want to specify the temperature.
2. Indicate the initial **Temperature** for the simulation.

## 4.7.5.2. Setting the Initial Layer Thickness for Polymer Blow Molding

### **Blow Molding > Initial Conditions > Initial Layer Thickness**

The initial layer thickness of the polymer sheet provides initial conditions for a blow molding simulation. Several layers of different materials may be specified by using multiple objects.

To initialize the solution, specify the following:

1. Select the **Location** (surface body) in your geometry to represent the polymer sheet or any part of it.
2. Select the **Layer material** making up the polymer sheet.
3. Enter the **Thickness** of the polymer sheet used to initialize the solution.

## 4.8. Interface Conditions

In AIM, you can set up interface conditions that enable interaction between bodies, physics regions, or physics tasks. Available types of interfaces are:

- [contacts](#) (between bodies),
- [joints](#) (between bodies),
- [region interface conditions](#) (between physics regions), and
- [physics coupling interfaces](#) (between physics tasks).

For information on identifying connections (contacts, joints, and springs), either for a selected body or bodies, or for a specific behavior, see [Identifying Connections](#) on page 370.

## Contacts

A contact is a type of connection between two solid, surface, or line bodies. Contacts can be generated manually, by directly setting up a contact, or automatically, through the interface generator.

To define a contact, you specify a set of options specific to that contact, then define a set of behaviors for that contact. When you create a contact, a default set of behaviors is created by default, which you can then modify. You can apply a set of contact behaviors to more than one contact.



[Generating Contacts](#)



[Nonlinear Contacts](#)

## Joints

A joint typically serves as a junction where bodies are joined together. Joint types are characterized by their rotational and translational degrees of freedom as being fixed or free.

To define a joint, you specify a set of options specific to that connection, then define a set of behaviors for that joint. When you create a joint, a default set of behaviors is created by default, which you can then modify. You can apply a set of joint behaviors to more than one joint.

# Region Interface Conditions

Fluid flow or polymer extrusion simulations may require [region interface conditions](#) to be defined in order to manage various types of discontinuities between different physics regions or within a single physics region.

## Physics Coupling

The [physics coupling interface](#) transfers data from a solved Physics Solution task into another Physics Solution task. On each physics coupling interface, you can transfer fluid force, temperature, or heat rate data between different types of simulations. For details, see [Data Transfers Possible Using the Physics Coupling Interface](#).

### 4.8.1. Beams

A beam model is a 1D representation of a 3D solid model in 3D space. When your design goals include rotation bending, using a beam element offers a computationally efficient solution in comparison to solid or surface bodies. A beam model is represented by a line body which consists entirely of edges and does not have a surface or volume.

#### 4.8.1.1. Beam Workflow

The workflow for using beams is as follows:

1. Import your model. You can apply the Structural template or import using the Geometry task. When importing in the Geometry task, ensure Line Bodies is selected. (This is handled automatically when using the template.)
2. For a line body, you must define minimal cross section properties, as well as length, offset and section orientation. You can do so in the model, or by picking beam profiles in the embedded AIM Geometry Modeler. If you define this information in the model, it is imported with the model into AIM. Additionally, you can define offset and orientation.

To edit the model using the AIM Geometry Modeler, click **Edit Geometry** in the **Geometry** task. You can then define cross-sections from the **Prepare** tab. For more information, see the section **Preparing Designs for Analysis > Beams > Profiles > Changing beam profiles** in the *SpaceClaim User's Guide*.

3. In the **Geometry** task, edit the geometry and use the **Share** tool to connect the beam bodies to make a single part model. If you edited any beam profiles or cross-sections, you must unshare parts and re-share; this must be the last operation in the AIM Geometry Modeler before setting up further physics. For more information, see the section **Workbench > Beams > Shared Topology > Sharing Topology** in the *SpaceClaim User's Guide*.
4. From the Physics panel, you can set the beam modeling options in the Physics Options. For more information, see Structural Physics Options.
5. Define your [loads and constraints](#) as you typically would. Beams support all structural conditions applicable to solid bodies, other than bolt pretension and pressure.
6. From the **Physics** panel, you can [define the end release for your beam](#). An end release will allow either or both ends of the beam to rotate about or translate along one or more of the local axes of the beam.
7. After solving, [view your results](#). Available are contour results of axial force, shear forces, bending moments, and torsional moment, in addition to typical structural variables.

## 4.8.1.2. Defining the Beam End Release

### Physics > Interface Conditions > Beam End Release

You can define the end release for your beam. An end release will allow either or both ends of the beam to rotate about or translate along one or more of the local axes of the beam.

To define a beam end release:

1. For **Vertex location**, select the vertex location where the end release is to be applied. This vertex must be shared by two or more beams.
2. For **Edges location**, select one or more edges of line bodies that are connected to the vertex selected as the Vertex location.

You can also choose a selection set.

3. If you specify more than one edge for the **Edges location**, select **Independent edges** to ensure that the edges are independent of one another.
4. Optionally, select or define a **Reference frame** for the directions of the releases. The default is the global reference frame.
5. Specify the translation and rotation for each degree of freedom as fixed or free.

## 4.8.2. Specifying Contacts

A contact is a type of connection between two solid, surface, or line bodies. Contacts can be generated automatically, through the [interface generator](#), or manually.

If you start your simulation with a template and choose the **Detect contact automatically** option, AIM automatically creates an interface generator and generates the contact objects and their associated [contact behavior](#) using the default tolerance value for the whole model.

You can also set up an interface generator for the model or for a group of bodies within the model and specify a tolerance. When you automatically generate contacts using a template or a specified interface generator, the contacts and contact behaviors are associated with the interface generator; therefore, if the interface generator is deleted, all associated contacts and contact behaviors are also deleted.

An automatically created contact and contact behavior come with intelligent default settings, but you may want to review them. If you want to modify an automatically generated contact, you can [convert it to manual](#), which disassociates the contact and its contact behavior from the interface generator.

To create a contact manually, [set up a contact](#), which can then be associated with a [contact behavior](#). The behavior defines all of the behavioral properties for the contact. Once you have defined a contact behavior, any number of contacts can refer to it.

When you re-generate automatic contacts, AIM retains any existing contact that uses the same source and target faces, as well as the associated contact behavior(s). However, if a retained contact does not have an associated contact behavior, a new one is not assigned.

For more information on identifying contacts, either for a selected body or bodies, or for a contact behavior, see [Identifying Connections](#) on page 370.

**Note:** If a contact is no longer required for your structural simulation, but may be needed in the future, you can [suppress it](#) rather than delete it entirely.

---

 [Generating Contacts](#)

---

 [Nonlinear Contacts](#)

---

## 4.8.2.1. Automatically Generating Contacts

### Physics > Interface Conditions > Interface Generator

You can use the interface generator tool to generate contacts.

**Note:** When you re-generate automatic contacts, AIM retains any existing contact that use the same source and target faces, as well as the associated contact behavior(s). However, if a retained contact does not have an associated contact behavior, a new one is not assigned.

To automatically generate contacts:

1. Define the location for the contacts.

Body is the only valid geometry type for automatically generated contact locations. This property can be specified directly through geometry entity selection or can be specified indirectly through a [selection set](#) that references one or more bodies. The default Location is the Physics Region.

2. For **Tolerance specification**, select if you want AIM to find contact pairs that fall within a suggested tolerance, or if you want to manually specify the tolerance value.
3. For **Contact behavior creation**, select if you want the interface generator to create one set of behavioral properties (contact behavior) that will be used by all generated contacts, or create a set of behavioral properties for each generated contact.
4.  Optionally, for **Detection types**, select the types of contacts you want to generate.
5. Click **Generate Interfaces**.

When you generate the interfaces, AIM automatically creates both the contacts for any located contact pairs and the associated contact behavior, which defines the behavioral properties for the contacts.

Depending on your settings, either all generated contact will use the same set of behavioral properties, or each will have its own associated contact behavior. If the contact behavior is shared by all generated contact, you can modify the behavioral properties for all generated contact by modifying that one contact behavior. If you want one or more of your generated contacts to have different behavioral properties, you can [add a new contact behavior for that contact](#).

**Note:** If a contact is no longer required for your structural simulation, but may be needed in the future, you can [suppress it](#) rather than delete it entirely.

### 4.8.2.1.1. Converting an Auto-Generated Contact to Manual

#### Physics > Interface Conditions > Contact

Contacts and contact behaviors generated by an interface generator are automatically associated with interface generator that created them. You can dissociate a contact and the contact behavior it references from its interface generator by converting the auto-generated contact to a manual one. Thus, when the interface generator is deleted or another interface generation is performed, the remaining auto-generated contacts are deleted but not the converted contact and the contact behavior it references.

Re-scoping an auto-generated contact automatically converts it to a manual contact and disassociates it from the interface generator object.

**CAUTION:** An auto-generated contact converted to manual cannot be converted back to auto-generated.

To convert an automatically generated contact to manual:

1. Navigate to the contact you want to convert.

2. For **Location definition method**, select **Manual**.

## 4.8.2.1.2. Modifying the Contact Behavior for a Generated Contact

### Physics > Interface Conditions > Contact

Contacts and contact behaviors generated by an interface generator are automatically associated with interface generator that created them. Every contact generated by an interface generator is assigned the same contact behavior if the "one shared by all contact" method is used to generate the contacts. Thus, if you modify the properties for this contact behavior, you modify the properties attached to all contact pairs generated with that contact pair.

If you don't want the changes to a contact behavior to apply to all generated contact, you can create a new contact behavior for a contact; this new contact behavior applies only to the contact from which you created it. You can then apply this new behavior to other contact.

To modify the contact behavior on a contact:

1. Navigate to the contact you want to modify.
2. Select **Contact behavior > Create New**.
3. Modify the [contact behavior properties](#).
4. Navigate to any other contact you want to use this contact behavior, and select the contact behavior from the **Contact behavior** option.

## 4.8.2.2. Manually Setting Up a Contact

### Physics > Interface Conditions > Contact

To set up a contact and assign a contact behavior:

1. For **Location 1**, define the location of the contact's first location.

You can select geometry or a Named Selection.

For contacts, select the geometries (faces, edges, or vertices) that are considered as the source for the contact. You can also choose a selection set. For a Face/Edge contact, the edge must be designated as the first location. If a contact pair is applied to flexible and rigid bodies, the flexible body must be the Location 1 side. If the Location 1 side of the contact pair is applied to multiple bodies, all of the bodies must have the same Stiffness Behavior, either Rigid or Flexible.

2. For **Location 2**, define the location of the contact's second location.

You can select geometry or a Named Selection.

For contacts, select the geometries (faces or edges) that are considered the target for the contact. For Face/Edge contact, the face must be designated as the second location. If the Location 1 side of the contact pair has a flexible Stiffness Behavior then the Location 2 side can be rigid. The selection of multiple rigid bodies for the Location 2 invalidates the Contact object and an error message is generated following the solution process.

3. Select **Allow self- contact** to allow the specified entity to be included in both the source and target locations.
4. For **Contact behavior**, create or select a [Contact Behavior](#).

A contact behavior defines the behavioral properties of a contact. You can create a contact behavior, then assign it to one or more contacts, or you can create a contact behavior directly from a Contact object.

Contact behaviors are designed to be independent of a specific contact, so that they may be applied to multiple contacts as needed for your physics solution.

**Note:** When adding a contact manually, you cannot choose a contact behavior created for auto-generated contacts.

The contact behavior is shown below.

5. For **Trim optimization**, select if you want to speed up the solution time by reducing the number of elements sent to the solver for consideration.

- **Program Controlled:** This is the default setting. The application chooses the appropriate setting.

Typically, the program sets **Trim Contact** to **On**. However, if the contacts are created manually, no trimming is performed by default.

- **On:** During the process of creating the solver input file, checking is performed to determine the proximity between the location 1 and location 2 elements. Elements which are not in close proximity (determined by a tolerance) are not written to the file and therefore ignored in the physics solution. If you select this option, enter a tolerance value greater than zero in the **Tolerance** field.

- **Off:** No contact trimming is performed.

The checking process is performed to identify if there is overlap between the bounding boxes of the elements involved. If the bounding box of an element does not overlap the bounding box of an opposing face or element set, that element is excluded from the solution. Before the elements are checked, the bounding boxes are expanded using the **Tolerance** property so that overlapping can be detected.

6.  Optionally, select **Apply symmetry** to use the symmetric contact in the analysis.

**Note:** If a contact is no longer required for your structural simulation, but may be needed in the future, you can **suppress it** rather than delete it entirely.

### 4.8.2.3. Defining Contact Behavior Properties

**Physics > Interface Conditions > Contact**

**Physics > Contact Behaviors**

You can create or modify a contact behavior under **Contact Behavior** on the **Contact** panel. A contact behavior defines the behavioral properties of a contact. Contact behaviors are designed to be independent of a specific contact, so that they may be applied to multiple contacts as needed for your physics solution.

AIM automatically creates associated contact behaviors using default properties when you use the template workflow with the **Detect structural contact automatically** option set, or when you use an interface generator to create contacts. You can modify the properties for one or more of these generated contacts on the **Contact Behavior** panel.

**Note:** By default, automatically generated contacts use the same behavior unless you modified the **Contact behavior creation** property, so any changes to the behavior for one contact will affect all other contacts. If you want to modify the properties for only one of your generated contact pairs, create a new behavior for that contact, rather than modifying the existing contact behavior.

When adding a new contact manually, you can either create a new contact behavior or use an existing contact behavior created manually from the **Contact** panel. If you create a new contact behavior from the **Contact** panel, it is automatically assigned to that contact. You can then use that contact behavior for subsequent new manual contacts or other existing contacts, including contacts that are auto-generated.

To set contact behavior properties:

1. Select the contact behavior you want to modify.

Or, to create a new contact behavior, select **Contact behavior > Create new** on the **Contact** panel.

2. For **Contact type**, select a [contact type](#).
3. For frictional contacts, set the dynamic friction coefficient under **Friction coefficient**.  
Specify this value as a non-negative coefficient.

4.  Specify the additional contact behavior settings as determined by your Contact Type, Formulation, and Physics Type setting.

If you want to...	then set...	More information:
Set the physics type(s) for this contact behavior.	<b>Physics to transfer</b>	When set to the default, <b>All relevant</b> , the application respects the physics type selected in the <b>Physics</b> region. You can also choose <b>Specified</b> , then set the physics type(s) for this contact behavior under <b>Transferred physics</b> . The selected physics type(s) will always apply to this contact behavior, even if the physics type for a physics solution is set to another value.
Select the <a href="#">algorithm</a> the software <b>Formulation</b> uses for a particular contact pair computation.		<p>The default setting is <b>Program controlled</b>. For Bonded and No Separation contacts, AIM selects the Augmented Lagrange formulation. For nonlinear solid body contact of faces, the application selects Pure Penalty for contact between two rigid bodies and Augmented Lagrange for all other contact situations. For information on formulation options, see <a href="#">Contact Formulation Algorithms</a> on page 347.</p> <p><b>Note:</b> Cases involving large gaps and faces bonded together can result in fictitious moments being transmitted across a boundary.</p>

If you want to...	then set...	More information:
Select the location of the <b>contact detection</b> used in the physics solution in order to obtain a good convergence.	<b>Detection method</b>	<p>This option is applicable to 3D face-face contacts. When set to the default, <b>Program controlled</b>, AIM determines the detection method based on the formulation selected:</p> <ul style="list-style-type: none"> <li>• MPC: Nodal point (Detect Nodal Normal to Target)</li> <li>• Pure Penalty: Gauss integration points (On Gauss Point)</li> <li>• Augmented Lagrange: Gauss integration points (On Gauss Point)</li> <li>• Normal Lagrange: Nodal point (Detect Nodal Normal to Target)</li> </ul> <p><b>Note:</b> For additional MAPDL specific information, see <i>Selecting Location of Contact Detection</i> in the section on "Surface-to-Surface Contact" in the Contact Technology Guide.</p>
Automatically include the effect of <b>Include shell thickness where</b> the surface body thickness during <b>possible</b> contact calculations	<b>Include shell thickness where</b> the surface body thickness during <b>possible</b> contact calculations	<p>The contact will be detected a distance of half the thickness away from the face.</p> <p><b>Note:</b> Shell thickness will be included only if the <b>Thickness Distribution</b> is set to <b>Equal from mid-surface</b>.</p>

If you want to...	then set...	More information:
Set the DOF (degree of freedom) to be used for the MPC formulation	<b>Constraint type</b>	<p>By default, AIM chooses the best option. However, you can also choose an option yourself.</p> <p>The option <b>Target normal, couple translation to rotation</b> represents the most common type of surface body contact. Constraints are constructed to couple the translational and rotational DOFs. In most types of surface body contact, an offset will exist. Due to this offset there will be a moment created. To get the correct moment, the rotation/displacement DOF's must be coupled together. If the program cannot detect any contact in the target normal direction, it will then search anywhere inside the pinball for contact.</p> <p>With <b>Target normal, uncouple translation to rotation</b>, the rotational and displacement constraints will not be coupled together. This option can model situations where the surface body edges line up well and a moment is not created from the physical surface body positions. Thus it is most accurate for the constraints to leave the displacements/rotations uncoupled. This provides an answer which is closer to a matching mesh solution. Using a coupled constraint causes artificial constraints to be added causing an inaccurate solution.</p> <p>For <b>Inside pinball, couple translation to rotation</b>, constraints are coupled and created anywhere to be found inside the pinball region. Thus the pinball size is important, as a larger pinball will result in a larger constraint set. This option is useful when you wish to fully constrain one contact side completely to another.</p>

If you want to...	then set...	More information:
Specify whether AIM detects asymmetry automatically.	<b>Detect asymmetry automatically</b>	This option is available if the <a href="#">Apply symmetry</a> property of any contacts that reference this contact behavior is set to <b>Yes</b> .
Set small sliding on or off	<b>Small sliding</b>	This option, when set to <b>On</b> , activates an assumption of relatively-small sliding (less than 20% of the contact length during the analysis). If small sliding is known to occur, such as when you have bonded contact, this feature can make your solution more efficient and robust. With the default option, <b>Program Controlled</b> , small sliding is generally turned on for linear contacts.
Define the contact search area, or <a href="#">Pinball pinball region</a> , as a physical value or factor.		With the default option, <b>Program controlled</b> , the program calculates the pinball region. You can also set the <a href="#">pinball region detection</a> to a factor or physical value, or, for auto-generated contacts, you can set the pinball region equal to the tolerance value used when generating the contacts.
Control the <a href="#">thermal or electric contact conductance value</a> used in a contact simulation.	<b>Thermal Conductance or Electric Conductance</b>	With the default setting, <b>Program controlled</b> , the program calculates the value for <a href="#">thermal or electric contact conductance</a> . The value will be set to a sufficiently high enough value (based on the thermal or electric conductivities and the model size) to model perfect contact with minimal thermal or electric resistance. You can also manually specify the thermal or electric contact conductance value.
Control the <a href="#">friction heat and heat distribution factors</a> .	<b>Heat Generation</b>	Thermal and electric contact conductance are not available for the MPC formulation.
		In order to model heat generation due to frictional dissipated energy, you need to model the transient effects. For more information, see <a href="#">Heat Generation</a> on page 351.

If you want to...	then set...	More information:
Control the <a href="#">stiffness settings</a> for the contact.	<b>Stiffness</b>	You can set the normal and tangential stiffness values and the stiffness update frequency. For more information, see <a href="#">Stiffness</a> on page 349.
Specify <a href="#">initial interface gap/overlap adjustment properties</a> .	<b>Interface Gap/Overlap Adjustment</b>	The <b>Initial interface treatment</b> property defines how the contact interface for the pair is treated. It becomes active when contact <b>Type</b> is set to <b>Frictionless</b> , <b>Rough</b> , or <b>Frictional</b> (nonlinear contact).  With the default setting, the handling of the interface gap/overlap is dependent on the setting for the <b>Initial interface treatment</b> in the <a href="#">Solver Settings</a> , but you can also select a specific option. For more information, see <a href="#">Interface Gap/Overlap Adjustment</a> on page 353.
Set the allowable <a href="#">elastic slip</a> value <b>Elastic Slip Tolerance</b> for a contact.		Available for penalty-based formulations. Not applicable to Frictionless or No Separation contact types.  You can specify the tolerance as a physical value or factor, or you can have the tolerance calculated by the application.  For more information, see <a href="#">Elastic Slip Tolerance</a> on page 351.
Set the <a href="#">penetration tolerance</a> value for penalty-based formulations.	<b>Penetration Tolerance</b>	Available for the Pure Penalty or Augmented Lagrange formulations.  You can specify the tolerance as a physical value or factor, or you can have the tolerance calculated by the application.  For more information, see <a href="#">Penetration Tolerance</a> on page 352.

If you want to...	then set...	More information:
Set <b>damping</b> , including the normal <b>Numerical Damping</b> and tangential stabilization damping factors.		You can specify a <b>normal and tangential damping factor</b> , and a value of 1 is usually appropriate. If this factor is 0 (default), the damping is activated only in the first load step. If its value is greater than 0, the damping is activated for all load steps.

You can now assign this contact behavior to one or more contacts defined on the **Contact** panel. For more information, see [Manually Setting Up a Contact](#) on page 339. If you defined the contact behavior as part of a setting up a contact, note that you can now assign this contact behavior to other contacts as well.

### 4.8.2.3.1. Contact Types

In the [Contact Behavior properties](#), you can select a contact type. Choosing the appropriate contact type depends on the type of problem you are trying to solve.

If modeling the ability of bodies to separate or open slightly is important and/or obtaining the stresses very near a contact interface is important, consider using one of the nonlinear contact types (**Frictionless**, **Rough**, **Frictional**), which can model gaps and more accurately model the true area of contact.

However, using these contact types usually results in longer solution times and may cause issues with convergence due to the contact nonlinearity. If convergence problems arise, or if determining the exact area of contact is critical, consider using a finer mesh (using the **Sizing** control available with manual meshing) on the contact faces or edges.

The available contact types are listed below. Most of the types apply to Contact Regions made up of faces only.

<b>Bonded</b>	This is the default configuration and applies to all contact regions (surfaces, solids, lines, faces, edges). If contact regions are bonded, then no sliding or separation between faces or edges is allowed. Think of the region as glued. This type of contact allows for a linear solution since the contact length/area will not change during the application of the load. If contact is determined on the mathematical model, any gaps will be closed and any initial penetration will be ignored.
<b>No separation</b>	This contact setting is similar to the Bonded case. It only applies to regions of faces (for 3D solids) or edges (for 2D plates). Separation of the geometries in contact is not allowed, but small amounts of frictionless sliding can occur along contact geometries.
<b>Frictionless</b>	This setting models standard unilateral contact; that is, normal pressure equals zero if separation occurs. Thus gaps can form in the model between bodies depending on the loading. This solution is nonlinear because the area of contact may change as the load is applied. A zero coefficient of friction is assumed, thus allowing free sliding. The model should be well constrained when using this contact setting.
<b>Rough</b>	Similar to the frictionless setting, this setting models perfectly rough frictional contact where there is no sliding. It only applies to regions of faces (for 3D solids) or edges (for 2D plates). By default, no automatic closing of gaps is performed. This case corresponds to an infinite friction coefficient between the contacting bodies.
<b>Frictional</b>	In this setting, the two contacting geometries can carry shear stresses up to a certain magnitude across their interface before they start sliding relative to each other. This state is known as "sticking." The model defines an equivalent shear stress at which sliding on the geometry begins as a fraction of the contact pressure. Once the shear stress is exceeded,

the two geometries will slide relative to each other. The coefficient of friction can be any nonnegative value.

## 4.8.2.3.2. Contact Formulation Algorithms

In the [Contact Behavior properties](#), you can specify which algorithm the software uses for a particular Contact pair computation by selecting from the  **Formulation** option. Not all Formulation algorithms are available for all Contact types.

Available formulations include:

<b>Augmented Lagrange</b>	A penalty-based method that usually leads to better conditioning and is less sensitive to the magnitude of the contact stiffness coefficient than the Pure Penalty method. However, in some analyses, the Augmented Lagrange method may require additional iterations, especially if the deformed mesh becomes too distorted.
<b>Pure penalty</b>	A basic contact formulation based on Penalty method that is useful for contacts that occur only on an edge or vertex.
<b>Multi-point constraint</b>	Prevents artificial stiffness when gaps exist between curved surfaces for <b>Bonded</b> and <b>No Separation</b> Contact types. (An alternative is to use fixed Joints.) Multi-point constraint is most sensitive to overconstraint, so it should be avoided when there are other contacts or boundary conditions that overlap. The Multi-point constraint option does not provide stiffness settings--it is a purely linear way to connect contacting bodies.
	MPC equations are created internally during the solve to tie the bodies together. This can be helpful if truly linear contact is desired, or if it's necessary to handle the nonzero mode issue for free vibration that can occur if a penalty function is used. Note that contact-based results (such as pressure) will be zero.
<b>Normal Lagrange</b>	Enforces zero penetration when contact is closed making use of a Lagrange multiplier on the normal direction and a penalty method in the tangential direction. It provides the highest accuracy of any of the formulations. This formulation enables contact with material nonlinearities, enables interference fit, and allows large sliding.  Normal Lagrange adds contact traction to the model as additional degrees of freedom, and, as a result, requires additional iterations to stabilize contact conditions. It often increases the computational cost compared to the Augmented Lagrange setting. The <a href="#">Iterative solver type setting</a> (under the <b>Solver Settings</b> ) cannot be used with this method, and <a href="#">Normal Stiffness</a> is not applicable.

For additional MAPDL specific information, see [in the Contact Technology Guide](#).

**CAUTION:** Cases involving large gaps and faces bonded together can result in fictitious moments being transmitted across a boundary.

## 4.8.2.3.3. Detection Method

In the [Contact Behavior properties](#), you can set the  **Detection Method**, which allows you to choose the location of contact detection used in the physics solution in order to obtain a good convergence. It is applicable to 3D face-face contacts.

Property options include:

<b>Program Controlled</b>	This is the default setting. The application uses Gauss integration points (On Gauss Point) when the formulation is set to Pure Penalty and
---------------------------	---

Augmented Lagrange. It uses nodal point (Detect Nodal Normal to Target) for the MPC formulation and Normal Lagrange formulations.

#### **Detect on Gauss Point**

The contact detection location is at the Gauss integration points. This option is not applicable to contacts with MPC or Normal Lagrange formulation.

#### **Detect Nodal Normal from Contact**

The contact detection location is on a nodal point where the contact normal is perpendicular to the contact surface.

#### **Detect Nodal Normal to Target**

The contact detection location is on a nodal point where the contact normal is perpendicular to the target surface.

#### **Detect Nodal Projected Normal from Contact**

The contact detection location is at contact nodal points in an overlapping region of the contact and target surfaces (projection-based method).

For additional MAPDL specific information, see [in the Contact Technology Guide](#).

### **4.8.2.3.4. Pinball**

In the [Contact Behavior properties](#), you can set the  **Pinball** region, which allows you to specify a contact search area commonly referred to as a pinball region.

Setting a pinball region can be useful in cases where initially, bodies are far enough away from one another that, by default, the program will not detect that they are in contact. You could then increase the pinball region as needed. Consider an example of a surface body that was generated by offsetting a face of a solid body, possibly leaving a large gap, depending on the thickness. Another example is a large deflection problem where a considerable pinball region is required due to possible large amounts of over penetration. In general though, if you want two regions to be bonded together that may be far apart, you should specify a pinball region that is large enough to ensure that contact indeed occurs.

For the bonded contact type, be cautious when specifying a large pinball region, as any regions found within the pinball region will be considered to be in contact. For other types of contacts, this issue is not as critical because additional calculations are performed to determine if the two bodies are truly in contact. The pinball region defines the searching range where these calculations will occur. Further, a large gap can transmit fictitious moments across the boundary.

When setting up the pinball region, you can leave the default option, **Program Controlled**, where the program calculates the pinball region, or you can set the pinball region detection to a factor or a physical value greater than 0.

If this contact behavior applies to contacts automatically generated by the Interface Generator, then the **Auto Detection Value** property is also available for pinball. With this option, the pinball region is equal to the tolerance value used in generating the contacts. The value is displayed as read-only in the **Auto Detection Value** field. Auto Detection Value is the recommended option for cases where the automatic contact detection region is larger than a Program Controlled region. In such cases, some contact pairs that were detected automatically may not be considered in contact for a solution.

### **4.8.2.3.5. Thermal and Electric Conductance**

In the [Contact Behavior properties](#), you can set thermal and electric conductance, which are available for penalty-based contact formulations.

For additional MAPDL specific information, see [, specifically the section of the Contact Technology Guide \(Multiphysics Contact\)](#).

## Thermal Conductance

To take into account the conductive heat transfer between contact and target surfaces, specify the thermal contact conductance (real constant thermal contact conductance) for a thermal contact simulation. Thermal contact conduction transfers heat between two contacting surfaces. If contact occurs, a small value of thermal contact conductance yields a measured amount of imperfect contact and a temperature discontinuity across the interface. For large values of thermal contact conductance, the resulting temperature discontinuity tends to vanish and perfect thermal contact is approached. When not in contact, however, the solver assumes that no heat is transferred across the interface.

To model contact conduction between two surfaces where a small gap exists, define either the “bonded contact” or “no-separation contact” options.

Property options include:

<b>Program Controlled</b>	This is the default setting. The program will calculate the value for the thermal contact conductance. The value will be set to a sufficiently high value (based on the thermal conductivities and the model size) to model perfect contact with minimal thermal resistance.
<b>Manual</b>	You specify the thermal contact conductance as a value greater than 0. The units for this value are based on the types of contact involved. For 3D faces and 2D edges, the units are $\text{HEAT} / (\text{TIME} * \text{TEMPERATURE} * \text{AREA})$ . For contact between 3D edges and vertices, the units are $\text{HEAT} / (\text{TIME} * \text{TEMPERATURE})$ . For more information about the units used for thermal contact conductance coefficient, see <a href="#">and</a> in the section of the Mechanical User's Guide.

## Electrical Conductance

To take into account the surface interaction for electric contact in an electric conduction physics solution, you need to specify the electric contact conductance (the real constant electric contact conductance).

Electric contact conductance is used to model current conduction between two contacting surfaces. To model contact interaction between two surfaces where a small gap exists, define either the “bonded contact” or “no-separation contact” options.

Property options include:

<b>Program Controlled</b>	This is the default setting. The program will calculate the value for the electric contact conductance. The value will be set to a sufficiently high enough value (based on the electric conductivities and the model size) to model perfect contact with minimal electric resistance.
<b>Manual</b>	You specify the electric contact conductance as a value greater than 0.

**Note:** The Electric Analysis result, Joule Heat, is not supported when generated by nonzero contact resistance.

### 4.8.2.3.6. Stiffness

In the [Contact Behavior properties](#), you can set the stiffness characteristics when the [Formulation](#) is set to [Augmented Lagrange](#) or [Pure Penalty](#).

Option	Description
 <b>Stiffness update frequency</b>	<p>Controls how often the solver updates the contact stiffness during the solution.</p> <p>If you choose any of these stiffness update settings, the solver will modify the stiffness (raise/lower/leave unchanged) based on the physics of the model (that is, the underlying element stress and penetration).</p>
	<p>With any of the "Each iteration" options, the solver automatically determines a stiffness that allows both convergence and minimal penetration. Also, if this setting is used, problems that would not otherwise converge may converge in a Newton-Raphson sense.</p>
	<p>Property options include:</p>
<b>Program controlled</b>	<ul style="list-style-type: none"> <li>For contacts between two rigid bodies: <b>Never</b>.</li> <li>For other contacts: <b>Each iteration (default range)</b>.</li> </ul>
<b>Each iteration (default range)</b>	<p>The solver updates the stiffness at the end of each equilibrium iteration. If you are not sure what stiffness value to use, select this option.</p>
<b>Each iteration (nominal refinement)</b>	<p>The solver updates stiffness at the end of each equilibrium iteration, with nominal changing of the value range.</p>
<b>Each iteration (aggressive refinement)</b>	<p>The solver updates the stiffness at the end of each equilibrium iteration, but permits the most aggressive changing of the value range.</p>
<b>Never</b>	<p>No stiffness update is performed.</p>
 <b>Normal Stiffness</b>	<p>Controls the amount of penetration between contact and target surfaces.</p> <ul style="list-style-type: none"> <li>Higher normal stiffness values decrease the amount of penetration, but can lead to ill-conditioning of the global stiffness matrix and to convergence difficulties.</li> <li>Lower normal stiffness values can lead to a certain amount of penetration and produce an inaccurate solution.</li> </ul>
	<p>Ideally, you want a high enough normal stiffness that the penetration is acceptably small, but a low enough normal stiffness that the problem will be well-behaved in terms of convergence.</p>
<b>Program controlled</b>	<p>This is the default setting. The solver determines the appropriate normal stiffness value.</p>
<b>Physical value</b>	<p>Enter the Normal Stiffness as a positive value, a function, or in a tabular form. This entry is in force per unit volume.</p>
<b>Factor</b>	<p>Enter the Normal Stiffness factor. The usual factor range is from 0.01-1.0.</p>
	<ul style="list-style-type: none"> <li>The default value of 1.0 is appropriate for bulk deformation.</li> <li>If bending deformation dominates, use a smaller value (0.1). A smaller value provides for easier convergence but with more penetration.</li> </ul>

Option	Description						
 <b>Tangential Stiffness</b>	<p>Controls the amount of slip in sticking contact.</p> <ul style="list-style-type: none"> <li>Higher tangential stiffness values decrease the amount of slip, but can lead to ill-conditioning of the global stiffness matrix and convergence difficulties.</li> <li>Lower tangential stiffness values can lead to a certain amount of slip and produce an inaccurate solution.</li> </ul> <p>Ideally, you want a high enough tangential stiffness that the slip is acceptably small, but a low enough stiffness that the problem will be well-behaved in terms of convergence.</p> <table> <tr> <td data-bbox="462 508 680 572"><b>Program Controlled</b></td> <td data-bbox="717 508 1457 572">This is the default setting. The solver determines the Tangential Stiffness factor.</td> </tr> <tr> <td data-bbox="462 593 680 656"><b>Physical Value</b></td> <td data-bbox="717 593 1457 656">Enter the Tangential Stiffness as a positive value. This entry is in force per unit volume.</td> </tr> <tr> <td data-bbox="462 677 680 741"><b>Factor</b></td> <td data-bbox="717 677 1457 741">Enter the Tangential Stiffness factor. The usual factor range is from 0.01-1.0.           <ul style="list-style-type: none"> <li>The default value of 1.0 is appropriate for bulk deformation.</li> <li>If bending deformation dominates, use a smaller value (0.1). A smaller value provides for easier convergence but with more penetration.</li> </ul> </td> </tr> </table>	<b>Program Controlled</b>	This is the default setting. The solver determines the Tangential Stiffness factor.	<b>Physical Value</b>	Enter the Tangential Stiffness as a positive value. This entry is in force per unit volume.	<b>Factor</b>	Enter the Tangential Stiffness factor. The usual factor range is from 0.01-1.0. <ul style="list-style-type: none"> <li>The default value of 1.0 is appropriate for bulk deformation.</li> <li>If bending deformation dominates, use a smaller value (0.1). A smaller value provides for easier convergence but with more penetration.</li> </ul>
<b>Program Controlled</b>	This is the default setting. The solver determines the Tangential Stiffness factor.						
<b>Physical Value</b>	Enter the Tangential Stiffness as a positive value. This entry is in force per unit volume.						
<b>Factor</b>	Enter the Tangential Stiffness factor. The usual factor range is from 0.01-1.0. <ul style="list-style-type: none"> <li>The default value of 1.0 is appropriate for bulk deformation.</li> <li>If bending deformation dominates, use a smaller value (0.1). A smaller value provides for easier convergence but with more penetration.</li> </ul>						

For additional MAPDL specific information, see the following sections within in the Mechanical APDL Contact Technology Guide:

- 
- 
- 

### 4.8.2.3.7. Heat Generation

In the [Contact Behavior properties](#), you can set the transient effects necessary to model heat generation due to frictional dissipated energy. Heat generation applies to frictional contacts in a thermal simulation, and can be accessed under the  **Heat Generation** heading.

Option	Description
<b>Friction heat factor</b>	Sets the fraction for frictional dissipated energy converted into heat. Specify a value between 0 and 1 inclusive.
<b>Heat distribution factor</b>	Sets a weight factor for the distribution of heat between contact and target surfaces. Specify a value between 0 and 1 inclusive.

### 4.8.2.3.8. Elastic Slip Tolerance

In the [Contact Behavior properties](#), you can specify the allowable elastic slip value for a contact using the  **Elastic Slip Tolerance** property. This option is available when the **Contact type** is set to **Rough**,

**Bonded**, or **Frictional** and the **Formulation** is set to **Pure Penalty**, **Normal Lagrange**, or **Augmented Lagrange**.

Set the **Define by** property to:

<b>Program controlled</b>	This is the default setting. The solver calculates the Elastic Slip Tolerance Value.
<b>Physical value</b>	Specify the Elastic Slip Tolerance distance.
<b>Factor</b>	Specify the Elastic Slip Tolerance Factor. This entry is unitless.

The value you specify for **Physical Value** is multiplied by -1 when it is written to the input file, and the value you specify for **Factor** is written out unchanged. As a result, a negative Physical Value will be interpreted as a factor by the solver and a negative Factor value will be interpreted as an absolute value in solver units. This behavior matches the MAPDL solver in which positive value is considered as a factor whereas a negative value is considered as an absolute value.

For additional information, see , specifically the section of the Contact Technology Guide.

### 4.8.2.3.9. Penetration Tolerance

In the [Contact Behavior properties](#), you can set the penetration tolerance value for a contact using the  **Penetration Tolerance** property. This option is available when

- The Contact type is **Frictionless**, **Rough**, or **Frictional** and the **Formulation** is set to **Program Controlled** or **Augmented Lagrange**
- The **Formulation** is set to **Pure Penalty** and the **Stiffness update frequency** is set to one of the **Each iteration** options

Set **Define by** to one of:

<b>Program controlled</b>	This is the default setting. The solver calculates the Penetration Tolerance.
<b>Physical value</b>	Specify the Penetration Tolerance distance.
<b>Factor</b>	Specify the Penetration Tolerance Factor.

The value you specify for **Physical Value** is multiplied by -1 when it is written to the input file, and the value you specify for **Factor** is written out unchanged. As a result, a negative Physical Value will be interpreted as a factor by the solver and a negative Factor value will be interpreted as an absolute value in solver units. This behavior matches the MAPDL solver in which positive value is considered as a factor whereas a negative value is considered as an absolute value.

For additional information, see , specifically the section of the Contact Technology Guide (Surface-to-Surface Contact).

### 4.8.2.3.10. Numerical Damping

When the contact's **Contact type** is set to **Rough**, **Frictionless**, or **Frictional**, you can set  **Numerical Damping** in the [Contact Behavior properties](#).

A contact you define may initially have a near open status due to small gaps between the element meshes or between the integration points of the contact and target elements. The contact will not get detected during the physics solution and can cause a rigid body motion of the bodies defined in the contact. The stabilization damping factors (normal stabilization and tangential stabilization) provide a certain resistance to dampen the relative motion between the contacting surfaces and prevents rigid body motion.

The damping is applied to each load step where the contact status is open. The value of the stabilization damping factor should be large enough to prevent rigid body motion but small enough to ensure a solution.

For the normal and tangential damping factors a value of 1 is usually appropriate. If this factor is 0 (default), the damping is activated only in the first load step. If its value is greater than 0, the damping is activated for all load steps.

For more information, see [in the Contact Technology Guide](#).

## 4.8.2.3.11. Interface Gap/Overlap Adjustment

 The **Initial interface treatment** property, available under **Interface Gap/Overlap Adjustment** in the [Contact Behavior properties](#), defines how the contact interface for the pair is treated. It becomes active when contact **Type** is set to **Frictionless**, **Rough**, or **Frictional** (nonlinear contact).

With the default setting, the handling of the interface gap/overlap is dependent on the setting for the **Initial interface treatment** in the [Solver Settings](#), but you can also select a specific option.

**Fix unintentional initial gap/overlap** Any initial gaps are closed and any initial overlaps are reduced, creating an initial stress free state. Contact pairs are "just touching".

This setting is useful to make sure initial contact occurs even if any gaps or overlaps are present (for the gap case). Without using this setting, the bodies may fly apart if any initial gaps exist. Although any initial gaps are ignored, gaps can still form during loading for the nonlinear contact types.

**Model initial gap/overlap (ramped)** Models the true contact gap/overlap plus the offset value you specify. A positive value moves the contact closer together (increase overlap/reduce gap) and a negative value moves the contact further apart.

This setting is the closest to the default contact setting used in the Mechanical APDL application except that the loading is ramped. Using this setting will not close gaps. Even a slight gap may cause bodies to fly apart. Should this occur, use a small contact offset to bring the bodies into initial contact.

**Model initial gap/overlap (no ramping)** This is the default setting. This option models the true contact gap/overlap plus the offset value you specify. However, in this case loading is not ramped. A positive value moves the contact closer together (increase overlap/reduce gap) and a negative value moves the contact further apart.

You can access the sub-stepping method through the command window. Sub-stepping allows you to specify if changes in contact behavior should control automatic time stepping. For more information, see [in the Workbench User's Guide](#).

## 4.8.2.4. Contact Best Practices

Increased solution times may result from the inclusion of nonlinear contact in a typical analysis. Poorly defined contact conditions may also create further instability, cause convergence issues, and require additional processing time. Given properly defined contact conditions, contact results converge more efficiently and results tend to be smoother.

See the following sections for more information:

- [Contact Setup](#) on page 354
- [Identifying Problematic Contacts](#) on page 358
- [Possible Remedies for Non-Convergence](#) on page 360

For information on identifying contacts, either for a selected body or bodies, or for a contact behavior, see [Identifying Connections](#) on page 370.

## 4.8.2.4.1. Contact Setup

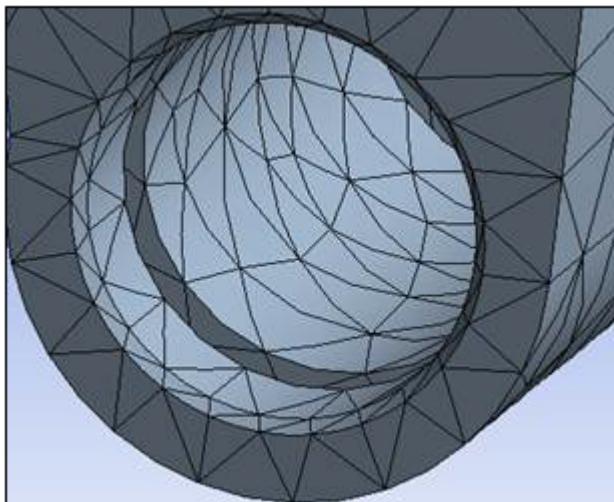
While AIM initially provides the default values and settings appropriate for typical contact problems, you may need to provide certain contact values and settings based on the type and nature of the problem. This section describes useful techniques for making sure that you establish robust contact settings and conditions prior to solving.

- Refine your [mesh quality and sizing](#).
- Use an [interface generator](#) with a specified tolerance value to automatically create contacts for a specified set of bodies that will all have the same contact properties. You can then automatically generate a separate group of contacts for a different set of bodies, perhaps using a different tolerance value and/or a different set of contact properties.
- [Manually create contacts](#) in cases where automatic contacts were not generated due to tolerance or other constraints.
- Ensure you are using the most appropriate [contact formulation](#).
- Modify the [contact stiffness](#).
- Avoid [over-constraining your model](#).
- Resolve [initial gaps and rigid body motion](#).

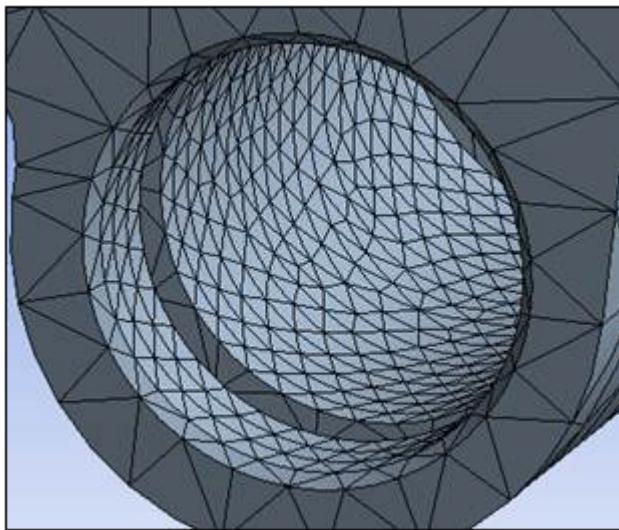
**Note:** Prior to solve, AIM reviews the model for bodies without contact, joints or constraints. If free bodies are detected, AIM then stops the solution and highlights the under-constrained bodies. For more information, see [Unconstrained Body Detection](#) on page 370.

### 4.8.2.4.1.1. Refine Mesh Quality and Sizing

Poor mesh quality can cause convergence problems, especially with the presence of nonlinear contacts. Examples of mesh quality are illustrated below. To use the approaches described here, you must choose to define the mesh manually. Understanding the use of [Local Mesh Sizing](#) can help you refine the mesh on your contact conditions. The contact surface shown here has a mesh that could be improved.



Whereas, this contact surface has a very good mesh quality.

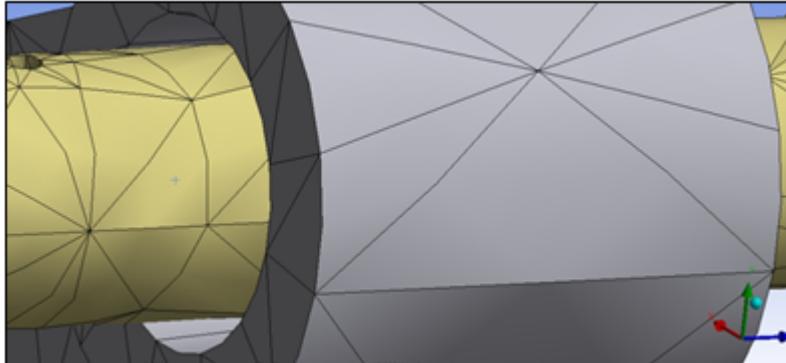


**Tip:** Try setting the **Shape Checking** property in your Meshing task to **Aggressive mechanical** for nonlinear contact models.

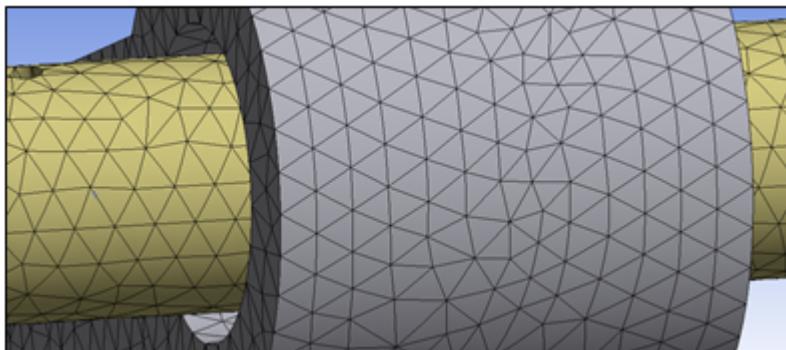
In order to create smooth results, make sure that the elements closely follow the curvature, and that your model has sufficient contact elements on curved surfaces, including straight surfaces that may become curved as the analysis proceeds. This practice is especially important for nonlinear contacts.

In addition, use similar mesh densities on both sides of the contact pair. You can improve element size and density using local mesh sizing.

This example illustrates a curve with not enough elements.



The parts are now shown below with improved element sizes and density for each contact side.



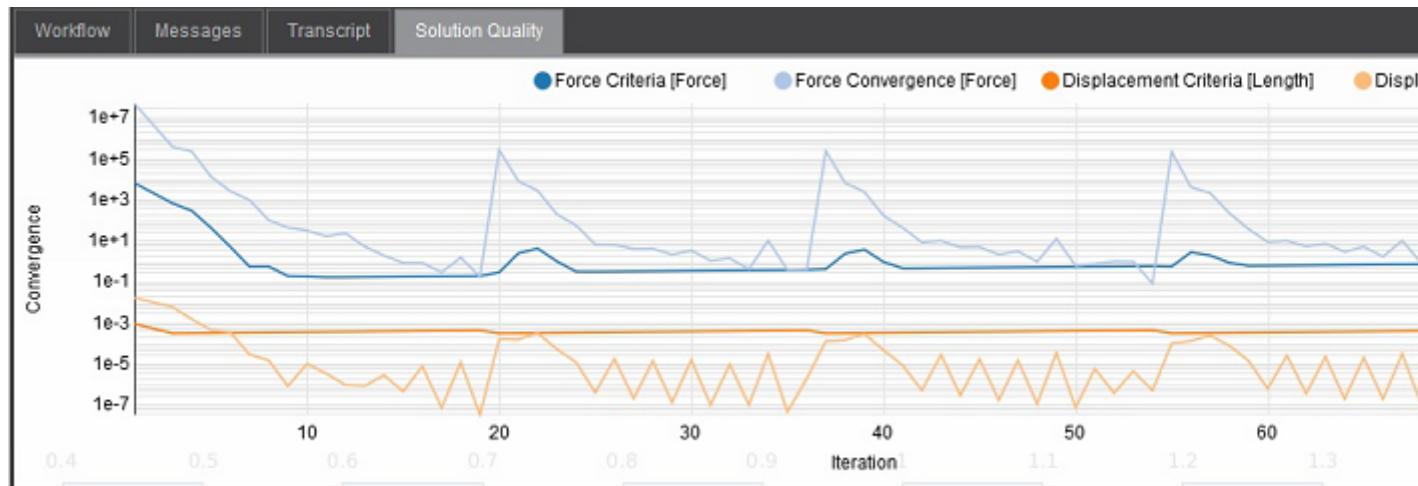
## 4.8.2.4.1.2. Modifying the Contact Stiffness

Using the [Normal Stiffness](#) property, you can manually increase the stiffness associated with a contact pair. A high stiffness setting can lead to reduced penetration and an increase in accuracy. However, it can also lead to ill-conditioning and divergence.

For contacts that lead to convergence difficulty, try lowering the stiffness.

And, when there is difficulty converging due to high penetration, you may need to increase the stiffness.

Here is an example of poor convergence. You can see the un-converged repeated pattern on the chart.



This example shows very good convergence as a result of the stiffness being reduced. With many fewer iterations, this contact analysis meets the convergence criteria and converges to a solution.



## 4.8.2.4.1.3. Avoid Over-Constraining the Model

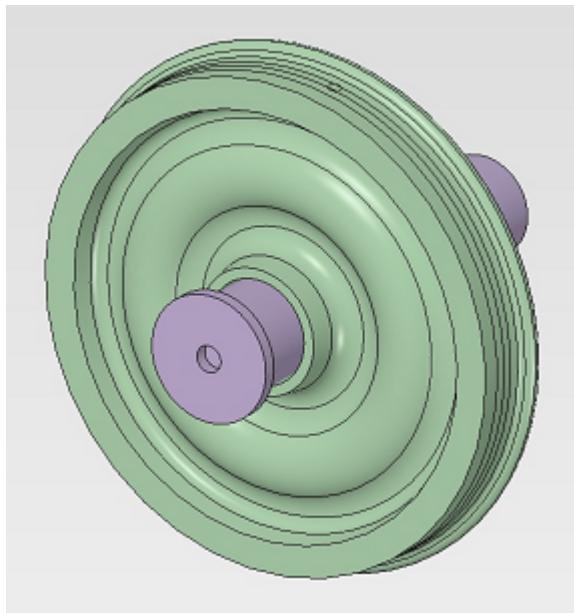
Care should be taken when a contact and a constraint type condition have overlapping topology. In this case, you should modify the [trimming tolerance specification](#) for your contacts.

If you cannot eliminate an over-constraint situation, avoid the MPC Formulation.

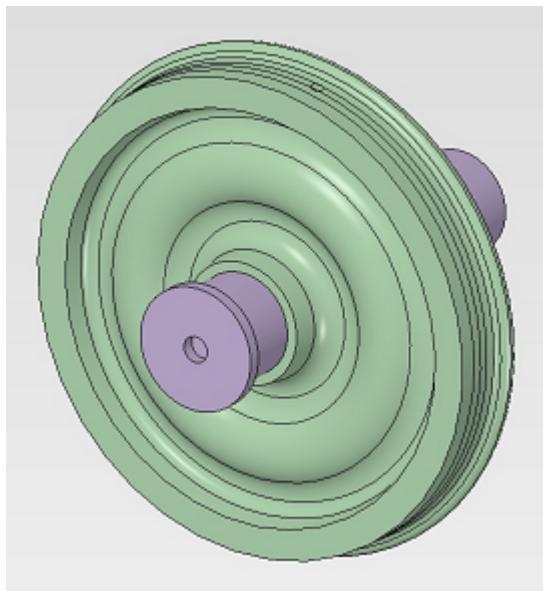
#### 4.8.2.4.1.4. Resolving Initial Gaps and Rigid Body Motion

There are times when parts with exterior constraints on them are dependent upon contact to prevent rigid body motion. In addition, when small gaps exist (especially for curved contact conditions), rigid body motion can occur at the beginning of the solution before contact engages.

When nonlinear contact is present, small gaps that are initially open can lead to rigid body motion (as shown below).



Changing the **Initial Interface Treatment** property to **Adjust to Touch** can be an effective means to resolve this gap. Mathematical adjustment to close the gap causes an open region to be visible when post-processing. This gap is simply offset by the contact elements. Note that for concentric cylinders, **Adjust to Touch** is not recommended. Instead, enter the offset manually.



Alternatively, you can employ **Numerical Damping**. Rather than ignoring the gap by offsetting the contact elements, the solver applies damping. Then the solver can more easily handle the approach of the bodies towards each other, leading eventually to closed contact. Note that you need to use care and verify that the damping does not negatively impact the accuracy of your analysis.

## 4.8.2.4.2. Identifying Problematic Contacts

There are two primary ways of determining the contacts that are causing non-convergence:

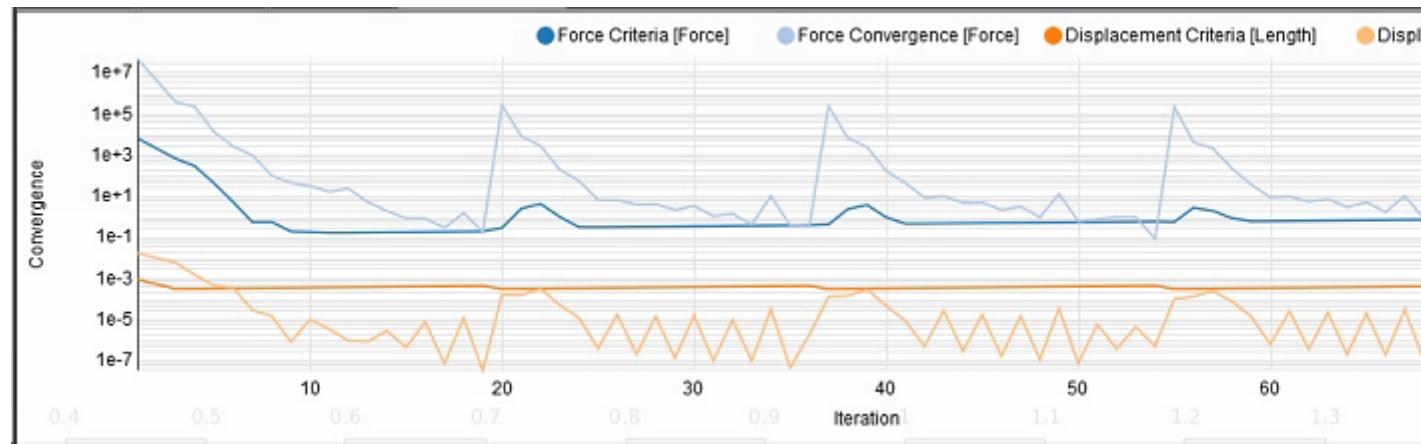
- View Contact Solution Monitoring Charts (including Force Convergence)
- Check Partial Solutions (especially Status and Penetration).

For general information on identifying contacts, either for a selected body or bodies, or for a contact behavior, see [Identifying Connections](#) on page 370.

**Note:** Prior to solve, AIM reviews the model for bodies without contact, joints or constraints. If free bodies are detected, AIM then stops the solution and highlights the under-constrained bodies. For more information, see [Unconstrained Body Detection](#) on page 370.

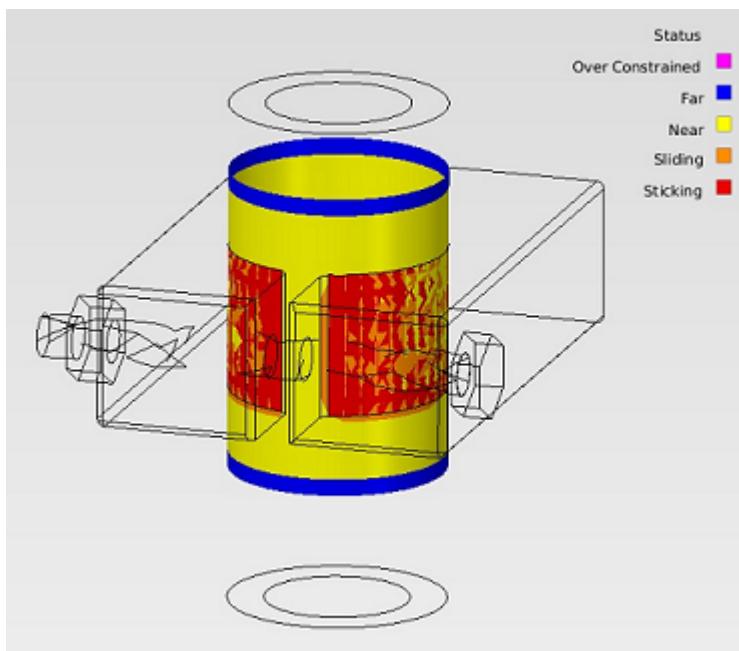
## Contact Solution Monitoring Charts

Contact solution monitor charts enable you to view contact information "live" as the solution processes. This enables you to observe trends that can help you diagnose problems. For example, a decreasing number of contact points indicate a trend towards a loss of contact.

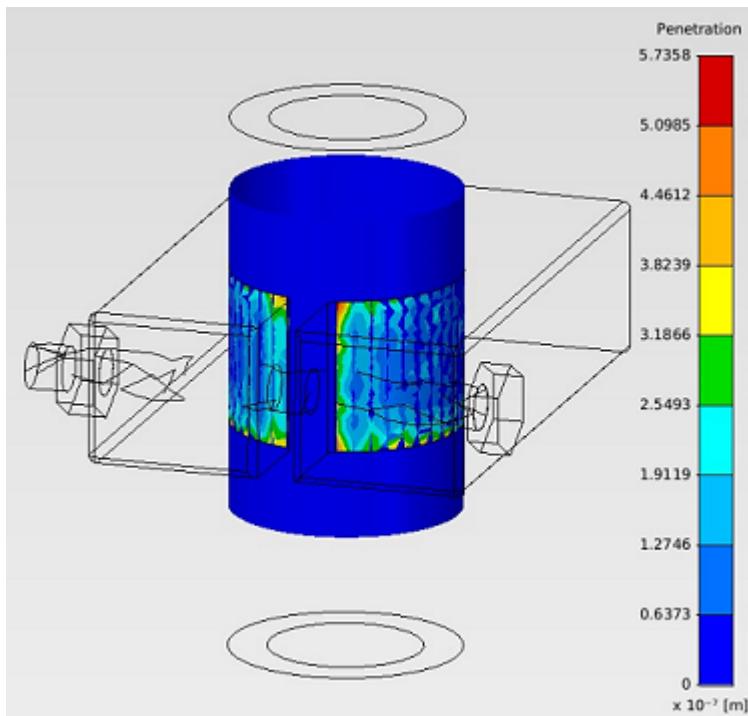


## Contact Results

When a solution fails to converge, you can review the results at the converged substeps, which can be invaluable for diagnosing problems. Begin by checking the Contact Status result. It can help you understand the global contact behavior of the model and enable you to find problem areas.



In addition, ensure that the contact penetration is small relative to local displacements. The reason is that any contact penetration is technically impossible, since bodies do not move into each other. A good check is to add the Contact contour result Penetration to the local displacement and compute a back-of-the-envelope strain calculation. This helps determine if the penetration is significantly affecting the stress calculations. An example is shown below.



Ways to reduce penetration include:

- Increasing the [stiffness](#).
- Reducing the [penetration tolerance](#).
- Selecting the [Normal Lagrange formulation](#).

## 4.8.2.4.3. Possible Remedies for Non-Convergence

The following are possible remedies for non-convergence due to contacts:

1. Make sure the model's units of measure are on an appropriate scale.
2. Increase the [pinball size in your Contact Behavior](#).
3. Adjust the [contact stiffness](#).
4. Use "nodal detection" if the problem is at a corner.
5. If a [large friction coefficient](#) is defined (>0.25), consider using an unsymmetric solver.
6. Adjust [initial interface treatment](#) for gaps in the contact behaviors.
7. If the **Contact type** is **Frictionless**, try setting it to **Rough** instead. This setting may help some problems to converge if any possible sliding is not constrained.
8. If symmetric contact is being used (by default the contact is symmetric), consider using [asymmetric contact pairs](#). This may help problems that experience oscillating convergence patterns due to contact chattering.
9. As a last resort: Add stabilization ([Contact](#) or [global](#)).

### 4.8.3. Specifying Contacts for Blow Molding Simulations

#### Blow Molding > Interface Conditions > Contact

A contact is a type of connection between the mold (structural region) and the fluid (polymer region). To set up a contact and assign a contact behavior:

1. Select the **Physics region on side 1** that represents the fluid (polymer region).
2. Specify **Location (side 1)** to be the faces of the polymer region that participate in the contact.
3. Select the **Physics region on side 2** that represents the mold (structural region).
4. Specify **Location (side 2)** to be the faces of the structural region that participate in the contact.
5. Specify the side of the (structural) physics region where the body of the mold is located. An arrow in the graphics window allows you to visualize the position of the mold body. The mold body sits on the same side of the structural region face as the arrowhead.
6. Create or select the [contact behavior](#) that defines the contact properties between the mold and the fluid.

## 4.8.3.1. Mold to Fluid Contact Behavior

Option	Description
<b>Allow contact release</b>	Contact release can occur under specific circumstances, for example, when the motion of a mold is reversed, or when the forming pressure results in the material being pulled away from a mold. By default, the possibility of contact release is disabled for contact problems. When it is enabled, the detachment of the free surface from the wall occurs if the local force is greater than the adhesion force.
<b>Adhesion force per unit area</b>	This is a physical parameter that depends on the interaction between the deformed melt and the material of the plug. The higher the adhesion force, the greater the pulling force must be to detach the fluid from the contact surface. Set the adhesion force density to zero if you do not have specifications to guide your definition.

Option	Description
<b>Penetration accuracy</b>	Penetration accuracy dictates the tolerance at contact. The smaller the tolerance, the more precise the contact is modeled at the expense of a higher CPU time. The tolerance is a distance equal to the penetration accuracy times the largest diagonal of the Cartesian box encompassing the whole geometry.
<b>Heat transfer coefficient</b>	A heat transfer coefficient should be defined when heat transfer occurs between the mold and the fluid in contact.
<b>Slipping coefficient</b>	If the slip coefficient and the penalty coefficient have the same value, then it is assumed that the fluid sticks to the mold when it comes into contact. Full slippage at the contact boundary is assumed if the slip coefficient is zero. Actual cases involving partial slippage will require a slip coefficient that typically ranges from 10e2 to 10e6, depending on the physics involved as well as on the system of units being used.
<b>Penalty coefficient</b>	This is the force applied to the fluid to stop its penetration into the mold. Increasing the penalty coefficient offsets the effects of an applied high pressure.
<b>Search settings</b>	A rigorous contact detection algorithm can be time consuming on large meshes. A faster, but less rigorous approach can be employed, however, if the geometrical complexity is high, then a safer and slower approach may be required. The default setting generally achieves the right balance, however, if you have a complex geometry or if the contact is not correctly detected, then increasing this setting may improve the solution.

#### 4.8.4. Setting Up a Joint

##### Physics > Interface Conditions > Joint

A joint typically serves as a junction where bodies are joined together. Joint types are characterized by their rotational and translational degrees of freedom as being fixed or free.

Joints can be defined as body-to-body or body-to-ground. If you select only one piece of geometry, AIM creates a body-to-ground joint by default. For body-to-body joints, you assign a base and mobile location.

For information on identifying joints, either for a selected body or bodies, or for a specific joint behavior, see [Identifying Connections](#) on page 370.



To set up a joint and assign a joint behavior:

**1. For Base location, define the location.**

When defining body-to-body joints, you can select a face, vertex, or edge for the base location of the joint. You can also choose a selection set.

**2. For Mobile location, define the location.**

You can select a face, vertex, or edge for the mobile side of the joint. You can also choose a selection set.

**3. For Joint behavior, create or select a joint behavior.**

A joint behavior defines the behavioral properties of the joint. You can create a joint behavior, then assign it to one or more joints, or you can create a joint behavior directly from the joint. Joint behaviors are designed to be independent of a specific joint, so that they may be applied to multiple joints as needed for your physics solution.

The selected joint behavior is shown at the bottom of the pane.

4. Optionally, select reference frames for orienting the reference and mobile sides of the joint under **Initial Reference Frames**.

By default, AIM creates one reference frame oriented to the defined Base location, which is then used to orient both the Base and Mobile sides of the joint. If desired, you can instead choose different reference frames for orienting the joint.

**Note:** If a joint is no longer required for your structural simulation, but may be needed in the future, you can suppress it rather than delete it entirely.

## 4.8.4.1. Specifying Joint Behaviors

### Physics > Interface Conditions > Joint

You can create or modify a joint behavior under **Joint Behavior** on the **Joint** panel. A joint behavior defines the behavioral properties of a joint. Joint behaviors are designed to be independent of a specific joint, so that they may be applied to multiple joints as needed for your physics solution.

When adding a new joint, you can either create a new joint behavior or use an existing joint behavior. If you create a new joint behavior from the **Joint** panel, it is automatically assigned to that joint. You can then use that joint behavior for other joints.

To set joint behavior properties:

1. To create a new joint behavior, select **Joint behavior > Create new** on the **Joint** panel.  
Or, select the joint behavior you want to assign or modify.
2. Select a **Joint type** or accept the default value of **Fixed**.  
See [Joint Types](#) on page 363 for more information.
3. For the General joint type, specify the degrees of freedom for translation and rotation under **Degrees of freedom**.  
All the degrees of freedom are fixed by default. You can set the **Translation** options to fixed or free, and the **Rotation** option to fixed, all free, or free about the X, Y, or Z axis.
4.  Optionally, for fixed joints, specify the [Formulation](#).

You can leave the formulation to be controlled by ANSYS AIM, or you can select a weld joint, rigid beam, or rigid link formulation. If the formulation is set to a rigid link or beam, you can then specify the reduction method.

For more information, see [Fixed Joint Formulation](#) on page 362.

### 4.8.4.1.1. Fixed Joint Formulation

You can leave the formulation to be controlled by ANSYS AIM, or you can select a weld joint, rigid beam, or rigid link formulation. If the formulation is set to a rigid link or beam, you can then specify the reduction method.

The reduction methods are:

- The **Direct elimination method**, wherein the kinematic constraints are imposed by internally generated constraint equations. The degrees of freedom of a dependent node in the equations are eliminated in favor of an independent node.

In general, you should use the direct elimination method when it is available since the degrees of freedom at the dependent nodes are eliminated, thereby reducing the problem size and solution time.

- The **Lagrange multiplier method**, wherein the kinematic constraints are imposed using Lagrange multipliers. In this case, all the participating degrees of freedom are retained.

Use the Lagrange multiplier method when the direct elimination method is not available or not suitable for the analysis.

The disadvantage of the Lagrange multiplier method is that the Lagrange multipliers are additional solution variables and, hence, the problem size and solution time become larger when compared with the direct elimination method.

## 4.8.4.1.2. Joint Types

You can create the following types of joints:

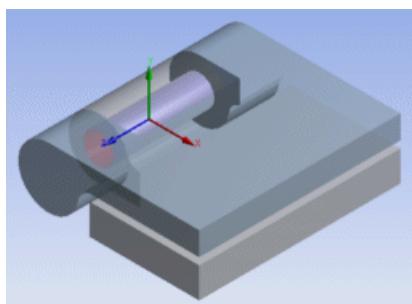
- Fixed
- Hinge
- Cylindrical
- Translational
- Slot
- Universal
- Spherical
- Planar
- General

## Fixed Joint

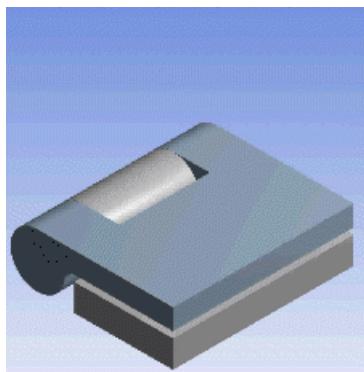
Fixed Joints support all degrees of freedom.

## Hinge Joint

**Constrained degrees of freedom:** UX, UY, UZ, ROTX, ROTY

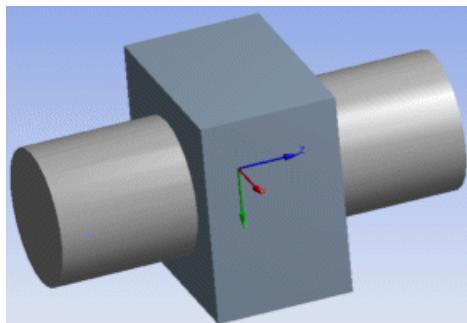


**Example:**

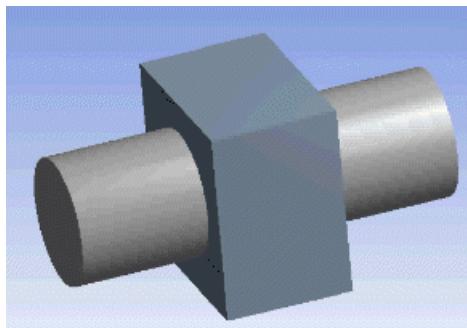


## Cylindrical Joint

**Constrained degrees of freedom:** UX, UY, ROTX, ROTY

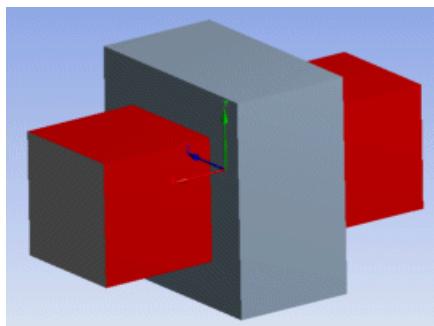


**Example:**

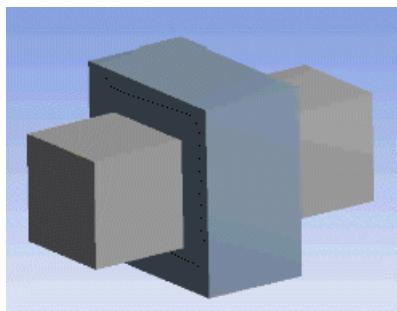


## Translational Joint

**Constrained degrees of freedom:** UY, UZ, ROTX, ROTY, ROTZ

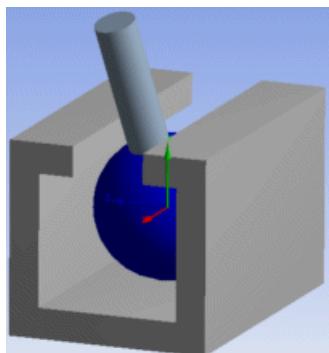


**Example:**

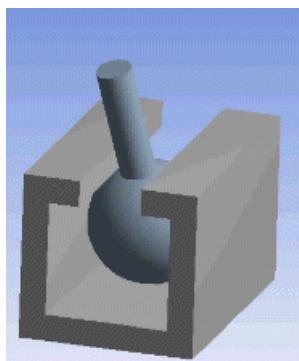


## Slot Joint

**Constrained degrees of freedom:** UY, UZ

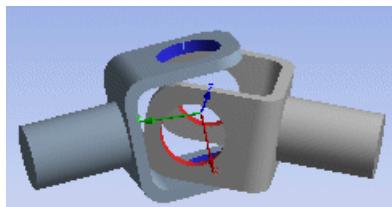


**Example:**

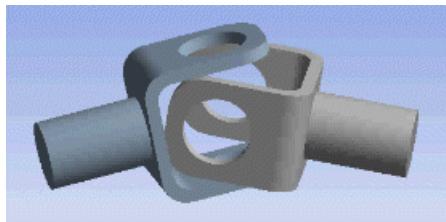


# Universal Joint

Constrained degrees of freedom: UX, UY, UZ, ROTY

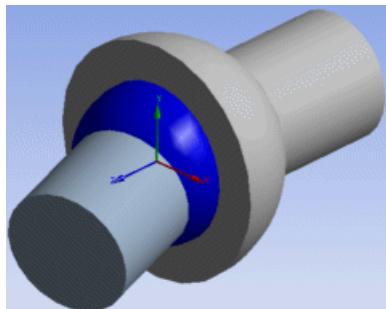


Example:

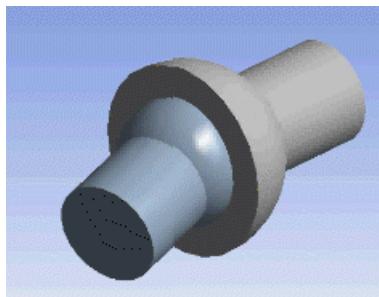


# Spherical Joint

Constrained degrees of freedom: UX, UY, UZ

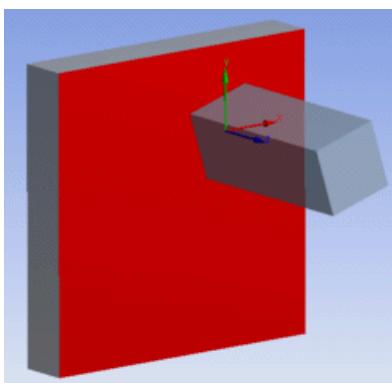


Example:

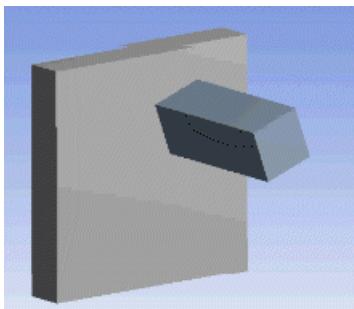


# Planar Joint

Constrained degrees of freedom: UZ, ROTX, ROTY



**Example:**



## General

With the General joint, you can set the constrained degrees of freedom and the rotation.

All the degrees of freedom are fixed by default. You can set the Translation options to fixed or free, and the Rotation option to fixed, all free, or free about the X, Y, or Z axis.

### 4.8.5. Specifying Structural Springs

A spring is an elastic element which is used to store mechanical energy and which retains its original shape after a force is removed. Springs are typically defined in a stress free or “unloaded” state. This means that no loading conditions exist unless preloading is specified for a 1-D spring, which creates an initial “loaded” state.

Springs connect two bodies together, which connects them and transfers the load from one body to the other. One of the ends can be set as grounded, which sets it as a fixed support. Longitudinal or torsional springs generate a force that depends on displacement or rotation, respectively.

To define a spring, you [specify a set of options specific to that connection](#), then [define a set of behaviors for that spring](#). You can apply a set of spring behaviors to more than one spring, so you can also select a pre-existing behavior.

**Note:** If a spring is no longer required for your structural simulation, but may be needed in the future, you can [suppress it](#) rather than delete it entirely.

You can also define a spring as 3-D by specifying stiffness values in any three orthogonal directions, as defined by the axes of a specified reference frame. AIM creates a default spring reference frame with a Z axis aligned along the line connecting the two end points of the spring, but you can select or define another reference frame. You can also define a spring as non-linear by specifying the stiffness as a stretch vs. force curve (for a longitudinal spring) or twist vs. moment curve (for a torsional spring). All spring results are supported, with the three component vectors and the resultant scalar available as individual results.

For information on identifying springs, either for a selected body or bodies, or for a specific spring behavior, see [Identifying Connections](#) on page 370.

## 4.8.5.1. Setting Up a Spring

### Physics > Interface Conditions > Spring

To define a spring and assign a spring behavior:

1. Define your location(s).
  - To define a body to body spring, define **Location 1** and **Location 2**. You can select a face, edge, or vertex. You can also choose a selection set.
  - To define a spring with one end grounded (essentially functioning as a fixed support), define **Location 1** or **Location 2** and then select the fixed location from **Ground spring**.
2. For **Spring behavior**, create or select a [spring behavior](#).

A spring behavior defines the behavioral properties of the spring. You can create a spring behavior, then assign it to one or more springs, or you can create a spring behavior directly from the spring. Spring behaviors are designed to be independent of a specific spring, so that they may be applied to multiple springs as needed for your physics solution.

The selected spring behavior is shown at the bottom of the pane.

3. If the selected spring behavior defines a 3D spring, select or define a frame of reference from **Relative to**.

AIM creates a default spring reference frame with a Z axis aligned along the line connecting the two end points of the spring, but you can choose the Global Reference Frame or select or define another reference frame.

**Note:** If a spring is no longer required for your structural simulation, but may be needed in the future, you can [suppress it](#) rather than delete it entirely.

## 4.8.5.2. Specifying the Behavior for a 1-D Spring

### Physics > Interface Conditions > Spring

#### Physics > Spring Behaviors

You can create or modify the behavior of a 1-D spring under **Spring Behavior** on the **Spring** panel. A spring behavior defines the behavioral properties of a spring and is designed to be independent of a specific spring, so that they may be applied to multiple springs as needed for your physics solution.

When adding a new spring, you can either create a new spring behavior or use an existing spring behavior. If you create a new spring behavior from the **Spring** panel, it is automatically assigned to that spring. You can then use that spring behavior for other springs.

To set spring behavior properties for a 1-D spring:

1. To create a new spring behavior, select **Spring behavior > Create new** on the **Spring** panel. Or, select the spring behavior you want to assign or modify.
2. Select the **Spring type**.
  - **Longitudinal** springs generate a force that displaces along the axis.
  - **Torsional** springs generate a force that rotates about the axis.
3. For **Behavior**, select **1-D**.

4. Enter the **Stiffness** as a constant value.

5.  Optionally, select a **Preload type** under **Spring Preload Settings**. You can define a preload as a length or specify a specific load.

For a Longitudinal spring:

- Select **Free length** to define the preload as a displacement length, then enter the length as a non-zero value in **Initial condition**. The actual length is calculated using the spring end points from the defined locations.
- Select **Load** to specify a load/force, then enter a non-zero value in **Initial condition**. When preload is specified as a load, a positive value creates tension and a negative value creates compression.

For a Torsional spring:

- Select **Torque** to define the preload as a force in the opposite direction, then enter a non-zero value in **Initial condition**.
- Select **Rotation** to define the preload as a rotational displacement, then enter a non-zero value in **Initial condition**.

### 4.8.5.3. Defining the Behavior of a 3-D Spring

**Physics > Interface Conditions > Spring**

**Physics > Spring Behaviors**

You can create or modify the behavior of a 3-D spring under **Spring Behavior** on the **Spring** panel. A spring behavior defines the behavioral properties of a spring and is designed to be independent of a specific spring, so that they may be applied to multiple springs as needed for your physics solution.

When adding a new spring, you can either create a new spring behavior or use an existing spring behavior. If you create a new spring behavior from the **Spring** panel, it is automatically assigned to that spring. You can then use that spring behavior for other springs.

To set spring behavior properties for a 3-D spring:

1. To create a new spring behavior, select **Spring behavior > Create new** on the **Spring** panel. Or, select the spring behavior you want to assign or modify.
2. Select the **Spring type**.
  - **Longitudinal** springs generate a force that displaces along the axis.
  - **Torsional** springs generate a force that rotates about the axis.
3. For **Behavior**, select **3-D**.
4. For **Stiffness**, enter a constant value for the stiffness coefficients in one or more of the three directions.

You can also define a spring as nonlinear by entering tabular data for a stretch vs. force curve (for a longitudinal spring) or a twist vs. moment curve (for a torsional spring).

**Note:** You can enter non-zero constant values for all stiffness coefficients or tabular data for all stiffness coefficients, but you cannot specify a non-zero constant for one stiffness value and tabular data for another.

AIM creates a default spring reference frame with a Z axis aligned along the line connecting the two end points of the spring, but you can [select or define another reference frame](#) from **Relative to** in the **Springs** panel.

#### 4.8.6. Identifying Connections

AIM includes several methods for identifying connections, either for a selected body or bodies, or a connection behavior. Connections include [contacts](#), [joints](#), and [springs](#). You can:

- Filter a connections list to show only the connections for a selected body or bodies, or the connections common to selected bodies.
- View a list of connections that utilize a specific connection behavior.

## Filtering Connections for a Body or Bodies

You can set a filter to display only the connections for a selected body or bodies. Additionally, you can also filter for connections common to the selected bodies.

1. View the list of connections you want to filter: [Contacts](#), [Joints](#), or [Springs](#).

2. Enable the **Filter based on selected bodies**  filter. You can then select one or more bodies on the geometry, or indirectly select a body by selecting a face, edge, or vertex.
3. You can further filter the list based on connections common to the selected bodies by enabling the **Show common objects for selected bodies**  filter.

You can also filter by right-clicking and selecting an option from the **Identify** menu. AIM will then display the filtered connections list.

## Identifying Connections for a Behavior

AIM displays the number of connections associated with that behavior as a link on the **Behavior** panel (Contact Behavior, Joint Behavior, or Spring Behavior) and under the **Behavior** heading (Contact Behavior, Joint Behavior, or Spring Behavior) on the panel for the specific connection type (Contact, Joint, or Spring). Click the link to view the list.

#### 4.8.7. Unconstrained Body Detection

Prior to solve, AIM reviews the model for bodies without contact, joints or constraints. If free bodies are detected, AIM then stops the solution and highlights the under-constrained bodies. You must then connect those bodies to ones already constrained or add a constraint to the free body to prevent rigid body motion.

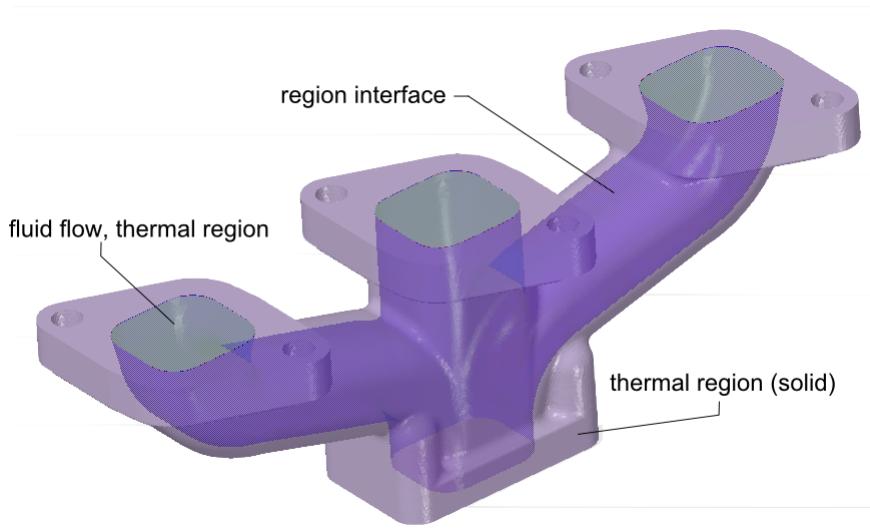
Your options depend in your analysis type.

1. For a structural analysis, either connect these bodies using a contact, joint, or spring, or constrain any topology of the free body using sufficient support or displacement.
2. For a thermal analysis, either connect these bodies using a contact or add a temperature, convection, or radiation constraint to the free body..
3. For an electric conduction and electrostatics analysis, either connect these bodies using a contact or add a voltage constraint to the free body.

#### 4.8.8. Region Interface Conditions

Region interface conditions can be defined within a single physics region where there is a mesh discontinuity, or when the topology is not connected. Region interfaces can also be defined between two physics regions (for example, between a fluid and a solid in a conjugate heat transfer problem). These interface conditions are defined by the specific physics regions and the faces on either side of the interface, where each face is allowed to participate in a single region interface. These region interfaces provide a way to handle various types of discontinuities between different physics regions or within a single physics region.

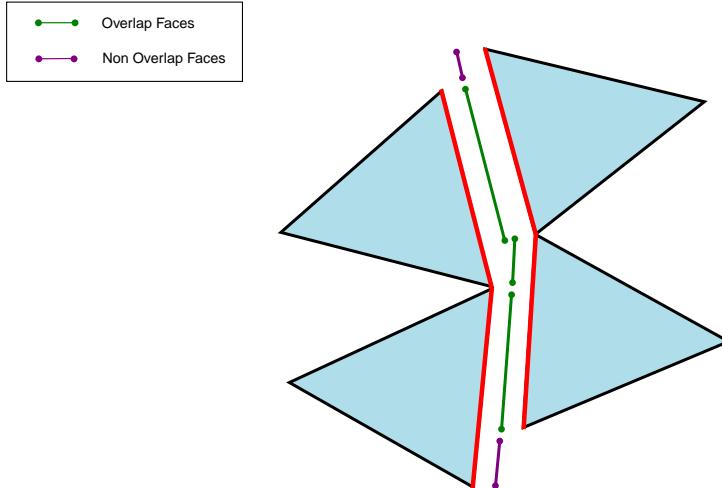
**Figure 4.8.8.1. Example of Multiple Physics Regions and a Region Interface**



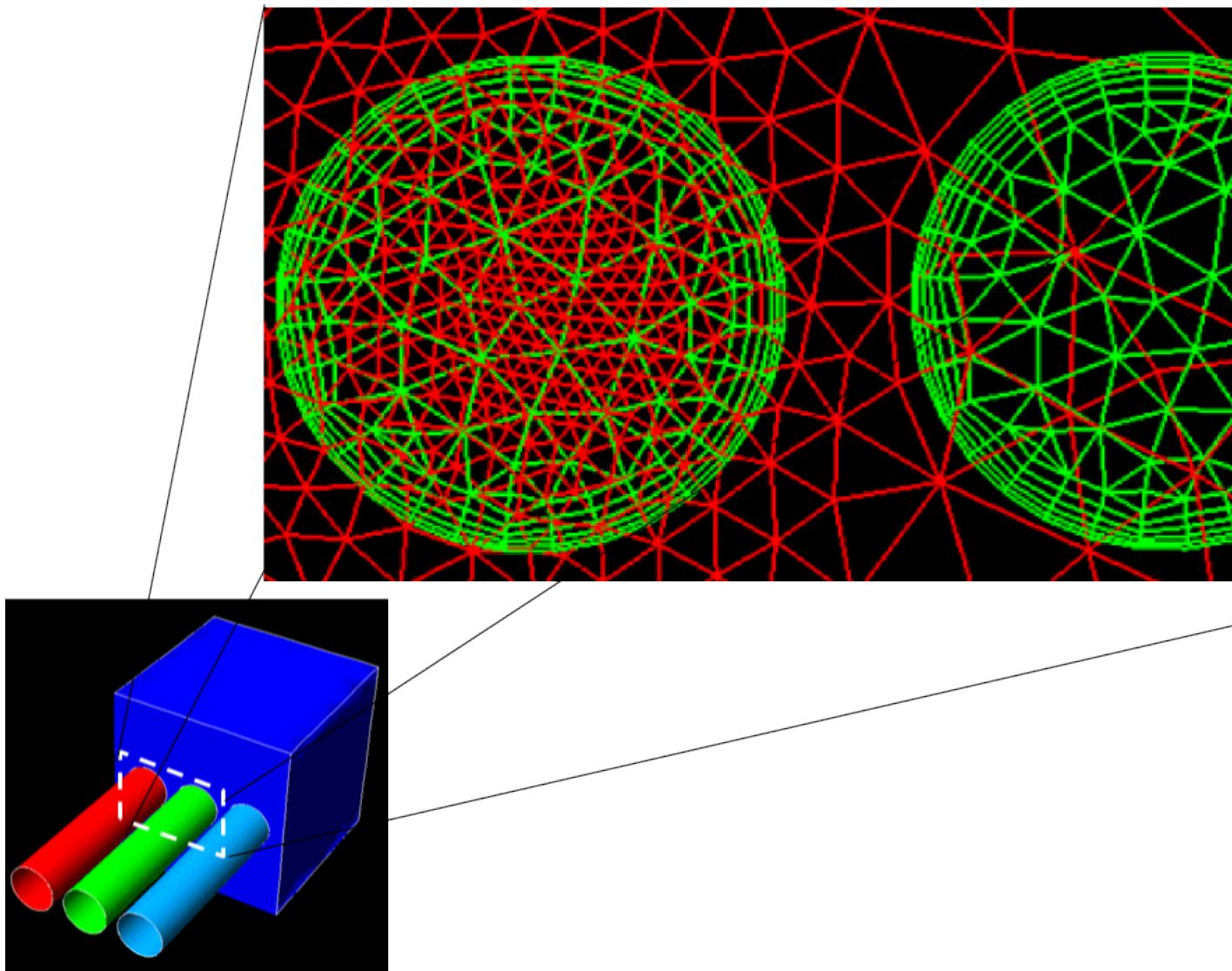
When adding a region interface condition, for either a [fluid flow](#) or [polymer extrusion](#) simulation, the physics regions on the two sides of the interface (the 'parent' regions) must be provided. The interface between the physics region connects the physics regions at the mesh level. If the mesh for each physics region at the interface is identical, then the interface is 'conformal'; if they differ, it is 'non-conformal', and AIM intersects the meshes on the two sides to connect them.

For fluid flow simulations with multiple physics regions, or even to create internal region interfaces for simulations with singular regions containing multiple parts, region interfaces can be automatically [generated](#). When the topology consists of multiple, disconnected, bodies being represented by a single physics region, you will need to create a region interface condition in order to allow for flow or heat transfer to occur between them; otherwise their adjacent bounding faces will be treated as walls.

**Note:** It is possible that some faces on one physics region do not have any face on the other side with which to intersect (either because there are no faces, or because the candidate faces are too far away or too twisted). These leftover, or non-overlap, faces are assigned a default non-overlap boundary condition of a no-slip insulated wall. The percentage of non-overlap is reported in the [region interface statistics](#).



The following figure illustrates an example of non-overlap. The enhanced figure shows the two 'parent' surface meshes on the region interface. The red mesh that does not intersect with the green faces are the non-overlap faces for the red side. There are no non-overlap faces for the green side in this example.



### 4.8.8.1. Adding Region Interface Conditions for Fluid Flow Simulations

#### Interface Conditions > Add > Region Interface

Before creating a region interface condition to your fluid flow simulation:

- If your simulation will have two or more distinct physics regions within the Physics task, those regions must already be defined.
- If your simulation will have only one physics region within the **Physics** task and you are creating an internal region interface, that region must already be defined.

Region interfaces provide a means to manage different types of discontinuities between one or more different physics regions within the same physics task.

To add a new physics region interface:

1. In the **Physics** task, select **Interface Conditions > Add > Region Interface** to open the **Region Interface** panel.

2. Use the drop-down list to select the physics regions on either side of the interface, and then, using either graphical selection or through the use of selection sets, select the topological locations on either side of the interface, filtered by the physics region selection.

**Note:** Each face can only be part of a single region interface. In the case where a face needs to be part of more than one interface, use your geometry editing tool to split that face so that there are individual faces for each region interface.

3. If you want to change the reference frame to one other than a global reference frame, select the appropriate option under **Reference frame**.
4. To model the effects of thermal conductance at a fluid-solid or solid-solid interface, expand **Interface Models > Energy Interface Conditions**.

For a coupled interface, under **Energy Interface Model**, select **Thermal conductance**. Thermal conductance between the regions is typically modeled as a result of:

- a thin material at a fluid-solid or solid-solid interface. The thermal conductance in this case is  $k/\Delta x$ , where  $k$  and  $\Delta x$  are the conductivity and material thickness.
- a thin film on a fluid-solid interface.
- contact conductance at a solid-solid interface.

Enter a constant value or expression in the **Thermal conductance** field; any expression must resolve to a constant.

5. To define the way your regions are connected, select the **Connection type**:

- **Standard** to connect adjacent meshes.
- **Periodic** to define translational or rotational periodicity between regions.

If the flow field is repeated in multiple identical regions, then only one region must be solved, but the boundaries are specified as periodic (via a rotation or translation). Select the **Periodicity type** to be either **Translational** or **Rotational**.

Option	Details
<b>Translational</b>	The two sides of the interface must be parallel to each other such that a single translation transformation can be used to map one region to the other. If two sides already coincide so that a translation of $<0,0,0>$ is used; this is equivalent to a standard connection. The translation vector is automatically computed by the solver.
<b>Rotational</b>	The two sides of the periodic interface can be mapped by a single rotational transformation about the local Z axis. This type of periodicity is commonly used in the analysis of a single blade passage in a rotating machine. The rotation angle is automatically computed by the solver.

6.  If you want to specify any changes to how intersecting faces are handled, select the appropriate option under **Intersection Control**.
  - If the geometries on the two sides of the interface match, select **Interface geometries match** to prevent the possibility of non-overlapping faces being generated during intersection.
  - The default tolerances are appropriate for most cases, but you can adjust the threshold values to meet your specifications. If the tolerance is exceeded, the intersection will be ignored. Entering a **Bound**

**extension factor** will create a bounding box which is used to search for and identify possible face intersections.

## 4.8.8.2. Adding Region Interface Conditions for Polymer Extrusion Simulations

### Interface Conditions > Add > Region Interface

Before creating a region interface condition for your polymer extrusion simulation:

- If your simulation will have two or more distinct physics regions within the Physics task, those regions must already be defined.
- If your simulation will have only one physics region within the **Physics** task and you are creating an internal region interface, that region must already be defined.
- When working with region interfaces in polymer extrusion simulations, you should be aware of some [considerations](#) regarding material assignments and non-overlap faces.

Region interfaces provide a means to connect two parts of a single physics region or two different physics regions within the same physics task that do not share topology.

To add a new physics region interface:

1. In the **Physics** task, select **Interface Conditions > Add > Region Interface** to open the **Region Interface** panel.
2. Use the drop-down list to select the physics regions on either side of the interface, and then, using either graphical selection or through the use of selection sets, select the topological locations on either side of the interface, filtered by the physics region selection.
- Note:** Each face can only be part of a single region interface. In the case where a face needs to be part of more than one interface, use your geometry editing tool to split that face so that there are individual faces for each region interface.
3. For polymer extrusions simulations involving conjugate heat transfer between a thermal, polymer extrusion physics region and a thermal physics region, click the **Create new** button, under **Fluid-solid interface behavior**, to create a new fluid-solid behavior.
4. Since faces that do not share topology can be connected when they are geometrically close to each other, if you want to change the criteria that define when faces are close enough, select the appropriate option under **Intersection Control**.
  - If the geometries on the two sides of the interface match, select **Interface geometries match**.
  - Along region interfaces, the polymer extrusion solver operates at an elemental level (mesh node, mid-face node, etc.) and creates a correspondence between the primary faces (sources) and the secondary faces (targets) of the two adjacent regions.



Thermal (temperature) and momentum (velocity) information that occurs between the source and target faces along the interface is controlled by the **Thermal transfer** and **Momentum transfer** fields.

Table 4.8.8.2.1. Thermal and Momentum Transfer Methods

Option	Description
<b>Program controlled</b>	Allows AIM to select the best approach and parameters to determine the optimal thermal and momentum transfer.

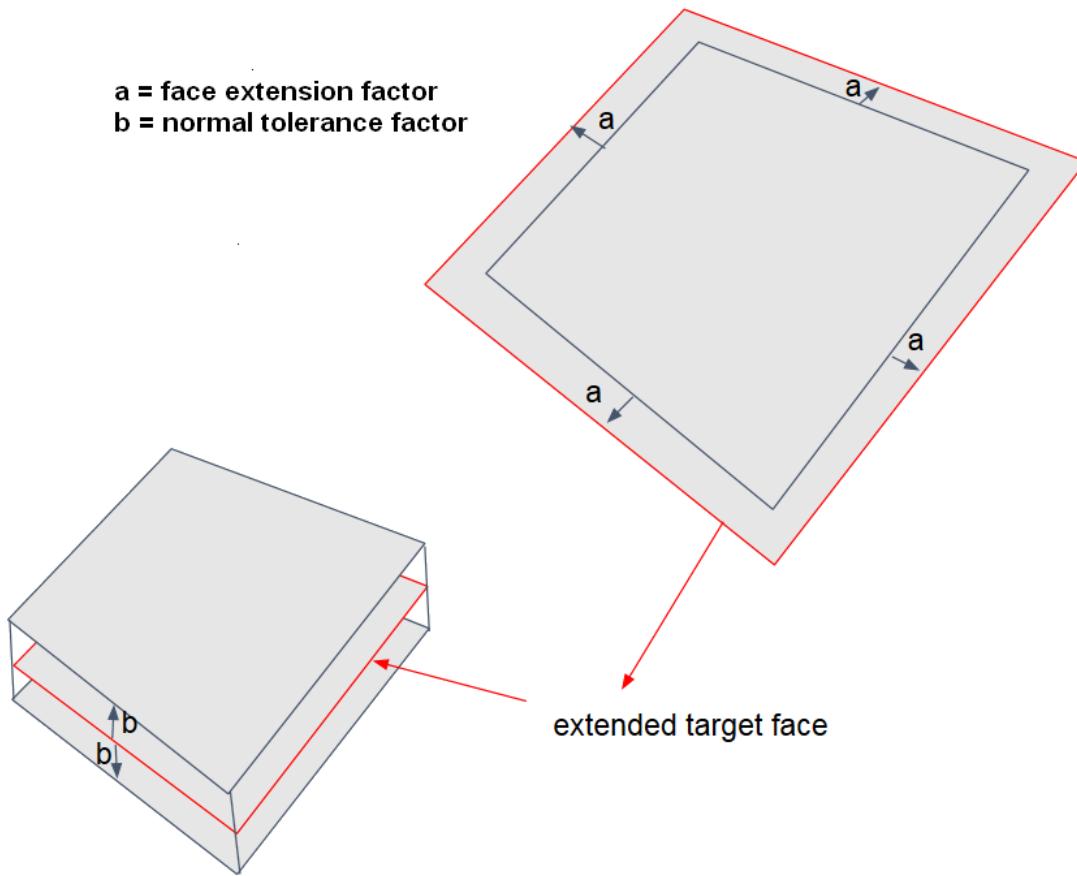
Option	Description
<b>Bidirectional</b>	Two passes are initiated: a correspondence is initiated from the source to the target if the source node is owned by an element smaller than the target; followed by a correspondence being initiated from the target to the source if the target node is found to be inside an element smaller than the source.
<b>Unidirectional</b>	Creates a correspondence from the source to the target if the source node is found to be inside an element on the target side.
<b>Complimentary</b>	Two passes are initiated: a correspondence is initiated from the source to the target if the source node is found to be inside an element on the target side; followed by a correspondence being initiated between target nodes for target nodes that were not included in the initial pass.
<b>Filtered</b>	Three passes are initiated: relationships are stored for any source node that is found to be inside an element on the target side; excessive relationships are then filtered out; then a correspondence is initiated based on the remaining relationships.

-  For a fluid-solid interface, use the **Thermal transfer** option to indicate how the temperatures on one side of the interface relate to temperatures on the other side of the interface.
  -  Use the **Smoothing factor** option to control the thermal transition between un-matched sides of the interface. Use a high value to obtain a smooth thermal transition.
-  For a fluid-fluid interface, use the **Momentum transfer** option to indicate how the velocities on one side of the interface relate to velocities on the other side of the interface.
  -  Use the **Spreading factor** option to control the momentum transition between un-matched sides of the interface.
-  When the **Thermal transfer** or the **Momentum transfer** options are set to anything other than **Program controlled**, additional information is required to make the face connections between regions easier to detect:
  -  Use the **Primary side** option to indicate the side of the physics region where the thermal transfer originates. For fluid-solid interfaces, ensure the temperature distribution along the interface is correct for cases where you have a problematic solution.

You can switch the sides assigned to be the source and the target by changing the value of the **Primary side** field. In doing so, you can create different correspondences along the interface. It is generally better to assign the face with the more refined mesh to be the source (or primary), and then assign the face with the coarsest mesh to be the target (or secondary).

  -  Extend the target face size by multiplying the average edge size by the **Face extension factor**.

-  Expand the extended target face in the normal direction to the face by multiplying the average edge size by the **Normal tolerance factor**, creating a 3D element.



#### 4.8.8.2.1. Considerations for Region Interfaces and Polymer Extrusion

When working with region interfaces in polymer extrusion simulations, you should be aware of some considerations regarding material assignments, non-overlap faces, and die deformations.

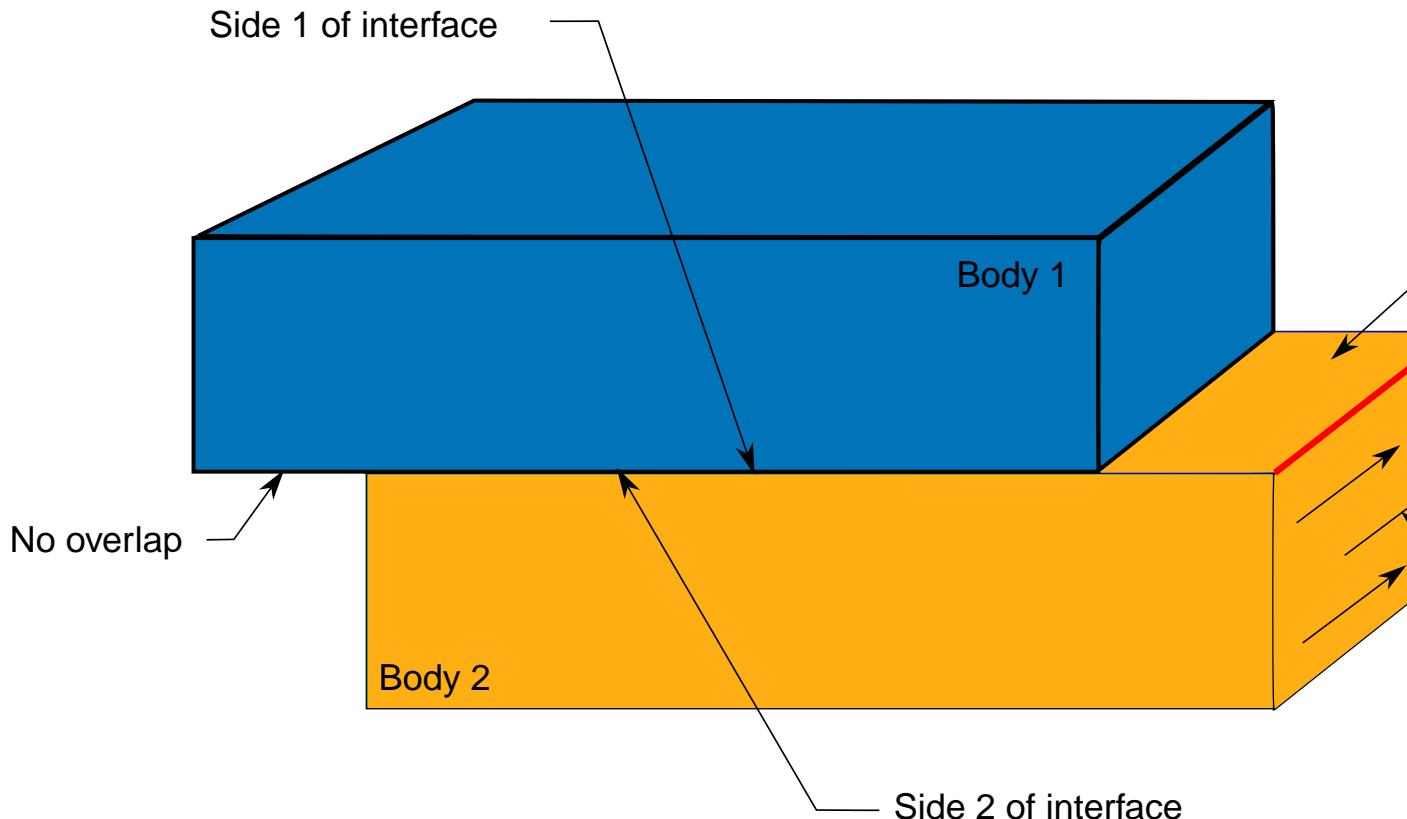
### Material Assignment and Material Models

As you work within your AIM study, you may find yourself having several versions, or instances, of the same material that have the same name and properties. Even though the materials look like they are the same, AIM views them as different material objects. So, when assigning materials to the regions on the two sides of a fluid-fluid interface, for example, make sure that you use the same instance of the material for both material assignments.

Likewise, you need to make sure that the material model used for each material assignment (Newtonian fluid, Generalized Newtonian fluid, etc.) is the same on each side of the interface.

## Non-Overlap Faces and Region Interfaces

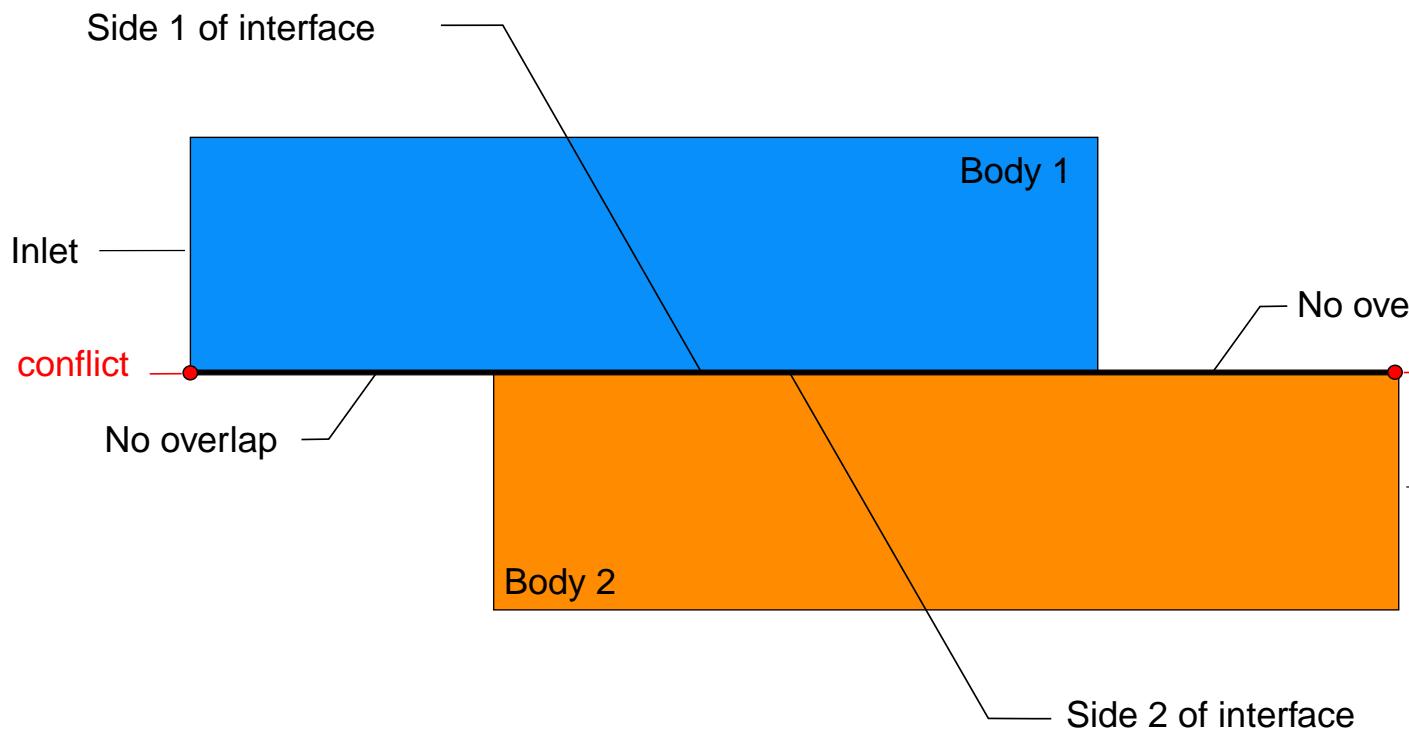
When the geometrical faces on both sides of the region interface do not match, a zero-velocity and/or an insulated condition is imposed on the areas that have no corresponding face on the other side of the region interface. This condition may conflict with conditions on neighboring faces, as illustrated in the following figure:



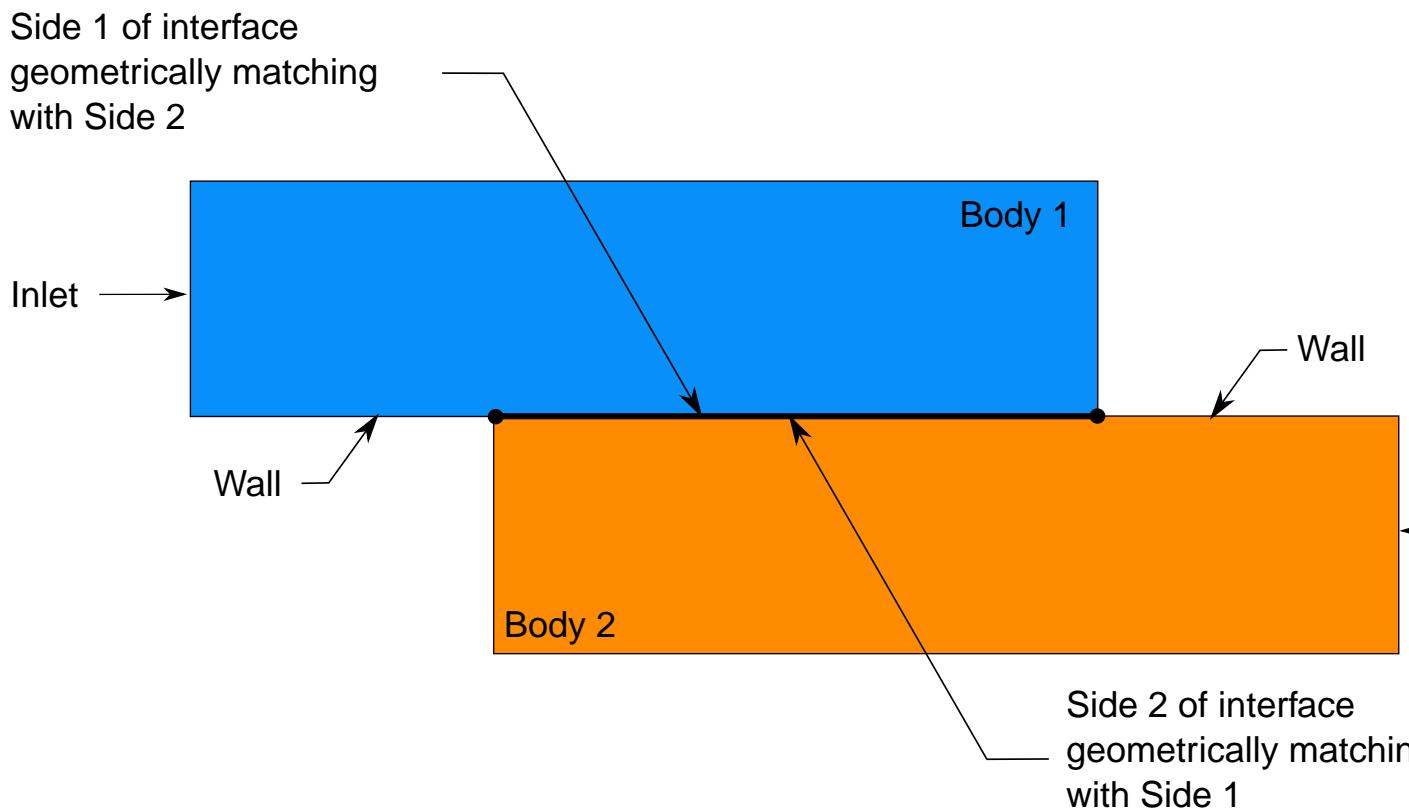
In order to avoid such conflicts, you should define a region interface on areas that match (or very nearly match). For example, you can use an external CAD system (such as SpaceClaim Direct Modeler) to edit the geometry accordingly and define matching faces.

In addition, you should avoid any intersection between a region interface and an extrude exit, a die deformation, a free surface, an inlet, or an outlet, as illustrated in the following figures:

**Figure 4.8.8.2.1.1. Close-Up View of Nonoverlapping Areas (Before Conflict Resolution)**



**Figure 4.8.8.2.1.2. Close-Up View of Nonoverlapping Areas (After Conflict Resolution)**



# Die Deformations

When a region interface is adjacent to a die deformation:

- For fluid-fluid interfaces, you will need to use an external CAD system (such as SpaceClaim Direct Modeler) to create a multi-body part with connected bodies (see [Region Connectivity and Polymer Extrusion](#) on page 215) and create a single extrusion physics region.
- For fluid-solid interfaces, you cannot model thermal physics regions on the other side of a die deformation, so you must remove the region interface and the thermal physics region.

## 4.8.8.2.2. Adding a Fluid-Solid Interface Behavior for Polymer Extrusion Simulations

Before creating a region interface behavior to your polymer extrusion simulation:

- If your simulation will have two or more distinct physics regions within the Physics task, those regions must already be defined.
- If your simulation will have only one physics region within the **Physics** task and you are creating an internal region interface, that region must already be defined.

In the **Fluid-Solid Interface Behavior** panel, you can assign details about the fluid and the solid properties.

1. For **Flow specification**, select **No slip**, **Partial slip**, or **Free slip**.
2. For **Wall velocity**, select **Stationary** or **Moving**.

These properties are similar to those of a [wall boundary](#) for polymer extrusion simulations.

## 4.8.8.3. Automatically Generating Region Interfaces for Fluid Flow Simulations

### Physics > Interface Conditions > Generator

You can use the interface generator tool to create interfaces between physics regions in fluid flow simulations, or to create interfaces between parts of a single region.

To automatically generate region interfaces:

1. Add an interface generator object to your simulation (**Physics > Interface Conditions > Add > Generator**). This displays the **Generator** panel that detects where any physics regions are connected based on a given location and tolerance.
2. For **Location**, specify one or more volumes or bodies, or use the drop-down list to select a corresponding pre-existing volume-based selection set. The expression `AllBodies()` is the default.
3. For **Tolerance**, specify the maximum distance between two face pairs from adjacent physics regions to be considered as candidates for a physics region interface.
4. Click **Generate Region Interfaces**.

When you generate the interfaces, AIM automatically creates the region interfaces based on face-pair detection between parts or by using shared topology. For recommendations and limitations on modeling Shared Topology with the Geometry Modeler in AIM, refer to [Model Editing](#) on page 92. For additional details, refer to the section **Workbench > Shared Topology** in the Geometry Modeling Help.

The following combinations of physics types are valid for generating region interfaces:

- **Fluid flow** region & **Thermal** region
- **Fluid flow** and **Thermal** region & **Thermal** region

- **Thermal** region & **Thermal** region
- and within any region of the above types

**Note:** Each face can only be part of a single region interface. In the case where a face must be part of more than one interface, use your geometry editing tool to split that face or create shared topology so that there are individual faces for each region interface.

If you create your simulation using the **Fluid-Solid Heat Transfer** template, and you use the **Detect interfaces automatically** option, an interface generator object is created as part of the simulation template. Update the generator object (using the **Generate Region Interfaces** button) to create the region interface objects automatically.

**Note:**

- Generated region interfaces are automatically associated with the corresponding generator object.
- Deleting an interface generator object removes all associated region interfaces.
- If you manually change a region interface that has been generated, then the association between the generator object and the altered region interface is removed. Therefore, you can regenerate or delete the generator object and still retain the altered region interface.
- The generator object will only need to be updated when either the location or tolerance is changed, or if the geometric model has changed.
- Multiple region interface generators can be added to a simulation, however, individual generators cannot be duplicated.
- Regenerating an interface creates a new interface based on current criteria, and removes all existing associated generated interfaces.
- Interface generation occurs regardless of the physics condition setup or any existing, manually defined, region interfaces.

## 4.8.8.4. Region Interface Statistics for Fluid Flow Simulations

To view region interface statistics, your simulation should contain [region interface conditions](#).

You can view a summary of the region interface statistics when solving the physics. This will provide information about non-conformal interfaces in the mesh. To access this information, go to the **Transcript** tab, as shown below.

Name	Type	Area	Side 0	Side 1
RegionInterface 1	NonConformal	7.229e-001	0.0	22.0
RegionInterface 2	NonConformal	7.328e-001	0.6	0.0

Notice: multiple flow passages found for pbns-cpld:  
Physics regions for connected-region-1:  
PhysicsRegion 3  
Physics regions for connected-region-2:  
PhysicsRegion 1

## 4.8.9. Physics Coupling Interface

The physics coupling interface transfers data to a physics region from a physics region in a separate Physics Solution task. On each physics coupling interface, you can transfer:

- [fluid force data](#) from surfaces in a fluid simulation onto surfaces in a structural simulation;
- [temperature data](#):
  - from solid bodies that are part of a fluid-solid heat transfer simulation into bodies in a structural simulation, and
  - from solid bodies that are part of a thermal simulation into bodies in an electromagnetic simulation; or
- [heat rate data](#) from solid bodies in an electromagnetics simulation into bodies in a thermal simulation.

**Note:** Only a single physics coupling interface can be created for the transfer of temperature data to an electromagnetics physics task.

For an exhaustive list of the data transfers that are available, see [Data Transfers Possible using the Physics Coupling Interface](#) on page 381.

When you update the Physics Solution task that is receiving data, the physics coupling interface is applied, and data from the source mesh is [mapped](#) to the target mesh. The resulting data is applied to the selected location, and used in the physics solution. Where applicable, single and multiple [simulation steps](#) are considered.

**Note:** In a time-dependent solution, data from the final time step is transferred.

The [physics coupling transcript](#) has information about this interface. Once the Physics Solution task associated with the physics coupling interface is updated, the transcript will have diagnostic information about the mapping and data transfer.

## 4.8.9.1. Data Transfers Possible using the Physics Coupling Interface

On each physics coupling interface, data is transferred to one physics region. This physics region on the target must have physics and calculation types that can receive data.

The physics region on the source is selected in the physics coupling interface, but it must exist in a separate Physics Solution task. This source physics region must have physics and calculation types that can send data.

This section shows the data transfers possible for each physics type that can be used as a target, indicating the types of data that it can receive from applicable source physics types. Only physics types that support transfers are included.

Supported calculation types are steady/static, time-dependent, and frequency response.

For the source and target locations on the physics coupling interface, select faces or bodies depending on the type of data being transferred:

- For force data, select faces; selection of faces on both solid and surface bodies is supported. Note that faces selected for source locations must be part of a [wall](#) or [region interface](#).
- For temperature data, select bodies; note that temperature cannot be transferred to surface bodies.
- For heat rate data, select bodies.

Table 4.8.9.1.1. Physics Regions that can receive data

Target Calculation Type	Target Physics Region	Can receive data from
<b>Steady State / Static</b>	<b>Structural</b>	Surface force from <b>Fluid Flow</b> * Surface force from <b>Fluid Flow, Thermal</b> * Volume temperature from <b>Thermal</b> (when the <b>Thermal</b> solid physics region is in the same Physics Solution task as a <b>Fluid Flow</b> physics region)*
	<b>Structural, Thermal</b>	Surface force from <b>Fluid Flow</b> * Surface force from <b>Fluid Flow, Thermal</b> * Volume heat rate from <b>Electromagnetics</b>
	<b>Structural, Electric Conduction</b>	Surface force from <b>Fluid Flow</b> * Surface force from <b>Fluid Flow, Thermal</b> * Volume temperature from <b>Thermal</b> (when the <b>Thermal</b> solid physics region is in the same Physics Solution task as a <b>Fluid Flow</b> physics region)*
	<b>Structural, Thermal, Electric Conduction</b>	Surface force from <b>Fluid Flow</b> * Surface force from <b>Fluid Flow, Thermal</b> * Volume heat rate from <b>Electromagnetics</b>
	<b>Thermal</b>	Volume heat rate from <b>Electromagnetics</b>
	<b>Thermal, Electric Conduction</b>	Volume heat rate from <b>Electromagnetics</b>
	<b>Electromagnetics</b>	Volume temperature from <b>Thermal</b> * (when the <b>Thermal</b> solid physics region is not in the same Physics Solution task as a <b>Fluid Flow</b> physics region)
<b>Time Dependent</b>	<b>Thermal</b>	Volume heat rate from <b>Electromagnetics</b>
<b>Frequency Response</b>	<b>Electromagnetics</b>	Volume temperature from <b>Thermal</b> * (when the <b>Thermal</b> solid physics region is not in the same Physics Solution task as a <b>Fluid Flow</b> physics region)

\* This source physics region supports the time-dependent calculation type. When the source physics region is time-dependent, data from the final time step is transferred.

## 4.8.9.1.1. Transfer Force Data using the Physics Coupling Interface

Transfer force data from surfaces in a fluid simulation onto surfaces in a structural simulation using the [physics coupling interface](#).

Before setting up the physics coupling interface, you must:

1. Set up the Physics Solution task for the fluid simulation from which you want to transfer forces. Ensure the [physics region](#)'s physics type includes **Fluid flow**, and [walls](#) or [region interfaces](#) are defined.

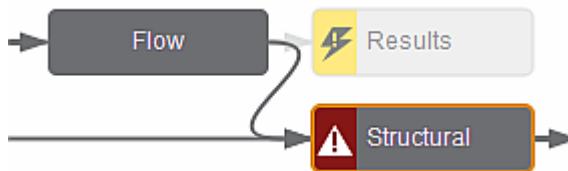
- Set up the Physics Solution task that is receiving the force data. Make sure that the physics region's physics type includes **Structural**.

### Physics Solution > Interface Conditions > Physics Coupling

To set up the physics coupling interface in the structural Physics Solution task:

- Select faces in the model for **Location (target)**. The faces you select will receive the fluid force data.
- For the **Physics region on source**, use the drop-down to select the fluid physics region from which you want to transfer fluid force data.

In the Workflow tab, the selected source region's Physics Solution task automatically connects to the structural Physics Solution task. This connector has to exist between the two Physics Solution tasks for the physics coupling interface to work.



- In the physics coupling panel, under **Location (source)**, the expression `AllCouplingSourceFaces()` automatically selects all of the source region's walls and region interfaces.

You can refine the source locations by selecting a wall, a region interface, or a selection set from the location drop-down. Selection sets with only faces that are part of walls or region interfaces are available in the drop-down.

- Once the physics coupling interface is Up-to-date, use the advanced mapping controls. Under **Mapping Control > Conservative algorithm**, you can see that the force transfer uses the [surface element projection algorithm](#). If you want to modify the overall gap tolerance for the force transfer, modify the [conservative mapping controls](#).

When you update the Physics Solution task, the physics coupling interface is applied and data from the source mesh is [mapped](#) to the structural mesh. The physics coupling transcript has [diagnostic information](#) about the mapping and data transfer.

## 4.8.9.1.2. Transfer Temperature Data using the Physics Coupling Interface

Use the [physics coupling interface](#) to transfer temperature data:

- from solid bodies that are part of a fluid-solid heat transfer simulation into bodies in a structural simulation, and
- from solid bodies that are part of a thermal simulation into bodies in an electromagnetic simulation.

**Note:** Only a single physics coupling interface can be created for the transfer of temperature data to the electromagnetics physics task.

Before setting up the physics coupling interface, you must:

- Set up the Physics Solution task for the fluid-solid heat transfer or steady thermal simulation from which you want to transfer temperatures. Ensure the solid region's physics type is set to **Thermal**.
- Set up the Physics Solution task that is receiving the temperature data.

- To transfer temperature data to a body in a structural simulation, make sure that the [physics region's physics type](#) is set to **Structural**.
- To transfer temperature data to a body in an electromagnetic simulation, make sure that the [physics region's physics type](#) is set to **Electromagnetics** and that [temperature-dependent materials](#) are used.

**Note:** Temperature values will be used to initialize the mesh, in the following priority: temperature values obtained from physics coupling; a [body temperature](#) defined as an electromagnetic condition; or the [ambient temperature](#) defined in physics options.

### Physics Solution > Interface Conditions > Physics Coupling

To set up the physics coupling interface in the structural or electromagnetics Physics Solution task:

1. For the **Location (target)**, select one or more solid bodies (i.e. not surface bodies) that belong to the target and to which temperature will be applied.

For electromagnetic simulations, only bodies that use [temperature-dependent materials](#) are valid target bodies. You can select one or more solid bodies that belong to the target region and use temperature-dependent materials.

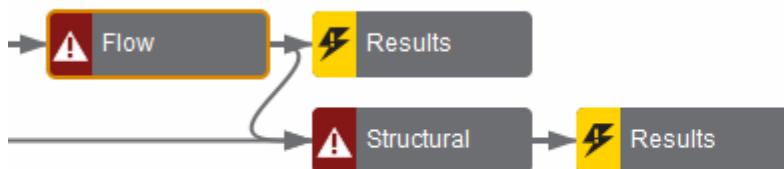
**Note:** Temperature values obtained using physics coupling will replace those specified as body or ambient temperatures.

2. For the **Physics region on source**, use the drop-down to select the physics region from which you want to transfer data. All physics regions capable of transferring temperature data will be available in the drop-down.

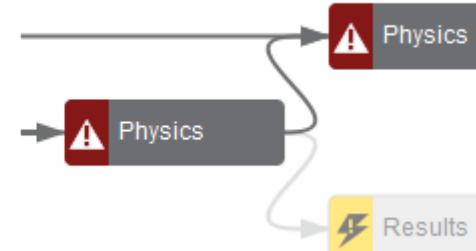
- For structural targets, select a solid region that is part of a fluid-solid heat transfer simulation.
- For electromagnetic targets, select a solid region that is part of a thermal simulation. Only regions with thermal physics are available in the drop-down.

In the Workflow tab, the selected source region's Physics Solution task automatically connects to the structural or electromagnetics Physics Solution task. This connector has to exist between the two Physics Solution tasks for the physics coupling interface to work.

### Fluid-Solid Heat Transfer to Structural



### Thermal to Electromagnetics



3. In the physics coupling panel, under **Location (source)**, the expression `AllCouplingSourceBodies()` automatically selects all of the source region's bodies that are capable of sending the reported data type.

You can refine the source locations by selecting selection sets. Selection sets with bodies that are part of the source region are the only ones available in the drop-down.

4. Once the physics coupling interface is **Up-to-date**, you can use the advanced mapping controls. If you want to modify the speed and accuracy of the mapping for the temperature transfer, modify the [mapping controls](#).

When you update the Physics Solution task, the physics coupling interface is applied and data from the source mesh is [mapped](#) to the structural or electromagnetics mesh. The physics coupling transcript has [diagnostic information](#) about the mapping and data transfer.

## 4.8.9.1.3. Transfer Heat Rate Data using the Physics Coupling Interface

Transfer heat rate data from a solid region that is part of an electromagnetics simulation into bodies in a thermal simulation using the [physics coupling interface](#). Note that the thermal simulation cannot be part of a fluid-solid heat transfer simulation.

Before setting up the physics coupling interface, you must:

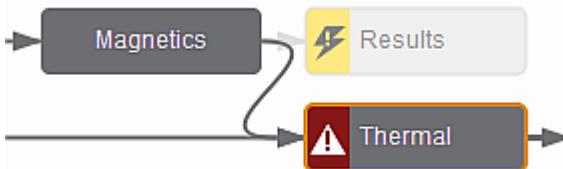
1. Set up the Physics Solution task for the electromagnetics simulation from which you want to transfer heat rate data. The following coupling-specific qualifications apply:
  - If the Current condition is modeling stranded conductors (i.e., the **Number of conductors** is set to a value greater than 1), a **Fill factor** must be specified as described in [Setting Current using a Cross-Section](#) on page 304 (default value is 1). This value is the adjustment/correction ratio the solver applies to the conductivity when calculating the losses for stranded conductors.
  - If the **Calculation Type** is set to **Frequency Response**, a [magnetic loss tangent](#) material property has been defined for one or more bodies, and/or one or more bodies may have [eddy effect computation](#) or [core loss computation](#) enabled.
2. Set up the Physics Solution task that is receiving the heat rate data. Make sure that the [physics region's](#) physics type includes **Thermal** and does not include **Fluid flow**.

### Physics Solution > Interface Conditions > Physics Coupling

To set up the physics coupling interface in the thermal Physics Solution task:

1. Select bodies in the model for **Location (target)**. The bodies you select will receive the heat rate data.  
**Important:** If you are using the [scaled nearest element](#) mapping algorithm, be careful to select bodies for the target location that intersect with the source location; this algorithm is sensitive to non-intersecting volumes on either the source or target locations. The [volume element intersection](#) mapping algorithm is recommended for cases without substantial intersection between source and target locations.
2. For the **Physics region on source**, use the drop-down to select the physics region from which you want to transfer data. Select a region that has an electromagnetics physics type.

In the Workflow tab, the selected source region's Physics Solution task automatically connects to the electromagnetics Physics Solution task. This connector has to exist between the two Physics Solution tasks for the physics coupling interface to work.



3. In the physics coupling panel, under **Location (source)**, the expression `AllCouplingSourceBodies()` automatically selects all of the source region's bodies that can send heat rate data.

If [mapping](#) issues occur, you can [define a selection set](#) for the source location. This will facilitate a comparison of the source and target geometries, enabling you to adjust the geometries to [improve mapping quality](#).

You can refine the source locations by selecting a selection set from the location drop-down. Selection sets with bodies that are part of the source physics region are the only selection sets available in the drop-down.

**Note:** Be careful to select bodies for the source location that intersect with the target location, especially if you are using the [scaled nearest element](#) mapping algorithm.

4. You can use the **Frequency (source)** drop-down to select the desired frequency from which you want to transfer data. This drop-down is available only if the **Physics region on source** refers to a region that is being solved with **Calculation type** set to **Frequency response**. The default value is determined by the upstream source participant's [frequency response calculation-type settings](#).
5.  Once the physics coupling interface is **Up-to-date**, open the advanced mapping controls. Under **Mapping Control > Conservative algorithm**, you can [select the conservative algorithm](#) to be used in the heat rate transfer.

When you update the Physics Solution task, the physics coupling interface is applied and data from the source mesh is [mapped](#) to the target mesh. The physics coupling transcript has [diagnostic information](#) about the mapping and data transfer.

## 4.8.9.2. Control the Mapping on the Physics Coupling Interface



Mapping controls are available on the [physics coupling interface](#) panel.

### Physics Solution > Interface Conditions > Physics Coupling > Mapping Control

You can control the speed and accuracy of the [mapping](#) by changing the mapping algorithm. Mapping controls are available once the physics coupling interface is **Up-to-date** or **Conditionally up-to-date**.

For more information, see:

[Setting Profile-Preserving Mapping Controls](#) on page 386

[Setting Conservative Mapping Controls](#) on page 387

### 4.8.9.2.1. Setting Profile-Preserving Mapping Controls



Mapping controls are available on the [physics coupling interface](#) panel.

### Physics Solution > Interface Conditions > Physics Coupling > Mapping Control

The **Mapping Control** for profile-preserving transfers appears for data transfers of non-conservative variables, such as temperature. To control the speed and accuracy of the profile-preserving mapping:

1. On the physics coupling panel, go to **Mapping Control > Profile-preserving algorithm**.
2. Select the mapping algorithm:
  - [Nearest node](#) – gives a fast mapping and is the default algorithm used.
  - [Radial basis function](#) – gives a mapping that is slower and more accurate. Choose this algorithm when more accuracy is desired, and especially if the source data distribution is non-uniform and the source and target meshes are not coincident.
3. If you selected [Radial basis function](#), select one of the available functions:

- **Thin plate spline** – the default and recommended option.
- **Gaussian** – allows you to modify the **Shape parameter**. The shape of the Gaussian function is controlled by this parameter. The parameter's default is 5, and you can change this to 0, or any positive value. The value can also be [parameterized](#). Larger values for the shape parameter will make the effects of the source nodes more localized, and will result in behavior close to that of the [nearest node algorithm](#).

## 4.8.9.2.2. Setting Conservative Mapping Controls



Mapping controls are available on the [physics coupling interface](#) panel.

### **Physics Solution > Interface Conditions > Physics Coupling > Mapping Control**

The **Mapping Control** for conservative transfers appears for data transfer of:

- force data, where the [surface element projection](#) algorithm is used, and
- heat rate data, where the [volume element intersection](#) or the [scaled nearest element](#) algorithm is used.

For more information, see:

[Setting Conservative Mapping Controls for Force Data Transfers](#) on page 387

[Setting Conservative Mapping Controls for Heat Rate Data Transfers](#) on page 387

### *4.8.9.2.2.1. Setting Conservative Mapping Controls for Force Data Transfers*

For force data transfers, the [surface element projection](#) algorithm is used. Additional mapping controls enable you to control the absolute gap tolerance for your physics coupling interface. To do so:

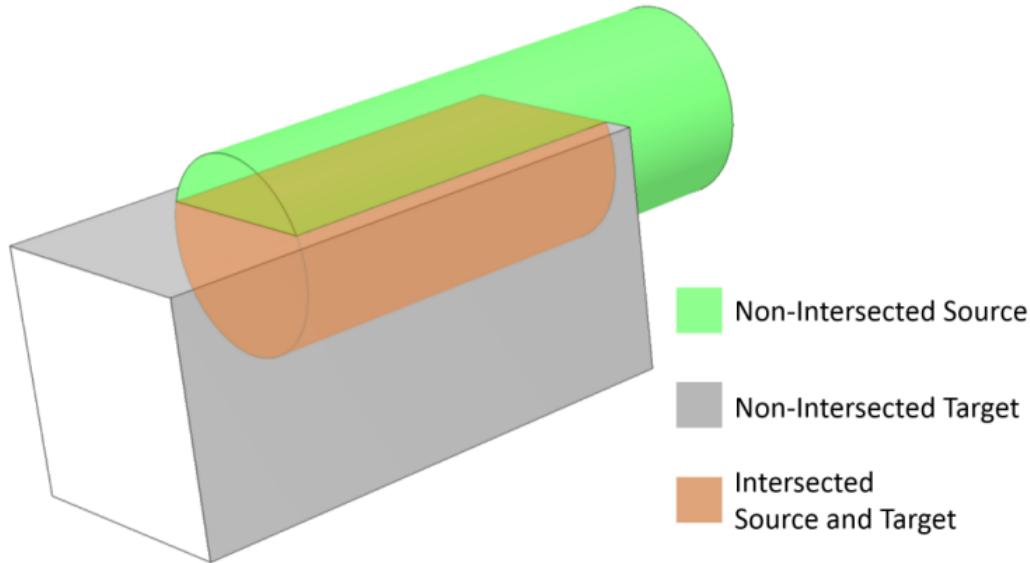
1. On the physics coupling panel, go to **Mapping Control > Conservative algorithm**.
2. Select the **Absolute gap definition** method:
  - **Program controlled** – by default, the program calculates the appropriate gap values automatically.
  - **Global value** – enables you to enter a single absolute gap tolerance value that is applied to all the regions in the physics coupling interface.
3. If you selected **Global value** tolerance definition, enter the **Absolute gap tolerance** value; this value will override the program-calculated values and will be applied to all regions in the physics coupling. To allow for the transfer of data, this value should not be smaller than the gap between the source and target locations of your physics coupling interface. For details on setting values to ensure the successful transfer of data, see [Correcting Gap Size and Gap Tolerance Mapping Issues](#).

### *4.8.9.2.2.2. Setting Conservative Mapping Controls for Heat Rate Data Transfers*

For heat rate data transfers, you can select the mapping algorithm most suited to your needs for speed and accuracy. To do so:

1. On the physics coupling panel, go to **Mapping Control > Conservative algorithm**.
2. Select the mapping algorithm:
  - [Volume element intersection](#) – gives the most accurate mapping. Use this algorithm when more accuracy is desired, and especially if there is a high percentage of volumetric non-intersection between source and target locations.

- **Scaled nearest element**—gives a mapping that is faster but less accurate when there are non-intersecting volumes in the source and target locations. Choose this algorithm when speed is a higher priority than accuracy and when there is substantial intersection between source and target locations.



### 4.8.9.2.2.3. Correcting Gap Size and Gap Tolerance Mapping Issues

With the [surface element projection](#) mapping algorithm, the **Absolute gap definition** property determines whether the absolute gap tolerance is defined automatically or manually. The **Absolute gap tolerance** value, visible when you have selected the **Global value** tolerance definition method, helps to determine the overall gap tolerance, which in turn controls what locations can be mapped. If the overall gap tolerance is smaller than the gap between the source location and the target location, they will not be mapped to one another and the data will not be transferred.

To verify that a data transfer occurred, solve the Physics task and review the [Conservative Mapping Diagnostics](#) table in the physics coupling transcript. If a large percentage of the target elements are mapped using **Zero Value**, then there may be issues with the gap size and its effect on mapping.

To address gap-related mapping issues, decrease the gap size and/or increase the absolute gap tolerance. You can do this in the following ways:

- **Geometry task: Decrease the gap size by editing the geometry.**

In the model editing (accessed through the Geometry task if model editing is enabled) or external geometry editing software, edit the geometry to reduce the gap between source and target surfaces. You can modify either the fluid body or the structural body.

Note that in order to modify a shell body to decrease gap size, you must change its thickness as described in [Defining Thickness in Geometry Modeler](#) or [Defining Thickness for Imported Geometry \(No Model Editing\)](#).

- **Mesh task: Decrease the gap size by editing the Mesh tasks.**

In both the fluid and structural Mesh tasks, set the source surface's mesh element size to approximately the same size as the gap. This will create an overall gap tolerance larger than the gap between the source surface and the target location. The size of the both the source and target mesh elements dictates the gap settings used by the mapping algorithm. To use this approach, you must choose to define the mesh manually. For more information, see [Mesh Size and Distribution](#).

- **Physics task: Increase the absolute gap tolerance in Physics Coupling.**

Under **Mapping Control** in the physics coupling interface, set **Absolute gap definition** to **Global** value and set **Absolute gap tolerance** to a value the same size or slightly larger than the gap. Ideally, the absolute gap tolerance should be as close to the gap size as possible; setting it to too large a tolerance could cause data to be transferred incorrectly.

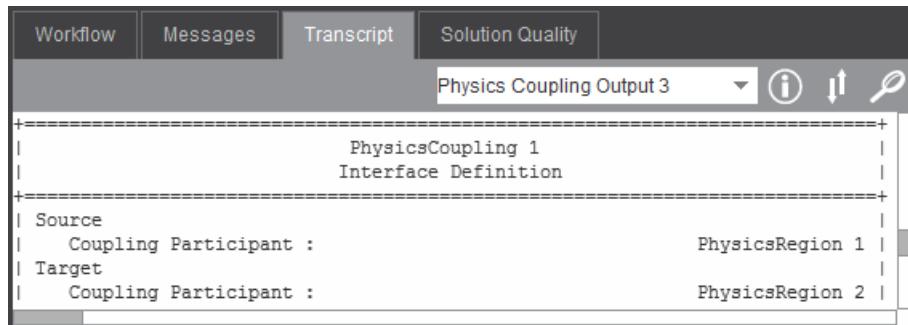
### 4.8.9.3. Physics Coupling Transcript

The physics coupling transcript is available in the **Transcript** tab on the View panel.

For each physics solution that has one or more physics coupling interfaces, one physics coupling transcript is automatically created. The physics coupling transcript keeps information from each update of the Physics Solution task. Information from the most recent update is at the bottom of the transcript file.

Once the physics solution has solved, the transcript has the following information for each physics coupling interface:

- The **Interface Definition** table with a summary of the participants that the interface connects, the variable transferred, and the [mapping controls](#) applied.
- The **Mapping Diagnostics** table with diagnostic information about the mapping on the interface. Use this information to assess:
  - the [accuracy of the force transfer](#),
  - the [accuracy of the temperature transfer](#), or
  - the [accuracy of the heat rate transfer](#).
- Error messages, if an error occurred with physics coupling.



#### 4.8.9.3.1. Assessing the Accuracy of the Force Transfer

Once the physics solution is solved, use the Conservative Mapping Diagnostics table to assess the accuracy of the conservative [mapping](#) that is used to [transfer force](#) on the physics coupling interface. The table from the most recent solution process is near the bottom of the [physics coupling transcript](#).

For more information, see:

- [Transcript Output for Force Transfers](#)
- [Interpreting the Transcript for Force Transfers](#)

##### 4.8.9.3.1.1. Transcript Output for Force Transfers

The transcript for the force transfer includes diagnostic information for both the source and the target.

The **Source** side of the interface has the following diagnostic information:

- The sum of each component of the data, for example **Force.x Sum [N]**, is used to assess the accuracy of a conservative data transfer (described below). The sum of the components on the source mesh is the sum of the data on the entire source mesh, including any elements that are not used in mapping.
- **Elements Used in Mapping [%]** is the percentage of the source mesh's elements that is matched with elements on the target mesh.

The **Target** side of the interface has the following diagnostic information:

- The sum of the components of the data, for example **Sum Force.x [N]**, is the sum of the data on the entire target mesh. This value is used to assess the accuracy of a conservative data transfer (described below).
- **Elements Mapped Using**
  - **Surface Element Projection [%]** is the percentage of target elements mapped to one or more source elements, and which receives data from those source elements. Ideally, the percentage of the target mesh using the [surface element projection algorithm](#) will be close to 100.
  - **Zero Value [%]** is the percentage of elements in the target mesh that do not overlap with the source mesh, and so receive a value of 0 [N].

### *4.8.9.3.1.2. Interpreting the Transcript for Force Transfers*

To assess the accuracy of this data transfer, review:

- The **Force Sum**.
  - A significant difference between the sum on the source and target means that the data may not be transferred accurately. A difference of 1 or 2 percent may be significant, but this range will depend on your specific case, especially if less than 100% of the source elements are used in mapping.
  - If 100 % of the source elements are used in mapping, then all of the source data is mapped onto the target mesh. This decreases the likelihood of a significant difference between the sum on the source and target.
  - If less than 100 % of the source elements are used in mapping, then only part of the source data is mapped onto the target mesh. This increases the likelihood of a significant difference between the sum on the source and target.
- The percentage of the source's **Elements Used in Mapping**.
  - Only these source elements will send data to the target mesh. [Geometry and mesh features](#) may cause less than 100 % of the source elements to be used in mapping.
- The percentage of elements receiving **Zero Value** on the target.
  - These elements in the target mesh do not overlap with the source mesh, which may be caused by [geometry and mesh features](#). These elements are not mapped to an element on the source side, and so receive a value of 0 [N].
  - If a large percentage of the target elements receive a value of 0, issues with the [gap size and gap tolerance](#) settings in the mapping algorithm may be preventing the data transfer.

To improve the accuracy of the data transfer, ensure that [geometry and mesh features](#) are not affecting mapping. You can also [control the mapping algorithm](#) used to transfer force data.

An example of the **Mapping Diagnostics** table:

=====	
	PhysicsCoupling 1: Force

Conservative Mapping Diagnostics		
Source		
Force.x Sum [N] :	4.205216E-02	
Force.y Sum [N] :	-8.647712E-06	
Force.z Sum [N] :	-4.431034E-02	
Elements Used in Mapping [%] :	86	
Target		
Force.x Sum [N] :	4.205215E-02	
Force.y Sum [N] :	-8.586374E-06	
Force.z Sum [N] :	-4.431029E-02	
Elements Mapped Using		
Surface Element Projection [%] :	78	
Zero Value [%] :	22	

## 4.8.9.3.2. Assessing the Accuracy of the Temperature Transfer

Once the physics solution is solved, use the Profile Preserving Mapping Diagnostics table to assess the accuracy of the profile-preserving [mapping](#) that is used to [transfer temperature](#) on each physics coupling interface. Information from the most recent solution process is at the bottom of the [physics coupling transcript](#).

For more information, see:

- [Transcript Output for Temperature Transfers](#)
- [Interpreting the Transcript for Temperature Transfers](#)

### 4.8.9.3.2.1. Transcript Output for Temperature Transfers

The transcript for the temperature transfer includes diagnostic information for both the source and the target.

The **Source** side of the interface has the following diagnostic information:

- The average of the data, for example **Temperature Average [K]**, is used to assess the accuracy of the data transfer (described below). The average of the temperature on the source mesh is the average of the data on the entire source mesh, including any nodes that do not send data to the target mesh.
- **Nodes Used in Mapping [%]** is the percentage of the source mesh's nodes that is matched with nodes on the target mesh.

The **Target** side of the interface has the following diagnostic information:

- The average of the data, for example **Temperature Average [K]**, is the average of the data on the entire target mesh. This value is used to assess the accuracy of a profile-preserving data transfer (described below).
- **Nodes Mapped Using**
  - **Radial Basis Function [%]** is the percentage of target nodes mapped to one or more source nodes using the [radial basis function algorithm](#). If the [mapping controls](#) are set to radial basis function, then as many nodes as possible are mapped with this algorithm, and the remaining nodes are mapped with the nearest node algorithm. [Geometry and mesh features](#) cause nodes to be mapped with the nearest node algorithm.
  - **Nearest Node [%]** is the percentage of target nodes mapped to one or more source node using the [nearest node algorithm](#). If the [mapping controls](#) are set to nearest node, then all of the target nodes are mapped with this algorithm.

## 4.8.9.3.2.2. Interpreting the Transcript for Temperature Transfers

To assess the accuracy of this data transfer, review:

- The **Temperature Average** on the source and target.
  - A significant difference between the average on the source and target may mean that the data is not being transferred accurately. A difference of 1 or 2 percent may be significant, but this range will depend on your specific case, especially if less than 100% of the source nodes are used in mapping.
- The percentage of the source's **Nodes Used in Mapping**.
  - Only these source nodes will send data to the target mesh. [Geometry and mesh features](#) may cause less than 100 % of the source nodes to be used in mapping.
- The nodes mapped using the **Nearest Node** algorithm.
  - If the [mapping controls](#) are set to radial basis function, then nodes mapped using the nearest node algorithm are nodes that were poorly mapped, most likely due to [geometry and mesh features](#).

To improve the accuracy of the data transfer, ensure that [geometry and mesh features](#) are not affecting mapping. You can also [control the mapping algorithm](#) used to transfer temperature data.

An example of the **Mapping Diagnostics** table:

PhysicsCoupling 1: Temperature Profile Preserving Mapping Diagnostics	
Source	
Temperature Average [K] :	6.300000E+01
Nodes Used in Mapping [%] :	74
Target	
Temperature Average [K] :	6.300000E+01
Nodes Mapped Using	
Radial Basis Function [%] :	90
Nearest Node [%] :	10

## 4.8.9.3.3. Assessing the Accuracy of the Heat Rate Transfer

Once the physics solution is solved, use the **Mapping Diagnostics** table to assess the accuracy of the conservative mapping that is used to [transfer heat rate](#) on the physics coupling interface. For heat transfers, the [volume element intersection](#) algorithm provides the most accurate mapping. The [scaled nearest element](#) algorithm, which is sensitive to non-intersecting volumes in either the source or target geometries, may be used when speed is a higher priority than accuracy. The table from the most recent solution process is near the bottom of the [physics coupling transcript](#).

For more information, see:

- [Transcript Output for Heat Rate Transfers](#)
- [Interpreting the Transcript for Heat Rate Transfers](#)

### 4.8.9.3.3.1. Transcript Output for Heat Rate Transfers

**Note:** Notation of “VEI” or “SNE” indicates that the diagnostic item is applicable to only the volume element intersection algorithm or only the scaled nearest element algorithm, respectively.

The transcript for the heat rate transfer includes diagnostic information for both the source and the target.

The **Source** side of the interface has the following diagnostic information:

- **Heat Rate Sum [W]** is the sum of the data on the entire source mesh, including any elements that are not used in mapping.
- **Elements Used in Mapping [%]** is the percentage of the source mesh’s elements that is matched with elements on the target mesh. These matched elements will contribute to the profile of the target data, but the scale factor ensures that [all source elements contribute to the data transferred](#) to the target mesh.

The **Target** side of the interface has the following diagnostic information:

- **Heat Rate Sum [W]** is the sum of the transferred data on the entire target mesh.
  - (VEI) When the source geometry is larger than the target geometry, data is transferred only from source elements intersecting a target element, so this value will always be smaller than the source’s **Heat Rate Sum [W]**.
  - (SNE) Due to the scale factor, this value will always be equal to the source’s **Heat Rate Sum [W]**.
- (SNE) **Scale Factor for Conservation** is the scale factor used to ensure that the target’s heat rate sum is equal to the source’s heat rate sum. The scale factor is the ratio of the sum of source data to sum of the unscaled target data.
- **Elements Mapped Using**
  - (SNE) **Scaled Nearest Element [%]** is the percentage of target elements mapped to the source elements. This percentage will always be 100, because [all target elements receive data from the source](#).
  - (VEI) **Volume Element Intersection [%]** is the percentage of target elements mapped to one or more source elements, and which receives data from those source elements. Ideally, the percentage of the target mesh using the [volume element intersection](#) algorithm will be close to 100.
  - (VEI) **Zero Value [%]** is the percentage of elements in the target mesh that do not intersect with the source mesh, and so receive no heat (i.e. a value of 0 [W]).

### 4.8.9.3.3.2. Interpreting the Transcript for Heat Rate Transfers

**Note:** Notation of “VEI” or “SNE” indicates that the diagnostic item is applicable to only the volume element intersection algorithm or only the scaled nearest element algorithm, respectively.

To assess the accuracy of the heat rate data transfer, review:

- (SNE) The **Scale Factor for Conservation**.
  - A scale factor far from 1 indicates that there were issues with the mapping. [Geometric non-intersection and uneven meshes](#) may be causing problems.
  - A scale factor equal or close to 1 may mean that mapping is accurate. Be careful, because issues caused by geometric non-intersection and uneven meshes [may still exist](#).
- The percentage of the source’s **Elements Used in Mapping**.

- [Geometry and mesh features](#) may cause less than 100 % of the source elements to be used in mapping.
- (SNE) If this percentage is less than 100, source elements exist that are not matched to the target mesh but are still contributing to the data transfer through the scale factor.

- The **Heat Rate Sum**.

- Ensure that the sum is in the expected range for your case.
- (SNE) The scale factor causes the heat rate sum on the source and target to be identical, and so comparing these values does not give any information about the mapping quality.
- (VEI) The more similar the heat rate sum on the source and target, the more accurate the mapping. These values should be the same unless there are issues because of non-intersecting elements.
- (VEI) The percentage of elements receiving **Zero Value** on the target. A high percentage of unmapped elements indicates poor mapping quality due to lack of intersection between source and target. This may be caused by [geometry and mesh features](#).

To improve the accuracy of the data transfer, ensure that [geometry and mesh features](#) are not affecting mapping. You can also [control the mapping algorithm](#) used to transfer heat rate data.

An example of the **Mapping Diagnostics** table for the volume element intersection algorithm:

Physics Coupling 1: Heat Rate Conservative Mapping Diagnostics		
Source		
Heat Rate Sum [W] :		8.620690E-01
Elements Used In Mapping [%] :		100
Target		
Heat Rate Sum [W] :		8.620690E-01
Elements Mapped Using		
Volume Element Intersection [%] :		97
Zero Value [%] :		3

An example of the **Mapping Diagnostics** table for the scaled nearest element algorithm:

Physics Coupling 1: Heat Rate Conservative Mapping Diagnostics		
Source		
Heat Rate Sum [W] :		1.737356E+00
Elements Used In Mapping [%] :		39
Target		
Heat Rate Sum [W] :		1.737356E+00
Scale Factor For Conservation :		0.626728987728
Elements Mapped Using		
Scaled Nearest Element [%] :		100

## 4.8.9.4. Mapping

Mapping is the process of using data on a source mesh to calculate data on a target mesh. Mapping takes place on each [physics coupling interface](#) during the update of the Physics Solution task that is receiving data.

There are two types of mapping:

- [Conservative mapping](#) and
- [Profile-preserving mapping](#).

To ensure mapping quality, you can address any [geometry- or mesh-based issues](#) that may exist.

After the mapping takes place, you can use the mapping diagnostics in the [physics coupling transcript](#) to assess:

- the [accuracy of the force data transfer](#),
- the [accuracy of the temperature data transfer](#), or
- the [accuracy of the heat rate transfer](#).

## 4.8.9.4.1. Addressing Geometry- and Mesh-Based Mapping Issues

When a simulation uses the [physics coupling interface](#) to transfer data from one physics task to another, geometry and mesh features have an impact on the quality of the [mapping](#) on that interface.

To ensure the accuracy of the mapping and resulting data transfer on the interface:

- Ensure that the source and target geometry overlap or intersect for the regions where you want data transferred. Non-overlapping or non-intersecting geometry can be used for physics coupling, but if the bodies or faces being mapped don't overlap or intersect with each other, the mapping will be less accurate.

If mapping issues occur because of dissimilarities between the source and target geometries and/or meshes, you can [define a selection set](#) for the source location instead of using the default

`AllCouplingSourceFaces()` or `AllCouplingSourceBodies()` expressions, which select all of the source surfaces or bodies capable of transferring the reported type of data. This will facilitate a comparison of the geometries and meshes, enabling you to adjust them to improve mapping quality.

- Ensure that the difference between the [gap size and overall gap tolerance](#) does not prevent the transfer of data. If the overall gap tolerance is smaller than the gap between the source and target locations, then data will not be transferred.
- Refine the meshes on the source and target surfaces or bodies. Coarse meshes on curved surfaces can create gaps or dissimilarities between the two meshes, which affect the mapping between the meshes.
- Ensure the local resolution of the meshes on the source and target of the physics coupling interface are as similar as possible. If the target mesh is too coarse, solution features that are resolved on the source mesh may be lost in the data transfer to the target mesh.

The mapping diagnostics in the [physics coupling transcript](#) will help you determine if geometry or mesh features may be affecting the quality of the mapping on the physics coupling interface.

## 4.8.9.4.2. Conservative Mapping Algorithms

Conservative mapping is used for conservative variables such as force and heat rate. The goal of the conservative mapping is to minimize the difference between the sum of the data (locally and globally) on the source and target meshes.

You can use the [conservative mapping controls](#) to set the conservative mapping algorithm used.

Conservative mapping controls include:

- the [surface element projection algorithm](#) used for force transfers, and
- the [volume element intersection](#) and [scaled nearest element](#) algorithms used for heat rate transfers.

## 4.8.9.4.2.1. Surface Element Projection Mapping Algorithm

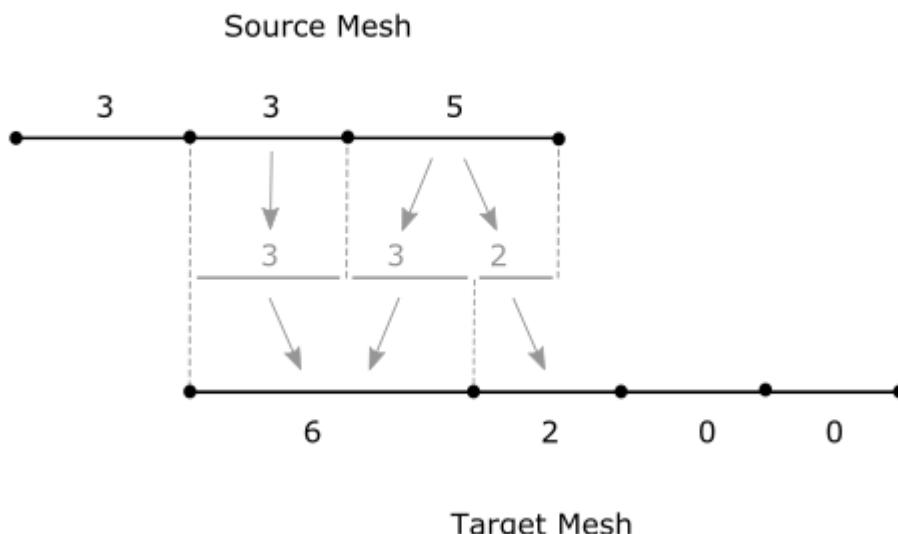
The surface element projection algorithm is used for conservative mapping in force data transfers. Data is scattered from the source side and collected on the target side of the interface. Mapping weights, derived from the area fractions of the source projected on the target mesh elements, are used to interpolate the data.

The [mapping diagnostics](#) have information about the mapping on each interface. [Geometry and mesh features](#) may affect the surface element projection algorithm's mapping.

- Ensure that the source and target geometry overlaps for the areas where you want data transferred and refine the meshes on the source and target surfaces.

If there are gaps between the meshes, such as those created by non-overlapping geometry or coarse meshes on curved surfaces, source and target elements may not be matched. Source elements that are matched to a target element are the only elements to transfer data. Target elements that are not matched to a source element mesh are assigned a value of zero. The percentage of the target mesh that is given a zero value is listed in the [mapping diagnostics](#).

The image below is a simplified example of data transferred using the surface element projection algorithm (mapping actually involves 3D meshes). In this example, areas of the mesh that don't overlap affect the quality of the mapping between the two meshes. The numbers on the source and target mesh are the data value on each element. The lines between the meshes represent the area fractions used to interpolate the data, and the numbers on these area fractions are the data being transferred. The arrows are the data transfers from the source to the target mesh.



The mapping diagnostics table below shows the information that would be reported in the transcript for this case. In the diagnostics, the difference in the sum on the source and target is due to the areas of the mesh that don't overlap.

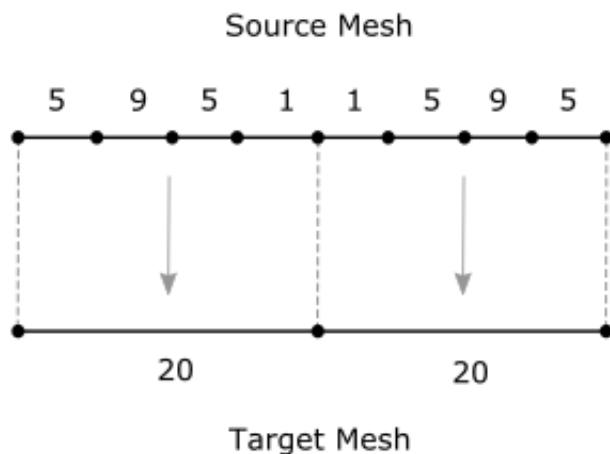
PhysicsCoupling 1: Force	
Conservative Mapping Diagnostics	
Source	

Force.x Sum [N] :	1.10E+01
Force.y Sum [N] :	0.00E+00
Force.z Sum [N] :	0.00E+00
Elements Used in Mapping [%] :	67
Target	
Force.x Sum [N] :	8.00E+00
Force.y Sum [N] :	0.00E+00
Force.z Sum [N] :	0.00E+00
Elements Mapped Using	
Surface Element Projection [%] :	50
Zero Value [%] :	50
=====	

- Ensure the local resolution of the meshes on the source and target of the physics coupling interface is as similar as possible.

If the target mesh is coarser than the source mesh, then multiple source elements may be mapped onto a larger target element. Details of the data distribution may be lost in the data transfer if the data on these areas of the source mesh isn't uniform.

The image below is a simplified example of data transferred using the surface element projection algorithm (mapping actually involves 3D meshes). In this example, a coarse target mesh affects the quality of the mapping between the two meshes. The numbers are the data value on each element, and the arrows are data transfers from the source to the target mesh. The detail of the interpolation using area fractions is not shown in this image.



In this case, the mapping diagnostics table would contain acceptable values, showing that data is being conserved. However, without similar mesh resolutions, detail of the data distribution can be lost if the target mesh resolution is coarser than the source mesh resolution. Keep in mind that this issue cannot be identified from a review of the transcript output.

- Ensure that the [absolute gap tolerance](#) is set appropriately.

The overall gap tolerance, defined as the relative gap tolerance plus the absolute gap tolerance, determines the locations that are mapped. The relative gap tolerance is calculated automatically by AIM and cannot be modified, but you can change the absolute gap tolerance using the physics coupling interface [mapping controls](#). If the overall gap tolerance is smaller than the gap between the source and target locations, then the source and target will not be mapped to one another and data will not be transferred.

For faces, the gap is the distance between the plane (as defined by its centroid and normal vector of the source face element) and the centroid of the target face element. For shells, the gap is the distance between the source mesh and the centroid of the target shell element.

The [mapping diagnostics](#) show the percentage of target elements that are not matched to a source and so are assigned a value of zero. A high percentage of mesh elements with zero value may indicate issues with the gap size and its effect on mapping.

## 4.8.9.4.2.2. Volume Element Intersection Mapping Algorithm

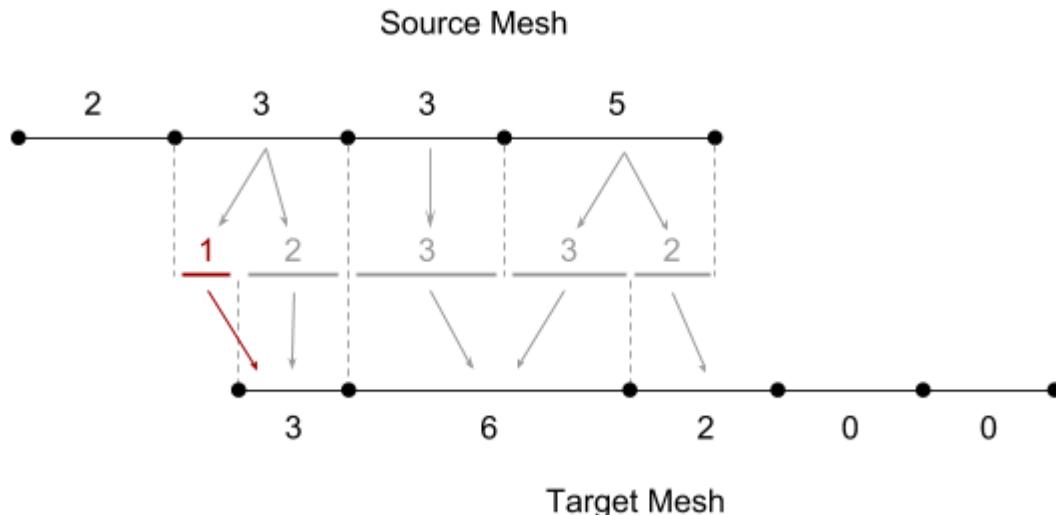
The volume element intersection mapping algorithm is used for conservative mapping in heat rate data transfers. Data is scattered from the source side and collected on the target side of the interface. Mapping weights, derived from the volume fractions of the source projected on the target mesh elements, are used to interpolate the data.

The [mapping diagnostics](#) have information about the mapping on each interface. [Geometry and mesh features](#) may affect the volume element intersection algorithm's mapping.

- Ensure that the elements in the source and target geometry intersect where you want data transferred and refine the meshes on the source and target surfaces.

If there are dissimilarities between the meshes, such as those created by non-intersecting geometry or coarse meshes on curved surfaces, source and target elements may not be matched. Source elements that are matched to a target element are the only elements to transfer data. Target elements that are not matched to a source element are assigned a value of zero. The percentage of the target mesh that is given a zero value is listed in the [mapping diagnostics](#).

The image below is a simplified example of data transferred using the volume element intersection algorithm (mapping actually involves 3D meshes). In this example, non-intersecting mesh elements affect the quality of the mapping between the two meshes. The numbers on the source and target mesh are the data value on each element. The lines between the meshes represent the volume fractions used to interpolate the data, and the numbers on these volume fractions are the data being transferred. The arrows are the data transfers from the source to the target mesh.



In the image above, note that target elements which do not intersect any source elements are assigned a value of zero. Source elements which do not intersect any target elements are excluded from the mapping.

However, source elements that partially intersect target elements are completely included in the mapping (shown in red).

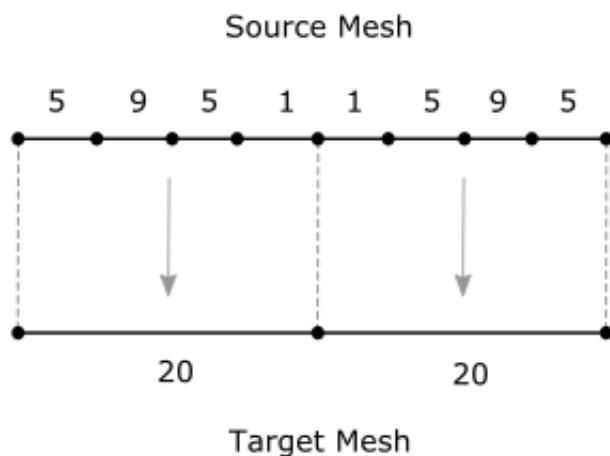
The mapping diagnostics table below shows the information that would be reported in the transcript for this case. In the diagnostics, the difference in the sum on the source and target is due to the elements of the mesh that don't intersect. A high percentage of mesh elements with a zero value may indicate that [geometry-or meshed-based issues](#) are affecting the mapping quality.

Physics Coupling 1: Heat Rate Conservative Mapping Diagnostics	
Source	
Heat Rate Sum [W] :	1.300000E+00
Elements Used In Mapping [%] :	75
Target	
Heat Rate Sum [W] :	1.100000E+00
Elements Mapped Using	
Volume Element Intersection [%] :	65
Zero Value [%] :	35

- Ensure the local resolution of the meshes on the source and target of the physics coupling interface is as similar as possible.

If the target mesh is coarser than the source mesh, then multiple source elements may be mapped onto a larger target element. Details of the data distribution may be lost in the data transfer if the data on these regions of the source mesh isn't uniform.

The image below is a simplified example of data transferred using the volume element intersection algorithm (mapping actually involves 3D meshes). In this example, a coarse target mesh affects the quality of the mapping between the two meshes. The numbers are the data value on each element, and the arrows are data transfers from the source to the target mesh. The detail of the interpolation using volume fractions is not shown in this image.



In this case, the mapping diagnostics table would contain acceptable values, showing that data is being conserved. However, without similar mesh resolutions, detail of the data distribution can be lost if the target mesh resolution is coarser than the source mesh resolution. Keep in mind that this issue cannot be identified from a review of the transcript output.

### 4.8.9.4.2.3. Scaled Nearest Element Mapping Algorithm

The scaled nearest element mapping algorithm is used for [conservative mapping](#) in heat rate data transfers. The algorithm ensures global conservation of data on the source and target, while capturing local data profiles. See the steps of the algorithm below.

Take care with the geometries selected for the source and target location, as this algorithm is sensitive to non-intersecting volumes on either the source or target locations.

**Note:** For cases where mapping will proceed with substantial regions of non-intersection between source and target, consider using the [volume element intersection](#) algorithm instead.

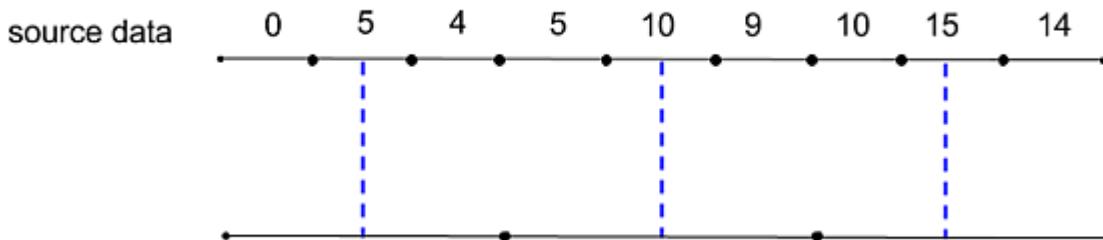
To ensure that geometry and mesh features are not affecting the mapping, review the following cases to understand how these features behave with the scaled nearest element algorithm:

- [Cases where the scale factor is not 1](#), and
- [Cases where the scale factor is equal to 1](#), but there are still issues with the quality of the mapping.

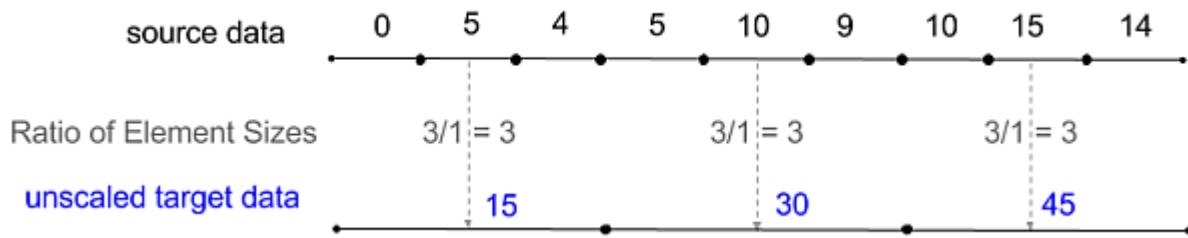
Steps of the scaled nearest element algorithm:

**Note:** The images below are simplified examples of data transferred using the scaled nearest element algorithm. Mapping actually involves 3D meshes.

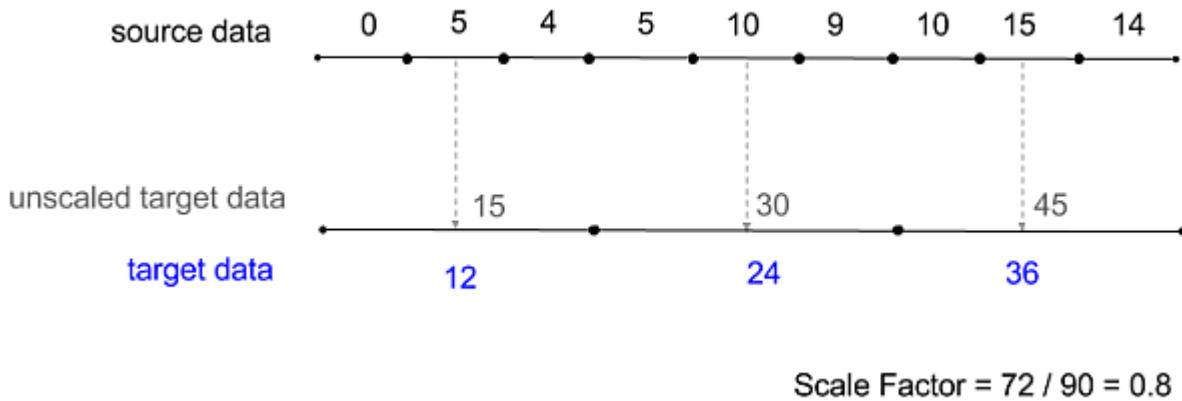
1. Each element on the target mesh is matched to the nearest element on the source mesh, determined by the distances between the elements' centroids.



2. The data from the source element is transferred to the target element using the ratio of the target element size to the source element size. This ensures that local data profiles are captured.



3. The data on the entire target mesh is then scaled to ensure that the total sum equals the sum over all the source elements. To do this, the scale factor is used, which is the ratio of the sum of source data to sum of the unscaled target data.



#### 4.8.9.4.2.3.1. Scale Factor not 1 Caused by Geometric Non-Intersection and Uneven Meshes

If the scale factor is not 1, this is an indication that geometry and/or mesh features are affecting the quality of the mapping.

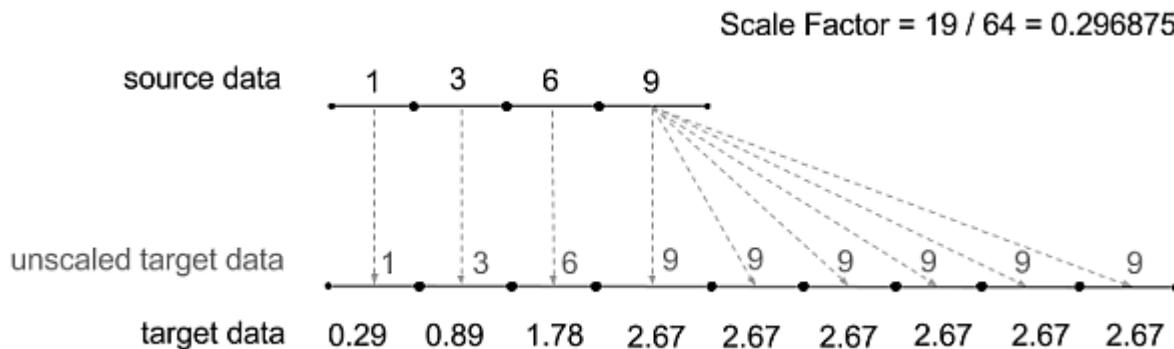
When mapping with the [scaled nearest element algorithm](#) (used for heat rate transfers), ensure that:

1. The source and target geometry intersect for the locations where you want data transferred. This algorithm is sensitive to non-intersection of either the source or target mesh. Review **Case 1** and **Case 2** below.
2. The local resolution of the meshes on the source and target of the physics coupling interface are as similar as possible. Review **Case 3** below.

Note that the images below are simplified examples of data transferred using the scaled nearest element algorithm. Mapping actually involves 3D meshes.

##### **Case 1: Geometric Non-Intersection of the Target**

When part of the target mesh doesn't intersect with the source mesh, all of the target elements are still matched to the nearest source elements, and some target elements may be matched to nearest source elements that are physically far away.



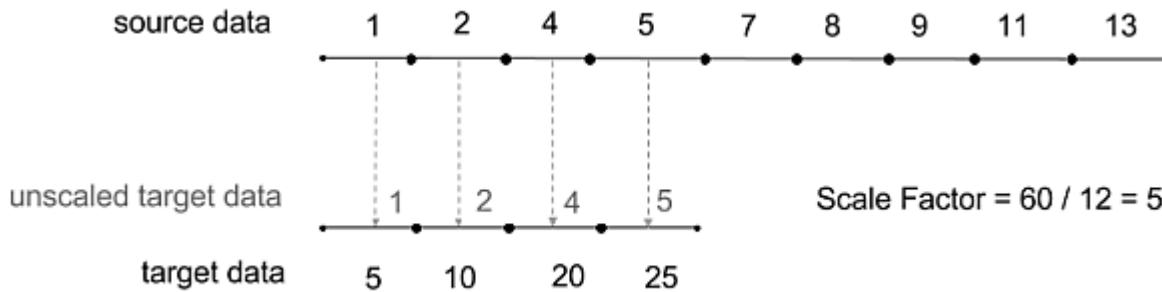
**Figure 1:** In this example, sections of the target mesh that don't intersect with the source mesh affect the quality of the mapping between the two meshes.

The mapping diagnostics table below shows the information that would be reported in the transcript for the case in Figure 1. In the diagnostics, the scale factor is far from unity, and is an indication that the mapping is affected.

Physics Coupling 1: Heat Rate	
Conservative Mapping Diagnostics	
<b>Source</b>	
Heat Rate Sum [W] :	6.400000E+01
Elements Used In Mapping [%] :	100
<b>Target</b>	
Heat Rate Sum [W] :	6.400000E+01
Scale Factor For Conservation :	0.2968750000
Elements Mapped Using	
Scaled Nearest Element [%] :	100

### Case 2: Geometric Non-Intersection of the Source

When the source mesh doesn't intersect with the target mesh, the data on the entire source mesh is still used to calculate the scale factor, and so the entire source mesh influences the data received on the target mesh.



**Figure 2:** In this example, sections of the source mesh that don't intersect with the target mesh influence the data on the target due to the scale factor.

The mapping diagnostics table below shows the information that would be reported in the transcript for the case in Figure 2. In the diagnostics, the scale factor is not 1 due to the non-intersecting meshes.

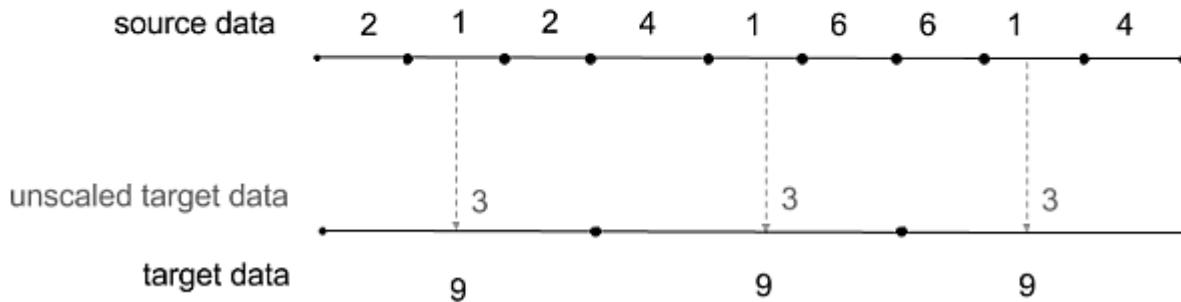
Physics Coupling 1: Heat Rate	
Conservative Mapping Diagnostics	
<b>Source</b>	
Heat Rate Sum [W] :	6.000000E+01
Elements Used In Mapping [%] :	44
<b>Target</b>	
Heat Rate Sum [W] :	6.000000E+01
Scale Factor For Conservation :	5.0000000000
Elements Mapped Using	
Scaled Nearest Element [%] :	100

### Case 3: Coarse Target Mesh

Ensure that the local resolution of the meshes on the source and target of the physics coupling interface are as similar as possible. If the target mesh is coarser than the source mesh, there will be source elements that

are not matched to any target elements. Details may be lost in the data transfer if the data on these sections of the source mesh is not uniform.

$$\text{Scale Factor} = 27 / 9 = 3$$



**Figure 3:** In this example, a coarse target mesh affects the quality of the mapping between the two meshes.

The mapping diagnostics table below shows the information that would be reported in the transcript for the case in Figure 3. In the diagnostics, the scale factor is not 1 due to a data profile on the source that is not captured on the coarser target mesh.

Physics Coupling 1: Heat Rate Conservative Mapping Diagnostics		
Source		
Heat Rate Sum [W] :		2.700000E+01
Elements Used In Mapping [%] :		33
Target		
Heat Rate Sum [W] :		2.700000E+00
Scale Factor For Conservation :		3.0000000000
Elements Mapped Using		
Scaled Nearest Element [%] :		100

#### 4.8.9.4.2.3.2. Scale Factor Equal to 1 with Poor Quality of Mapping

If the scale factor is equal to or close to 1, this could indicate that the data transfer is accurate. However, it is possible that issues still exist and that geometric and mesh features are interacting to produce a scale factor close to 1.

When mapping with the [scaled nearest element algorithm](#) (used for heat rate transfers), ensure that:

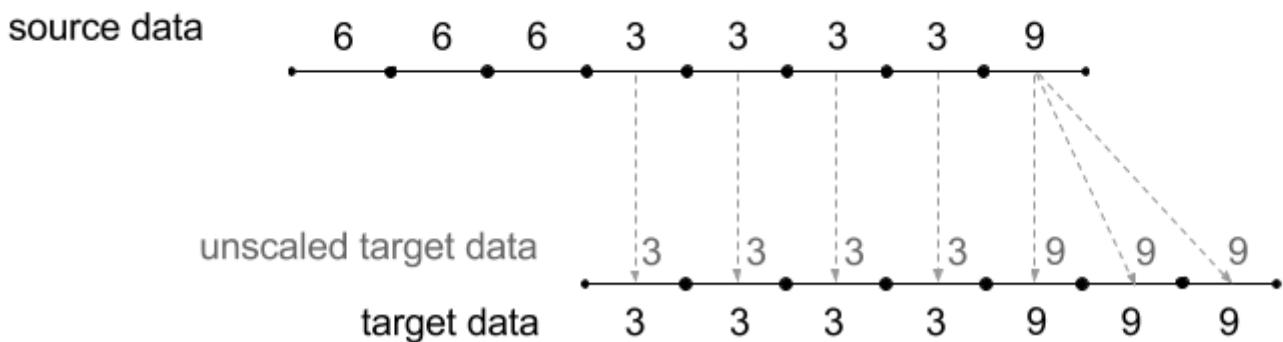
- The source and target geometry intersect for the locations where you want data transferred. This algorithm is sensitive to non-intersection of either the source or target mesh. Review **Case 1** below.
- The local resolution of the meshes on the source and target of the physics coupling interface are as similar as possible.

Note that the image below is a simplified example of data transferred using the scaled nearest element algorithm. Mapping actually involves 3D meshes.

#### Case 1: Geometric Non-Intersection with Scale Factor of 1

The non-intersecting target mesh receives data from the nearest source element. The non-intersecting source mesh influences the data received on the target mesh through the scale factor.

Scale Factor = 39 / 39



**Figure 1:** In this example, geometric non-intersection affects the quality of the mapping between the two meshes. The combination of source and target non-intersection produces a scale factor of 1.

The mapping diagnostics table below shows the information that would be reported in the transcript for the case in Figure 1. In the diagnostics, the scale factor is 1 even though there are non-intersecting meshes.

Physics Coupling 1: Heat Rate Conservative Mapping Diagnostics	
Source	
Heat Rate Sum [W] :	3.900000E+01
Elements Used In Mapping [%] :	63
Target	
Heat Rate Sum [W] :	3.900000E+01
Scale Factor For Conservation :	1.0000000000
Elements Mapped Using	
Scaled Nearest Element [%] :	100

### 4.8.9.4.3. Profile-Preserving Mapping Algorithms

Profile-preserving mapping is used for non-conservative quantities such as temperature. The goal of this mapping is to minimize the difference between the profile, or distribution of data, on the source and target meshes.

You can use the [profile mapping controls](#) set the profile-preserving algorithm used.

- Profile-preserving mapping is used for non-conservative quantities such as temperature. The goal of this mapping is to minimize the difference between the profile, or distribution of data, on the source and target meshes. which include:
  - the [nearest node algorithm](#) and
  - the [radial basis function algorithm](#).

#### 4.8.9.4.3.1. Nearest Node Mapping Algorithm

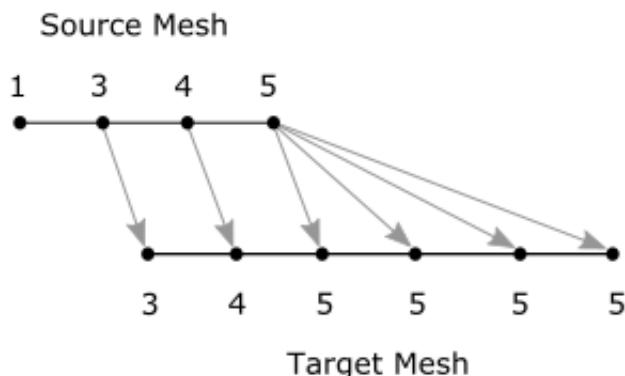
The nearest node algorithm is used in [profile-preserving mapping](#), where data profiles on the source side are used as bases to calculate data on the target side of the interface. Each node on the target mesh is matched to the nearest node on the source mesh. The target node is given the value of the data on this matched source node.

The [mapping diagnostics](#) have information about the mapping on each interface. Geometry and mesh features may affect the nearest node algorithm's mapping.

- Ensure that the source and target geometry overlaps for the areas where you want data transferred.

When the source and target meshes don't overlap, all of the target nodes are still matched to the nearest source nodes. Some target nodes may be matched to nearest source nodes that are physically far away.

The image below is a simplified example of data transferred using the nearest node algorithm (mapping actually involves 3D meshes). In this example, areas of the mesh that don't overlap affect the quality of the mapping between the two meshes. The numbers are the data value on each node, and the arrows are data transfers from the source to the target mesh.



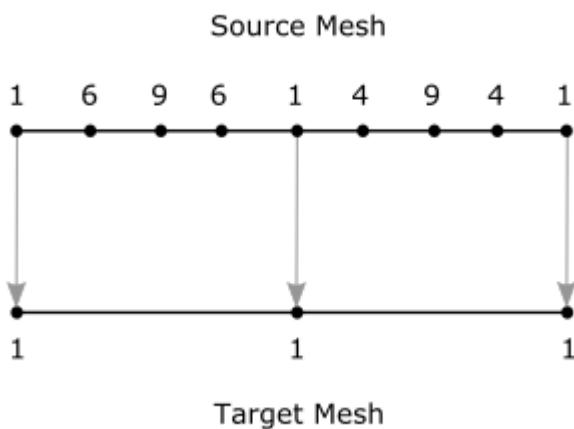
The mapping diagnostics table below shows the information that would be reported in the transcript for this case. In the diagnostics, the difference in the average on the source and target is due to the areas of the mesh that don't overlap.

PhysicsCoupling 1: Temperature Profile Preserving Mapping Diagnostics		
Source		
Temperature Average [K] :	3.25E+00	
Nodes Used in Mapping [%] :	75	
Target		
Temperature Average [K] :	4.50E+00	
Nodes Mapped Using		
Nearest Node [%] :	100	

- Ensure the local resolution of the meshes on the source and target of the physics coupling interface are as similar as possible.

If the target mesh is coarser than the source mesh, there will be source nodes that are not matched to any target nodes. Details may be lost in the data transfer if the data on these areas of the source mesh is not uniform.

The image below is a simplified example of data transferred using the nearest node algorithm (mapping actually involves 3D meshes). In this example, a coarse target mesh affects the quality of the mapping between the two meshes. The numbers are the data value on each node, and the arrows are data transfers from the source to the target mesh.



The mapping diagnostics table below shows the information that would be reported in the transcript for this case. In the diagnostics, the difference in the average on the source and target is due to data from the source that is not transferred to the target.

PhysicsCoupling 1: Temperature Profile Preserving Mapping Diagnostics	
Source	
Temperature Average [K] :	4.55E+00
Nodes Used in Mapping [%] :	33
Target	
Temperature Average [K] :	1.00E+00
Nodes Mapped Using	
Nearest Node [%] :	100

You can use the [mapping controls](#) to change the algorithm to [radial basis function](#) for increased accuracy.

#### 4.8.9.4.3.2. Radial Basis Function Mapping Algorithm

The radial basis function algorithm is used in [profile-preserving mapping](#), where data profiles on the source side are used as bases to calculate data on the target side of the interface. Multiple source nodes can contribute to the data transferred to a target node, so the radial basis function algorithm is more accurate than the [nearest node algorithm](#).

With this algorithm, each node on the target mesh is matched to a source element and then to the source nodes that define that element. A function and the distances between the nodes determine how these matched source nodes contribute to the data that is transferred to the target node. In the [mapping controls](#), there are two options available for this function: thin plate spline (default and recommended), and Gaussian (which involves a shape parameter that can be modified in the mapping controls).

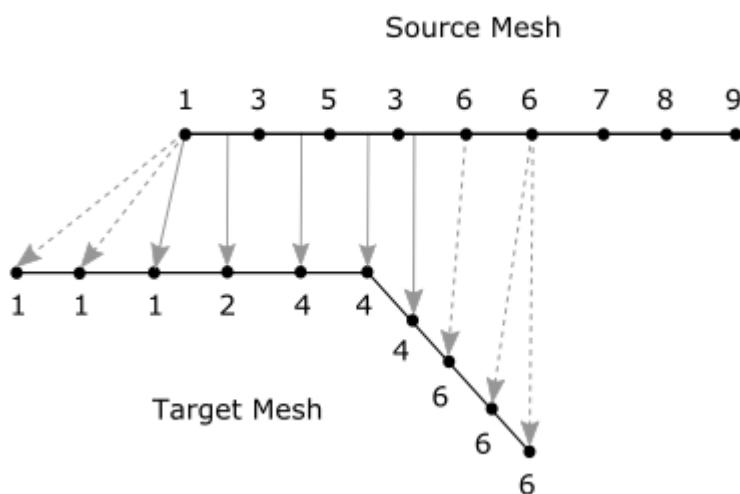
In a profile-preserving data transfer, all target nodes are mapped. Due to tolerances set for this algorithm, target nodes may not all be mapped with the radial basis function algorithm. If the nodes on the target mesh are not matched to source nodes using this algorithm, then that target node is mapped using the [nearest node algorithm](#). The percentages of targets nodes mapped using each algorithm are reported in the [mapping diagnostics](#).

The [mapping diagnostics](#) have information about the mapping on each interface. Geometry and mesh features may affect the radial basis function algorithm's mapping.

- Ensure that the source and target geometry overlaps for the areas where you want data transferred and refine the meshes on the source and target surfaces.

The radial basis function's tolerance that may be exceeded if the source and target meshes don't overlap, or if there are gaps between the meshes (for example, gaps due to coarse meshes on curved surfaces). When the tolerance is exceeded, the [nearest node algorithm](#) will be used rather than the radial basis function to map those sections of the mesh. Review the [mapping diagnostics](#) to see if a percentage of the mesh was mapped using the nearest node algorithm.

The image below is a simplified example of data transferred using the radial basis function algorithm (mapping actually involves 3D meshes). In this example, non-overlapping meshes and gaps between the meshes affect the quality of the mapping between the two meshes. The numbers are the data value on each node, and the arrows are data transfers from the source to the target mesh. Solid arrows are data transfers using the radial basis function algorithm. Dashed arrows are data transfers using the nearest node algorithm due to tolerances being exceeded.



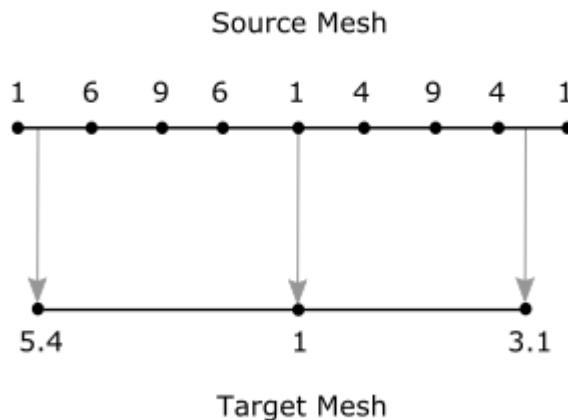
The mapping diagnostics table below shows the information that would be reported in the transcript for this case. In the diagnostics, the difference in the average on the source and target is due to the exceeded tolerances caused by gaps between the meshes and areas of the mesh that don't overlap.

PhysicsCoupling 1: Temperature Profile Preserving Mapping Diagnostics		
Source		
Temperature Average [K] :		5.33E+00
Nodes Used in Mapping [%] :		66
Target		
Temperature Average [K] :		3.50E+00
Nodes Mapped Using		
Radial Basis Function [%] :		50
Nearest Node [%] :		50

- Ensure the local resolution of the meshes on the source and target of the physics coupling interface are as similar as possible.

If the target mesh is coarser than the source mesh, there will be source nodes that are not matched to any target nodes. Detail may be lost in the data transfer if the data on these areas of the source mesh is not uniform.

The image below is a simplified example of data transferred using the radial basis function algorithm (mapping actually involves 3D meshes). In this example, a coarse target mesh affects the quality of the mapping between the two meshes. The numbers are the data value on each node, and the arrows are data transfers from the source to the target mesh.



The mapping diagnostics table below shows the information that would be reported in the transcript for this case. In the diagnostics, the difference in the average on the source and target is due to data from the source that is not transferred to the target.

PhysicsCoupling 1: Temperature Profile Preserving Mapping Diagnostics	
Source	
Temperature Average [K] :	4.55E+00
Nodes Used in Mapping [%] :	56
Target	
Temperature Average [K] :	3.16E+00
Nodes Mapped Using	
Radial Basis Function [%] :	100

[Nearest node](#) is the default algorithm used for profile-preserving mapping. The radial basis function is chosen using the [mapping controls](#).

## 4.9. Solver Options

You can control the solution of the physics simulation for the following physics types:

- [Fluid flow](#) (steady-state and time-dependent)
- [Polymer extrusion](#)
- [Polymer blow molding](#)
- [Structural](#) (static and modal)
- [Thermal](#) (steady-state and time-dependent)

- Electric conduction
- Electromagnetic
- Electrostatic

With the exception of electromagnetic, the physics types listed above support the use of fidelity refinement to identify localized geometric details of a specific size that you want to capture in the solution. This enables you to increase the accuracy of the solution while optimizing solution time by setting the [global solution fidelity](#) to a lower setting, and then applying [fidelity refinement](#) to specific topologies only.

#### 4.9.1. Fluids Solver Options

The solver settings control various details of the discretization and solution control. The default settings have been selected to provide good accuracy and robustness for most problems. However, it may sometimes be appropriate to fine-tune the following settings:

- Discretization details (affect the converged solution):
  - [Gradient method](#)
  - [Advection scheme](#)
  - [Pressure scheme](#)
  - [Temporal discretization](#) (time-dependent calculations)
- Solution controls (affect convergence behavior but not the converged solution):
  - [Coupling control](#) (for example, [pressure-velocity coupling](#))
  - [Advancement control](#) (relaxations applied prior to linear solution; for example, implicit relaxation and pseudo-transient details)
  - [Explicit relaxation](#) (relaxations applied after linear solution)
- Boundary refinement:
  - [Boundary refinement](#) provides an optional input that lets you refine the boundary mesh generated by automatic physics-aware meshing.
- Solution fidelity refinement:
  - [Fidelity refinement](#) provides an optional input that lets you identify localized geometric details of a specific size that you want to capture in the solution. This enables you to increase the accuracy of the solution while optimizing solution time by setting the [global solution fidelity](#) to a lower setting, and then applying fidelity refinement to specific topologies only.

#### 4.9.1.1. Launching the Solver on Multiple Processes

##### Physics > Solver Options > Solver Settings > Launch Controls

Launch controls allow you to control the computational resources used by a simulation when running in parallel. The launch controls for the solver are populated with default settings and values.

To change the default launch control settings:

Increase the **Number of processes** if you want to run on more than 2 processes.



**Solver files directory** displays the path and the organization of files and subdirectories beneath the application directory.

**Note:** You will need additional AIM (or HPC) licenses if specifying more than four processor cores.

## 4.9.1.2. Solving Steady-State Fluid Flow Simulations

### Physics > Solver Options > Solution Controls

In the **Physics** task, the **Calculation type** should be set to **Steady/Static** in order to solve a steady-state fluid flow simulation.

During the solution process you can [monitor convergence](#) by checking residuals. The default settings are a good start and are generally acceptable in most cases.

In the **Solution Controls** panel:

1. Under **Calculation Control** adjust the **Solution advancement exponent** towards conservative (a negative value) or aggressive (a positive value) to control the rate at which the solver converges toward the solution.
2.  Under **Residual Control** you can modify the defaults for the way in which [residuals](#) are calculated and the frequency with which they are reported.
3. Under **Iteration Control**, enter the **Maximum number of iterations** you would like the solver to run through. A default value is set as a starting point. If your solution does not converge, choose a larger number of iterations or modify the convergence criteria.
-  You can also force the solver to perform a **Minimum number of iterations**.
4. Under **Convergence Criteria** you can change the residual tolerance. You can also change it per equation under  **Equation Class Tolerance**. Increasing the tolerance will lead to a smaller number of iterations, but the solution may not be sufficiently converged. Decreasing the tolerance will require a larger number of iterations. [Judging Convergence](#) on page 424 provides tips and guidance on how to obtain better solution convergence.
5. Under **Solver Meshing Control**, make any adjustments as needed to improve solver efficiency. See [Adjusting Solver Meshing Controls](#) on page 413 for more information.
6. In the **Physics** task, click **Solve Physics** () to update and generate a solution.

**Note:** If your solution has reached the **Maximum number of iterations** before reaching convergence, you can:

- **Mark as Partially Up-to-Date** and review your results and monitors to decide if the solution has converged enough. You can also change the solution settings, such as increasing the **Maximum number of iterations**, and solve again to try to reach [convergence](#).
- Accept the partially updated solution if you are satisfied with it by selecting **Mark as Up-to-Date**.

If you [interrupted your solution](#) before reaching the specified **Maximum number of iterations**, and you decide that the solution is sufficient for you to proceed with your simulation, you can accept the partially updated solution by right-clicking on the physics task in the workflow view and selecting **Mark as Up-to-Date**.

## 4.9.1.3. Solving Time-Dependent Fluid Flow Simulations

### Physics > Solver Options > Solution Controls

Time-dependent fluid flow analyses are used to simulate systems where the flow behavior varies over time. Enabling time dependence is sometimes useful when attempting to [solve steady-state](#) problems that tend toward instability (for example, natural convection problems in which the Rayleigh number is close to the

transition region). It is possible in many cases to reach a steady-state solution by integrating the time-dependent equations.

To run a [time-dependent simulation](#):

1. Navigate to the **Physics** panel and select **Time-dependent** as the **Calculation type**.
2.  Choose to enter **Time** or **Time steps** for the **Duration specification**. By default, you will enter the time under **Duration**.

In the **Solution Controls** panel:

1. Under **Calculation Control > Adaptive Time Stepping**, you can set the time step size to adapt based on the transient truncation error. The time-dependent calculation starts using the **Initial Time Step Size**, and the time step size for each step is controlled to keep the truncation error close to a specified tolerance. If the truncation error for a time step exceeds the tolerance, that time step may be repeated with a smaller time step size. The time step controller involves a number of settings, and the default settings are appropriate for most cases.

Option	Description
 <b>Error tolerance</b>	The target normalized truncation error tolerance. A time step may be repeated with a smaller time step size if the truncation error exceeds this value. The default of 0.001 is acceptable for many cases. A smaller value will result in a smaller time step size, higher transient accuracy, and a longer computation time.
 <b>Minimum/Maximum time step factor</b>	Limits the adaptive time step size by the factor times whatever timestep is calculated using the Intial Time Step Size. The maximum time step size cannot be larger than (Maximum time step factor) * (timestep computed by Initial Time Step Size).
 <b>Minimum/Maximum time step size</b>	The upper and lower limits for the time step size.
<b>Number of initial time steps</b>	The number of time steps using the time step size determined by the <b>Initial Time Step Size</b> which are performed before adaptive time stepping begins.
 <b>Minimum/Maximum change factor</b>	Limits the rate at which the time step size may change.

2. Under **Calculation Control > Initial Time Step Size**, select the **Time stepping** method used to track the progress of real time during the simulation. If **Adaptive Time Stepping** is turned off, then settings will be made under **Calculation Control > Time Step Size**.
  - **Automatic** is the default and it chooses a time step size from various relevant physical timescales for the problem, including advection, diffusion, buoyancy, and compressibility. Each physical time scale is estimated by dividing a geometrical length scale by an estimate of the velocity scale relevant for that process. The automatic time step size is then calculated as a fraction (1/50) of the smallest of these physical timescales.
    - **Timestep scaling factor**: Since the automatic timescale calculation is somewhat heuristic, you may adjust it using this scaling factor.
    - **Length scale calculation method**: The length scale calculation used in the automatic time step size calculation can be tricky for the solver to estimate. The default **Conservative** approach sets the length scale to the cube root of the geometry volume. This works well for many situations, particularly internal

flows. In some cases (such as a 2D geometry modeled as a thin 3D geometry) this estimate can give an excessively small time step size, and the **Aggressive** approach (which sets the length scale to the maximum extent) may be better. Alternatively, if you know a relevant length scale (e.g., chord length for an external aerodynamic calculation), use the **User specified** length scale option.

- **Time periodic** allows you to specify the **Period** and the **Time steps per period**, and the solver will use those values to compute the time step size. This option is useful if there is a large-scale transient oscillation of known frequency.
- **User specified** allows you to directly specify the **Time step size**.

3.  Under **Residual Control** you can [scale and/or normalize the residuals](#) as well as specify the monitoring frequency of the residual reporting.
4. Under **Iteration Control**, enter the **Maximum/Minimum iterations per time step** you would like the solver to run through. The maximum number of iterations per time step may not always be reached if the residual target level is achieved first.
5. Under **Convergence Criteria** you can change the residual tolerance. You can also change it per equation under **Equation Class Tolerance**. Increasing the tolerance will lead to a smaller number of iterations per time step, but the solution may not be sufficiently converged. Decreasing the tolerance will require a larger number of iterations per time step.
6. Under **Solver Meshing Control**, make any adjustments as needed to improve solver efficiency. See [Adjusting Solver Meshing Controls](#) on page 413 for more information.
7. In the **Physics** panel, click **Solve Physics** () to update the Physics Solution task and generate a solution.

**Note:** If your solution is interrupted or has reached the maximum number of iterations for a given time

 **Partially updated** point, it will become  [to allow you to review results to that point, but you will not be able to mark it as up-to-date as it has not solved for the defined duration.](#)

## 4.9.1.4. Controlling Residuals

### Physics > Solver Options > Solution Controls > Residual Control

 Click  to show all properties.

Residuals are an indication of how well the solution satisfies the discrete governing equations. As the solution converges to the correct discrete solution on the current mesh, the residuals will drop accordingly.

Residuals can be scaled and/or normalized. The default setting of local scaling and no normalization is recommended.

1. In the **Solution Controls** panel, under **Residual Control**, set **Residual scaling** to choose how the residuals are nondimensionalized.

Option	Description
<b>Local scaling</b>	This is the recommended approach for judging convergence. The residual in each cell is scaled based on the local central coefficient and global solution range, and the display value takes the RMS average over the solution domain. It is also possible to display and base convergence on the maximum rather than RMS average. Typically the maximum residual is one or two orders of magnitude larger than the RMS average.
<b>Global scaling</b>	The residuals are scaled based on a global central coefficient and solution scale. The continuity residual is also normalized, leading to a residual which depends on the initial guess.
<b>None</b>	Allows you to monitor raw residuals rather than scaled residuals, which may be useful for some diagnostics.

2. Select whether or not you want to apply residual **Normalization**.

Option	Description
<b>None</b>	Will not apply residual normalization and therefore the residual drop will not start at a value of 1 at startup.
<b>Automatic</b>	Normalizes the residuals to have a value of 1 at startup (based on the maximum residual during the first N iterations, where N=5 by default). This is a useful option if you would like to see your residuals drop from a value of 1, but has the disadvantage that the normalized residuals depend on the initial guess, so the convergence criteria may need to be adjusted accordingly.

3. Under **Monitor frequency**, if you chose to monitor the residuals, enter the **Iteration interval** at which you want the residual reporting to happen.

## 4.9.1.5. Adjusting Solver Meshing Controls

### Physics > Solver Options > Solution Controls

Before solving the physics, you can make adjustments to the way in which the mesh and certain aspects of the geometry are manipulated in order to make the solution run more efficiently.

In the **Solution Controls** panel, under **Solver Meshing Control** you can:

1. Perform a **Mesh check** by selecting one of the following options:

Option	Description
<b>Basic</b> (default)	Checks for the most common mesh issues, such as left-handed faces or negative elements.
<b>None</b>	Skips the mesh check entirely.

Option	Description
<b>Full</b>	Checks for additional errors in the mesh, such as incorrect topology.
<b>Verbose</b>	Checks the same items as <b>Full</b> , but with more verbose output.

2. Allow for **Polyhedral conversion** by keeping the default selection of **Convert**. This will result in the conversion of tetrahedral and prism cells in the mesh to polyhedral cells, which leads to improved solver convergence.
3. Perform a **Physics based partition** to ensure that the partitioning of the model takes into account the number of equations being solved in a region, providing better balance across the compute engine processes.
4. Merge bodies and faces.

For simulations with complex geometries you may observe long project load and solution update times, slow post processing, as well as high memory usage (especially in parallel processing). To improve performance in simulations containing a large number of geometric faces and bodies:

- Enable **Merge faces within conditions** to combine faces within a fluid flow or solid thermal condition.
- Enable **Merge bodies within regions** to combine bodies within a fluid or solid thermal physics region.

There are several items worth noting when using the merging options. See [Considerations when Merging Faces and Bodies](#) on page 414 for more information.

## 4.9.1.5.1. Considerations when Merging Faces and Bodies

[Merging of faces and bodies](#) may produce unexpected results. The following list details what you can expect when using the merge option.

- Merging will generally help all simulations, but is most beneficial if you have 100 or more volumetric bodies and/or 1000 or more surfaces.
- If you load a project from a previous release and then enable merging, the mesh will be re-sent to the fluids solver where merging will occur. To see the full benefit, you will need to re-solve.
- Polyhedral conversion times are improved when faces/bodies are merged. If you have deactivated polyhedral conversion as a workaround then the merge option should help.
- Faces and bodies can be merged independently, however, enabling both options provides the full performance benefit.
- Merging will result in post processing objects behaving differently when applied to an individual geometric face or body:
  - Contours and vectors: An individual face may be selected in the geometry, but the displayed plot and any summary data is computed for the entire condition in which the face is present.
  - Calculated values: For any selected individual face or body the calculated value is evaluated for the group of merged bodies/faces.
  - Solution monitoring and output parameters will be based on the group of merged bodies/faces containing your selected location.
- Merging of faces and/or bodies can affect a physics coupling solution if the physics coupling interface's target location is a selection set defining a subset of faces in a fluid flow condition or bodies in a region, as the target location will be extended to all faces or bodies in the condition or region.

- There is no special treatment for selection sets, with bodies or faces treated the same as if they were selected directly (i.e. selection sets will extend to all merged bodies/faces).
- If you want to gain the benefits of the automatic merging, but have areas where you wish to have specific results or physics coupling, consider using individual conditions to prevent any merging, e.g. create a separate wall for a specific number of faces of a larger wall if you want a result only on those faces.

## 4.9.1.6. Solving Particle Tracks

### Physics > Solver Options > Solution Controls

In the **Solution Controls** panel, under **Particle Control** you can modify the settings to solve for particle tracking.

The default settings are acceptable and do not need to be modified, but if you need to fine tune your solution you can adjust the settings under **Particle Control**.

**Note:** For any duplicated physics task containing particles, the **Particle Control** settings will not be copied over. Make sure you review all the settings in the duplicated task to ensure the setup is correct.

Option	Details
 <b>Particle summary verbosity</b>	The default value of 2 is recommended. Higher verbosity levels will print out a more detailed transcript, which may help troubleshoot any problems.
 <b>Tracking scheme</b>	To solve equations of motion for the particles, select one of the available numerical schemes. The default <b>Backward euler</b> scheme is appropriate for most options.
<b>Maximum tracking distance</b>	This is the maximum distance over which particles are integrated. It should be large enough to enable a particle to be tracked through the geometry, but not too large or the computational cost of tracking particles which may become trapped in recirculation zones could become exceedingly large. The default value is 100 [m].
<b>Maximum tracking time</b>	This is the maximum time for which the particles are integrated. This should be set to a time long enough for a particle to be tracked through the geometry but not too large or the computational cost of tracking particles that may become trapped in recirculation zones could become exceedingly large. The default value is 1000 [s].
<b>Maximum number of integration steps</b>	This can be used to terminate tracking of particles that may become trapped in recirculation zones. The default value is 100,000. You can increase this number if you believe it will be exceeded by a particle following a normal route through the simulation.

Option	Details
 <b>Fluid variable interpolation</b>	The fluid flow variables (velocity, temperature) are interpolated to the particle position for calculations like drag and heat convection using one of the available options: <b>Cell center and gradient</b> (default), <b>Cell center</b> values, or <b>Cell nodes</b> . If <b>Cell nodes</b> is selected, you can set the <b>Interpolation kernel</b> , which controls the weighting from the nodes. The default setting of <b>Laplacian</b> is appropriate for most cases.
 <b>Time Step Scheme &gt; Option</b>	By default, <b>Adaptive</b> is selected to allow for the size of the time step to change as the calculation proceeds. <b>Standard</b> chooses a time step to achieve a set number of integration steps per cell.
 <b>Time Step Scheme &gt; Minimum time step</b>	Specify the lower limit for the size of the time step.
 <b>Time Step Scheme &gt; Adaptive time step maximum error</b>	This is a tolerance used in adaptive time stepping.

## 4.9.1.7. Choosing a Gradient Method

**Physics > Solver Options > Numerical Controls > Discretization Control > Gradient method**

Gradients are needed for constructing values of a scalar at the cell faces, as well as for computing secondary diffusion terms and velocity derivatives.

1. In the **Numerical Controls** panel, expand **Discretization Control** and select a **Gradient method**.
2. Use the guidelines below to select the most appropriate gradient method for your case. Note that any gradient method can be applied to all mesh types, but accuracy and computational expense may be affected.

Option	Description
<b>Least squares cell based</b>	Recommended for hexahedral and polyhedral meshes. Computation expense: Medium
<b>Green gauss node based</b>	Recommended for tetrahedral/wedge meshes and meshes with highly skewed and distorted cells. Computation expense: High
<b>Green gauss cell based</b>	Not recommended for industrial problems. Computation expense: Low
<b>Automatic</b>	Selects <b>Green gauss node based</b> for mesh regions having tetrahedra and/or wedges, and <b>Least squares cell based</b> otherwise.

## 4.9.1.8. Choosing an Advection Scheme

### Physics > Solver Options > Numerical Controls > Discretization Control > Advection

The advection scheme controls the details of how the solver interpolates solution values to faces when discretizing the advection operator.

1. In the **Numerical Controls** panel, expand **Discretization Control > Advection** to expose the advection scheme settings.
2. If you choose to change the advection scheme, you can make the change either at the global level or per  **Advection Equation**.

The default advection scheme is **Second order upwind** for all equations except turbulence, which uses **First order upwind**. These settings are usually appropriate. In some cases, accuracy can be further improved by changing the turbulence equations to **Second order upwind**, although this may also make convergence more difficult to achieve. (The advection scheme tends to be less important for turbulence than other equations, because turbulence tends to be source-dominated.)

3.  **First to higher order blending** can be enabled if you want to blend between the first and second order upwind schemes. The degree of blending is controlled by the **Blend factor**, where the value of 0 recovers first order upwind and 1 recovers second order upwind. Using a value less than 1 will reduce accuracy, but in some cases may lead to improved convergence.

## 4.9.1.9. Choosing a Pressure Scheme

### Physics > Solver Options > Numerical Controls > Discretization Control > Pressure Scheme

The pressure scheme controls how the solver interpolates pressure to faces when discretizing the pressure gradient term in momentum. The default setting of **Automatic** chooses the best available scheme based on the flow physics and displays this as the **Current pressure scheme**.

1. In the **Numerical Controls** panel, expand **Discretization Control > Pressure Scheme** to expose the settings.
2. To change from the default setting of **Automatic**, use the following guidelines when selecting a scheme:

Option	Description
<b>First order</b>	Interpolates the face pressure using a first order central differencing scheme and momentum-weighting. In general it has first order accuracy, particularly adjacent to boundaries.
<b>Second order</b>	Interpolates the face pressure using a second order central differencing scheme. It is the default because it provides good accuracy and convergence behavior for a wide range of flow situations.
<b>PRESTO</b>	An alternative second order scheme, which is often useful when there are strong body forces present such as high swirl or high-Rayleigh-number natural convection.
<b>Body force weighted</b>	An alternative second order scheme useful for flows with large body forces.

3.  **Implicit extrapolation** is active by default and improves the pressure discretization accuracy at walls when the mesh is coarse.

## 4.9.1.10. Applying Temporal Discretization

### Physics > Solver Options > Numerical Controls

For time-dependent simulations, the governing equations must be discretized in both space and time. The **spatial discretization** for the time-dependent equations is identical to the steady-state case. Temporal discretization involves discretizing the transient term in the differential equations.

The default settings under **Discretization Control > Temporal Discretization** in the **Numerical Controls** panel are adequate and do not typically need to be modified. However, information about each of the settings is provided should you need to modify them.

1. Keep the default settings or select the **Transient Scheme** that best defines the discretization algorithm for the transient term.

Option	Details
<b>First order backward euler</b>	An implicit time-stepping scheme that is first-order accurate. Although it may be useful for initial studies, its use is not recommended for production runs.
<b>Second order backward euler</b> (default)	An implicit time-stepping scheme that is second-order accurate. It is applicable for constant and variable time step sizes. This scheme is recommended for most transient runs.
<b>Bounded second order backward euler</b>	An implicit time-stepping scheme that is second order accurate. It is applicable for constant and variable time steps sizes. It is the same as the <b>Second order backward euler</b> scheme except for the energy equation, for which a modification is introduced to ensure new extrema do not appear in the temperature field.

If you choose to change the transient scheme, you can make the change either at the global level or per  **Transient Equation**.

2. Choose between **Iterative** and **Non iterative** for the **Time Advancement Method**.

Option	Details
<b>Iterative</b> (default)	Iterations within the time step continue until the equations reach their convergence target or the maximum number of iterations is met. The iterations ensure that all inter-equation couplings are resolved, leading to a robust method, particularly for large time step sizes.

Option	Details
<b>Non iterative</b>	A single iteration per time step is performed, which significantly speeds up transient simulations. However, the splitting error which results from not fully resolving inter-equation couplings means that the method may be less robust, particularly for large time step sizes. In addition, it is not applicable for some types of physics, such as high-speed compressible flow. Two noniterative methods are available: the <b>Fractional step</b> method (faster), and <b>PISO</b> (can be more beneficial with complex physics).

## 4.9.1.11. Choosing a Pressure-Velocity Coupling Scheme

**Physics > Solver Options > Numerical Controls > Solution Control > Coupling Control**

The choice for pressure velocity coupling controls how the pressure and velocity fields are advanced in order to achieve a converged solution.

1. In the **Numerical Controls** panel, expand **Solution Control > Coupling Control** and select a **Pressure velocity coupling** method.
2. The default setting of **Coupled** is usually the most robust and efficient algorithm. However, you can select one of the segregated schemes. The table below can help guide you in making a choice:

Option	Description
Coupled	Very robust and recommended for most simulations.
SIMPLE	Convergence rate can be sensitive to pressure and velocity relaxation factors.
SIMPLEC	Allows a more aggressive relaxation factor to be used for pressure. A <b>Skewness correction</b> of 1 is typically applied to improve convergence on skewed meshes.
PISO	Allows a more aggressive relaxation factor to be used for pressure. For transient flows, PISO can maintain a stable calculation with a larger timestep and under-relaxations of 1 for both momentum and pressure. A <b>Neighbor correction</b> of 1 is usual, and a <b>Skewness correction</b> of 1 is typically applied to improve convergence on skewed meshes.

[Controlling Advancement](#) on page 419 describes additional controls that affect convergence behavior.

## 4.9.1.12. Controlling Advancement

**Physics > Solver Options > Numerical Controls > Solution Control > Advancement Control**

You can fine-tune the solution advancement process by choosing the type and magnitude of under-relaxation applied to the discretized equations.

- In the **Numerical Controls** panel, under **Solution Control > Coupling Control**, if you are using the default **Coupled** pressure-velocity coupling scheme, under **Advancement Control** you can under-relax the flow equations either using the **Pseudo transient** method or a **Courant number**.

The pseudo transient method applies a form of under-relaxation which is similar to the transient term discretization for a time-dependent calculation, using a global timescale for all cells. It therefore approximates the physical transient behavior of the flow. The Courant number uses a local timescale, which potentially differs in each cell.

By default, the choice of under-relaxation for energy and turbulence equations is the same as that for flow. Other choices can be made for energy and turbulence by turning filtering off . The available choices are **Pseudo transient**, **Courant number**, and **Relaxation factor**. **Courant number** and **Relaxation factor** are alternative methods of expressing implicit relaxation and are related by the equation:  

$$\text{Relaxation factor} = 1/(1+1/\text{Courant number})$$

Option	Description
<b>Pseudo transient</b>	<p>This method generally gives the best convergence behavior. The solver requires a pseudo-transient timescale. A timescale on the order of 1/10th of the residence time (time for fluid to pass through the system) will usually provide a good balance between speed and robustness. Increasing the timescale may accelerate convergence and more rapidly reduce large global imbalances, while a smaller timescales may stabilize localized numerical instabilities. A very small timestep is not recommended as it will effectively "freeze" the solution.</p> <p>The timescale can be specified directly or estimated by the solver. See the <i>Pseudo-Transient Settings</i> table below for more details and guidance.</p>
<b>Courant number</b>	<p>In some situations this method may give better convergence than the pseudo-transient approach, particularly if there is a wide range of timescales active at the same time. On the other hand, convergence with the Courant number approach may be adversely affected if there are large aspect ratios in the mesh or significant nonlinear source terms.</p>

- Use the table below for guidance on applying the pseudo transient options.

If using the pseudo-transient method, the solver requires a global time scale to apply the under-relaxation. The global timescale can often be estimated by taking a relevant physical length scale divided by a velocity scale.

Table 4.9.1.12.1. Pseudo-Transient Settings

Option	Description
<b>Automatic vs. User specified</b>	Choose <b>User specified</b> if you know what relevant global timescale can be applied. The <b>Automatic</b> option will compute a global timescale by dividing a characteristic length scale by a characteristic velocity, derived from a combination of boundary conditions and average velocity over the physics region.

Option	Description
<b>Timestep scaling factor</b>	The automatic pseudo timestep can sometimes be conservative. It is often possible to accelerate convergence by choosing a scaling factor larger than 1, particularly if the convergence plots show smooth behavior. Alternatively, if the convergence plots show rapid oscillations, decreasing the scaling factor may improve convergence.
 <b>Verbosity</b>	Set this to 1 to display the pseudo timestep calculated by the solver. You can only enter an integer value of 0 or 1.
 <b>Length scale calculation method</b>	The automatic pseudo timestep calculation requires a length scale. The default <b>Conservative</b> approach sets the length scale to the cube root of the geometry volume. This works well for many situations, particularly internal flows. In some cases (such as a 2D geometry modeled as a thin 3D geometry) this estimate can give slow convergence, and the <b>Aggressive</b> approach (which sets the length scale to the maximum extent) may be better. Alternatively, if you know a relevant length scale (e.g., chord length for an external aerodynamic calculation), use the <b>User specified</b> length scale calculation.

3. If you are using one of the segregated pressure-velocity coupling schemes (**SIMPLE**, **SIMPLEC**, or **PISO**), the default value for the implicit relaxation factor is acceptable for most problems. If you encounter convergence difficulties, you can try lowering either the **Implicit relax factor** under **Advancement Control > Flow Control**, or the **explicit relaxation factor** for pressure.

### 4.9.1.13. Setting Explicit Relaxation Factors

**Physics > Solver Options > Numerical Controls > Solution Control > Explicit Relaxation**

After the linear system of equations is solved, the solution fields are corrected. The corrections may be under-relaxed using explicit relaxation. The default under-relaxations depend on the pressure velocity coupling scheme and have been chosen to give robust and rapid convergence for a wide range of cases.

When using the coupled solver, altering these relaxation factors is generally not recommended; the pseudo-timescale or Courant number should be changed instead, as described in [Controlling Advancement](#) on page 419.

When using the segregated solver, if your solution is unstable or diverging, it may be helpful to reduce the under-relaxation factors. In some situations it may also be helpful to reduce the under-relaxation factor for temperature or density (e.g., flows with strong density variations).

### 4.9.1.14. Applying High Order Term Relaxation

**Physics > Solver Options > Numerical Controls > Solution Control > High Order Term Relaxation**

 Click  to show all properties.

**High Order Term Relaxation** is a form of relaxation which often improves robustness and convergence rates when using second order spatial discretization schemes. The default under-relaxation of 0.5 for steady simulations and 1.0 for time-dependent simulations is generally appropriate .

Additionally, high order term relaxation can be controlled per equation.

## 4.9.1.15. Using the Linear Solver

**Physics > Solver Options > Numerical Controls > Solution Control > Linear Solver**

Click  to show all properties.

The linear equation solver uses an algebraic multigrid (AMG) method to iteratively solve the linear system of equations. To understand the idea behind AMG, we note that a simple iterative solver can effectively reduce high wavelength errors only. But the remaining low wavelength errors can be made to appear as high wavelength errors by agglomerating the original equations to form a ‘coarse level’ equation set. Doing this to a hierarchy of coarse levels, together with a strategy to cycle between the levels, ensures that all error modes are effectively reduced. The details of the multigrid solver are controlled by a number of settings as described below, but there will usually be no need to modify them.

Option	Description
Verbosity	Controls the amount of information that is printed out by the linear solver. For example, setting the verbosity to 1 prints the equation name, tolerance, and residuals, while setting it to 2 will print even more information.
Cycle type	Controls how the solver cycles between the different multigrid levels. The choices are Flexible Cycle, V-Cycle, W-Cycle, and F-Cycle.
Termination Value	Multigrid cycles are performed until the linear solver residuals are reduced by this factor.
Restriction Value	Residual reduction criterion controls when to visit a coarser level when using the Flexible cycle.
Stabilization	Activates a Krylov linear solver preconditioned by AMG. Available when the Cycle type is V, W, or F.
Fixed cycle smoother	<b>Gauss seidel</b> is a fast, simple smoother while <b>ILU</b> is more powerful, especially for block-coupled systems.
Factor	Sets the relaxation factor for ILU smoothing sweeps.
Pre sweeps	Sets the number of sweeps to perform on a particular level before moving to a coarser level when using V, W, or F cycles.
Post sweeps	Sets the number of sweeps to perform after coarser level corrections have been applied to this finer level when using V, W, or F cycles.
Sweeps	Sets the number of sweeps performed when using the Flexible cycle.
Maximum coarse levels	The maximum number of coarse levels that will be built by the multigrid solver. Setting this to 0 turns off the multigrid solver.

Option	Description
Maximum cycles	Sets the maximum number of multigrid cycles to be performed. The linear solver will continue to solve the set of equations until either the maximum number of cycles has been performed, or the <b>Termination value</b> is reached.
Coarsening rate	Target coarsening rate when forming each coarse level from its finer level.

## 4.9.1.16. Setting Solution Limits

Physics > Solver Options > Numerical Controls > Solution Control > Solution Limits

Click  to show all properties.

In some difficult situations, the solver may predict solution fields which are physically unrealistic, particularly prior to convergence. If this occurs, you can apply limits to the solution fields. This should not be necessary for most simulations.

1. In the **Numerical Controls** panel, expand **Solution Control > Solution Limits** to expose the default settings.
2. Set the **Minimum/Maximum temperature**.  
These are used if the analysis includes thermal effects.
3. Set the **Maximum turbulence viscosity ratio**, **Minimum turbulence kinetic energy**, and **Minimum turbulence eddy frequency**.

These are used for turbulent flow simulations. The turbulence viscosity is not permitted to exceed the molecular viscosity multiplied by the **Maximum turbulence viscosity ratio**.

## 4.9.1.17. Setting Output Controls for Time-Dependent Fluid Flows

Physics > Solver Options > Output Control

**Output Control** is used to manage the way data is stored during the solution process. If a time-dependent simulation is running, you can control how frequently the data is stored. All stored data will be persisted with the AIM project.

You can set the following:

1. If the **Output Control** panel has not been made readily available to you, you can access it from the **Solver Options** panel. Click the **Add Solver Options** drop-down menu and select **Output Control**.
2. For the **Output frequency**:
  - Select **Low**, **Medium** or **High** for recommended output intervals. A higher frequency will consume more computational resource when it comes to viewing results, but will equally give a higher fidelity.
  - Select **Time interval** if you want output to be created at a specific **Time interval**.
  - Select **Time steps** if you want output to be created after a specific **Number of time steps**.

## 4.9.1.18. Judging Convergence

The nonlinear equations governing fluid flow are solved using an iterative solution method. Convergence refers to the point at which further iterations are no longer necessary. An important part of determining whether your solution is appropriate is to judge whether the solution has converged to an acceptable level.

There are two types of monitor charts which assist with judging convergence:

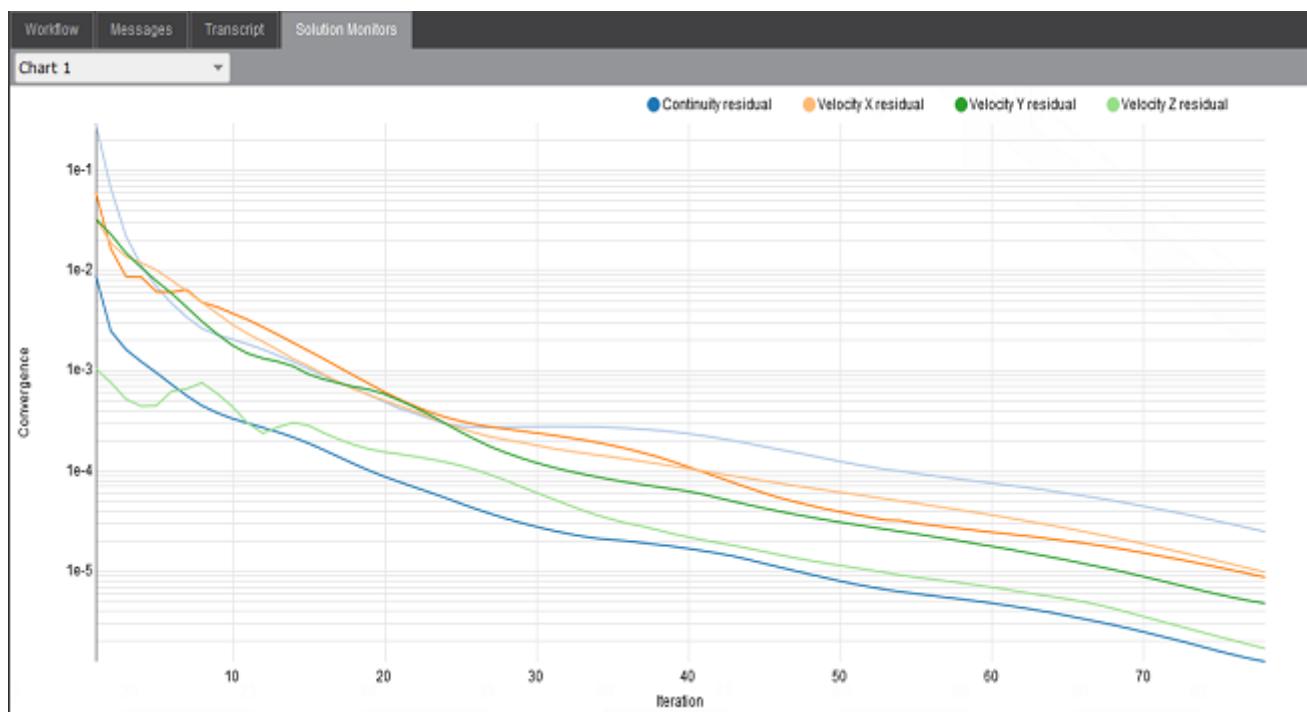
- **Residual monitor charts.** By default, a residual chart is created for you and is displayed in the [Solution Monitor tab](#). [Residuals](#) measure the amount by which a particular solution does not satisfy the discretized equations. If all goes well, the residuals will drop toward zero as more iterations are performed. Iterations are terminated when the residual convergence target has been reached for relevant equations or when the maximum number of iterations have been performed.

**Note:** The turbulence equations (tke and tef) are excluded from the convergence check by default. If you would like to include them (or any other equations) in your convergence plots, you can create a chart and add the monitored quantities as described in [Creating a Monitor Chart](#). Make sure to update the solution to view the new plots.

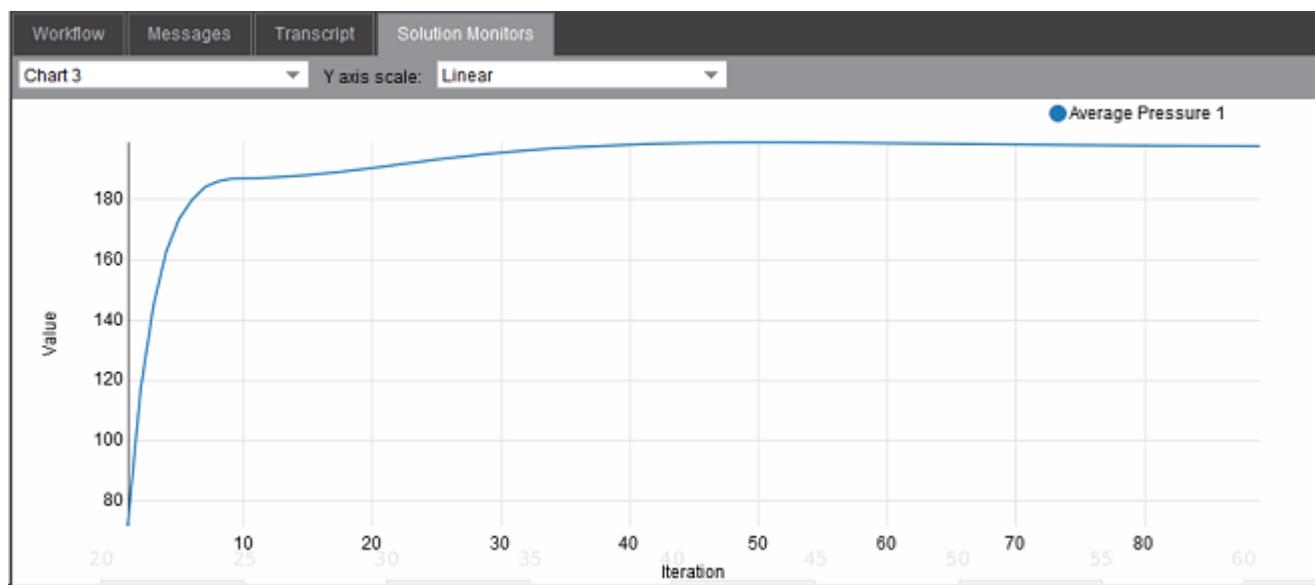
- **Calculated value monitor charts.** [Creating a calculated value](#) and [monitoring](#) it is another powerful tool to help judge convergence, particularly in situations where residuals do not fully converge due to a localized instability. The solution may still be reliable provided relevant quantities of interest have stabilized. [Examples of calculated value monitors](#) which may be useful include:
  - Mass flow rate through a pressure-specified outflow boundary
  - Average pressure at velocity-specified or mass flow-specified inflow boundary
  - Force components on a wall
  - Minimum, maximum, or average temperature in a body
  - Heat flow through a boundary face

Residual and calculated value monitors for a simulation which has good convergence behavior are shown in Figure 1 and Figure 2. Figure 1 shows that the residuals have reached the default target of 1.e-5 in 78 iterations, and Figure 2 shows that the average pressure on the inflow boundary is no longer changing.

**Figure 4.9.1.18.1. Solution Quality Monitor Displaying Residuals for a Converged Solution**



**Figure 4.9.1.18.2. Calculated Value Monitor of Average Pressure on Inlet Boundary**



You can also judge convergence using the Transcript tab, which shows the residual values in numerical format. The transcript will indicate whether the residuals have converged for a solution.

**Figure 4.9.1.18.3. Transcript Displaying Residuals for a Converged Solution**

Workflow    Messages    Transcript    Solution Monitors

Transcript 1

```

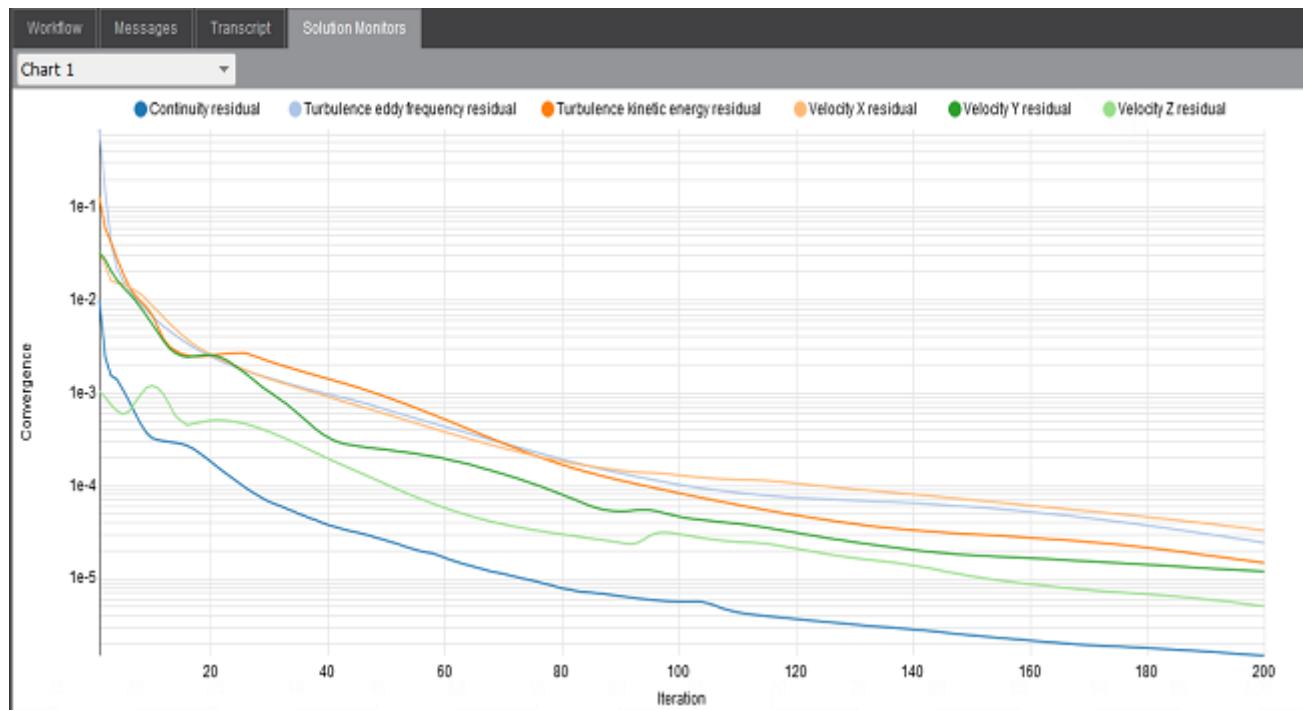
62  4.2524e-06  3.1802e-05  1.3559e-05  8.0977e-06  2.2554e-05  8.6522e-05
63  4.0138e-06  2.9903e-05  1.4441e-05  5.7597e-06  2.1527e-05  6.5280e-05
64  3.7818e-06  2.8074e-05  1.3518e-05  5.4832e-06  2.0524e-05  6.2036e-05
65  3.5376e-06  2.6298e-05  1.2657e-05  5.1886e-06  1.9666e-05  5.8907e-05
66  3.2988e-06  2.4608e-05  1.1767e-05  4.8835e-06  1.8809e-05  5.5873e-05
67  3.0794e-06  2.2971e-05  1.0961e-05  4.5506e-06  1.7871e-05  5.2806e-05
68  2.8624e-06  2.1396e-05  1.0192e-05  4.1829e-06  1.6928e-05  4.9777e-05
69  2.6510e-06  1.9891e-05  9.4263e-06  3.8107e-06  1.5963e-05  4.6776e-05
70  2.4398e-06  1.8454e-05  8.6942e-06  3.4682e-06  1.5003e-05  4.3806e-05
71  2.2370e-06  1.7081e-05  7.9997e-06  3.1574e-06  1.4057e-05  4.0949e-05
72  2.0544e-06  1.5777e-05  7.3541e-06  2.8726e-06  1.3205e-05  3.8371e-05
73  1.8816e-06  1.4569e-05  6.7736e-06  2.6176e-06  1.2352e-05  3.5816e-05
74  1.7136e-06  1.3437e-05  6.2228e-06  2.3862e-06  1.1503e-05  3.3267e-05
75  1.5614e-06  1.2378e-05  5.7436e-06  2.1775e-06  1.0700e-05  3.0839e-05
iter continuity xvelocity yvelocity zvelocity tke tef
76  1.4350e-06  1.1389e-05  5.3355e-06  1.9884e-06  9.9492e-06  2.8559e-05
77  1.3212e-06  1.0472e-05  4.9872e-06  1.8193e-06  9.2445e-06  2.6421e-05
78  1.2361e-06  9.6307e-06  4.6989e-06  1.6674e-06  8.5842e-06  2.4455e-05
Convergence criteria have been met.

```

In some cases, convergence is not achieved within the specified number of iterations. If this occurs, follow the recommendations below based on the behavior you observe.

- **Residuals are decreasing but do not reach target**

**Figure 4.9.1.18.4. Residuals Decreasing But Not Reaching Target**



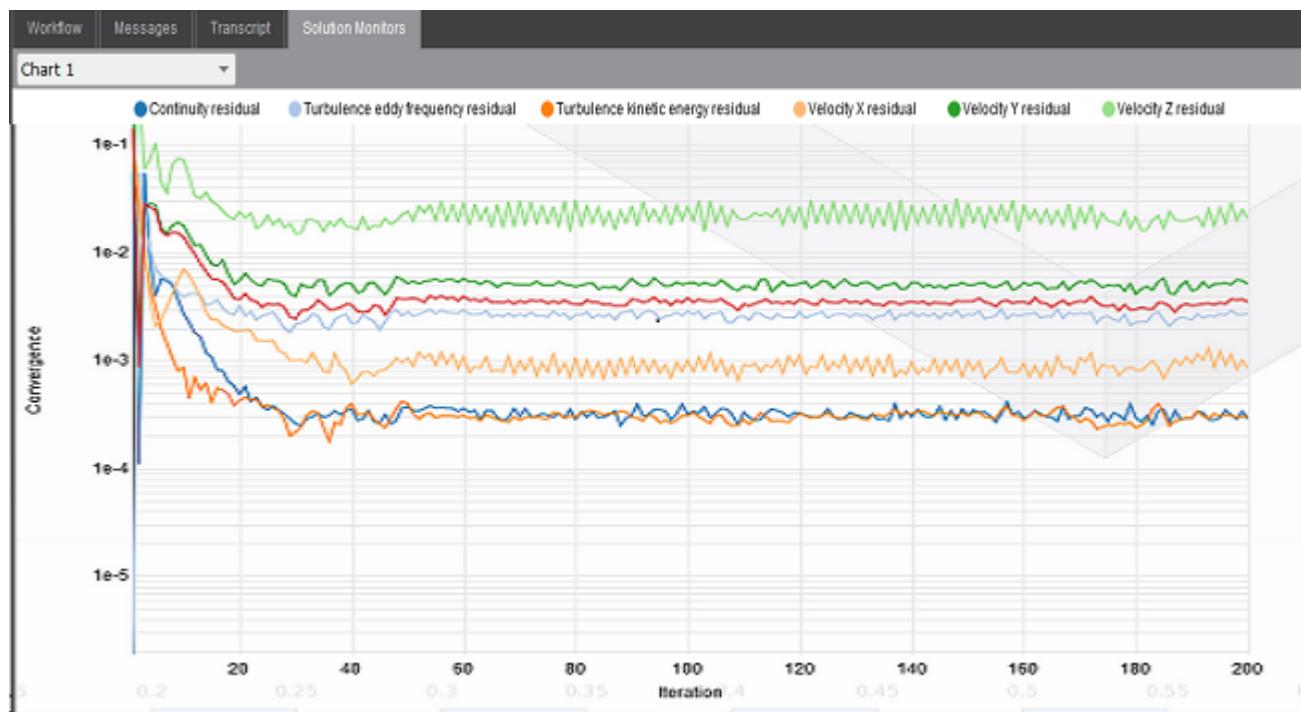
**Recommendation:** You can either increase the [maximum number of iterations](#) under **Iteration Control** in the **Solution Controls** panel, or adjust the **Solution advancement exponent** towards aggressive (a positive value such as 0.5 or 1) to increase the rate at which the solver converges toward the solution. Then, solve the flow physics task again and the solution will be re-calculated with the new settings. Repeat these steps until the solution has converged.

Alternatively, if relevant calculated value monitors have leveled off, or your solution is close to being converged and is good enough, you can accept the solution as it is by right-clicking on the physics task in the workflow view and selecting **Mark as Up-to-Date**. Note that if your solution is currently running, you do not have to wait until the maximum number of iterations is reached, instead you can interrupt it and then **Mark as Up-to-Date**.

- **Residuals are stalled or oscillating**

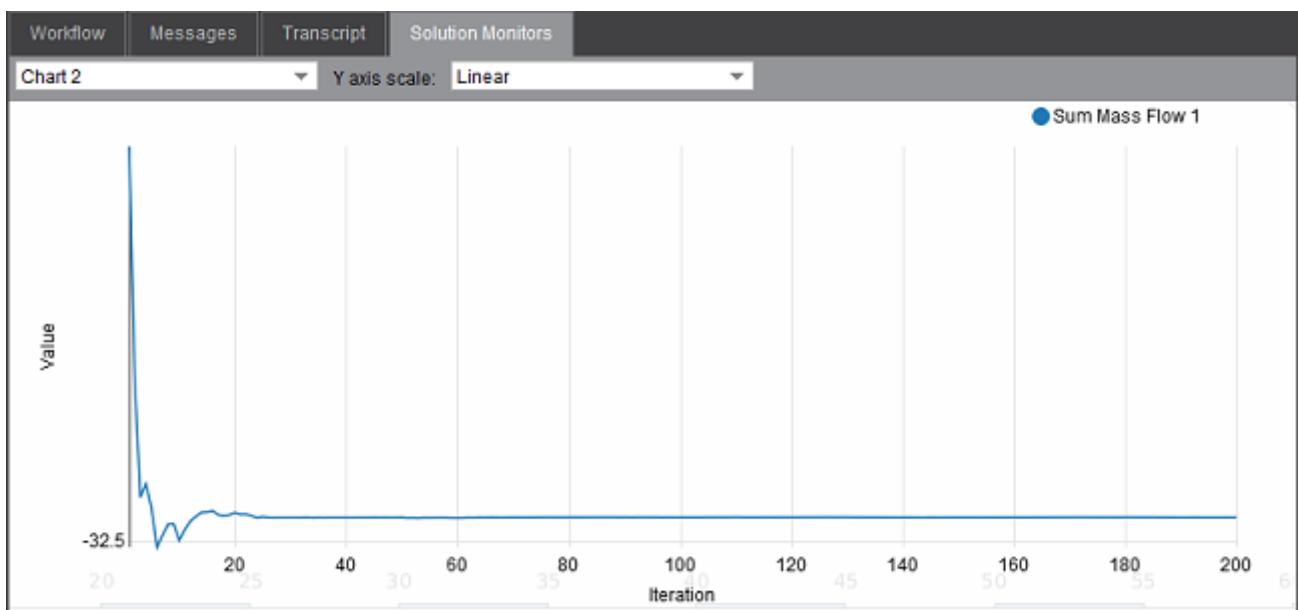
Residuals that are stalled or oscillating indicate that a solution cannot be obtained with current settings in some areas. There may be physical and/or numerical reasons for the lack of convergence. It is not always easy to isolate the cause, but the nature of the convergence plot may give some hints. For example, sawtooth oscillations or stalled residuals such as in the figure below suggest a convergence issue or transient behavior localized to a small number of cells.

**Figure 4.9.1.18.5. Residuals Stalled or Oscillating Due to Insufficient Relaxation**



The calculated value monitor of mass flow rate through the outflow boundary shown in the figure below is stable, despite the fact that residuals have not converged. The solution might therefore still be reliable.

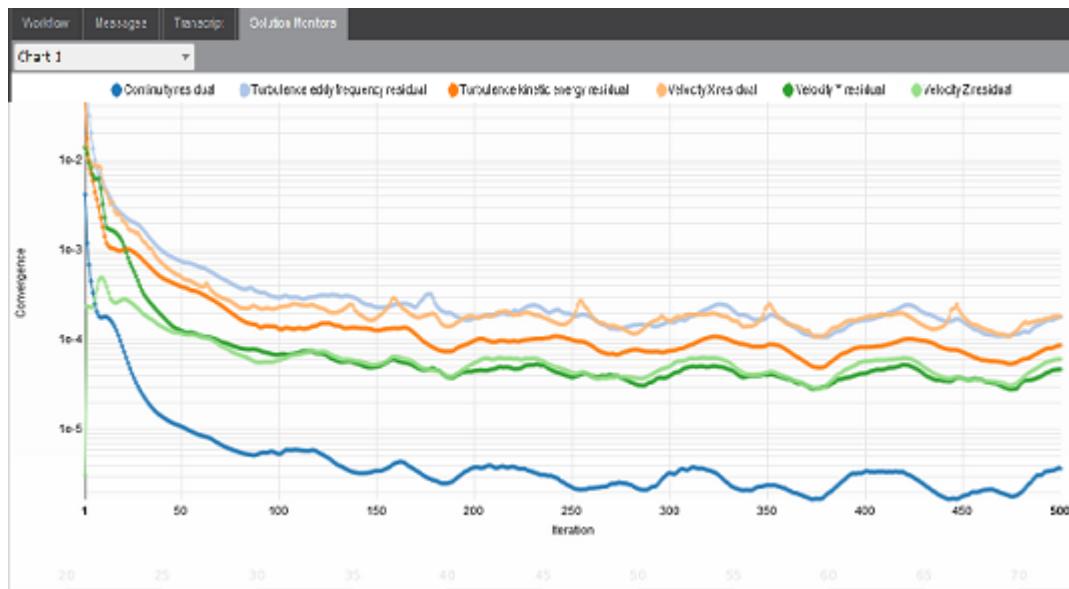
**Figure 4.9.1.18.6. Calculated Value Monitor of Mass Flow Through Outflow Boundary**



- ° **Recommendation:** If the lack of convergence is numerical in origin, adjust the **Solution advancement exponent** in the **Solution Controls** panel towards conservative (a negative value, such as -0.5 or -1.0). Alternatively, if relevant calculated value monitors have leveled off, you can also accept the solution as it is.

Larger oscillations, such as those displayed in the figure below, often indicate a larger-scale transient effect, such as an unsteady recirculation zone.

**Figure 4.9.1.18.7. Residuals Stalled or Oscillating Due to Physical Unsteadiness**



- ° **Recommendation:** If the lack of convergence is because the flow is physically unsteady, then it will not be possible to achieve a converged steady state solution. If the degree of unsteadiness is small, such that relevant calculated value monitors are stable, you can also accept the solution as it is.

Additionally, it is helpful to evaluate the solution and mesh in the area where the maximum residuals are located. The centroid location of each maximum residual is printed to the transcript at the end of the

iterations. In the figure below, the maximum residual of 4.861e-02 is for the equation pbns-cpld/velocity-0 at the location [-3.977e+01, 1.580e+01, 3.846e-02].

**Figure 4.9.1.18.8. Transcript Displaying Maximum Residuals**

The screenshot shows the ANSYS Fluent interface with the 'Transcript' tab selected. The transcript displays two tables: 'Residual Summary' and 'Maximum Residual Centroid'. The 'Residual Summary' table lists equations, RMS residuals, maximum residuals, and global IDs. The 'Maximum Residual Centroid' table lists equations and their corresponding centroid coordinates where the maximum residual was found.

Equation	RMS Res	Max Res	Global ID
pbns-cpld/pressure	9.819e-06	1.528e-03	122914
pbns-cpld/velocity-0	3.168e-04	4.861e-02	290043
pbns-cpld/velocity-1	7.998e-05	8.713e-03	106704
pbns-cpld/velocity-2	8.326e-05	8.715e-03	115324
turb-seg/tke	2.367e-04	4.442e-02	115324
turb-seg/tef	4.275e-04	3.441e-02	106810

Equation	Centroid
pbns-cpld/pressure	[-1.900e+01 1.420e+01 9.615e-01]
pbns-cpld/velocity-0	[-3.977e+01 1.580e+01 3.846e-02]
pbns-cpld/velocity-1	[-1.380e+01 1.404e+01 3.846e-02]
pbns-cpld/velocity-2	[-1.323e+01 1.412e+01 1.154e-01]
turb-seg/tke	[-1.323e+01 1.412e+01 1.154e-01]
turb-seg/tef	[-1.316e+01 1.404e+01 1.923e-01]

Inspect the mesh and solution in the vicinity of these points to check for potential issues. For example:

- A steady solution may not be achievable. If a transient flow phenomenon (such as vortex shedding past a circular cylinder or unsteady separation) is modeled as a steady-state simulation, the solution will not converge.
- Incorrect physical models or boundary conditions.
- Poor [mesh quality](#), particularly high skewness, can make convergence difficult or impossible.
- [Nonoptimal advancement settings](#) or [explicit relaxation factors](#). The default values have been found to work well for a wide range of problems, but in some cases they may need to be fine-tuned, particularly if the physics model is numerically stiff (hard to solve).
- [Poor initial guess](#). The default Hybrid initialization scheme works well in general, but in some special cases, User Specified initialization may be required.

- **Residuals are increasing or diverging**

Divergence is indicated by strongly oscillating or increasing residuals. Residuals which suddenly drop toward zero in a single iteration can also indicate divergence. This is because residuals are [scaled](#) by the solution range, which grows rapidly during divergence.

**Figure 4.9.1.18.9. Residuals Increasing or Diverging**



**Recommendation:** Adjust the **Solution advancement exponent** towards conservative (a negative value, such as -0.5 or -1.0). Additionally, it is helpful to evaluate the solution and mesh in the area where the maximum residuals are located, as described above.

#### 4.9.2. Structural Solver Options

##### Physics > Solver Options

When you create your physics solution using a template, a number of solver options are set up using standard defaults. You can modify the default values for a given physics solution on the **Solver Settings**, **Launch Controls**, **Output Controls**, **Solution Progression**, **Boundary Refinement**, and **Fidelity Refinement** panels.

## Setting General Solver Options

On the **Solver Settings** panel, the settings available define the solver, such as the solver unit system and solver type. For a Modal solution, you also define the number of modes to extract. The default is to extract the first six natural modes. For a nonlinear simulation, you can set the global nonlinear options for all solution steps.

## Specifying Launch Controls

On the **Launch Controls** panel, you can control how the solver is launched and how the solver data is managed. Properties include solver file locations, file names, and distributed solve controls.

## Defining Output Specifications

In the **Output Controls** panel, you can control how the solution data is written to the output file by defining one or more output specifications, which set the solution location, output type, and frequency. These output specifications are processed sequentially based on the order they are listed in the Output Controls panel.

By default, three output specifications are created for a static structural physics solution and one for a modal structural physics solution.

If you want to add Strain or other output types to the result file, you can create additional Output Controls objects for the same location.

**Note:** In a modal structural physics solution, the solution consists of a deformed shape scaled by an arbitrary factor. The actual magnitudes of the deformations and any derived quantities, such as strains and stresses, are therefore meaningless. Only the relative values of such quantities throughout the model should be considered meaningful. The arbitrary scaling factor is numerically sensitive to slight perturbations in the analysis; choosing a different unit system, for example, can cause a significantly different scaling factor to be calculated.

## Setting Nonlinear Options

You can apply global nonlinear controls for all solution steps, such as the Newton-Raphson setting, on the **Solver Settings** panel. On the **Solution Progression** panel, you can apply nonlinear settings that are applied per solution step.

## Specifying Boundary Refinement

[Boundary refinement](#) provides an optional input that lets you refine the boundary mesh generated by automatic physics-aware meshing.

## Specifying Solution Fidelity Refinement

[Fidelity refinement](#) provides an optional input that lets you identify localized geometric details of a specific size that you want to capture in the solution. This enables you to increase the accuracy of the solution while optimizing solution time by setting the [global solution fidelity](#) to a lower setting, and then applying fidelity refinement to specific topologies only.

### 4.9.2.1. Configuring the Solver for a Static Structural Solution

#### Physics > Solver Options > Solver Settings

Default solver settings are generated when you set up your physics regions, however you may want to modify some of the settings.

To change the default solver settings:

1. For **Equation solver type**, select the solver type.

The default is **Program controlled**, which lets the program choose the type of solver appropriate for your model. However, you can explicitly select one of the following:

- **Direct**: More effective with thin flexible models.
- **Iterative**: Best for bulky models.

2. For **Newton Raphson method**, select the method you want to use.

- **Full**: The solver uses the full Newton-Raphson procedure, in which the stiffness matrix is updated at every equilibrium iteration.
- **Modified**: The solver uses the modified Newton-Raphson technique, in which the tangent stiffness matrix is updated at each substep. The matrix is not changed during equilibrium iterations at a substep. This option is not applicable to large-deformation analyses. Adaptive descent is not available.

- **Initial stiffness:** The solver uses the initial stiffness matrix in every equilibrium iteration. This option can be less likely to diverge than the full option, but it often requires more iterations to achieve convergence. It is not applicable to large-deformation analyses. Adaptive descent is not available.
- **Unsymmetrical:** The program uses the full Newton-Raphson procedure, in which the stiffness matrix is updated at every equilibrium iteration. In addition, it generates and uses unsymmetric matrices that you can use for any of the following:
  - If you are defining an unsymmetric material model, you would need this method to fully use the property you defined.
  - If you are running a contact physics solution, an unsymmetric contact stiffness matrix would fully couple the sliding and the normal stiffnesses. You should first try the Full method, then try this method only if you experience convergence difficulties. (Using an unsymmetric solver requires more computational time to obtain a solution than if using a symmetric solver.)

For more information, see [and](#) in the [.](#)

### 3. Set the **Initial interface treatment** for nonlinear contacts.

- To ensure AIM will warn you of any possible gap/penetration issues before the solve, select **Check for gaps/overlaps before solve**.
- To direct AIM to automatically try to fix initial gaps/overlaps for all nonlinear contacts introduced due to poor geometry or coarse mesh, select **Fix unintentional initial gap/overlap**.

**Note:** This option requires that the **Initial interface treatment** property in the **Contact Behavior** be set to **Program controlled**.

- To include/consider any initial gap/overlap issues and proceed with the solve, select **Model initial gap/overlap**.

**Note:** Additionally, the behavior of individual nonlinear contacts can be controlled through the **Interface Gap/Penetration Adjustment** property in the **Contact Behavior**.

### 4. Select an option for **Large deflection** to specify if the solver should take into account large deflection, large rotation or large strain.

With the default setting, AIM sets large deflection on when it finds nonlinearities (such as nonlinear contacts) in the solution but otherwise leaves it off.

You can set it to **On** if you expect large deflections (as in the case of a long, slender bar under bending) or large strains (as in a metal-forming problem).

When using hyperelastic material models, you must set it to **On**.

### 5. Additionally, you can modify the following settings:

If you want to...	then set...	More information:
Select a unit system for the solver.	<b>Solver unit system</b>	<p>The selection of a consistent unit system guarantees that all quantities, inputs and outputs to the solver can be interpreted correctly in terms of the units in the system.</p> <p>You can leave the default setting, <b>Active Unit System</b>, or choose one of the standard unit systems defined in the <b>Unit Systems</b> window. When you leave the default, <b>Active Unit System</b>, the active unit system chosen in the <b>Unit Systems</b> window is used. Select <b>File icon &gt; Units</b> to confirm that the units displayed are what you expect.</p> <p>If you do not specify a unit system, in either the <b>Unit Systems</b> window or here in the solver settings, AIM selects one of six possible standard unit systems and converts all quantities into that system. The standard unit systems include Metric and the Consistent systems (CGC, NMM, uMKS, BIN, BFT); see the <b>Unit Systems</b> window for a listing.</p>
Select if you want the solver to calculate accelerations to counterbalance the applied loads.	<b>Inertia relief</b>	<p>Select whether you want inertia relief on or off. When inertia relief is on, the solver calculates accelerations to counterbalance the applied loads. Displacement constraints on the structure should only be those necessary to prevent rigid-body motions (six for a 3-D structure). The sum of the reaction forces at the constraint points will be zero. Accelerations are calculated from the element mass matrices and the applied forces. Data needed to calculate the mass (such as density) must be input. Both translational and rotational accelerations may be calculated. This option applies only to the linear static structural analyses.</p> <p>Nonlinearities, elements that operate in the nodal coordinate system, and axisymmetric or generalized plane strain elements are not allowed. Models with both 2-D and 3-D element types or with symmetry boundary constraints are not recommended. Loads may be input as usual. Displacements and stresses are calculated as usual. Symmetry models are not valid for inertia relief.</p>

## 4.9.2.2. Configuring the Solver for a Modal Structural Solution

### Physics > Solver Options > Solver Settings

Default solver settings are generated when you set up your physics regions. You can modify these values on the **Solver Settings** panel. These options are set for a given calculation run of a solver and not tied to a topology.

To change the default solver settings:

1. For **Max modes to find**, specify the number of natural frequencies to solve for in a modal physics solution. The default is to extract the first six natural frequencies.

The number of modes can be specified in two ways:

- First N modes( $N > 0$ )

- First N modes in a selected range of frequencies

- 2.**  For **Limit search to range**, select whether you want to specify a frequency range within which to find the natural frequencies. By default, this option is set to **Yes** for the Supernode equation solver and **No** for other solvers.

If this option is set to **Yes**, enter minimum and maximum frequency range values in the **Range minimum** and **Range maximum** fields. Within that range, the solver will strive to extract as many frequencies as possible subject to a maximum specified in the **Max modes to find** field.

If this option is set to **No**, the solver will find all of the possible modes without any restrictions on the frequency range.

- 3.**  For **Solver unit system**, select a unit system for the solver. You can leave the default setting, **Active Unit System**, or choose one of the standard unit systems defined in the **Unit Systems** window.

When you leave the default, **Active Unit System**, the active unit system chosen in the **Unit Systems** window is used. If you have not specifically chosen a unit system in the **Unit Systems** window, AIM selects one of six possible standard unit systems and converts all quantities into that system. The selection of a consistent unit system guarantees that all quantities, inputs and outputs to the solver can be interpreted correctly in terms of the units in the system. The standard unit systems include Metric and the Consistent systems (CGC, NMM, uMKS, BIN, BFT); see the **Unit Systems** window for a listing.

- 4.** For **Equation solver type**, select the solver type. The default is **Program Controlled**, which lets the program choose the type of solver appropriate for your model. However, you can explicitly select one of the following:

- **Direct**: More effective with thin flexible models.
- **Iterative**: Best for bulky models.
- **Supernode**: Recommended for extracting a large number of modes.
- **Subspace**: Reasonably accurate but relatively faster than the Direct solver. In addition, the Subspace solver allows you to take advantage of a distributed architecture to perform faster computations.

For more information, see [Choosing a Modal Solver Type](#) on page 434.

- 5.** Set the **Initial interface treatment** for nonlinear contacts.

- To ensure AIM will warn you of any possible gap/penetration issues before the solve, select **Check for gaps/overlaps before solve**.
- To direct AIM to automatically try to fix initial gaps/overlaps for all nonlinear contacts introduced due to poor geometry or coarse mesh, select **Fix unintentional initial gap/overlap**.

**Note:** This option requires that the **Initial interface treatment** property in the **Contact Behavior** be set to **Program controlled**.

- To include/consider any initial gap/overlap issues and proceed with the solve, select **Model initial gap/overlap**.

**Note:** Additionally, the behavior of individual nonlinear contacts can be controlled through the [Interface Gap/Penetration Adjustment](#) property in the **Contact Behavior**.

## 4.9.2.2.1. Choosing a Modal Solver Type

By default, AIM selects the optimal solver for your modal physics solution. However, you can also explicitly select the Direct, Iterative, Subspace, or Supernode solver type.

All four solver types – Direct, Iterative, Subspace, and Supernode – are used to solve a modal system that does not include any damping effects.

These solver types are intended to solve Eigen solutions with symmetric mass and stiffness. For a large model, the Iterative solver is preferred over the Direct solver for its efficiency in terms of solution time and memory usage.

The Supernode solver is recommended for extracting a large number of modes. Selecting **Supernode** as the **Equation Solver type** automatically sets the **Limit search to range** property to **Yes**. This selection also displays the **Range minimum** and **Range maximum** properties and requires a range maximum frequency entry. Alternatively, you may reset the **Limit search to range** property to **No** to find all of the possible modes without any restrictions on the frequency range.

The Subspace solver is relatively faster than Direct solver and also has reasonable accuracy.

## 4.9.2.2.2. Eigenvalues for a Modal Solution

Once you have solved your physics, the eigenvalues for your solution are automatically created as a **Table** result on the **Results** panel. You can parameterize results based on the eigenvalues selected for a given result.

**Note:** In a Modal physics solution, the solution consists of a deformed shape scaled by an arbitrary factor. The actual magnitudes of the deformations and any derived quantities, such as strains and stresses, are therefore meaningless. Only the relative values of such quantities throughout the model should be considered meaningful. The arbitrary scaling factor is numerically sensitive to slight perturbations in the analysis; choosing a different unit system, for example, can cause a significantly different scaling factor to be calculated.

## 4.9.2.3. Configuring the Solver for an Eigenvalue Buckling Solution

### Physics > Solver Options > Solver Settings

Default solver settings are generated when you set up your physics regions, however you may want to modify some of the settings.

To change the default solver settings:

1. For **Max modes to find**, specify the number of buckling load factors (and therefore the corresponding buckling mode shapes of interest). Typically the first (lowest) buckling load factor is of interest. The default value for this field is 2.
2. For **Equation solver type**, select the solver type. The default is **Program Controlled**, which lets the program choose the type of solver appropriate for your model. However, you can explicitly select one of the following:
  - **Direct**: More effective with thin flexible models.
  - **Subspace**: Reasonably accurate but relatively faster than the Direct solver. In addition, the Subspace solver allows you to take advantage of a distributed architecture to perform faster computations.
3. For **Newton Raphson method**, select the method you want to use.
  - **Full**: The solver uses the full Newton-Raphson procedure, in which the stiffness matrix is updated at every equilibrium iteration.
  - **Modified**: The solver uses the modified Newton-Raphson technique, in which the tangent stiffness matrix is updated at each substep. The matrix is not changed during equilibrium iterations at a substep. This option is not applicable to large-deformation analyses. Adaptive descent is not available.
  - **Initial stiffness**: The solver uses the initial stiffness matrix in every equilibrium iteration. This option can be less likely to diverge than the full option, but it often requires more iterations to achieve convergence. It is not applicable to large-deformation analyses. Adaptive descent is not available.

- **Unsymmetrical:** The program uses the full Newton-Raphson procedure, in which the stiffness matrix is updated at every equilibrium iteration. In addition, it generates and uses unsymmetric matrices that you can use for any of the following:
  - If you are defining an unsymmetric material model, you would need this method to fully use the property you defined.
  - If you are running a contact physics solution, an unsymmetric contact stiffness matrix would fully couple the sliding and the normal stiffnesses. You should first try the Full method, then try this method only if you experience convergence difficulties. (Using an unsymmetric solver requires more computational time to obtain a solution than if using a symmetric solver.)

For more information, see [and](#) in the [.](#)

#### 4. Set the **Initial interface treatment** for nonlinear contacts.

- To ensure AIM will warn you of any possible gap/penetration issues before the solve, select **Check for gaps/overlaps before solve**.
- To direct AIM to automatically try to fix initial gaps/overlaps for all nonlinear contacts introduced due to poor geometry or coarse mesh, select **Fix unintentional initial gap/overlap**.

**Note:** This option requires that the **Initial interface treatment** property in the **Contact Behavior** be set to **Program controlled**.

- To include/consider any initial gap/overlap issues and proceed with the solve, select **Model initial gap/overlap**.

**Note:** Additionally, the behavior of individual nonlinear contacts can be controlled through the **Interface Gap/Penetration Adjustment** property in the **Contact Behavior**.

#### 5. Select an option for **Large deflection** to specify if the solver should take into account large deflection, large rotation or large strain.

With the default setting, AIM sets large deflection on when it finds nonlinearities (such as nonlinear contacts) in the solution but otherwise leaves it off.

You can set it to **On** if you expect large deflections (as in the case of a long, slender bar under bending) or large strains (as in a metal-forming problem).

When using hyperelastic material models, you must set it to **On**.

#### 6. Select whether to include **Negative load multipliers**. By default, the solver determines this value based on the requirements of the simulation, but you can also select **Yes** to extract both the negative and positive eigenvalues (load multipliers) or **No** to extract only positive eigenvalues (load multipliers).

#### 7. Additionally, you can modify the following settings:

If you want to...	then set...	More information:
Select a unit system for the solver.	<b>Solver unit system</b>	<p>The selection of a consistent unit system guarantees that all quantities, inputs and outputs to the solver can be interpreted correctly in terms of the units in the system.</p> <p>You can leave the default setting, <b>Active Unit System</b>, or choose one of the standard unit systems defined in the <b>Unit Systems</b> window. When you leave the default, <b>Active Unit System</b>, the active unit system chosen in the <b>Unit Systems</b> window is used. Select <b>File icon &gt; Units</b> to confirm that the units displayed are what you expect.</p> <p>If you do not specify a unit system, in either the <b>Unit Systems</b> window or here in the solver settings, AIM selects one of six possible standard unit systems and converts all quantities into that system. The standard unit systems include Metric and the Consistent systems (CGC, NMM, uMKS, BIN, BFT); see the <b>Unit Systems</b> window for a listing.</p>
Select if you want the solver to calculate accelerations to counterbalance the applied loads.	<b>Inertia relief</b>	<p>Select whether you want inertia relief on or off. When inertia relief is on, the solver calculates accelerations to counterbalance the applied loads. Displacement constraints on the structure should only be those necessary to prevent rigid-body motions (six for a 3-D structure). The sum of the reaction forces at the constraint points will be zero. Accelerations are calculated from the element mass matrices and the applied forces. Data needed to calculate the mass (such as density) must be input. Both translational and rotational accelerations may be calculated. This option applies only to the linear static structural analyses.</p> <p>Nonlinearities, elements that operate in the nodal coordinate system, and axisymmetric or generalized plane strain elements are not allowed. Models with both 2-D and 3-D element types or with symmetry boundary constraints are not recommended. Loads may be input as usual. Displacements and stresses are calculated as usual. Symmetry models are not valid for inertia relief.</p>

## 4.9.2.4. Setting Solver Launch Options for a Structural Solution

### Physics > Solver Options > Launch Controls

Launch controls allow you to control the computational resources used by a simulation. The launch controls for the solver are populated with default settings, file names, and values. Review these settings and ensure that the default entries are acceptable for your model.

To change the default launch control settings:

1. For **Number of processors**, enter the desired number of processors.
2.  Additionally, you can modify the following settings:

If you want to...	then set...	More information:
Run the solver live or in batch mode.	<b>Solver mode</b>	With <b>Live</b> , the solver runs in RPC mode. It is run as a server and clients may communicate to the solver while it is running. The solver can also run in <b>Batch</b> mode, which is typical of running from a command line. In this mode, AIM is not communicating directly with the solver, but communicates instead through input and output files.
Use a distributed solver.	<b>Distributed solve</b>	
Remove stale files from a previous solve at the next solve.	<b>Delete work files before solve</b>	
Modify a default file name.	The file name for any of the following: <ul style="list-style-type: none"><li>• <b>Input data file</b></li><li>• <b>Mesh data file</b></li><li>• <b>Component data file</b></li><li>• <b>Connection elements data file</b></li><li>• <b>Load elements data file</b></li><li>• <b>Output data file</b></li></ul>	Enter the name of file you want to use instead of the default.
Change the name of the job.	<b>Job name</b>	
Output the contact pair information to a file other than the default output file.	<b>Output contact pairs to Jobname.cnm</b>	

## 4.9.2.5. Controlling Output Data for a Structural Solution

### Physics > Solver Options > Output Controls

The output specification set in the **Output Controls** panel determines how the solution data is written to the output file. By default, AIM creates three output specifications for a Static Structural solution and one for a Modal Structural solution. The default output types are set to:

Physics Solution	Output Type	Which Stores
<b>Static Structural</b>	Nodal DOF Solution	Displacement
	Nodal Reaction Loads	Force Reaction
	Element Nodal Stresses	Stress
<b>Modal Structural</b>	Nodal DOF Solution	Mode Shape

The **Frequency** is set by default to **All Time Points**. However, if you want to add other output types to the result file, you can modify these output specifications or create additional Output Controls for the same location.

1. Define the location within the model for which you want the solution information written in the output file.

The default is `Everywhere()`. You can also specify the `AllBodies()` expression for all the bodies defined in the Physics Region, or select one or more bodies from the model.

2. For **Output type**, select a solution type.

The options available are based on the physics and calculation types.

3.  For **Frequency**, select how often the solver should write the solutions results to the output file. Available options are:

- **All Time Points**: Writes the solution information for every sub step. This is the default option.
- **Last Time Point**: (Static only) Writes the solution information only for the last sub step.
- **Equally Spaced Time Points**: (Static only) Writes the solution information for up to a specified number of equally spaced solutions. This option only applies to Static or Transient analyses when the Automatic Time Stepping is enabled. For this option, specify a nonzero positive number for the equally spaced time points.
- **Specified Recurrence Rate**: (Static only) Writes the solution information at every specified interval and at the last sub step of each load step. For this option, specify a nonzero positive number for the recurrence rate.
- **Never**: Suppresses writing of the specified solution type for all sub steps.

4. Repeat the previous steps for additional output specifications.

## 4.9.2.6. Setting Nonlinear Controls in a Static Structural Simulation

When solving nonlinear simulations, AIM carries out an iterative procedure (equilibrium iterations) at each substep, successfully solving the simulation only when the out-of-balance loads are less than the specified convergence criteria. On the **Solution Progression** panel, you can define the number of substeps in a solution step. You can then define the convergence controls and other solution progression properties.

For **Substepping**, you can define a range or a fixed number, or you can leave it up to the solver.

AIM displays the convergence criteria appropriate for each physics type. For a static structural physics solution, the convergence criteria consist of Force convergence, Moment convergence, Displacement convergence, and Rotation convergence. You can set the convergence controls for each of these criteria.

 Under **Additional Convergence Controls**, you can also specify the number of equilibrium iterations at each substep, activate a predictor on the degree-of-freedom solution for the first equilibrium iteration of each substep, and control the line search program.

For **Stabilization**, you can specify the key for controlling nonlinear stabilization. Convergence difficulty due to an unstable problem is usually the result of a large displacement for small load increments. Nonlinear stabilization technique can help achieve convergence. Nonlinear stabilization can be thought of as adding artificial dampers to all of the nodes in the system. Any degree of freedom that tends to be unstable has a large displacement causing a large damping/stabilization force. This force reduces displacements at the degree of freedom so stabilization can be achieved.

## 4.9.2.6.1. Specifying Solution Progression for Static Structural

### Physics > Solver Options > Solution Progression

To set the controls for complete criteria:

- From **Substepping**, set time stepping to a range or a fixed number, or leave it to be controlled by the solver.

If substepping is controlled by the solver, a check is performed on non-convergent patterns, and the physics of the simulation is also taken into account. In case of **Adaptive**, the solver marches through the solution based on the initial, minimum, and maximum number of substeps.

- If you choose **Adaptive**, specify the initial, minimum, and maximum number of substeps.
- If you choose **User specified**, specify the substep count.

- For each convergence type under **Convergence Controls**, specify how you want convergence handled.

- The recommended setting, **Program controlled**, automatically calculates the value based on external forces, including reactions.
- If set to **On**, you can input convergence values in the fields that appear.
- If set to **Off**, convergence is not checked.

- If a convergence type is set to **On**, do the following:

- For **Tolerance (%)**, specify the tolerance value as a percentage.

**Tolerance** times **Value** determines the convergence criterion.

- For **Reference value specification**, select if you want the solver to use a **Reference value** when establishing convergence, or have the solver control the convergence based on a **Minimum reference value**.

The **Minimum reference value** option is useful for analyses where the external forces tend to zero. This can happen, for example, with free thermal expansion where rigid body motion is prevented.

- Specify any additional desired criteria.

If you want to...	then set...	More information:
Activate a predictor in a nonlinear analysis on the degree-of-freedom solution for the first equilibrium iteration of each substep	<b>Predictor-corrector</b>	You can set this to: <ul style="list-style-type: none"> <li>• Program controlled</li> <li>• Off</li> <li>• On after the first substep</li> <li>• On for all substeps</li> </ul>
Specify the number of equilibrium iterations at each substep.	<b>Equilibrium iterations</b>	You can specify whether the number of equilibrium iterations carried out at each substep is: <ul style="list-style-type: none"> <li>• Automatically set by the solver</li> <li>• Set to a value of 1</li> <li>• Set to a number to be specified by you in the <b>Number equilibrium iterations</b> field</li> </ul>

If you want to...	then set...	More information:
Control the line search program.	<b>Line search</b>	Line search can be useful for enhancing convergence, but it can be expensive (especially with plasticity). For more information, see <a href="#">Specifying Line Search for Static Structural</a> on page 441.
Control nonlinear stabilization.	<b>Stabilization &gt; Stabilization key</b>	Nonlinear stabilization technique can help achieve convergence. For more information, see <a href="#">Activating Stabilization for Static Structural</a> on page 441.

## 4.9.2.6.1.1. Specifying Line Search for Static Structural

### Physics > Solver Options > Solution Progression

Line search can be useful for enhancing convergence, but it can be expensive (especially with plasticity). You might consider setting line search on for the following cases:

- When your structure is force-loaded (as opposed to displacement-controlled).
- If you are analyzing a "flimsy" structure which exhibits increasing stiffness (such as a fishing pole).
- If you notice (from the program output messages) oscillatory convergence patterns.

You can set Line Search to:

- Program controlled
- On
- Off

## 4.9.2.6.1.2. Activating Stabilization for Static Structural

### Physics > Solver Options > Solution Progression > Stabilization

For **Stabilization**, you can specify the key for controlling nonlinear stabilization. Convergence difficulty due to an unstable problem is usually the result of a large displacement for small load increments. Nonlinear stabilization technique can help achieve convergence. Nonlinear stabilization can be thought of as adding artificial dampers to all of the nodes in the system. Any degree of freedom that tends to be unstable has a large displacement causing a large damping/stabilization force. This force reduces displacements at the degree of freedom so stabilization can be achieved.

1. Under **Stabilization**, select the **Stabilization key**.

#### Option

<b>Constant energy dissipation</b>	Activate stabilization and use the energy dissipation ratio as the control. Then specify in the <b>Energy dissipation ratio</b> field the ratio of work done by stabilization forces to element potential energy. This value is usually a number between 0 and 1. The default value is 1.0e-4.
<b>Reduced energy dissipation</b>	Activate stabilization and reduce the energy dissipation ratio linearly from the specified or calculated value to zero at the end of the load step. Then specify in the <b>Energy dissipation ratio</b> field the ratio of work done by stabilization forces to element potential energy. This value is usually a number between 0 and 1. The default value is 1.0e-4.

Option	
<b>Constant damping factor</b>	<p>Activate stabilization and use the damping factor as the control.</p> <p>Then specify in the <b>Damping factor</b> field the value that the solver uses to calculate stabilization forces for all subsequent substeps. This value is usually a value greater than 0.</p> <p><b>Note:</b> The <b>Damping factor</b> value is dependent on the active unit system and may influence the results if unit systems are changed.</p>
<b>Reduced damping factor</b>	<p>Activate stabilization and reduce the damping factor linearly from the specified or calculated value to zero at the end of the load step.</p> <p>Then specify in the <b>Damping factor</b> field the value that the solver uses to calculate stabilization forces for all subsequent substeps. This value is usually a value greater than 0.</p> <p><b>Note:</b> The <b>Damping factor</b> value is dependent on the active unit system and may influence the results if unit systems are changed.</p>

2. For **Substep option**, select an option for when stabilization is activated.
  - No - Stabilization is not activated for the first substep even when it does not converge after the minimal allowed time increment is reached (default setting).
  - Yes - Stabilization is activated for the first substep. Use this option if stabilization was active for the previous load step.
  - If not converged - Stabilization is activated for the first substep if it still does not converge after the minimal allowed time increment is reached. Use this option for the first load step only.
3. For **Force limit**, specify the stabilization force check as a number between 0 and 1. The default value is 0.2. To omit a stabilization force check, set this value to 0.

#### 4.9.3. Thermal Solver Options

##### Physics > Solver Options

When you create your physics solution using a template, a number of solver options are set up using standard defaults. You can modify the default values for a given physics solution on the **Solver Settings**, **Launch Controls**, **Output Controls**, **Solution Progression**, **Boundary Refinement**, and **Fidelity Refinement** panels.

## Setting General Solver Options

On the **Solver Settings** panel, the settings available define the solver, such as the solver unit system and solver type. For a nonlinear simulation, you can set the global nonlinear options for all solution steps.

## Specifying Launch Controls

On the **Launch Controls** panel, you can control how the solver is launched and how the solver data is managed. Properties include solver file locations, file names, and distributed solve controls.

# Output Controls

In the **Output Controls** panel, you can control how the solution data is written to the output file by defining one or more output specifications, which set the solution location, output type, and frequency. These output specifications are processed sequentially based on the order they are listed in the Output Controls panel.

By default, two output specifications are created in the Output Controls, with the output types set to Nodal DOF Solution and Nodal Reaction Loads, which respectively store Temperature and Heat Reactions in the result file. If you want to add other output types to the result file, you can modify these output specifications or create additional Output Controls for the same location.

## Setting Nonlinear Options

You can apply global nonlinear controls for all solution steps, such as the Newton-Raphson setting, on the **Solver Settings** panel. On the **Solution Progression** panel, you can apply nonlinear settings that are applied per solution step.

## Specifying Boundary Refinement

[Boundary refinement](#) provides an optional input that lets you refine the boundary mesh generated by automatic physics-aware meshing.

## Specifying Solution Fidelity Refinement

[Fidelity refinement](#) provides an optional input that lets you identify localized geometric details of a specific size that you want to capture in the solution. This enables you to increase the accuracy of the solution while optimizing solution time by setting the [global solution fidelity](#) to a lower setting, and then applying fidelity refinement to specific topologies only.

### 4.9.3.1. Configuring the Solver for a Thermal Solution

#### Physics > Solver Options > Solver Settings

Default solver settings are generated when you set up your physics regions, however you may want to modify some of the settings.

To change the default solver settings:

1. For **Equation solver type**, select the solver type.

The default is **Program controlled**, which lets the program choose the type of solver appropriate for your model. However, you can explicitly select one of the following:

- **Direct**: More effective with thin flexible models.
- **Iterative**: Best for bulky models.

2. For **Newton Raphson method**, select the method you want to use.

- **Full**: The solver uses the full Newton-Raphson procedure, in which the stiffness matrix is updated at every equilibrium iteration.
- **Modified**: The solver uses the modified Newton-Raphson technique, in which the tangent stiffness matrix is updated at each substep. The matrix is not changed during equilibrium iterations at a substep. This option is not applicable to large-deformation analyses. Adaptive descent is not available.

- **Initial stiffness:** The solver uses the initial stiffness matrix in every equilibrium iteration. This option can be less likely to diverge than the full option, but it often requires more iterations to achieve convergence. It is not applicable to large-deformation analyses. Adaptive descent is not available.
- **Unsymmetrical:** The program uses the full Newton-Raphson procedure, in which the stiffness matrix is updated at every equilibrium iteration. In addition, it generates and uses unsymmetric matrices that you can use for any of the following:
  - If you are defining an unsymmetric material model, you would need this method to fully use the property you defined.
  - If you are running a contact physics solution, an unsymmetric contact stiffness matrix would fully couple the sliding and the normal stiffnesses. You should first try the Full method, then try this method only if you experience convergence difficulties. (Using an unsymmetric solver requires more computational time to obtain a solution than if using a symmetric solver.)

For more information, see [and](#) in the [.](#)

3.  Additionally, you can select a unit system for the solver from **Solver unit system**.

The selection of a consistent unit system guarantees that all quantities, inputs and outputs to the solver can be interpreted correctly in terms of the units in the system.

You can leave the default setting, **Active Unit System**, or choose one of the standard unit systems defined in the **Unit Systems** window. When you leave the default, **Active Unit System**, the active unit system chosen in the **Unit Systems** window is used. Select **File icon > Units** to confirm that the units displayed are what you expect.

If you do not specify a unit system, in either the **Unit Systems** window or here in the solver settings, AIM selects one of six possible standard unit systems and converts all quantities into that system. The standard unit systems include Metric and the Consistent systems (CGC, NMM, uMKS, BIN, BFT); see the **Unit Systems** window for a listing.

**Note:** The **Initial interface treatment** property is only applicable to non-linear contacts.

### 4.9.3.2. Setting Solver Launch Options for a Thermal Solution

#### Physics > Solver Options > Launch Controls

Launch controls allow you to control the computational resources used by a simulation. The launch controls for the solver are populated with default settings, file names, and values. Review these settings and ensure that the default entries are acceptable for your model.

To change the default launch control settings:

1. For **Number of processors**, enter the desired number of processors.
2.  Additionally, you can modify the following settings:

If you want to...	then set...	More information:
Run the solver live or in batch mode.	<b>Solver mode</b>	With <b>Live</b> , the solver runs in RPC mode. It is run as a server and clients may communicate to the solver while it is running. The solver can also run in <b>Batch</b> mode, which is typical of running from a command line. In this mode, AIM is not communicating directly with the solver, but communicates instead through input and output files.
Use a distributed solver.	<b>Distributed solve</b>	
Remove stale files from a previous solve at the next solve.	<b>Delete work files before solve</b>	
Modify a default file name.	The file name for any of the following: <ul style="list-style-type: none"><li>• <b>Input data file</b></li><li>• <b>Mesh data file</b></li><li>• <b>Component data file</b></li><li>• <b>Connection elements data file</b></li><li>• <b>Load elements data file</b></li><li>• <b>Output data file</b></li></ul>	Enter the name of file you want to use instead of the default.
Change the name of the job.	<b>Job name</b>	
Output the contact pair information to a file other than the default output file.	<b>Output contact pairs to Jobname.cnm</b>	

### 4.9.3.3. Controlling Output Data for a Thermal Solution

#### Physics > Solver Options > Output Controls

The output specification set in the **Output Controls** panel determines how the solution data written is to the output file. By default, two output specifications are created in the Output Controls object. The output types for these two output specifications are set to:

Output Type	Which Stores
Nodal DOF Solution	Temperature
Nodal Reaction Loads	Heat Reactions

If you want to add other output types to the result file, you can modify these output specifications or create additional Output Controls for the same location.

To set up an output specification:

1. [Define the location](#) within the model for which you want the solution information written in the output file.

The default is `Everywhere()`. You can also specify the `AllBodies()` expression for all the bodies defined in the Physics Region, or select one or more bodies from the model.

## 2. For **Output type**, select a solution type.

The options available are based on the physics and calculation types.

### 3. For **Frequency**, select how often the solver should write the solutions results to the output file. Available options are:

- **All Time Points**: Writes the solution information for every sub step. This is the default option.
- **Last Time Points**: Writes the solution information only for the last sub step.
- **Equally Spaced Time Points**: Writes the solution information for up to a specified number of equally spaced solutions. This option only applies to Static or Transient analyses when the Automatic Time Stepping is enabled. For this option, specify a nonzero positive number for the equally spaced time points.
- **Specified Recurrence Rate**: Writes the solution information at every specified interval and at the last sub step of each load step. For this option, specify a nonzero positive number for the recurrence rate.
- **Never**: Suppresses writing of the specified solution type for all sub steps.

## 4.9.3.4. Setting Nonlinear Controls in a Steady-State Thermal Simulation

When solving nonlinear simulations, AIM carries out an iterative procedure (equilibrium iterations) at each substep, successfully solving the simulation only when the out-of-balance loads are less than the specified convergence criteria. On the **Solution Progression** panel, you can define the number of substeps in a solution step. You can then define the convergence controls and other solution progression properties.

For **Substepping**, you can define a range or a fixed number, or you can leave it up to the solver.

AIM displays the convergence criteria appropriate for each physics type. For a steady-state thermal physics solution, the convergence criteria consist of Temperature and Heat convergence. You can set the convergence controls for each of these criteria.

 Under **Additional Convergence Controls**, you can also specify the number of equilibrium iterations at each substep, activate a predictor on the degree-of-freedom solution for the first equilibrium iteration of each substep, and control the line search program.

For **Stabilization**, you can specify the key for controlling nonlinear stabilization. Convergence difficulty due to an unstable problem is usually the result of a large displacement for small load increments. Nonlinear stabilization technique can help achieve convergence. Nonlinear stabilization can be thought of as adding artificial dampers to all of the nodes in the system. Any degree of freedom that tends to be unstable has a large displacement causing a large damping/stabilization force. This force reduces displacements at the degree of freedom so stabilization can be achieved.

## 4.9.3.4.1. Specifying Solution Progression for Steady-State Thermal

### Physics > Solver Options > Solution Progression

To set the controls for complete criteria:

1. From **Substepping**, set time stepping to a range or a fixed number, or leave it to be controlled by the solver.

If substepping is controlled by the solver, a check is performed on non-convergent patterns, and the physics of the simulation is also taken into account. In case of specified range, the solver marches through the solution based on the initial, minimum, and maximum number of substeps.

- If you choose **Specified range**, specify the initial, minimum, and maximum number of substeps.
- If you choose **Fixed number**, specify the substep count.

2. For each convergence type under **Convergence Controls**, specify how you want convergence handled.
  - The recommended setting, **Program controlled**, automatically calculates the value based on external forces, including reactions.
  - If set to **On**, you can input convergence values in the fields that appear.
  - If set to **Off**, convergence is not checked.
3. If a convergence type is set to **On**, do the following:
  - a) For **Tolerance (%)**, specify the tolerance value as a percentage.  
**Tolerance** times **Value** determines the convergence criterion.
  - b) For **Reference value specification**, select if you want the solver to use a **Reference value** when establishing convergence, or have the solver control the convergence based on a **Minimum reference value**.  
The **Minimum reference value** option is useful for analyses where the external forces tend to zero. This can happen, for example, with free thermal expansion where rigid body motion is prevented.
4. Specify any additional desired criteria.

If you want to...	then set...	More information:
Activate a predictor in a nonlinear analysis on the degree-of-freedom solution for the first equilibrium iteration of each substep	<b>Predictor-corrector</b>	You can set this to: <ul style="list-style-type: none"> <li>• Program controlled</li> <li>• Off</li> <li>• On after the first substep</li> <li>• On for all substeps</li> </ul>
Specify the number of equilibrium iterations at each substep.	<b>Equilibrium iterations</b>	You can specify whether the number of equilibrium iterations carried out at each substep is: <ul style="list-style-type: none"> <li>• Automatically set by the solver</li> <li>• Set to a value of 1</li> <li>• Set to a number to be specified by you in the <b>Number equilibrium iterations</b> field</li> </ul>
Control the line search program.	<b>Line search</b>	Line search can be useful for enhancing convergence, but it can be expensive (especially with plasticity). For more information, see <a href="#">Specifying Line Search for Steady-State Thermal</a> on page 449.
Control nonlinear stabilization.	<b>Stabilization &gt; Stabilization key</b>	Nonlinear stabilization technique can help achieve convergence. For more information, see <a href="#">Specifying Nonlinear Stabilization Controls for Steady-State Thermal</a> on page 448.

## 4.9.3.4.2. Specifying Nonlinear Stabilization Controls for Steady-State Thermal

### Physics > Solver Options > Solution Progression > Stabilization

For **Stabilization**, you can specify the key for controlling nonlinear stabilization. Convergence difficulty due to an unstable problem is usually the result of a large displacement for small load increments. Nonlinear stabilization technique can help achieve convergence. Nonlinear stabilization can be thought of as adding artificial dampers to all of the nodes in the system. Any degree of freedom that tends to be unstable has a large displacement causing a large damping/stabilization force. This force reduces displacements at the degree of freedom so stabilization can be achieved.

1. Under **Stabilization**, select the **Stabilization key**.

#### Option

##### Constant energy dissipation

Activate stabilization and use the energy dissipation ratio as the control.

Then specify in the **Energy dissipation ratio** field the ratio of work done by stabilization forces to element potential energy. This value is usually a number between 0 and 1. The default value is 1.0e-4.

##### Reduced energy dissipation

Activate stabilization and reduce the energy dissipation ratio linearly from the specified or calculated value to zero at the end of the load step.

Then specify in the **Energy dissipation ratio** field the ratio of work done by stabilization forces to element potential energy. This value is usually a number between 0 and 1. The default value is 1.0e-4.

##### Constant damping factor

Activate stabilization and use the damping factor as the control.

Then specify in the **Damping factor** field the value that the solver uses to calculate stabilization forces for all subsequent substeps. This value is usually a value greater than 0.

**Note:** The **Damping factor** value is dependent on the active unit system and may influence the results if unit systems are changed.

##### Reduced damping factor

Activate stabilization and reduce the damping factor linearly from the specified or calculated value to zero at the end of the load step.

Then specify in the **Damping factor** field the value that the solver uses to calculate stabilization forces for all subsequent substeps. This value is usually a value greater than 0.

**Note:** The **Damping factor** value is dependent on the active unit system and may influence the results if unit systems are changed.

2. For **Substep option**, select an option for when stabilization is activated.

- No - Stabilization is not activated for the first substep even when it does not converge after the minimal allowed time increment is reached (default setting).
- Yes - Stabilization is activated for the first substep. Use this option if stabilization was active for the previous load step.
- If not converged - Stabilization is activated for the first substep if it still does not converge after the minimal allowed time increment is reached. Use this option for the first load step only.

- For **Force limit**, specify the stabilization force check as a number between 0 and 1. The default value is 0.2. To omit a stabilization force check, set this value to 0.

## 4.9.3.4.3. Specifying Line Search for Steady-State Thermal

### Physics > Solver Options > Solution Progression

Line search can be useful for enhancing convergence, but it can be expensive (especially with plasticity). You might consider setting line search on for the following cases:

- When your structure is force-loaded (as opposed to displacement-controlled).
- If you are analyzing a "flimsy" structure which exhibits increasing stiffness (such as a fishing pole).
- If you notice (from the program output messages) oscillatory convergence patterns.

You can set Line Search to:

- Program controlled
- On
- Off

## 4.9.3.5. Setting Nonlinear Controls in a Time-Dependent Thermal Simulation

When solving nonlinear simulations, AIM carries out an iterative procedure (equilibrium iterations) at each time step, successfully solving the simulation only when the out-of-balance loads are less than the specified convergence criteria. On the **Solution Progression** panel, you can define the number of time steps in a solution step. In addition to defining time steps in terms of a fixed number or minimum/maximum/initial time step values, you can also define time steps at fixed time intervals or initial/min/max time intervals. You can then define the convergence controls and other solution progression properties.

For **Time Stepping**, you can define a range or a fixed number, or you can leave it up to the solver.

AIM displays the convergence criteria appropriate for each physics type. For a thermal physics solution, the convergence criteria consist of Temperature and Heat convergence. You can set the convergence controls for each of these criteria.

 Under **Additional Convergence Controls**, you can also specify the number of equilibrium iterations at each time step, activate a predictor on the degree-of-freedom solution for the first equilibrium iteration of each time step, and control the line search program.

For **Stabilization**, you can specify the key for controlling nonlinear stabilization. Convergence difficulty due to an unstable problem is usually the result of a large displacement for small load increments. Nonlinear stabilization technique can help achieve convergence. Nonlinear stabilization can be thought of as adding artificial dampers to all of the nodes in the system. Any degree of freedom that tends to be unstable has a large displacement causing a large damping/stabilization force. This force reduces displacements at the degree of freedom so stabilization can be achieved.

## 4.9.3.5.1. Specifying Solution Progression for Time-Dependent Thermal

### Physics > Solver Options > Solution Progression

To set the controls for nonlinear criteria:

- From **Time Stepping**, choose the method AIM uses to determine the length of the time step. By default, the solver attempts to set the time step length that captures the transient effects within your simulation. Adaptive will vary the time step length within the range you specify. User specified will use the fixed time step length you define.

The rate of loading could be important in a time-dependent thermal analysis if the material properties vary rapidly with temperature. When such nonlinearities are present, you may need to apply the loads in small increments and perform solutions at these intermediate loads to achieve convergence.

- If you choose **Adaptive**, specify the length of the first time step for **Initial time step**, the shortest time step that the program should use in **Minimum time step**, and the longest time step the program should use in **Maximum time step**.
- If you choose **User specified**, specify a fixed length for all time steps in **Time step**.

- To include transient effects, select **On** from **Time integration**.
- For each convergence type under **Convergence Controls**, specify how you want convergence handled.
  - The recommended setting, **Program controlled**, automatically calculates the value based on external forces, including reactions.
  - If set to **On**, you can input convergence values in the fields that appear.
  - If set to **Off**, convergence is not checked.
- If a convergence type is set to **On**, do the following:
  - For **Tolerance (%)**, specify the tolerance value as a percentage.  
**Tolerance** times **Value** determines the convergence criterion.
  - For **Reference value specification**, select if you want the solver to use a **Reference value** when establishing convergence, or have the solver control the convergence based on a **Minimum reference value**.  
The **Minimum reference value** option is useful for analyses where the external forces tend to zero. This can happen, for example, with free thermal expansion where rigid body motion is prevented.
- Specify any additional desired criteria.

If you want to...	then set...	More information:
Activate a predictor in a nonlinear analysis on the degree-of-freedom solution for the first equilibrium iteration of each substep	<b>Predictor-corrector</b>	You can set this to: <ul style="list-style-type: none"> <li>Program controlled</li> <li>Off</li> <li>On after the first substep</li> <li>On for all substeps</li> </ul>
Specify the number of equilibrium iterations at each substep.	<b>Equilibrium iterations</b>	You can specify whether the number of equilibrium iterations carried out at each substep is: <ul style="list-style-type: none"> <li>Automatically set by the solver</li> <li>Set to a value of 1</li> <li>Set to a number to be specified by you in the <b>Number equilibrium iterations</b> field</li> </ul>
Control the line search program.	<b>Line search</b>	Line search can be useful for enhancing convergence, but it can be expensive (especially with plasticity). For more information, see <a href="#">Specifying Line Search for Steady-State Thermal</a> on page 449.

If you want to...	then set...	More information:
Control nonlinear stabilization.	<b>Stabilization &gt; Stabilization key</b>	Nonlinear stabilization technique can help achieve convergence. For more information, see <a href="#">Specifying Nonlinear Stabilization Controls for Steady-State Thermal on page 448</a> .

## 4.9.3.5.2. Specifying Nonlinear Stabilization Controls for Time-Dependent Thermal

### Physics > Solver Options > Solution Progression > Stabilization

For **Stabilization**, you can specify the key for controlling nonlinear stabilization. Convergence difficulty due to an unstable problem is usually the result of a large displacement for small load increments. Nonlinear stabilization technique can help achieve convergence. Nonlinear stabilization can be thought of as adding artificial dampers to all of the nodes in the system. Any degree of freedom that tends to be unstable has a large displacement causing a large damping/stabilization force. This force reduces displacements at the degree of freedom so stabilization can be achieved.

1. Under **Stabilization**, select the **Stabilization key**.

#### Option

<b>Constant energy dissipation</b>	Activate stabilization and use the energy dissipation ratio as the control. Then specify in the <b>Energy dissipation ratio</b> field the ratio of work done by stabilization forces to element potential energy. This value is usually a number between 0 and 1. The default value is 1.0e-4.
<b>Reduced energy dissipation</b>	Activate stabilization and reduce the energy dissipation ratio linearly from the specified or calculated value to zero at the end of the load step. Then specify in the <b>Energy dissipation ratio</b> field the ratio of work done by stabilization forces to element potential energy. This value is usually a number between 0 and 1. The default value is 1.0e-4.
<b>Constant damping factor</b>	Activate stabilization and use the damping factor as the control. Then specify in the <b>Damping factor</b> field the value that the solver uses to calculate stabilization forces for all subsequent substeps. This value is usually a value greater than 0. <b>Note:</b> The <b>Damping factor</b> value is dependent on the active unit system and may influence the results if unit systems are changed.
<b>Reduced damping factor</b>	Activate stabilization and reduce the damping factor linearly from the specified or calculated value to zero at the end of the load step. Then specify in the <b>Damping factor</b> field the value that the solver uses to calculate stabilization forces for all subsequent substeps. This value is usually a value greater than 0. <b>Note:</b> The <b>Damping factor</b> value is dependent on the active unit system and may influence the results if unit systems are changed.

2. For **Substep option**, select an option for when stabilization is activated.

- No - Stabilization is not activated for the first substep even when it does not converge after the minimal allowed time increment is reached (default setting).
  - Yes - Stabilization is activated for the first substep. Use this option if stabilization was active for the previous load step.
  - If not converged - Stabilization is activated for the first substep if it still does not converge after the minimal allowed time increment is reached. Use this option for the first load step only.
3. For **Force limit**, specify the stabilization force check as a number between 0 and 1. The default value is 0.2. To omit a stabilization force check, set this value to 0.

### 4.9.3.5.3. Specifying Line Search for Time-Dependent Thermal

#### Physics > Solver Options > Solution Progression

Line search can be useful for enhancing convergence, but it can be expensive (especially with plasticity). You might consider setting line search on for the following cases:

- When your structure is force-loaded (as opposed to displacement-controlled).
- If you are analyzing a "flimsy" structure which exhibits increasing stiffness (such as a fishing pole).
- If you notice (from the program output messages) oscillatory convergence patterns.

You can set Line Search to:

- Program controlled
- On
- Off

### 4.9.4. Electric Conduction Solver Options

#### Physics > Solver Options

When you create your physics solution using a template, a number of solver options are set up using standard defaults. You can modify the default values for a given physics solution on the **Solver Settings**, **Launch Controls**, **Output Controls**, **Solution Progression**, **Boundary Refinement**, and **Fidelity Refinement** panels.

## Setting General Solver Options

On the **Solver Settings** panel, the settings available define the solver, such as the solver unit system and solver type. For a nonlinear simulation, you can set the global nonlinear options for all solution steps.

## Specifying Launch Controls

On the **Launch Controls** panel, you can control how the solver is launched and how the solver data is managed. Properties include solver file locations, file names, and distributed solve controls.

## Output Controls

In the **Output Controls** panel, you can control how the solution data is written to the output file by defining one or more output specifications, which set the solution location, output type, and frequency. These output specifications are processed sequentially based on the order they are listed in the Output Controls panel.

By default, three output specifications are created for an electric conduction physics solution:

- Nodal DOF Solution (which stores Voltage)
- Nodal Reaction Loads (which stores Current Reaction)
- Nodal Element Solution (which stores Current Density, Electric Field, and Joule Heat Generation)

If you want to add other output types to the result file, you can modify these output specifications or create additional Output Controls for the same location.

## Setting Nonlinear Options

You can apply global nonlinear controls for all solution steps, such as the Newton-Raphson setting, on the **Solver Settings** panel. On the **Solution Progression** panel, you can apply nonlinear settings that are applied per solution step.

## Specifying Boundary Refinement

[Boundary refinement](#) provides an optional input that lets you refine the boundary mesh generated by automatic physics-aware meshing.

## Specifying Solution Fidelity Refinement

[Fidelity refinement](#) provides an optional input that lets you identify localized geometric details of a specific size that you want to capture in the solution. This enables you to increase the accuracy of the solution while optimizing solution time by setting the [global solution fidelity](#) to a lower setting, and then applying fidelity refinement to specific topologies only.

### 4.9.4.1. Configuring the Solver for an Electric Conduction Solution

#### Physics > Solver Options > Solver Settings

Default solver settings are generated when you set up your physics regions, however you may want to modify some of the settings.

To change the default solver settings:

1. For **Equation solver type**, select the solver type.

The default is **Program controlled**, which lets the program choose the type of solver appropriate for your model. However, you can explicitly select one of the following:

- **Direct**: More effective with thin flexible models.
- **Iterative**: Best for bulky models.

2. For **Newton Raphson method**, select the method you want to use.

- **Full**: The solver uses the full Newton-Raphson procedure, in which the stiffness matrix is updated at every equilibrium iteration.
- **Modified**: The solver uses the modified Newton-Raphson technique, in which the tangent stiffness matrix is updated at each substep. The matrix is not changed during equilibrium iterations at a substep. This option is not applicable to large-deformation analyses. Adaptive descent is not available.

- **Initial stiffness:** The solver uses the initial stiffness matrix in every equilibrium iteration. This option can be less likely to diverge than the full option, but it often requires more iterations to achieve convergence. It is not applicable to large-deformation analyses. Adaptive descent is not available.
- **Unsymmetrical:** The program uses the full Newton-Raphson procedure, in which the stiffness matrix is updated at every equilibrium iteration. In addition, it generates and uses unsymmetric matrices that you can use for any of the following:
  - If you are defining an unsymmetric material model, you would need this method to fully use the property you defined.
  - If you are running a contact physics solution, an unsymmetric contact stiffness matrix would fully couple the sliding and the normal stiffnesses. You should first try the Full method, then try this method only if you experience convergence difficulties. (Using an unsymmetric solver requires more computational time to obtain a solution than if using a symmetric solver.)

For more information, see [and](#) in the [.](#)

3.  Additionally, you can select a unit system for the solver from **Solver unit system**.

The selection of a consistent unit system guarantees that all quantities, inputs and outputs to the solver can be interpreted correctly in terms of the units in the system.

You can leave the default setting, **Active Unit System**, or choose one of the standard unit systems defined in the **Unit Systems** window. When you leave the default, **Active Unit System**, the active unit system chosen in the **Unit Systems** window is used. Select **File icon > Units** to confirm that the units displayed are what you expect.

If you do not specify a unit system, in either the **Unit Systems** window or here in the solver settings, AIM selects one of six possible standard unit systems and converts all quantities into that system. The standard unit systems include Metric and the Consistent systems (CGC, NMM, uMKS, BIN, BFT); see the **Unit Systems** window for a listing.

**Note:** The **Initial interface treatment** property is only applicable to non-linear contacts.

## 4.9.4.2. Setting Solver Launch Options for an Electric Conduction Solution

### Physics > Solver Options > Launch Controls

Launch controls allow you to control the computational resources used by a simulation. The launch controls for the solver are populated with default settings, file names, and values. Review these settings and ensure that the default entries are acceptable for your model.

To change the default launch control settings:

1. For **Number of processors**, enter the desired number of processors.
2.  Additionally, you can modify the following settings:

If you want to...	then set...	More information:
Run the solver live or in batch mode.	<b>Solver mode</b>	With <b>Live</b> , the solver runs in RPC mode. It is run as a server and clients may communicate to the solver while it is running. The solver can also run in <b>Batch</b> mode, which is typical of running from a command line. In this mode, AIM is not communicating directly with the solver, but communicates instead through input and output files.
Use a distributed solver.	<b>Distributed solve</b>	
Remove stale files from a previous solve at the next solve.	<b>Delete work files before solve</b>	
Modify a default file name.	The file name for any of the following: <ul style="list-style-type: none"><li>• <b>Input data file</b></li><li>• <b>Mesh data file</b></li><li>• <b>Component data file</b></li><li>• <b>Connection elements data file</b></li><li>• <b>Load elements data file</b></li><li>• <b>Output data file</b></li></ul>	Enter the name of file you want to use instead of the default.
Change the name of the job.	<b>Job name</b>	
Output the contact pair information to a file other than the default output file.	<b>Output contact pairs to Jobname.cnm</b>	

### 4.9.4.3. Controlling Output Data for an Electric Conduction Solution

#### Physics > Solver Options > Output Controls

The output specification set in the **Output Controls** panel determines how the solution data written is to the output file. By default, three output specifications are created for a steady-state electric conduction solution, with the frequency is set to All Time Points and the output types set as follows:

Output Type	Which Stores
Nodal DOF Solution	Voltage
Nodal Reaction Loads	Current Reaction
Element Solution	Current Density, Electric Field, and Joule Heat Generation

However, you can modify these output specifications or create additional Output Controls objects for the same location if you want to add other output types to the result file.

To set up an output specification:

1. Define the location within the model for which you want the solution information written in the output file.

The default is Everywhere(). You can also specify the AllBodies() expression for all the bodies defined in the Physics Region, or select one or more bodies from the model.

2. For Output type, select a solution type.

The options available are based on the physics and calculation types.

3.  For Frequency, select how often the solver should write the solutions results to the output file. Available options are:

- **All Time Points:** Writes the solution information for every sub step. This is the default option.
- **Last Time Points:** Writes the solution information only for the last sub step.
- **Equally Spaced Time Points:** Writes the solution information for up to a specified number of equally spaced solutions. This option only applies to Static or Transient analyses when the Automatic Time Stepping is enabled. For this option, specify a nonzero positive number for the equally spaced time points.
- **Specified Recurrence Rate:** Writes the solution information at every specified interval and at the last sub step of each load step. For this option, specify a nonzero positive number for the recurrence rate.
- **Never:** Suppresses writing of the specified solution type for all sub steps.

## 4.9.4.4. Setting Nonlinear Controls in an Electric Conduction Simulation

When solving nonlinear simulations, AIM carries out an iterative procedure (equilibrium iterations) at each substep, successfully solving the simulation only when the out-of-balance loads are less than the specified convergence criteria. On the **Solution Progression** panel, you can define the number of substeps in a solution step. You can then define the convergence controls and other solution progression properties.

For **Substepping**, you can define a range or a fixed number, or you can leave it up to the solver.

AIM displays the convergence criteria appropriate for each physics type. For a steady-state thermal physics solution, the convergence criteria consist of Current and Voltage convergence. You can set the convergence controls for each of these criteria.

 Under **Additional Convergence Controls**, you can also specify the number of equilibrium iterations at each substep, activate a predictor on the degree-of-freedom solution for the first equilibrium iteration of each substep, and control the line search program.

For **Stabilization**, you can specify the key for controlling nonlinear stabilization. Convergence difficulty due to an unstable problem is usually the result of a large displacement for small load increments. Nonlinear stabilization technique can help achieve convergence. Nonlinear stabilization can be thought of as adding artificial dampers to all of the nodes in the system. Any degree of freedom that tends to be unstable has a large displacement causing a large damping/stabilization force. This force reduces displacements at the degree of freedom so stabilization can be achieved.

### 4.9.4.4.1. Specifying Solution Progression for an Electric Conduction Simulation

**Physics > Solver Options > Solution Progression**

To set the controls for complete criteria:

- From **Substepping**, set time stepping to a range or a fixed number, or leave it to be controlled by the solver.

If substepping is controlled by the solver, a check is performed on non-convergent patterns, and the physics of the simulation is also taken into account. In case of specified range, the solver marches through the solution based on the initial, minimum, and maximum number of substeps.

- If you choose **Specified range**, specify the initial, minimum, and maximum number of substeps.
- If you choose **Fixed number**, specify the substep count.

- For each convergence type under **Convergence Controls**, specify how you want convergence handled.

- The recommended setting, **Program controlled**, automatically calculates the value based on external forces, including reactions.
- If set to **On**, you can input convergence values in the fields that appear.
- If set to **Off**, convergence is not checked.

- If a convergence type is set to **On**, do the following:

- For **Tolerance (%)**, specify the tolerance value as a percentage.

**Tolerance** times **Value** determines the convergence criterion.

- For **Reference value specification**, select if you want the solver to use a **Reference value** when establishing convergence, or have the solver control the convergence based on a **Minimum reference value**.

The **Minimum reference value** option is useful for analyses where the external forces tend to zero. This can happen, for example, with free thermal expansion where rigid body motion is prevented.

- Specify any additional desired criteria.

If you want to...	then set...	More information:
Activate a predictor in a nonlinear analysis on the degree-of-freedom solution for the first equilibrium iteration of each substep	<b>Predictor-corrector</b>	You can set this to: <ul style="list-style-type: none"> <li>Program controlled</li> <li>Off</li> <li>On after the first substep</li> <li>On for all substeps</li> </ul>
Specify the number of equilibrium iterations at each substep.	<b>Equilibrium iterations</b>	You can specify whether the number of equilibrium iterations carried out at each substep is: <ul style="list-style-type: none"> <li>Automatically set by the solver</li> <li>Set to a value of 1</li> <li>Set to a number to be specified by you in the <b>Number equilibrium iterations</b> field</li> </ul>
Control the line search program.	<b>Line search</b>	Line search can be useful for enhancing convergence, but it can be expensive (especially with plasticity). For more information, see <a href="#">Specifying Line Search for an Electric Conduction Simulation</a> on page 459.

If you want to...	then set...	More information:
Control nonlinear stabilization.	<b>Stabilization &gt; Stabilization key</b>	Nonlinear stabilization technique can help achieve convergence. For more information, see <a href="#">Specifying Nonlinear Stabilization Controls for an Electric Conduction Simulation</a> on page 458.

## 4.9.4.4.1.1. Specifying Nonlinear Stabilization Controls for an Electric Conduction Simulation

### Physics > Solver Options > Solution Progression > Stabilization

For **Stabilization**, you can specify the key for controlling nonlinear stabilization. Convergence difficulty due to an unstable problem is usually the result of a large displacement for small load increments. Nonlinear stabilization technique can help achieve convergence. Nonlinear stabilization can be thought of as adding artificial dampers to all of the nodes in the system. Any degree of freedom that tends to be unstable has a large displacement causing a large damping/stabilization force. This force reduces displacements at the degree of freedom so stabilization can be achieved.

- Under **Stabilization**, select the **Stabilization key**.

#### Option

<b>Constant energy dissipation</b>	Activate stabilization and use the energy dissipation ratio as the control. Then specify in the <b>Energy dissipation ratio</b> field the ratio of work done by stabilization forces to element potential energy. This value is usually a number between 0 and 1. The default value is 1.0e-4.
<b>Reduced energy dissipation</b>	Activate stabilization and reduce the energy dissipation ratio linearly from the specified or calculated value to zero at the end of the load step. Then specify in the <b>Energy dissipation ratio</b> field the ratio of work done by stabilization forces to element potential energy. This value is usually a number between 0 and 1. The default value is 1.0e-4.
<b>Constant damping factor</b>	Activate stabilization and use the damping factor as the control. Then specify in the <b>Damping factor</b> field the value that the solver uses to calculate stabilization forces for all subsequent substeps. This value is usually a value greater than 0. <b>Note:</b> The <b>Damping factor</b> value is dependent on the active unit system and may influence the results if unit systems are changed.
<b>Reduced damping factor</b>	Activate stabilization and reduce the damping factor linearly from the specified or calculated value to zero at the end of the load step. Then specify in the <b>Damping factor</b> field the value that the solver uses to calculate stabilization forces for all subsequent substeps. This value is usually a value greater than 0. <b>Note:</b> The <b>Damping factor</b> value is dependent on the active unit system and may influence the results if unit systems are changed.

- For **Substep option**, select an option for when stabilization is activated.

- No - Stabilization is not activated for the first substep even when it does not converge after the minimal allowed time increment is reached (default setting).
- Yes - Stabilization is activated for the first substep. Use this option if stabilization was active for the previous load step.
- If not converged - Stabilization is activated for the first substep if it still does not converge after the minimal allowed time increment is reached. Use this option for the first load step only.

3. For **Force limit**, specify the stabilization force check as a number between 0 and 1. The default value is 0.2. To omit a stabilization force check, set this value to 0.

## 4.9.4.4.1.2. Specifying Line Search for an Electric Conduction Simulation

### Physics > Solver Options > Solution Progression

Line search can be useful for enhancing convergence, but it can be expensive (especially with plasticity). You might consider setting line search on for the following cases:

- When your structure is force-loaded (as opposed to displacement-controlled).
- If you are analyzing a "flimsy" structure which exhibits increasing stiffness (such as a fishing pole).
- If you notice (from the program output messages) oscillatory convergence patterns.

You can set Line Search to:

- Program controlled
- On
- Off

## 4.9.5. Troubleshooting Structural, Thermal, and Electric Conduction Simulations

In some cases, your structural, thermal, or electric conduction simulation may not converge, or may be subject to element distortions, exaggerated distortions, disk space errors, or other issues. The following sections address recommendations for the following issues:

- Solver engine unable to converge
- Element distortions
- Large deformations in comparison to the model bounding box
- Insufficient disk space errors
- Errors when starting to solve
- Unable to find requested modes
- Internal solution magnitude limit exceeded
- Iterative solver required excessive number of iterations to converge
- Large deformation results active
- License manager server down

Additional information is also available in the [Contact Best Practices](#) on page 353 section.

## 4.9.5.1. Solver Unable to Converge

If the solver is unable to converge on a solution of a nonlinear problem, follow the recommendations below.

**Note:** Additional information is also available in the [Contact Best Practices](#) on page 353 section, particularly [Possible Remedies for Non-Convergence](#) on page 360.

- In a structural simulation, check for sufficient supports to prevent rigid body motion or that contact with other parts will prevent rigid motion..
- In a thermal simulation, check for thermal material curves or convection curves which rise and/or fall sharply over the temperature range.
- Check your [mesh quality](#).
- Make sure the model's units of measure are on an appropriate scale.
- Increase [substepping](#) before and during the onset of the divergence.
- If you encounter a convergence error during a thermal analysis that is using contact, consider modifying the [Thermal Conductance](#) property.
- Check that the loading is of a reasonable nature. Unlike linear problems whose results will scale linearly with the loading, nonlinear problems that include nonlinear contact may result in convergence problems if the loading is too big or small in a real world setting.

## 4.9.5.2. Element Distortions

Excessive distortion of elements is usually a symptom indicating the need for corrective action elsewhere.

- Try incrementing the load more slowly (increase the number of substeps or decrease the time step size).
- In the Mesh task, [change the setting for the Aggressive mechanical control](#) that will create better aspect ratios. This approach is applicable only if you are using manual meshing.
- Consider the behavior of materials and review the contact pairs.

## 4.9.5.3. Large Deformation

When the calculated nodal deformations are large compared to the solver limits, the model mechanics depart from linear behavior in response to the applied boundary conditions. In this case, review the load magnitudes, surface body thickness, and contact options, if applicable. If these conditions are intended, consider enabling a nonlinear analysis by turning on [Large Deflection](#) in the [Solver Settings](#).

## 4.9.5.4. Insufficient Disk Space

If an error occurs while solving due to insufficient disk space, try the following recommendations:

- Verify that there is sufficient free disk space on the drive where the solver directory resides. You may be running out of disk space due to writing large solution files. The location of the solver directory is listed in the [Launch Controls](#).
- Make sure you have write permissions to the solver directory.
- Try setting **Delete work files before solve** in the [Launch Controls](#), which ensures that stale files from a previous solve are removed when you solve again.

## 4.9.5.5. Error While Starting Solver

If the solver fails to start, try the following recommendations:

- Increase the allocation of virtual memory (total paging file size) on your system.
- Ensure you have enough disk space or move to an area where you have enough. You need enough disk space to support the increase in virtual memory and the temporary files that are created during the solve.
- Try de-installing and re-installing the product. You may have a corrupt product installation.
- Verify your license. Your license request may have been rejected.
- Verify the startup directory for `cmd.exe` hasn't been overridden by the `AUTORUN` option, which causes the solver to be unable to locate the solver input files.

## 4.9.5.6. Unable to Find Requested Modes

This message is issued during a modal analysis when a frequency search range is specified but no natural frequencies were found within the specified range. Consider [increasing the search range](#) or specifying that the first  $N$  frequencies be found.

## 4.9.5.7. Internal Solution Magnitude Limit Was Exceeded

In most cases this message occurs if your model is improperly constrained or if extremely large load magnitudes are applied relative to the model size. First, check that the applied loads and constraints are correct. In some cases, you may want loads that are self-equilibrating with no support. To help these cases, consider turning on the inertia relief.

For Thermal simulations, verify that your setup has at least one heat sink and one heat source as well as closed contact regions.

## 4.9.5.8. Large Number of Iterations Necessary for Convergence

If, when using an iterative solver, a large number of iterations were needed in order to get a converged answer, you may want to instead select a direct solver.

By default, the program will either choose a direct or iterative solver based on the analysis type and geometric properties. In general, thin models perform better with a direct solver while bulky models perform better with an iterative solver. However, sometimes the iterative solver is chosen when the direct solver would have performed better. In such cases, you may want to force the use of the direct solver.

## 4.9.5.9. Concerns with Active Large Deformation Effects

In a large deformation analysis, the solver updates the nodal coordinates as the solution progresses towards the final configuration. The displacement load is susceptible to large deformation effects. As a result, supports that fix only some of the degrees of freedom of a node may become invalid as the model's nodal coordinates, and thus nodal rotation angles, are updated. The imposed DOF displacement directions do not change even though rotation angles change. This may or may not be a desirable situation.

A classic example is a simple torsion of a rod. Initially the nodes at zero degrees have a circumferential direction of UY but after a twist of 90 degrees, have a circumferential direction of UX.

You'll have to determine if any nodal rotation at the support is significant enough to cause undesired results.

The displacement boundary condition is susceptible to large deformation effects.

## 4.9.5.10. License Manager Server Is Down

This message is issued in a one-server environment if your license manager has quit running when the server is down. In a three-license server environment, the ANSYS license manager must be running on at least two of the three license server machines at all times. If two of the license server machines go down, or two of the machines are not running the license manager, this error message will appear in the Messages box.

The program will continue to run for  $nn$  minutes to allow the license manager to be restarted or to be started on a second machine if using redundant servers. When the first message displays with  $nn = 60$ , the message

then reappears every five minutes with *nn* displaying the elapsed time at each 5 minute increment (45, 50, 55, etc) until the connection is established.

Start the license manager on the other machines designated as license servers. If you find that the license manager is still running, and it is in a one-server environment, then the IP address of the license server machine was changed while the application is running (this is usually caused by connecting to or disconnecting from an Internet Service Provider (ISP) that dynamically allocates IP addresses). To correct this situation, you must return the IP address to the same address that the license server had when the application was started. If the IP address changes after you start the application (either because you connected to or disconnected from your ISP), you can correct the error by restarting the application. You should not need to restart the license manager.

You can avoid this problem by remaining connected to or disconnected from the ISP the entire time you are running the application.

#### 4.9.6. User Commands for the MAPDL Solver

You can use the **User Command** feature to write user-defined commands to the Solver Settings file (input.dat).

Any command that you enter must be included in the proper section in order to execute properly. To make sure that the application does not overwrite your command entries, the application writes your entries to the solver file at the very end of the section (also called a “module”).

The solver runs all input file commands, including the ones that may have existed as a part of the regular solution process. Some commands require variables and/or parameters to be active for execution or to produce correct results.

Refer to the [User Commands Reference](#) for more specific information about command usage and specification.

### 4.9.6.1. Specifying User Commands Options

To enter a **User Command**:

1. On the **Physics** panel, for **Solvers Options**, select **Add > User Commands**.
2. Select one of the following sections under **Solver Command Sections**:

- **Begin**
- **/PREP7**
- **/SOLU Start**
- **Just Before Solve**
- **/SOLU End**
- **/POST1**

3. Enter commands in the right panel.

No right-click options are available in the entry field, but you can use keyboard shortcut options to copy (CTRL + C) and paste (CTRL + V) command content into the fields.

Command text cannot contain characters outside of the standard US ASCII character set because the text must follow the rules for the Mechanical APDL application commands and input files. Use of languages other than English for the command text may cause erratic behavior.

The application automatically updates the panel. It is always Up-To-Date.

4. Once your entries are complete, continue your physics solution. The **Solver Options** panel now contains a **User Commands** link.

**Caution:** The application does not perform any validation checks on your command entries.

## 4.9.6.2. User Commands Settings

Option	Description
Begin	These commands are prior to the preprocessing commands. It is often used to perform some type of initialization, such as defining the unit system.
/PREP7	These commands are used to create and set up the model. They are written prior to solution commands. This section often includes commands to change inputs such as element key options, nodal rotations, or coordinate systems.
/SOLU	These commands are written at the beginning of solution section and are used to load and solve the model.
Just Before Solve	These commands are written immediately before the solution process. It is a section where you may wish to make changes such as modifying a load or constraint value, or a solver setting, or an output control, or change a time step option.
/POST1	These commands are used to postprocess the results with the database processor.

## 4.9.7. Electromagnetic Solver Options

### Physics > Solver Options

When you create your physics solution using a template, a number of solver options are set up using standard defaults. However, it may sometimes be appropriate to fine-tune the following settings:

- Use the **Solver Settings** panel to tune the solution performance and control the degree of curved surfaces mesh refinement applied to the initial mesh.
- Use the **Launch Controls** panel to control the number of processors used by the solver.
- For frequency response designs, induced currents allow magnetic fields to penetrate conductors only to a certain depth, known as the skin depth. Due to the skin concentration of current, AC inductances and resistances are not equal to their DC equivalents. This affects the inductance and resistance values computed during impedance computations. To help improve accuracy in these regions, depending on the use case, you can add one or more **Skin Depth Resolution** panels to control the mesh resolution applied to selected locations.

Optionally, to aid in deriving an accurate solution, or deriving a converged solution more quickly, if the physics setup is valid, and if no initial mesh has yet been generated you can [generate an initial mesh](#) for viewing on a contour plot.

The solution quality monitor, which is displayed in the [Solution Monitor](#) tab, can also help with [judging convergence for electromagnetics solutions](#).

## 4.9.7.1. Launching the Solver on Multiple Processes

### Physics > Solver Options > Launch Controls

The **Launch Controls** data panel enables you to specify and control the computational resources used when solving the physics.

Set the desired number of processors in **Number of processors**. The number supported is determined by the number of AIM licenses available for your installation.

## 4.9.7.2. Controlling the Magnetics Solution

### Physics > Solver Options > Solver Settings

In the **Solver Settings** you can refine the settings that control the magnetics solution computations.

- Set the **Solution performance tuning** slider to one of the eight settings to control the number of passes the adaptive mesh refinement process can make to achieve a converged solution, or to achieve results of a desired accuracy.

Settings closer to **Accuracy** increase the number of adaptive passes and the accuracy of the solution, but at the cost of increased simulation time.

Settings closer to **Speed** decrease the number of solver passes, reducing simulation time at the cost of decreased accuracy.

- To control the maximum number of adaptive passes, set the **Maximum passes option** to **User defined**. Then, specify the **Maximum adaptive passes** for your simulation to run in order to achieve a converged solution, or to achieve results of a desired accuracy.
- To control the desired solution accuracy, set the **Error percentage option** to **User defined**. Then specify the **Error percentage**. Smaller values produce more accurate (but slower) solutions; larger values produce less accurate (but faster) solutions.

At each step in the adaptive process, the energy and error energy are computed, and you can visualize results even when convergence has not yet been achieved.

After the mesh is refined, the solution is calculated on the refined mesh. The relative change between the previous solution and the current solution is then computed using this specified error percentage. The energy difference is also calculated. A small delta indicates that further mesh refinement will probably not change the solution significantly.

- Use the **Curved surface meshing** slider to control the degree of curved surface mesh refinement applied to the initial mesh. The finer the setting, the better the mesh approximates curved surfaces, but the longer it takes to generate the initial mesh.

-  5. Optionally:

If you want to...	Then set...	Additional information
Control the nonlinear residuals	<b>Nonlinear residuals option &gt; User defined</b>	Then specify the <b>Nonlinear residuals</b> value, which is used as stopping criteria for the iterative process of approximating nonlinear materials.
Control the mesh refinement	<b>Mesh refinement option &gt; User defined</b>	Then specify the <b>Mesh Refinement</b> value, which determines how many tetrahedra are added at each iteration of the adaptive refinement process. The tetrahedra with the highest error are refined.

### 4.9.7.3. Specifying Skin Depth Resolution

**Skin Depth Resolution** is available for Physics Regions with the Physics type set to Electromagnetics and the Physics Calculation type set to Frequency response.

**Physics > Solver Options > Skin Depth Resolution**

You can add a **Skin Depth Resolution** physics option to generate a mesh that's more dense for selected locations near the surfaces of conductors, where Eddy current effects are more significant, thus avoiding the need to generate a very large, dense mesh for all surfaces.

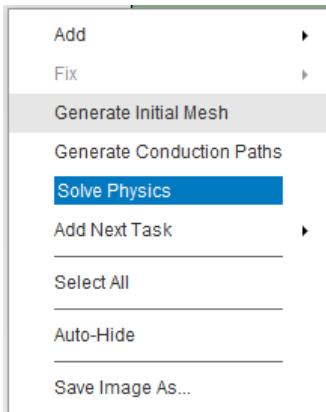
1. Select one or more faces for the **Surface location** for which you want to specify a skin depth resolution.
2. Set the **Resolution** slider to one of the eight settings to control the initial mesh resolution for the selected location.

In general, the finer the setting, the denser the initial mesh will be, and the more accurate the solution, but at the expense of increased computation time.

## 4.9.7.4. Generating an Initial Mesh for Viewing on a Contour Plot

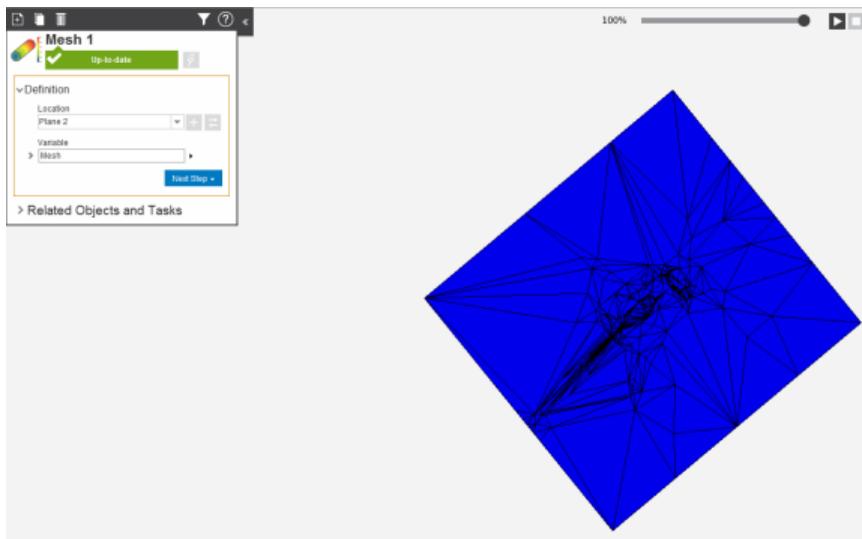
Electromagnetics designs - both static and frequency response - are solved using an adaptive iterative solution method. When you **Solve Physics**, the solver first generates an initial mesh. This mesh is then used by the solver to perform the adaptive iterative solution using the settings chosen on the **Solver Settings** panel.

Optionally, to aid in deriving an accurate solution, or deriving a converged solution more quickly, if the physics setup is valid, and if no initial mesh has yet been generated (**Solve Physics** has not been done), you can right-click anywhere on the model and select **Generate Initial Mesh** to instruct the solver to generate just the initial mesh without completing the simulation (the Physics task will indicate that the solution is only partially updated).



**Note:** **Generate Initial Mesh** is no longer available in the menu after the initial mesh has been generated.

After generating the initial mesh, you can create a Contour plot of the mesh similar to the example below to see if the mesh quality at the area of interest is as desired. If not, it can be further refined by applying **Skin Depth Resolution** settings at locations where you are expecting significant eddy current.



## 4.9.7.5. Judging Convergence for Electromagnetics Solutions

Electromagnetics designs are solved using an adaptive iterative solution method. When an adaptive analysis is performed:

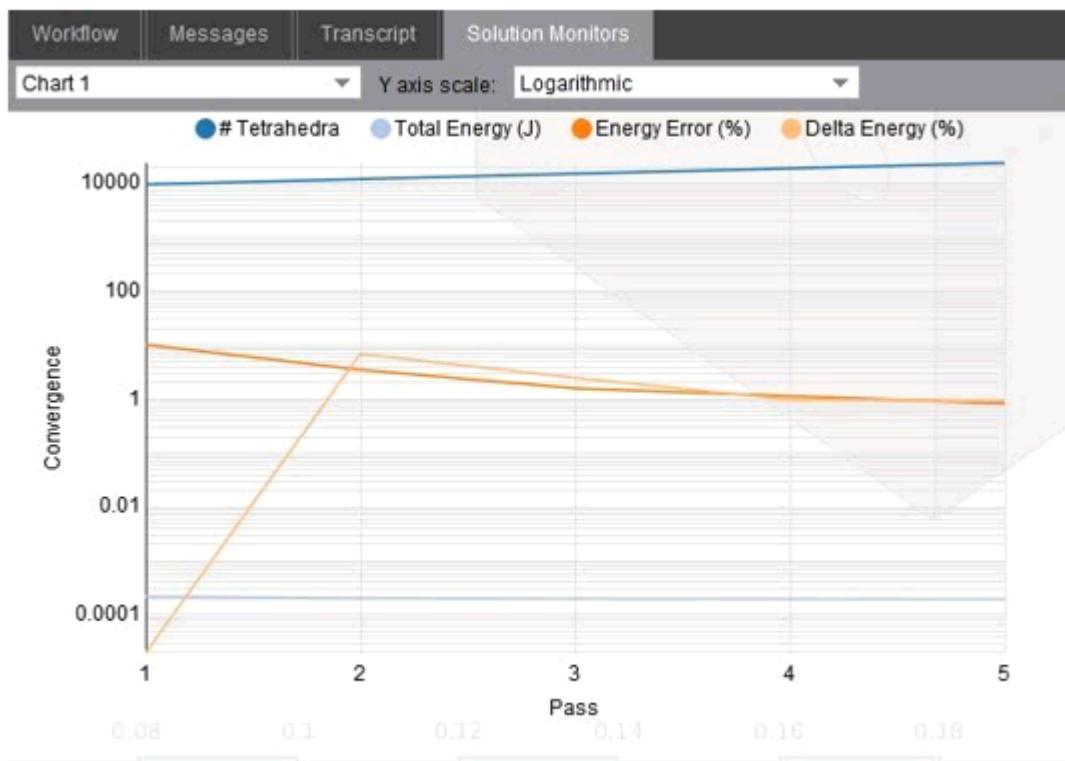
- The solver generates a field solution using the initial mesh, which is composed of tetrahedral elements.
- The solver then analyzes the accuracy of the solution by calculating an energy value based on the error in the solution. The exact mechanism for evaluating the error varies by solution type. For example, in a magnetostatic design the solver calculates the current density and then subtracts all input currents and other sources. For a perfect solution the result would be zero. However, for a real, finite mesh the result is some amount of residual current density. The energy value calculated from this residual current density is called the error energy. The **Energy Error %** is the error energy as a percentage of the **Total Energy (J)** (calculated with the original sources).
- If more than one pass has been completed, the solver also calculates the change in total energy from the previous pass. The percentage difference is the **Delta Energy (%)**.
- When the error targets are not satisfied, the mesh is refined. This is generally done by subdividing the mesh elements (**Tetrahedra**) with the highest error energy into smaller elements. The solver then generates a new field solution.
- This adaptive refinement process continues either until convergence is achieved (both the **Energy Error %** and the **Delta Energy (%)** are below the target error), or until the solver reaches the maximum number of adaptive passes requested.

The **Solution performance tuning** slider on the **Solver Settings** panel determines how strict the convergence criteria are. The slider controls the maximum number of adaptive passes the solver can make to achieve a converged solution, and also controls the target error goal. The solution process stops (convergence is achieved) when both the **Energy Error %** and the **Delta Energy (%)** fall below the goal. Setting the slider to **Speed** decreases the number of solver passes, and reduces simulation time, but at the cost of decreased accuracy. Setting the slider to **Accuracy** increases the number of adaptive passes, and increases the accuracy of the solution, but at the cost of increased simulation time.

The primary tool for judging convergence is the solution quality monitor, which is displayed in the **Solution Monitor** tab. By default, a chart is created for you with a set of standard quantities to monitor. For Electromagnetics designs, the monitor tracks the values of four parameters during the solution iterations - **Tetrahedra**, **Total Energy**, **Energy Error (%)**, and **Delta Energy (%)**. Iterations are terminated when the

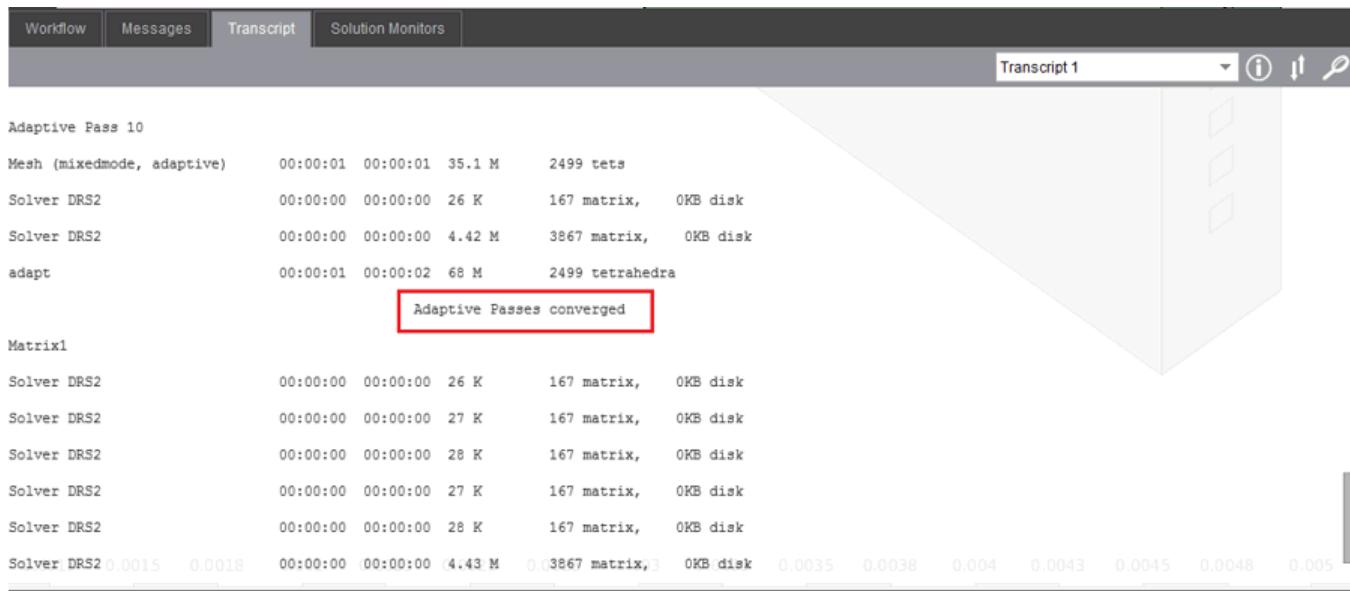
convergence target has been reached, or when the maximum number of iterations have been performed. A solution that has converged is illustrated in the figure below.

**Figure 4.9.7.5.1. Example of Multiple Physics Regions and a Region Interface**



You can also judge convergence using the **Transcript** tab, which shows the process in numerical format, and also indicates whether or not a solution has converged.

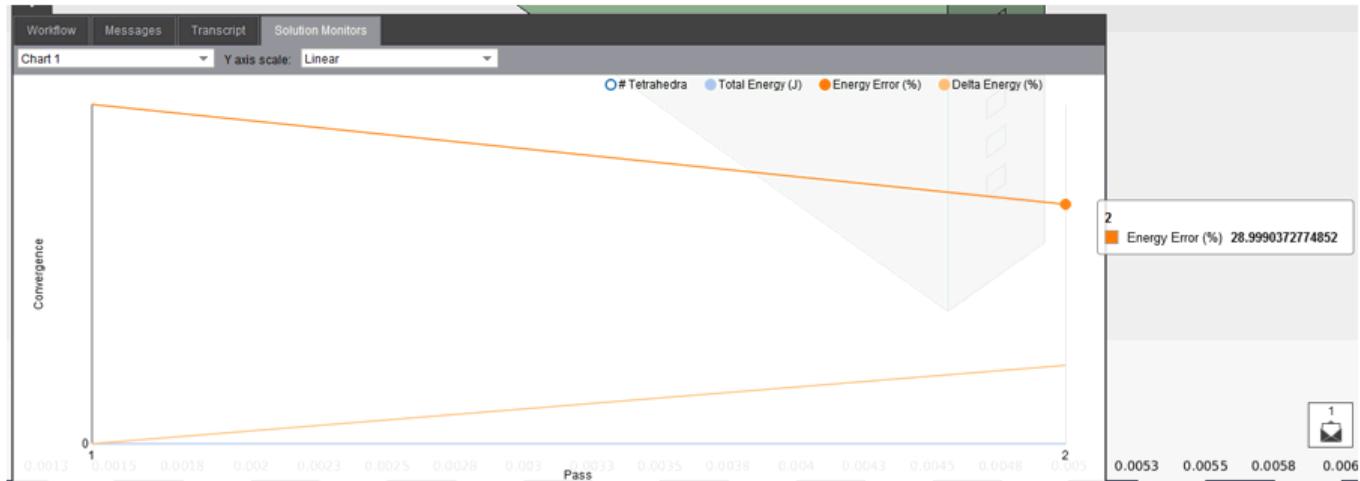
**Figure 4.9.7.5.2. Transcript Displaying Results for a Converged Solution**



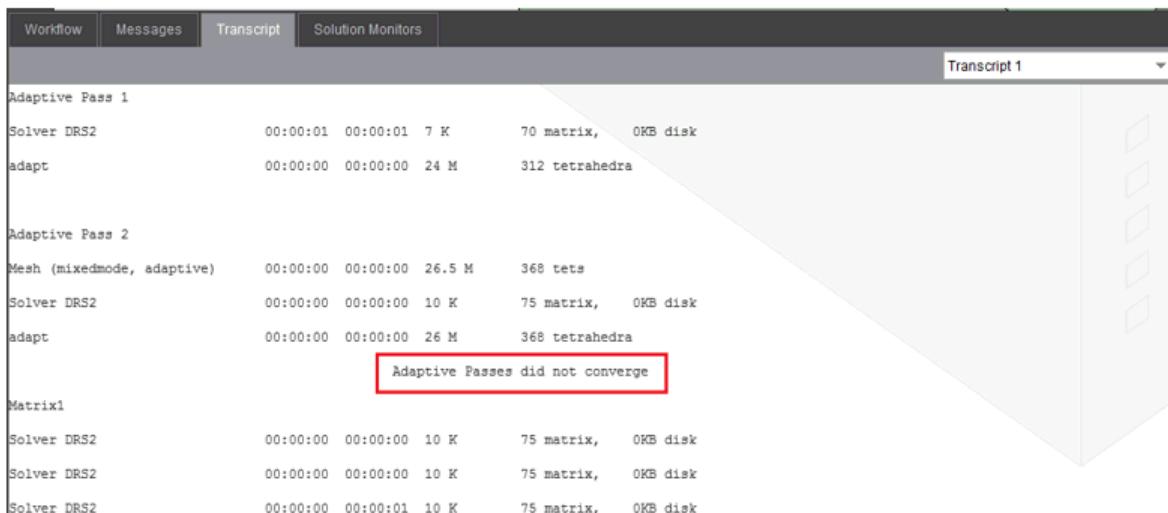
In some cases, convergence is not achieved within the specified number of iterations. When a solution does not converge, a warning message is displayed in the **Messages** tab. The Transcript also displays a non-convergence message. Follow the recommendations below if a solution fails to converge.

#### Energy Error (%) and Delta Energy (%) are decreasing but do not reach target value

**Figure 4.9.7.5.3. Results Decreasing But Not Reaching Target**



**Figure 4.9.7.5.4. Transcript Displaying Results for a Non-Converged Solution**



- **Recommendation:** The **Solution performance tuning** slider on the **Solver Settings** panel determines how strict the criteria are. When it's set to **Speed**, the criteria are looser. When it's set to **Accuracy**, the criteria are higher.



For non-converged cases, you can try:

- adjusting the accuracy level of the **Solution performance tuning** slider.
- increasing the number of adaptive passes, by changing the **Maximum passes option** to **User defined**, and specifying a greater number of passes.
- increasing the **Curved surface meshing** slider to make it finer.
- For frequency response designs only, in addition to the above suggestions, you can try increasing **Skin Depth Resolution**.
- Solve the physics task again and the solution will be re-calculated with the new settings.
- Repeat these steps until the solution has converged.

#### 4.9.8. Electrostatic Solver Options

**Physics > Solver Options**

When you create your physics solution using a template, a number of solver options are set up using standard defaults. You can modify the default values for a given physics solution on the **Solver Settings**, **Launch Controls**, **Output Controls**, **Solution Progression**, **Boundary Refinement**, and **Fidelity Refinement** panels.

## Setting General Solver Options

On the **Solver Settings** panel, the settings available define the solver, such as the solver unit system and solver type. For a nonlinear simulation, you can set the global nonlinear options for all solution steps.

## Specifying Launch Controls

On the **Launch Controls** panel, you can control how the solver is launched and how the solver data is managed. Properties include solver file locations, file names, and distributed solve controls.

## Output Controls

In the **Output Controls** panel, you can control how the solution data is written to the output file by defining one or more output specifications, which set the solution location, output type, and frequency. These output specifications are processed sequentially based on the order they are listed in the Output Controls panel.

By default, three output specifications are created for an electrostatic physics solution:

- Nodal DOF Solution (Voltage)
- Nodal Reaction Loads (Charge Reaction)

- Nodal Element Solution (Electric Field and Electric Potential)

If you want to add other output types to the result file, you can modify these output specifications or create additional Output Controls for the same location.

## Setting Nonlinear Options

You can apply global nonlinear controls for all solution steps, such as the Newton-Raphson setting, on the **Solver Settings** panel. On the **Solution Progression** panel, you can apply nonlinear settings that are applied per solution step.

## Specifying Boundary Refinement

[Boundary refinement](#) provides an optional input that lets you refine the boundary mesh generated by automatic physics-aware meshing.

## Specifying Solution Fidelity Refinement

[Fidelity refinement](#) provides an optional input that lets you identify localized geometric details of a specific size that you want to capture in the solution. This enables you to increase the accuracy of the solution while optimizing solution time by setting the [global solution fidelity](#) to a lower setting, and then applying fidelity refinement to specific topologies only.

### 4.9.8.1. Configuring the Solver for an Electrostatic Solution

#### Physics > Solver Options > Solver Settings

Default solver settings are generated when you set up your physics regions, however you may want to modify some of the settings.

To change the default solver settings:

**1. For Equation solver type**, select the solver type.

The default is **Program controlled**, which lets the program choose the type of solver appropriate for your model. However, you can explicitly select one of the following:

- **Direct**: More effective with thin flexible models.
- **Iterative**: Best for bulky models.

**2. For Newton Raphson method**, select the method you want to use.

- **Full**: The solver uses the full Newton-Raphson procedure, in which the stiffness matrix is updated at every equilibrium iteration.

- **Modified**: The solver uses the modified Newton-Raphson technique, in which the tangent stiffness matrix is updated at each substep. The matrix is not changed during equilibrium iterations at a substep. This option is not applicable to large-deformation analyses. Adaptive descent is not available.

- **Initial stiffness**: The solver uses the initial stiffness matrix in every equilibrium iteration. This option can be less likely to diverge than the full option, but it often requires more iterations to achieve convergence. It is not applicable to large-deformation analyses. Adaptive descent is not available.

- **Unsymmetrical**: The program uses the full Newton-Raphson procedure, in which the stiffness matrix is updated at every equilibrium iteration. In addition, it generates and uses unsymmetric matrices that you can use for any of the following:

- If you are defining an unsymmetric material model, you would need this method to fully use the property you defined.
- If you are running a contact physics solution, an unsymmetric contact stiffness matrix would fully couple the sliding and the normal stiffnesses. You should first try the Full method, then try this method only if you experience convergence difficulties. (Using an unsymmetric solver requires more computational time to obtain a solution than if using a symmetric solver.)

For more information, see [and](#) in the [.](#)

-  3. Additionally, you can select a unit system for the solver from **Solver unit system**.

The selection of a consistent unit system guarantees that all quantities, inputs and outputs to the solver can be interpreted correctly in terms of the units in the system.

You can leave the default setting, **Active Unit System**, or choose one of the standard unit systems defined in the **Unit Systems** window. When you leave the default, **Active Unit System**, the active unit system chosen in the **Unit Systems** window is used. Select **File icon > Units** to confirm that the units displayed are what you expect.

If you do not specify a unit system, in either the **Unit Systems** window or here in the solver settings, AIM selects one of six possible standard unit systems and converts all quantities into that system. The standard unit systems include Metric and the Consistent systems (CGC, NMM, uMKS, BIN, BFT); see the **Unit Systems** window for a listing.

**Note:** The **Initial interface treatment** property is only applicable to non-linear contacts.

## 4.9.8.2. Setting Solver Launch Options for an Electrostatic Solution

### Physics > Solver Options > Launch Controls

Launch controls allow you to control the computational resources used by a simulation. The launch controls for the solver are populated with default settings, file names, and values. Review these settings and ensure that the default entries are acceptable for your model.

To change the default launch control settings:

1. For **Number of processors**, enter the desired number of processors.

-  2. Additionally, you can modify the following settings:

If you want to...	then set...	More information:
Run the solver live or in batch mode.	<b>Solver mode</b>	With <b>Live</b> , the solver runs in RPC mode. It is run as a server and clients may communicate to the solver while it is running. The solver can also run in <b>Batch</b> mode, which is typical of running from a command line. In this mode, AIM is not communicating directly with the solver, but communicates instead through input and output files.
Use a distributed solver.	<b>Distributed solve</b>	

If you want to...	then set...	More information:
Remove stale files from a previous solve at the next solve.	<b>Delete work files before solve</b>	
Modify a default file name.	The file name for any of the following: <ul style="list-style-type: none"><li>• <b>Input data file</b></li><li>• <b>Mesh data file</b></li><li>• <b>Component data file</b></li><li>• <b>Connection elements data file</b></li><li>• <b>Load elements data file</b></li><li>• <b>Output data file</b></li></ul>	Enter the name of file you want to use instead of the default.
Change the name of the job.	<b>Job name</b>	
Output the contact pair information to a file other than the default output file.	<b>Output contact pairs to Jobname.cnm</b>	

### 4.9.8.3. Controlling Output Data for an Electrostatic Solution

#### Physics > Solver Options > Output Controls

The output specification set in the **Output Controls** panel determines how the solution data written is to the output file. By default, three output specifications are created for a steady-state electric conduction solution, with the frequency is set to All Time Points and the output types set as follows:

Output Type	Which Stores
Nodal DOF Solution	Voltage
Nodal Reaction Loads	Charge Reaction
Element Solution	Electric Field and Electric Potential

However, you can modify these output specifications or create additional Output Controls objects for the same location if you want to add other output types to the result file.

To set up an output specification:

1. Define the location within the model for which you want the solution information written in the output file.

The default is `Everywhere()`. You can also specify the `AllBodies()` expression for all the bodies defined in the Physics Region, or select one or more bodies from the model.

2. For **Output type**, select a solution type.

The options available are based on the physics and calculation types.

3.  For **Frequency**, select how often the solver should write the solutions results to the output file. Available options are:

- **All Time Points:** Writes the solution information for every sub step. This is the default option.
- **Last Time Points:** Writes the solution information only for the last sub step.

- **Equally Spaced Time Points:** Writes the solution information for up to a specified number of equally spaced solutions. This option only applies to Static or Transient analyses when the Automatic Time Stepping is enabled. For this option, specify a nonzero positive number for the equally spaced time points.
- **Specified Recurrence Rate:** Writes the solution information at every specified interval and at the last sub step of each load step. For this option, specify a nonzero positive number for the recurrence rate.
- **Never:** Suppresses writing of the specified solution type for all sub steps.

## 4.9.8.4. Setting Nonlinear Controls in an Electrostatic Simulation

When solving nonlinear simulations, AIM carries out an iterative procedure (equilibrium iterations) at each substep, successfully solving the simulation only when the out-of-balance loads are less than the specified convergence criteria. On the **Solution Progression** panel, you can define the number of substeps in a solution step. You can then define the convergence controls and other solution progression properties.

For **Substepping**, you can define a range or a fixed number, or you can leave it up to the solver.

AIM displays the convergence criteria appropriate for each physics type. For a steady-state thermal physics solution, the convergence criteria consist of Current and Voltage convergence. You can set the convergence controls for each of these criteria.

 Under **Additional Convergence Controls**, you can also specify the number of equilibrium iterations at each substep, activate a predictor on the degree-of-freedom solution for the first equilibrium iteration of each substep, and control the line search program.

For **Stabilization**, you can specify the key for controlling nonlinear stabilization. Convergence difficulty due to an unstable problem is usually the result of a large displacement for small load increments. Nonlinear stabilization technique can help achieve convergence. Nonlinear stabilization can be thought of as adding artificial dampers to all of the nodes in the system. Any degree of freedom that tends to be unstable has a large displacement causing a large damping/stabilization force. This force reduces displacements at the degree of freedom so stabilization can be achieved.

### 4.9.8.4.1. Specifying Solution Progression for an Electrostatic Simulation

#### Physics > Solver Options > Solution Progression

To set the controls for complete criteria:

1. From **Substepping**, set time stepping to a range or a fixed number, or leave it to be controlled by the solver.

If substepping is controlled by the solver, a check is performed on non-convergent patterns, and the physics of the simulation is also taken into account. In case of specified range, the solver marches through the solution based on the initial, minimum, and maximum number of substeps.

- If you choose **Specified range**, specify the initial, minimum, and maximum number of substeps.
- If you choose **Fixed number**, specify the substep count.

2. For each convergence type under **Convergence Controls**, specify how you want convergence handled.
  - The recommended setting, **Program controlled**, automatically calculates the value based on external forces, including reactions.
  - If set to **On**, you can input convergence values in the fields that appear.

- If set to **Off**, convergence is not checked.
3. If a convergence type is set to **On**, do the following:
- a) For **Tolerance (%)**, specify the tolerance value as a percentage.  
**Tolerance** times **Value** determines the convergence criterion.
  - b) For **Reference value specification**, select if you want the solver to use a **Reference value** when establishing convergence, or have the solver control the convergence based on a **Minimum reference value**.  
The **Minimum reference value** option is useful for analyses where the external forces tend to zero. This can happen, for example, with free thermal expansion where rigid body motion is prevented.
4. Specify any additional desired criteria.

If you want to...	then set...	More information:
Activate a predictor in a nonlinear analysis on the degree-of-freedom solution for the first equilibrium iteration of each substep	<b>Predictor-corrector</b>	You can set this to: <ul style="list-style-type: none"> <li>• Program controlled</li> <li>• Off</li> <li>• On after the first substep</li> <li>• On for all substeps</li> </ul>
Specify the number of equilibrium iterations at each substep.	<b>Equilibrium iterations</b>	You can specify whether the number of equilibrium iterations carried out at each substep is: <ul style="list-style-type: none"> <li>• Automatically set by the solver</li> <li>• Set to a value of 1</li> <li>• Set to a number to be specified by you in the <b>Number equilibrium iterations</b> field</li> </ul>
Control the line search program.	<b>Line search</b>	Line search can be useful for enhancing convergence, but it can be expensive (especially with plasticity). For more information, see <a href="#">Specifying Line Search for an Electric Conduction Simulation</a> on page 459.
Control nonlinear stabilization.	<b>Stabilization &gt; Stabilization key</b>	Nonlinear stabilization technique can help achieve convergence. For more information, see <a href="#">Specifying Nonlinear Stabilization Controls for an Electric Conduction Simulation</a> on page 458.

#### 4.9.8.4.1.1. Specifying Line Search for an Electrostatic Simulation

##### Physics > Solver Options > Solution Progression

Line search can be useful for enhancing convergence, but it can be expensive (especially with plasticity). You might consider setting line search on for the following cases:

- When your structure is force-loaded (as opposed to displacement-controlled).

- If you are analyzing a "flimsy" structure which exhibits increasing stiffness (such as a fishing pole).
- If you notice (from the program output messages) oscillatory convergence patterns.

You can set Line Search to:

- Program controlled
- On
- Off

## 4.9.8.4.1.2. Specifying Nonlinear Stabilization Controls for an Electrostatic Simulation

### Physics > Solver Options > Solution Progression > Stabilization

For **Stabilization**, you can specify the key for controlling nonlinear stabilization. Convergence difficulty due to an unstable problem is usually the result of a large displacement for small load increments. Nonlinear stabilization technique can help achieve convergence. Nonlinear stabilization can be thought of as adding artificial dampers to all of the nodes in the system. Any degree of freedom that tends to be unstable has a large displacement causing a large damping/stabilization force. This force reduces displacements at the degree of freedom so stabilization can be achieved.

1. Under **Stabilization**, select the **Stabilization key**.

#### Option

##### Constant energy dissipation

Activate stabilization and use the energy dissipation ratio as the control.

Then specify in the **Energy dissipation ratio** field the ratio of work done by stabilization forces to element potential energy. This value is usually a number between 0 and 1. The default value is 1.0e-4.

##### Reduced energy dissipation

Activate stabilization and reduce the energy dissipation ratio linearly from the specified or calculated value to zero at the end of the load step.

Then specify in the **Energy dissipation ratio** field the ratio of work done by stabilization forces to element potential energy. This value is usually a number between 0 and 1. The default value is 1.0e-4.

##### Constant damping factor

Activate stabilization and use the damping factor as the control.

Then specify in the **Damping factor** field the value that the solver uses to calculate stabilization forces for all subsequent substeps. This value is usually a value greater than 0.

**Note:** The **Damping factor** value is dependent on the active unit system and may influence the results if unit systems are changed.

##### Reduced damping factor

Activate stabilization and reduce the damping factor linearly from the specified or calculated value to zero at the end of the load step.

Then specify in the **Damping factor** field the value that the solver uses to calculate stabilization forces for all subsequent substeps. This value is usually a value greater than 0.

**Note:** The **Damping factor** value is dependent on the active unit system and may influence the results if unit systems are changed.

2. For **Substep option**, select an option for when stabilization is activated.

- No - Stabilization is not activated for the first substep even when it does not converge after the minimal allowed time increment is reached (default setting).
  - Yes - Stabilization is activated for the first substep. Use this option if stabilization was active for the previous load step.
  - If not converged - Stabilization is activated for the first substep if it still does not converge after the minimal allowed time increment is reached. Use this option for the first load step only.
3. For **Force limit**, specify the stabilization force check as a number between 0 and 1. The default value is 0.2. To omit a stabilization force check, set this value to 0.

#### 4.9.9. Topology Optimization Solver Options

##### Optimize > Solver Options

When you create your topology optimization using the template, a number of solver options are set up using standard defaults.

The **Solver Settings** are distinctive to topology optimization, and are discussed in this section. For more information on the **Launch Controls**, **Output Controls**, and **Solution Progression** panels, see [Structural Solver Options](#) on page 430.

### 4.9.9.1. Configuring the Solver for a Topology Optimization

#### Physics > Solver Options > Solver Settings

Default solver settings are generated when you set up your physics regions, however you may want to modify some of the settings.

To change the default solver settings:

1. Specify the **Maximum number of iterations** performed for the topology optimization analysis.  
The solution process continues until the application achieves convergence accuracy or reaches the maximum number of iterations.
2. Set the **Convergence accuracy** of the topology optimization analysis by moving the slider or entering a value.  
The solution process continues until the application achieves convergence accuracy or reaches the maximum number of iterations. This value must be greater than 0% but not greater than 2%.
3. For **Equation solver type**, select the solver type.

The default is **Program controlled**, which lets the program choose the type of solver appropriate for your model. However, you can explicitly select one of the following:

- **Direct**: More effective with thin flexible models.
- **Iterative**: Best for bulky models.

4.  Additionally, you can modify the following options:

If you want to...	Then set	More information
Specify the optimization method	<b>Optimization method</b>	<p>By default, AIM chooses the optimization method appropriate for your model. However, you can also select <b>Sequential convex programming</b> or <b>Optimality criteria</b>.</p> <p>The <b>Sequential convex programming</b> method is an extension of the method for moving asymptotes (MMA). This method requires the derivatives of all functions present in the topology optimization analysis.</p> <p>The <b>Optimality criteria</b> method can be used to solve topology optimization problems with a simple compliance objective that uses a volume or mass constraint.</p>
Specify the minimum normalized density to view more or fewer poor elements	<b>Minimum normalized density</b>	Enter a threshold value between zero and 0.125 (default) below which you want elements removed. The purpose of this setting is to remove the bad element represented by poor element densities (between 0-0.125). You can further reduce this value if you want to see more of poor elements.
Specify the maximum number of files created during solve	<b>Max number of intermediate files</b>	This setting caps the number of files produced by the solver during the solve.

If you want to...	Then set	More information
Select a unit system for the solver	<b>Solver unit system</b>	<p>The selection of a consistent unit system guarantees that all quantities, inputs and outputs to the solver can be interpreted correctly in terms of the units in the system.</p> <p>You can leave the default setting, <b>Active Unit System</b>, or choose one of the standard unit systems defined in the <b>Unit Systems</b> window. When you leave the default, <b>Active Unit System</b>, the active unit system chosen in the <b>Unit Systems</b> window is used. Select <b>File icon &gt; Units</b> to confirm that the units displayed are what you expect.</p> <p>If you do not specify a unit system, in either the <b>Unit Systems</b> window or here in the solver settings, AIM selects one of six possible standard unit systems and converts all quantities into that system. The standard unit systems include Metric and the Consistent systems (CGC, NMM, uMKS, BIN, BFT); see the <b>Unit Systems</b> window for a listing.</p>

#### 4.9.10. Polymer Extrusion Solver Options

##### Extrusion > Physics Solution > Solver Options

When you create your physics solution using a template, a number of solver options are set up using standard defaults.

- Use the **Launch Controls** panel to [specify launch controls](#) and manage the computational resources used by the polymer extrusion solver.
- Use the **Numerical Controls** panel to [set numerical controls](#) and manage how the polymer extrusion solver interpolates numerical values.
- Use the **Solution Controls** panel to [set solution controls](#) and manage convergence and divergence criteria and the numerical progression of the polymer extrusion simulation.
- Use the **Output Controls** to [set output controls](#) and indicate specifics for the output of the polymer extrusion simulation.
- Use the **Boundary Refinement** panel to [refine the boundary mesh](#) generated by automatic physics-aware meshing.
- Use the **Fidelity Refinement** panel to [identify localized geometric details](#) of a specific size that you want to capture in the solution.

## 4.9.10.1. Specifying Launch Controls for the Polymer Solver

### Extrusion/Blow Molding > Physics Solution > Solver Options > Launch Controls

Launch controls allow you to control the computational resources used by a simulation. The launch controls for the solver are populated with default settings and values.

To change the default launch control settings:

1. Modify the **Number of processes** if you want to run on more or less than 2 processor cores.  
Using more than 2 processor cores requires additional AIM (or HPC) licenses.
2.  (optional) Use the **Solver arguments** field to provide command-line arguments to the polymer extrusion or blow molding solver. For example, to achieve greater graphics performance you can enter a command that will [activate the GPU accelerator](#). ANSYS Customer Support can provide more recommendations when using this field.
3.  (optional) Use the **Solver keywords** field to pass options to the polymer extrusion or blow molding solver. Using this field is not recommended unless suggested by ANSYS Customer Support.

## 4.9.10.1.1. Activating the GPU Accelerator Capability

### Extrusion/Blow Molding > Physics Solution > Solver Options > Launch Controls

Activating the GPU accelerator capability is achieved through a command line argument.

1. Before activating the GPU accelerator capability, you must have at least one supported GPU card with the proper driver level and proper HPC licensing.
2. In the **Launch Controls** panel, enter the following command in the **Solver arguments** field:

```
-acc nvidia -na N
```

The `-na` command line option followed by a number ( $N$ ) indicates the number of GPU accelerator devices to use for the simulation. If only the `-acc` option is specified, the program uses a single GPU device for the simulation by default (that is, `-na 1`). You may specify a value for  $N$  that is greater than 1.

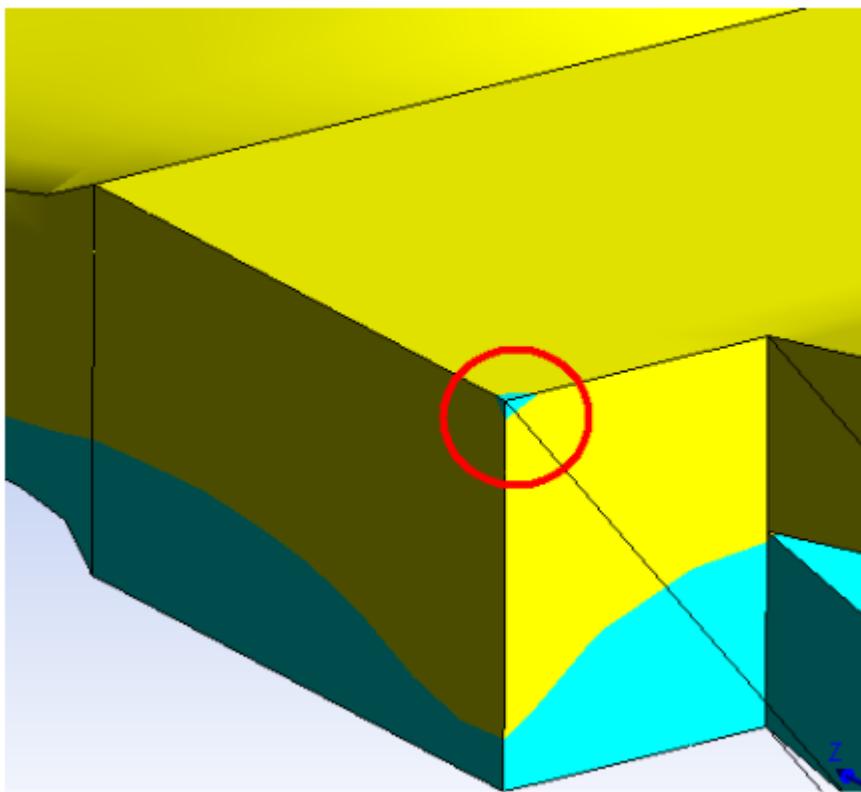
With the GPU accelerator capability, the acceleration obtained by using the parallelism on the GPU hardware occurs only during the solution operations. Operational randomness and numerical round-off inherent to any parallel algorithm can cause slightly different results between runs on the same machine when using or not using the GPU hardware to accelerate the simulation.

## 4.9.10.2. Setting Numerical Controls for Polymer Extrusion Flows

### Extrusion > Physics Solution > Solver Options > Numerical Controls

Numerical controls allow you to control how the polymer extrusion [solver](#) interpolates values for velocity, pressure, and temperature. The numerical controls for the solver are populated with default settings and values.

1. Activate **Picard iterations** to help convergence when the selected nonconstant viscosity model (such as the [Generalized Newtonian fluid](#)) exhibits important variations. Picard iterations reduce the convergence rate but are more robust than the default pure Newton-Raphson iterations.
  
2.  The pressure stabilization is required with linear interpolation of velocity. When the velocity under **Interpolation** is defined as being nonlinear, the **Pressure stabilization** option can be activated to smooth the pressure over the elements by introducing a slight compressibility of the fluid. This compressibility is controlled by the **Relative compression factor**. A factor that is too small will introduce pressure peaks. If the factor is too large, the fluid is too compressible and all pressure gradients will be smoothed out, leading to an unrealistically low pressure prediction. The default value of 0.01 has been shown to be the best choice for this factor.
  
3.  The **Fluid fraction diffusivity** option (in the co-extrusion of multiple materials) allows you to have more control over the artificial diffusivity used for the transport of fluid fractions, in order to avoid small distortions that may appear when displaying fluid fraction results.



4. When **restrictors** are used in polymer extrusion, an **Overlap threshold** needs to be defined. The overlap quantity is a representation of the number of elements that are connected to a specific node that actually reside in the restrictor. The overlap quantity varies between 0 and 1. If the overlap quantity lies above the overlap threshold, then the node is considered to be inside the restrictor and its velocity is set to 0. If the overlap quantity is below the threshold, then the node lies in the flow domain, where the governing Navier-Stokes equations will be solved.
  
5. Under **Interpolation**, control the [numerical interpolation](#) as needed by selecting an appropriate option for **Velocity and pressure** and **Pressure at slip wall**.

## 4.9.10.2.1. Numerical Interpolation for Polymer Extrusion Flows

The polymer extrusion solver in AIM uses a series of finite element velocity-pressure interpolation functions where linear, quadratic, and/or mini-elements representation of the velocity field are combined with a constant, linear, and/or discontinuous representations of the pressure field.

Table 4.9.10.2.1.1. Velocity and Pressure Interpolation Options

Velocity/Pressure Interpolation Option	Description	Information
<b>Linear velocity, constant pressure</b>	A linear representation of the velocity field with a constant pressure in each element.	The linear representation is a computationally inexpensive method that gives satisfactory results. In some configurations it also leads to smoother velocity profiles compared to the mini-element representation, which is especially important in nonisothermal cases.  To avoid spurious pressure modes, the pressure is stabilized by an artificial compressibility term.
<b>Mini-element velocity, constant pressure</b>	A mini-elements representation of the velocity field with a constant pressure for each element.	The mini-element is implemented as a correction to the linear element: a degree of freedom is introduced on the mid-faces of the elements, but only for the normal part of the velocity vector, that better satisfies the conservation of mass at the level of the element and allows you to skip the pressure stabilization.  The cost of the mini-element is similar to that of the simpler linear element.
<b>Mini-element velocity, linear pressure</b>	A mini-elements representation of the velocity field, with a linear representation of the pressure for each element.	While the mini-element approach provides more accuracy for 3D flows, this option avoids the mini-element low pressure accuracy and greatly improves the pressure representation. This uses a "bubble" velocity shape functions at element centers.

Velocity/Pressure Interpolation Option	Description	Information
<b>Quadratic velocity, linear discontinuous pressure</b>	A discontinuous representation for the pressure, which then becomes a first-order polynomial in each element.	May be more accurate for free surface or thermal convection problems. However, using the quadratic representation on a dense 3D finite-element mesh can be computationally expensive and require a significant amount of memory.
<b>Quadratic velocity, linear pressure</b>	A quadratic representation of the velocity field and a linear, continuous representation of the pressure field.	Most accurate and the most expensive of all representations.

For non-isothermal flows, several types of interpolation are available to calculate the temperature field, based on the Péclet number that is the ratio of the advection (or transport) of heat by the flow to the diffusion (or conduction) of heat:

Table 4.9.10.2.1.2. Temperature Interpolation Options

Temperature Option	Description	Information
Linear	Use linear elements (the default) for temperature instead of the default quadratic elements.	Useful for low Péclet-number simulations limits the CPU time and problem size.
Quadratic	Use quadratic elements for the temperature.	Use when the Péclet number is low.
2x2	Use a linear 2x2 arrangement of sub-elements.	Use for moderate Péclet numbers.

When slipping conditions are involved, the solver uses a pressure field along the slipping wall to enforce the zero normal velocity condition. This pressure field has a specific interpolation. The **Program controlled** option is recommended unless unphysical results are obtained, in which case, you should try other interpolations and check your results.

Table 4.9.10.2.1.3. Pressure Field at Slip Wall Interpolation Options

Pressure at Slip Wall Option	Description
<b>Constant per element</b>	The interpolation is constant at the level of the elements.
<b>Same as pressure field</b>	The interpolation is the same as the pressure field.
<b>Same as velocity field</b>	The interpolation is the same as the velocity field.
<b>Program controlled</b>	Allows the solver to select the interpolation according to the interpolation of the velocity field and slipping parameters (the default).

## 4.9.10.3. Setting Solution Controls for Polymer Extrusion Flows

### Extrusion > Physics Solution > Solver Options > Solution Controls

The **Solution Controls** panel allows you to manage convergence and divergence criteria and the numerical progression of the polymer extrusion simulation. The solution controls for the solver are populated with default settings and values.

To change the default solution control settings:

1. If you need to increase or decrease the total number of iterations required for the solution, make the appropriate changes to the **Maximum number of iterations** field.
2.  Use the **Convergence criterion** and **Divergence criterion** fields to better [control the convergence of the solution](#).
3.  For **Solver unit system**, if you require the solution to be provided in metric units, specify **Metric (kg, m, s, C)** rather than the default value of **Consistent CGS (g, cm, s, C)**.
4.  For **Solver type**, indicate which [linear solver](#) will be used to resolve the system of equations.
5.  Use the **Convergence Strategies** and the **Progression Controls** fields to better [control the progression of the solution](#). If you do not require the use of a convergence strategy, then, under **Progression Controls**, disable the **Use substepping** option.

### 4.9.10.3.1. Convergence Controls for Polymer Flows

Convergence of your polymer simulation can be managed using several different approaches:

Convergence option	Description
<b>Maximum number of iterations</b>	Used to increase or decrease the acceptable number of iterations required for the solution.
 <b>Convergence criterion</b>	Controls when the convergence to a stable solution is assumed. You can let AIM control the convergence by setting this option to <b>Program controlled</b> .
 <b>Convergence tolerance</b>	The convergence tolerance is based on relative error. For each type of field, the modification at every node between two successive iterations is compared to the maximum value of the field at the current iteration. You can control the convergence yourself by setting the <b>Convergence criteria</b> option to <b>User-specified</b> and set the <b>Convergence tolerance</b> to something other than the default value of 0.001.
 <b>Divergence criterion</b>	As the solution diverges, errors rapidly increase with each iteration. Since there is usually no hope of convergence when the relative error has become too great, you can let AIM control the divergence by setting this option to <b>Program controlled</b> .

Convergence option	Description
 <b>Divergence tolerance</b>	<p>The maximum value for the relative error at which AIM stops the iterations (but not the evolution or time-marching scheme).</p> <p>You can control the divergence yourself by setting the <b>Divergence criteria</b> option to <b>User-specified</b> and set the <b>Divergence tolerance</b> to something other than the default value of 1000.</p>

## 4.9.10.3.2. Solver Options for Polymer Flows

 In the **Solution Controls** panel, for **Solver type**, you can indicate which polymer solver is available for your simulation.

For polymer application problems, the basic problem that AIM must solve at a given time step or iteration is a linear algebraic system of the following form:

$$Ax = b$$

where the matrix A is typically large, sparse, and asymmetric.

Solver type	Description
<b>Classic direct solver</b>	A frontal, direct method to solve the linear algebraic system of equations based on Gaussian elimination. Generally the slowest approach and should be used only when troubles are encountered with the AMF option.
<b>AMF direct solver</b>	The default solver is an algebraic multi-frontal (AMF) direct solver (rather than mesh-based) where the critical variable ordering is based on the assembled equations, having the advantage of taking into account more sparsity in many systems of equations (for example, in multi-mode viscoelastic flows where stress modes are uncoupled).
<b>AMF direct solver and secant</b>	A "modified Newton" iterative process that takes advantage of the system regularity between iterations in order to factorize the system matrix only from time to time. This technique is known to work in most problems with potentially significant savings (as the CPU-dominant system solution step is performed less frequently and system assembly can be done at a reduced cost). In very rare cases, however, the secant iteration may direct iterations along a wrong path, therefore, this option is not the default. Note that this technique requires slightly more disk space for storing the factors than is the case for the pure Newton-Raphson scheme, so do not expect memory savings from this technique.
	Allows faster iterations (recommended when <a href="#">using the Picard scheme</a> ).

### 4.9.10.3.3. Convergence Strategies and Solution Progression

The computation of polymer extrusion fluid dynamics is intrinsically non-linear.

Typically, the solution of a nonlinear problem is surrounded by a domain of convergence. This means that the solver cannot reach the final solution if the final solution is too far away from the starting solution. The starting point being, in most cases, where all fields are equal to zero apart from coordinates that come from the mesh and the initialized temperature. When the final solution is not close enough to the starting solution, the strategy is to calculate an initial solution that is close to the starting solution, a second solution close to the first solution, a third solution close to the second solution, and so on, up to the final solution. Practically, this means that you modify parameters (such as material data, fluid flow condition, etc.) starting from an easy solution and you progressively change them back to their nominal values.

The key is to isolate a parameter that is largely responsible for the nonlinearity, and to devise a series of solutions based on the parameter and a defined substep. If the initial solution converges, then a new solution is devised with a larger substep size, and the process continues. If a solution diverges, then the previous solution is revised with a smaller substep, and the process continues until the final solution is achieved.

For example, polymers have a low heat conductivity and an important heat capacity. This leads to high temperature gradients in a polymer flow. A classical strategy is to start with a high thermal conductivity leading to a much more uniform temperature and to progressively decrease it down to its nominal value.

In AIM, the sub-stepping ("evolution") implements this progressive approach of the nominal solution. The substepping can be tuned using the **Progression Controls** in the **Solution Controls** panel.

Table 4.9.10.3.3.1. Using the Progression Controls

If you want to...	then ...	Information
Change the starting value for the size of the substep.	Specify a value for the <b>Initial substep size</b> parameter.	
Change the lower bound for the size of the substep.	Specify a value for <b>Minimum substep size</b> parameter.	If the substep size is too large, the problem obtained will not converge. When this happens, the substep size will be decreased, and a new problem tried. To prevent an infinite loop, you should select a lower bound for the substep size here. If this lower bound is reached, the evolution process is terminated.
Change the upper bound for the size of the substep.	Specify a value for the <b>Maximum substep size</b> parameter.	
Change the upper bound on the number of steps used to reach the original problem.	Specify a value for the <b>Maximum number of substeps</b> parameter.	This number allows you to limit the total CPU time spent on the simulation.

Time-dependent flow problems can be represented using the following approximation:

$$\mathbf{X}_{n+1} = \mathbf{X}_n + \Delta t_n \dot{\mathbf{X}} = \mathbf{X}_n + \Delta t_n (\theta f(\mathbf{X}_{n+1}) + (1 - \theta) f(\mathbf{X}_n))$$

where  $\mathbf{X}$  is the vector of nodal unknowns such as velocity, pressure, temperature, viscoelastic extra stresses, and free surface location, the subscript  $n$  refers to the time step, and different values of  $\theta$  result in different integration methods with different accuracy and stability attributes.

For polymer extrusion flows, AIM has several integration schemes available, each with varying degrees of precision.

In an evolution or substepping process, one uses a pseudo-time. The same integration schemes can be used even if the derivatives with respect to the pseudo-time have no physical meaning.

The default integration scheme is **Implicit Euler** that usually, due to the prediction of the solution at the following step, requires slightly fewer substeps than the **Zero order** scheme.

Table 4.9.10.3.3.2. Evolution Integration Schemes

Integration Scheme	$\theta$	Precision
<b>Zero order</b>	$\theta = 0$	$O(\Delta t)$
<b>Crank-Nicolson</b>	$\theta = \frac{1}{2}$	$O(\Delta t^2)$
<b>Galerkin</b>	$\theta = \frac{2}{3}$	$O(\Delta t)$
<b>Implicit Euler</b>	$\theta = 1$	$O(\Delta t)$

It is important to identify the parameters that govern any nonlinearities in the simulation. For polymer extrusion flows, AIM has several predefined strategies choices:

Table 4.9.10.3.3.3. Predefined Evolution/Convergence Strategies Available in AIM

Evolution/Convergence Strategies	Description
<b>Free surfaces</b>	For each <b>Free Surface</b> condition, the solver progressively switches from a no force condition to an actual free surface.  Recommended when at least one free surface is defined.  Any results that are available <a href="#">if the solve step fails</a> are not quite physical but give an idea of the way the free surface is deforming.
<b>Viscosity</b>	The solver applies numerical techniques to cope with complex viscosity models.  Recommended when the fluid model is not Newtonian.  Any results that are available <a href="#">if the solve step fails</a> are valid and physical but correspond to different viscosity model, closer to a Newtonian model than the nominal viscosity law.
<b>Thermal effects</b>	The solver progressively switches from a quasi-isothermal setup to a defined non-isothermal setup.  Recommended for non-isothermal setup.

Evolution/Convergence Strategies	Description
<b>Friction</b>	For each <b>Wall</b> condition with an imposed partial slip, the solver progressively switches from a full slip condition to the desired wall defined with a partial slip condition.
	Any results that are available <a href="#">if the solve step fails</a> are valid and physical but correspond to a lower value of the friction coefficient, to a flow with more slipping than prescribed in the setup.
<b>Multiple materials</b>	Viscosity parameters are progressively modified (from the same Newtonian model for all fluids to the nominal parameters of each specific material).
	Recommended for co-extrusion setups.
 <b>Flow at inlet</b>	For each <b>Inlet</b> condition with an imposed flow rate or normal velocity, the solver progressively increases the flow rate or normal velocity at the inlet.
	Any results that are available <a href="#">if the solve step fails</a> are valid and physical but correspond to a lower value of the flow rate.
 <b>Flow at outlet</b>	For each <b>Outlet</b> condition with an imposed flow rate, the solver progressively increases the flow rate at the outlet.
	Any results that are available <a href="#">if the solve step fails</a> are valid and physical but correspond to a lower value of the flow rate.
 <b>Take-up at extrudate exit</b>	For each <b>Extrudate exit</b> with an imposed take-up force or velocity, the solver progressively increases the take-up force or velocity at the exit.
	Any results that are available <a href="#">if the solve step fails</a> are valid and physical but correspond to a lower value of the take-up force (per unit area) or velocity. Those results give a valuable indication of the way the extrudate deforms and allow to diagnose mesh (deformation) issue.

#### 4.9.10.3.4. Troubleshooting Solution Convergence

In some polymer extrusion cases, a solution substep may not converge. In most cases, the results are still valid, however, they correspond to the previous substep (as described in [Convergence Strategies and Solution Progression](#) on page 485). If a solution substep does not converge, follow the recommendations below.

Possible Cause	Recommendation
The selected viscosity model hampers convergence.	Activate the convergence strategy on viscosity in the <b>Solution Controls</b> panel ( <b>Convergence Strategies &gt; Viscosity</b> ).

Possible Cause	Recommendation
The convergence strategy on viscosity does not reach the end of sub-stepping.	Activate <b>Picard iterations</b> in the <b>Numerical Controls</b> panel.
When the flow rate is imposed at inlet(s) and outlet(s), calculation errors may lead to mass balance issues.	Replace the flow rate or the velocity condition at the outlet with a condition on pressure.
Incorrect boundary conditions.	Check and correct boundary conditions.
The mesh is too coarse or of bad quality.	Improve the mesh using the Meshing task.
Inlet/outlet where the flow rate or the velocity is imposed is not perpendicular to the adjacent boundary.	Check that the <b>Allow non-zero tangential velocity</b> option is enabled for the inlet/outlet.
The mesh is too distorted.	Try to design a mesh that can better cope with the deformation.  You can also locate the <code>bad_mesh.stl</code> file (found within your AIM working project's <a href="#">folder</a> ) and load it into your CAD system to re-create the geometry and the mesh on the deformed configuration.
The deformation of the extrudate or the die is too great due to an unbalanced velocity profile at the die exit.	Improve the balance of the flow by modifying the geometry of the feeding channels.

## 4.9.10.4. Setting Output Controls for Polymer Extrusion Flows

### Extrusion > Physics Solution > Solver Options > Output Controls

Output controls allow you to manage particular aspects of the simulation's output. For instance, you can determine how detailed the solution transcript should be, or what types of quantities are available to use when analyzing the results of the simulation.

To change the default output control settings:

1. Decrease the **Transcript verbosity** if you want to include less details in the solution transcript.
2. Under **Derived Quantities**, select an appropriate option that you would like to have available when analyzing the results of the polymer extrusion simulation.

Derived quantity	Description
<b>Viscous heating</b>	The heat generation per unit time and per unit volume.
<b>Stress tensor</b>	The total extra-stress tensor in the domain, composed of the sum of the viscous (or the (generalized) Newtonian) components and the viscoelastic components.

3.  If you require additional analysis of your polymer simulation results, there are several output types available under **More Outputs**:

- **STL** (StereoLithography): this output file (.stl) contains the facets describing the boundaries of the domains of the geometry, such that each boundary of each region corresponds to a STL solid.
- **CFD-Post**: this output file (cfx.res) contains the geometry and the computed fields for use with ANSYS CFD-Post.
- **Polyflow**: these output files (res.msh, res) contain the mesh and the computed fields for use with ANSYS Polyflow.
- **CSV**: this output file (csv) is a common format for tabulated data that can be read into spreadsheet programs such as Excel. It can also be read in ANSYS Polyflow for initial or boundary conditions.
- **EnSight**: these output files (.case and several other files) contain the geometry and computed fields for use with EnSight.
- **FieldView**: this output file (.uns) contains the geometry and the computed fields for use with FieldView.

#### 4.9.11. Polymer Blow Molding Solver Options

**Blow Molding > Physics Solution > Solver Options**

When you create your physics solution using a template, a number of solver options are set up using standard defaults.

- Use the **Launch Controls** panel to [specify launch controls](#) and manage the computational resources used by the polymer blow molding solver.
- Use the **Numerical Controls** panel to define the thickness interpolation. In simulations that use the shell element, only the linear interpolation scheme is available for the temperature, velocity, and position variables. The constant interpolation scheme (**Constant per face**) is enabled by default for the thickness variable.
- Use the **Solution Controls** panel to [set solution controls](#) and manage convergence and divergence criteria and the numerical progression of the polymer blow molding simulation.
- Use the **Output Controls** panel to control how frequently the computed data is stored. Select **Low**, **Medium** or **High** for recommended output intervals. A higher frequency will consume more computational resource when it comes to viewing results, but will produce higher quality output. Alternatively, you can choose to store data at a certain **Time interval** or at a specified number of **Time steps**.
- Use the **Boundary Refinement** panel to [refine the boundary mesh](#) generated by automatic physics-aware meshing.
- Use the **Fidelity Refinement** panel to [identify localized geometric details](#) of a specific size that you want to capture in the solution.

#### 4.9.11.1. Setting Solution Controls for Polymer Blow Molding Flows

**Blow Molding > Physics Solution > Solver Options > Solution Controls**

The **Solution Controls** panel allows you to manage convergence and divergence criteria and the numerical progression of the polymer blow molding simulation. The solution controls for the solver are populated with default settings and values.

To change the default solution control settings:

1. If you need to increase or decrease the total number of iterations required for the solution, make the appropriate changes to the **Maximum number of iterations** field.
2.  Use the **Convergence criterion** and **Divergence criterion** fields to better [control the convergence of the solution](#).
3.  For **Solver unit system**, if you require the solution to be provided in metric units, specify **Metric (kg, m, s, C)** rather than the default value of **Consistent CGS (g, cm, s, C)**.

4.  For **Solver type**, indicate which [linear solver](#) will be used to resolve the system of equations.
5. **Adaptive remeshing techniques** are used to refine the mesh and improve mesh quality and contact accuracy. You can exclude **Adaptive remeshing** to keep the same mesh throughout, however, mesh quality and contact accuracy may be compromised.
6. Use the **Progression Controls** field to better control the [progression of the solution](#).

## 4.9.11.1.1. Adaptive Remeshing

In a transient simulation, such as blow molding, it is possible for the mesh-quality criteria to change over time, such that a mesh that was acceptable at the start of the simulation is no longer adequate later on in the simulation. To address these issues, enable **Adaptive remeshing** in the **Solution Controls** panel and adjust the settings or define how you want to refine the mesh, as described in the table below.

**Note:** When using adaptive remeshing, the mesh is progressively refined according to the parison to mold distance, which decreases during inflation. As the results are being displayed, you will not see this progressive refinement because all intermediate results are re-mapped on the most refined mesh (the latest one).

Option	Description
<b>Adaptive remeshing frequency</b>	Specifies the frequency of the meshing. A value of 1 indicates that adaptive meshing will be performed at each time step. A value of 5 is usually sufficient for contact and remeshing simulations. A value below 3 is not recommended.
<b>Maximum number of adaptive passes</b>	This is the maximum number of times an element may be subdivided. The subdivision of a quadrilateral leads to 4 elements having a quarter of the area of the original element. If a quadrilateral element undergoes 3 subdivisions, the area of the new elements will be 64 times smaller.
<b>Adapt initial mesh</b>	For blow molding and thermoforming simulations that involve both contact and adaptive meshing, you can request that an additional adaptive meshing step is performed before the start of the time-dependent simulation. This allows you to start with a mesh that is coherent with the defined mesh quality criteria.
<b>Permit non-conformal meshing</b>	Allows the mesh to contain both conformal and non-conformal elements. A conformal mesh eliminates all the constraints that are needed to guarantee the continuity of the fields for mid-segment nodes and is recommended.
<b>Mesh refinement</b>	The refinement of the mesh can be done automatically ( <b>Program controlled</b> ) or by specifying the parameters to be used in the mesh refinement ( <b>User defined</b> ). To refine the mesh manually, you can modify the following: <ul style="list-style-type: none"> <li>• The <b>Minimum size</b> and <b>Maximum size</b> specifications represent, respectively, the global minimum and global maximum allowable element size.</li> <li>• The <b>Threshold distance</b> is the distance to mold under which refinement takes place. As long as an element of the parison is farther away, it is not subdivided.</li> <li>• The <b>Maximum gap to mold</b> is the final gap between the mold and the parison that you can accept. This gap dictates the size of an element of the parison; for a given gap, a large radius of curvature of the mold leads to large elements in the parison, while a small radius of curvature leads to small elements.</li> </ul>

## 4.9.11.1.2. Solution Progression

To control the manner in which the solution progresses, the time step can be fine-tuned as described in the table below.

Option	Description
<b>Initial time step</b>	This sets the initial time increment. The first time step can be recalculated with a smaller time increment if the calculation fails.
<b>Minimum time step</b>	This sets the minimum value of time increment. The time increment varies according to convergence and/or accuracy of the transient scheme. The calculation stops if the time increment goes below the minimum time step.
<b>Maximum time step</b>	This sets the maximum value of time increment. The time increment increases when convergence and accuracy are achieved. The upper limit of the time increment is the maximum time step.
<b>Maximum number of time steps</b>	This sets the number of time steps to complete the simulation. The calculation will stop as soon as this number is reached.
<b>Tolerance</b>	The default value of 0.01 is acceptable for many cases. A smaller value will result in a smaller time step size, higher transient accuracy, and a longer computation time.
 <b>Integration scheme</b>	Several integration schemes are available, each with varying degrees of precision. <b>Implicit Euler</b> is the default integration scheme because of its better stability.

## 4.9.11.2. Setting Output Controls for Polymer Blow Molding

### Blow Molding > Physics Solution > Solver Options > Output Controls

Output controls allow you to control how frequently the computed data is stored.

To change the default output control settings:

1. For **Output frequency**, select **Low**, **Medium** or **High** for recommended output intervals. A higher frequency will consume more computational resource when it comes to viewing results, but will produce higher quality output. Alternatively, you can choose to store data at a certain **Time interval** or at a specified **Steps interval**.
2. Decrease the **Transcript verbosity** if you want to include less details in the solution transcript.
3.  If you require additional analysis of your polymer simulation results, there are several output types available under **More Outputs**:
  - **STL** (StereoLithography): this output file (.stl) contains the facets describing the boundaries of the domains of the geometry, such that each boundary of each region corresponds to a STL solid.
  - **CFD-Post**: this output file (cfx.res) contains the geometry and the computed fields for use with ANSYS CFD-Post.
  - **Polyflow**: these output files (res.msh, res) contain the mesh and the computed fields for use with ANSYS Polyflow.
  - **CSV**: this output file (csv) is a common format for tabulated data that can be read into spreadsheet programs such as Excel. It can also be read in ANSYS Polyflow for initial or boundary conditions.

- **EnSight**: these output files (`.case` and several other files) contain the geometry and computed fields for use with EnSight.
- **FieldView**: this output file (`.uns`) contains the geometry and the computed fields for use with FieldView.

#### 4.9.12. Boundary Refinement

Boundary refinement is an optional input that enables you to refine the boundary mesh generated by automatic physics-aware meshing.

A typical application for boundary refinement is when you've used automatic physics-aware meshing and the mesh fails due to the automatic boundary layer specification. In other cases, the mesh may be successful but the number of boundary layers is inappropriate for the physics of the problem, such as a conjugate heat transfer or compressible flow problem.

In these cases, you can use boundary refinement [options](#) not only to control the number of boundary layers but also to provide additional fidelity control for automatic physics-aware meshing.

### 4.9.12.1. Specifying Boundary Refinement

**Physics > Solver Options > Boundary Refinement**

Boundary refinement is an optional input that enables you to refine the boundary mesh generated by automatic physics-aware meshing.

1. Select the location in your geometry that you want to refine.

You may select faces on which to create boundary layers, or enable **Automatically defined** to create boundary layers on all faces except those in Selection Sets.

2. For **Define by**, choose a method for defining the boundary layers.

3. Depending on the selected definition method, specify other [options](#) to further refine the boundary layers.

#### 4.9.13. Solution Fidelity Refinement

Fidelity refinement is an optional input that enables you to increase the fidelity of the solution by [capturing localized geometric details](#) of the model that may be missed by the [global solution fidelity setting](#).

A typical application for fidelity refinement is when you've used automatic physics-aware meshing, and the mesh that was generated automatically did not provide enough focus on an area of interest in the model.

Typically, the use of fidelity refinement will optimize the solution time, because it lets you set the global solution fidelity to a lower setting, and then apply fidelity refinement only on specific topologies to capture finer details locally.

In addition, if you set global solution fidelity to its highest setting and the desired geometric details are still not being captured, you can apply fidelity refinement to very specific areas of the model where you want to capture even finer details for greater solution accuracy. The more details that are captured, the longer the solution will take, so you should specify a value that captures only those details that are important to your design.

When you specify fidelity refinement, you identify a **Location** and set a **Detail capture** value. At a minimum, AIM refines the solution to capture geometric details of the size you've specified for **Detail capture**. However, in some cases, it may capture even smaller details, based on the global **Solution fidelity** setting or to ensure a stable solution.

Internally, AIM generates a mapping of **Fidelity Refinement** objects to mesh **Sizing** controls. Fidelity refinement is not available if you are defining the mesh manually. If you opt to define the mesh manually, you can use mesh **Sizing** controls instead of fidelity refinement. If you define fidelity refinement and

subsequently switch to manual meshing, a Mesh task is inserted into the workflow and the **Fidelity Refinement** objects are replaced by **Sizing** controls in the Mesh task.

## 4.9.13.1. Specifying Solution Fidelity Refinement

### Physics > Solver Options > Fidelity Refinement

Fidelity refinement enables you to control the fineness of the detail capture and solution accuracy locally.

1. Select the location in your geometry that you want to refine.

You may select a topology, or use an expression or selection set to specify the location. The location may consist of multiple entities, but they must be of a single type (that is, all edges, all faces, or all bodies).

2. For **Detail capture**, specify a length to define the size of the geometric details you want the solution to capture in the refinement area.

For example, to capture 1 mm details, specify 1 [mm].

## 4.9.13.2. Solution Fidelity Refinement Considerations

The following are important considerations for using fidelity refinement.

- At a minimum, AIM refines the solution to capture geometric details of the size you've specified for **Detail capture**. However, in some cases, it may capture even smaller details, based on the global **Solution fidelity** setting or to ensure a stable solution.
- If more than one **Fidelity Refinement** object is applied to the same topology (whether the topology is an edge, face, or body), the smallest value specified for **Detail capture** is respected.
- If you define fidelity refinement and subsequently switch to manual meshing mode, a Mesh task is inserted into the workflow and the **Fidelity Refinement** objects are replaced by **Sizing** controls in the Mesh task. You can delete the **Fidelity Refinement** objects, or retain them in case you later decide to delete the Mesh task and return to automatic physics-aware meshing.

## 4.10. Simulation Steps

You can specify multiple steps for a structural simulation.



Expand **Multi-Step Simulation Settings** under **Settings** in the Physics panel. Click to open the **Simulation Step Manager**. This button also appears in the toolbar of conditions that appear in the Simulation Steps Manager.

The Simulation Steps tab is shown. From here you can view the initial simulation step(s) set up by default, and define additional simulation steps.

### 4.10.1. Defining Simulation Steps

Your solution can consist of multiple steps that specify a different status for each condition (forces, pressures, supports, etc.) in the project. There is one default simulation step defined on the **Simulation Steps** tab. To set up your simulation steps:

1. Click **Add Step** to create a new simulation step. The new step will be inserted with the status of each condition set the same as in the last step. Add as many steps as needed.
2. Set the status of each condition in each step by toggling the icon for that condition.

-  (default for spatial conditions) activates the condition.
-  (default for numerical conditions) applies a constant factor to the condition. Enter a **Factor** value for this option. Factor defaults to 1.

**Note:** For a temperature quantity, the factor is applied to the temperature value in degrees K regardless of the units displayed, and a negative factor is invalid.

**Note:** If you change the factor for a condition, you will see a button in the condition's panel next to the output field, indicating that there is a factor other than 1 applied to the output during the simulation. Click



to see a chart showing the value of the output for each simulation step.

-  suppresses the condition.
-  locks the condition (only valid for certain conditions).

- If you need to delete steps, select the step(s) you need to delete and click the trash can icon that appears on the toolbar above the table, then confirm the deletion when prompted.
- If you need to insert a step between existing steps, select a step then click **Insert Step Before** or **Add Step After**. The new step will be inserted with the status of each condition set the same as in the selected step.

#### 4.10.2. Defining Steps using Status Groups

You can group conditions and define condition statuses for those groups, rather than setting up the condition statuses for each step. To do this, you must first select  from the right side of the toolbar, and then designate:

- Condition groups** - define groups of conditions that will be changed simultaneously during simulation steps.
- Status groups** - define different combinations of statuses for each condition group.
- Simulation steps - set the status group used for each condition group in each step, as shown below.

To define your simulation steps:

- Click **Add Step** as many times as needed to create your simulation steps.
- The **Substeps** field contains a link that reads either Program Controlled or shows the number of user-defined steps. Click the link to edit the associated **Simulation Step** panel, which specifies the solution progression used to define the step control and convergence controls. By default, each simulation step will be associated with the first solution progression.
- All of the defined condition groups are listed. You can specify the status group used for each condition group during each simulation step. Select one of the following options for each condition group in each simulation step:

Option	Description
<b>Off</b>	Use the options specified in the Off status group for this condition group during this simulation step.
<b>On</b>	Use the options specified in the On status group for this condition group during this simulation step.
<b>&lt;status group&gt;</b>	Use the options specified in the selected status group for this condition group during this simulation step.

4. If you need to add a step before or after an existing step, select a step then choose **Insert Step Before** or **Add Step After** from the toolbar above the table.
5. If you need to delete steps, select the step(s) you need to delete and click the trash can icon that appears on the toolbar above the table, then confirm the deletion when prompted.
6. If you want to return to the view where you can see and set the status of all conditions for each step



individually, press You will be warned that the status groups and condition groups that you defined may be deleted. Press **OK** to view all of the individual condition statuses for each step.

To view the status of every condition for every simulation step, open the **Step Details** tab. If the status of a



condition in any step is in conflict because of the statuses defined by the various status groups, is shown as the status for that condition. Review the messages either in the **Messages** tab of the View panel or at the bottom of the **Physics** panel for information about the conflict. Review the status groups specified for the condition groups in each step and correct the conflicting configuration.

## 4.10.2.1. Creating Condition Groups

A condition group should contain all of the conditions that need to be modified for a particular simulation step. You can create as many condition groups as you need, and a particular group can be used to define the behavior of multiple simulation steps. The default All Conditions group is created automatically and contains all conditions defined for that Physics. To create a new condition group:

1. Click **Add Condition Group** on either the **Simulation Steps** or **Condition Groups** tab of the Simulation Step Manager. If you start from the **Simulation Steps** tab, you will need to switch to the **Condition Groups** tab to edit the group.

The new condition group column appears on the **Condition Groups** tab. All conditions are excluded from the new group by default.

If you want to rename the condition groups, you must go to the **Simulation Steps** tab. The condition groups are shown in the first column and can be renamed there.

2. Click the icons for the conditions that you want to include in this group, then click **Apply**.

## 4.10.2.2. Defining Status Groups

Status groups allow you to specify the status of each condition in a particular condition group. There will be three automatically created status groups for the All Conditions group, one with all conditions active (On), one with all conditions suppressed (Off), and a Custom group where you can set the status of each condition as needed.

1. On the **Status Groups** tab, select a **Condition Group**. Each condition group will automatically have On and Off status groups, and an additional status group that you can modify as needed. You can add as many status groups as you need for that condition group. You can add a status group and then modify it, or add a number of them and then go back and modify each one.
2. Set the status of each condition for each status group by toggling the icon for that condition.



- (default for spatial conditions) activates the condition.



- (default for numerical conditions) applies a constant factor to the condition. Enter a **Factor** value for this option. Factor defaults to 1.

**Note:** For a temperature quantity, the factor is applied to the temperature value in degrees K regardless of the units displayed, and a negative factor is invalid.

**Note:** If you change the factor for a condition, you will see a button in the condition's panel next to the output field, indicating that there is a factor other than 1 applied to the output during the simulation. Click  to see a chart showing the value of the output for each simulation step.

-  suppresses the condition.
  -  locks the condition (only valid for certain conditions).
3. Delete any unwanted status groups by selecting the box to the left of the status group(s) and clicking the trash icon that appears, then confirm the deletion when prompted.

### 4.10.2.3. Configuring Simulation Steps

When a simulation step is created, its solution progression defaults to the first solution progression (which is created by default for the solution). To configure a simulation step:

1. On the **Simulation Steps** tab of the **Simulation Step Manager**, click the link in the **Substeps** cell of a simulation step.  
The associated **Simulation Step** panel will open.
2. Select the **Solution Progression** that you want to use for this simulation step.
3. If you need to create a new solution progression, click the **Solution Progression** field and select **Create New**. A new **Solution Progression** panel will open, where you can set the fields as desired.
4. The attributes of the solution progression you have chosen are displayed in the **Simulation Step** panel. You can change the attributes for the selected solution progression from this panel, but if you do so, the new settings will be in effect for any object that uses the selected solution progression.

## 4.11. Output Monitors

You can add monitor charts to your Physics task. For most types of analyses, a default monitor chart is provided. You can create additional charts to monitor any quantities exposed for that analysis type.

Monitor charts should be added before you solve. If you add a monitor chart after the solution is complete, you will have to re-solve to display that chart. In addition, if you make changes to a calculated value that is displayed on a chart after the solution has completed, you will have to re-solve to display that chart.

To improve the drawing speed of the chart, filtering takes place once there are more than 2000 points, where the number of points is calculated as the number of iterations multiplied by the number of trackers in the chart. When filtering is activated, redundant points are removed from the plot. For example, if multiple points describe a straight line, then some of the points will be eliminated, as only the first and last of these points is required.

Table 4.11.1. Default Charts for Physics Type

Physics Type	Default Monitor Chart	More Information
Fluids	For a Fluids physics, the default monitor chart tracks the values of residuals during the solution iterations.	<p>Residuals measure the amount by which a particular solution does not satisfy the discretized equations. If all goes well, the residuals will drop toward zero as more iterations are performed. Iterations are terminated when the residual convergence target has been reached for relevant equations or when the maximum number of iterations have been performed.</p> <p>For more information on evaluating convergence charts in a fluid physics, see <a href="#">Judging Convergence</a> on page 424.</p>
Polymer Extrusion	For a polymer extrusion simulation, AIM automatically creates a single convergence chart with two variables: velocities convergence and pressure convergence. For thermal simulations, temperature convergence is provided. Similarly, for co-extrusion simulations, fluid fraction convergence is provided.	<p>Automatically created convergence charts are plotted as function of cumulative iterations (x-axis of the chart). All automatically created charts are also normalized with respect to their target values and also plotted in log scale. This allows multiple convergence variables to be visualized via a single chart.</p> <p>For more information about convergence in polymer extrusion simulations, see <a href="#">Convergence Strategies and Solution Progression</a> on page 485</p>
Polymer Blow Molding	For a polymer blow molding simulation, AIM automatically creates a single convergence chart with three variables: coordinates, velocities, and thickness convergences. For thermal simulations, temperature convergence is provided.	<p>Automatically created convergence charts are plotted as function of cumulative iterations (x-axis of the chart). All automatically created charts are also normalized with respect to their target values and also plotted in log scale. This allows multiple convergence variables to be visualized via a single chart.</p>
Optimization	For a topology optimization, AIM automatically creates a single convergence chart with two variables, stiffness convergence and mass convergence, plotted as a function of the iteration (as there is no concept of iterations in Topology Optimization).	<p>Automatically created convergence charts are all plotted as function of cumulative iterations (x-axis of the chart). All automatically created charts are also normalized with respect to their target values and also plotted in log scale. This allows multiple convergence variables to be visualized via a single chart with a single target line at 1.0 parallel to x axis.</p> <p>You can also create a custom chart and pick any number of solution variables to plot on the same chart, including convergence for stiffness/frequency, mass, volume, stress, and min and size limit.</p>

Physics Type	Default Monitor Chart	More Information
Electromagnetics	For an electromagnetics simulation, AIM automatically creates a single convergence chart displaying four variables--the number of tetrahedra for the current mesh, total energy (J), energy error (%), and delta energy (%) for each solver iteration--plotted as a function of cumulative iterations.	Automatically created convergence charts are plotted as function of cumulative iterations (x-axis of the chart). All automatically created charts are also normalized with respect to their target values and also plotted in log scale. This allows multiple convergence variables to be visualized via a single chart.
Static Structural	For a structural simulation, AIM automatically creates a single convergence chart with three variables--force convergence, displacement convergence, and a load fraction-- plotted as a function of cumulative iterations.	For these physics types, you can also create a custom chart and pick any number of solution variables to plot on the same chart, including Max DOF Increment, Time Increment, line search, load step, substep etc.  Automatically created convergence charts are all plotted as function of cumulative iterations (x-axis of the chart). All automatically created charts are also normalized with respect to their target values and also plotted in log scale. This allows multiple convergence variables to be visualized via a single chart with a single target line at 1.0 parallel to x axis.
Steady-State Thermal	For a steady-state thermal simulation, AIM automatically creates a single convergence chart with two variables, heat convergence and a load fraction, plotted as a function of cumulative iterations.	For more information on judging convergence in these physics types, see <a href="#">Identifying Problematic Contacts</a> on page 358 in the <a href="#">Contact Best Practices</a> on page 353 section.
Time-Dependent Thermal	For a transient thermal simulation, AIM automatically creates a single convergence chart with two variables, heat convergence and solution progression, plotted as a function of time.	You can also create a custom chart and pick any number of solution variables to plot on the same chart.  For more information on judging convergence, see <a href="#">Identifying Problematic Contacts</a> on page 358 in the <a href="#">Contact Best Practices</a> on page 353 section.

Physics Type	Default Monitor Chart	More Information
Steady-State Electric Conduction	For a steady-state electric conduction simulation, AIM automatically creates a single convergence chart with two variables, current convergence and a load fraction, plotted as a function of cumulative iterations.	<p>You can also create a custom chart and pick any number of solution variables to plot on the same chart, including Max DOF Increment, Time Increment, line search, load step, substep etc.</p> <p>Automatically created convergence charts are all plotted as function of cumulative iterations (x-axis of the chart). All automatically created charts are also normalized with respect to their target values and also plotted in log scale. This allows multiple convergence variables to be visualized via a single chart with a single target line at 1.0 parallel to x axis.</p> <p>For more information on judging convergence in these physics types, see <a href="#">Identifying Problematic Contacts</a> on page 358 in the <a href="#">Contact Best Practices</a> on page 353 section.</p>

#### 4.11.1. Creating a Monitor Chart

Monitor charts can be created from the **Physics Solution** section of the **Physics** panel, or by using the right mouse button in the graphics window and selecting **Add > Monitors > Chart**.

A list of all existing charts can be found by opening **Monitors** under **Physics Solution** and clicking on **Chart**. Charts can also be added from the **Monitors** or **Chart** summary panels.

Once a Chart object is created, to define the monitor chart:

1. Click **Add Monitored Quantity**.

A list of quantities available for plotting is shown. This may include convergence indicators and solution variables.

2. From the resulting list, choose a quantity that you want to monitor. You can individually add to a chart any number of quantities to monitor.

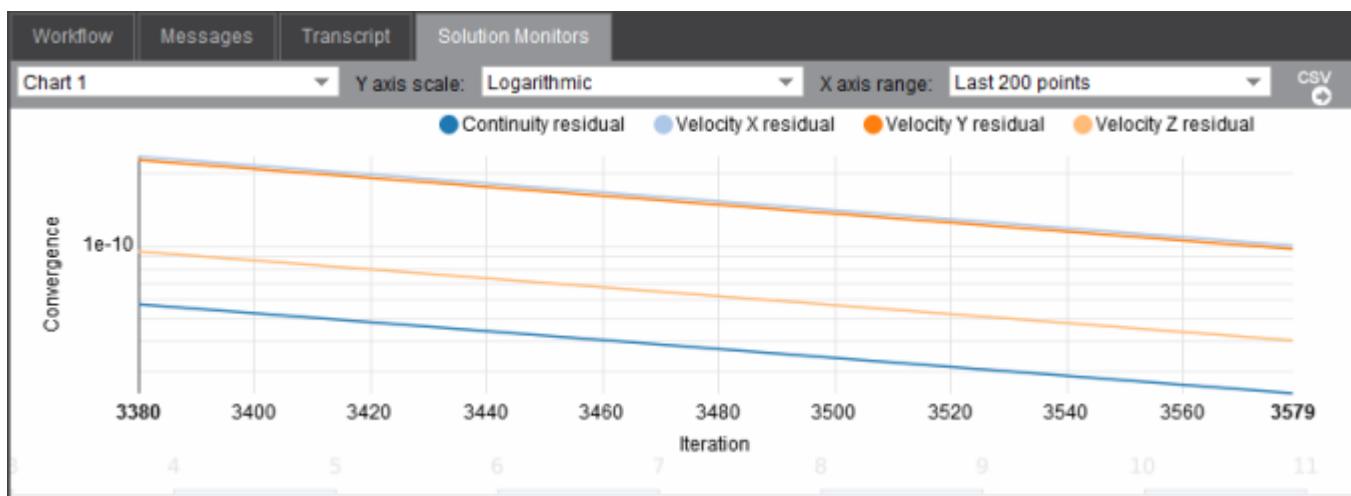
The selected quantity is added to the **Monitored Quantity** table. You can delete quantities from the table by selecting the check box next to the quantity name and clicking on the trash can that appears above the table.

3. For some Physics types, you can also choose **Create > Calculated Value** from the list to create a new [Calculated Value](#) and add it to the current chart.

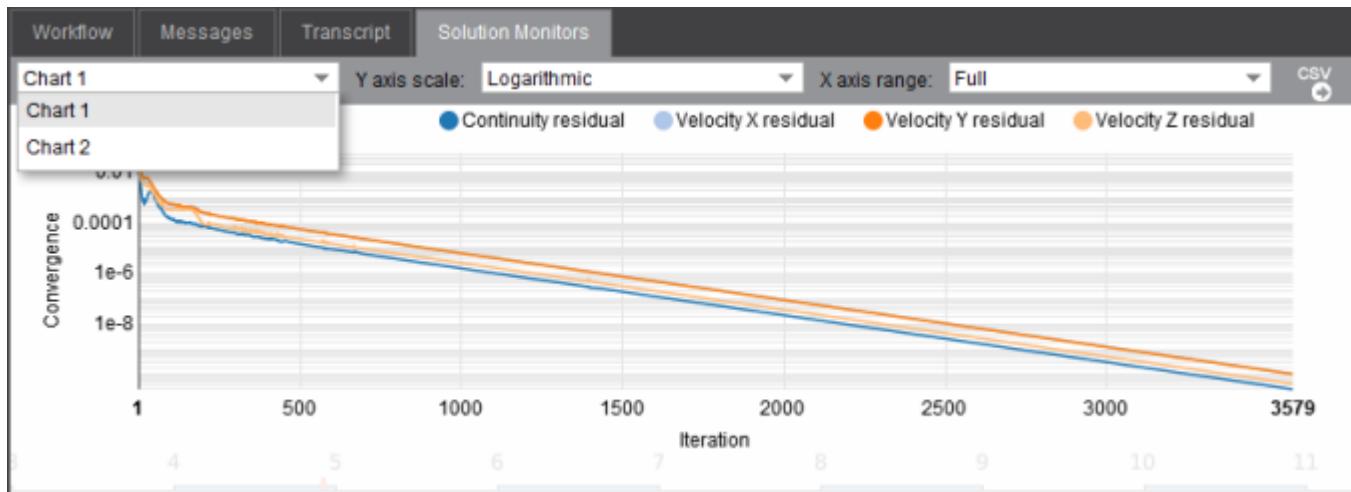
#### 4.11.2. Viewing Monitor Charts

The **Solution Monitors** tab of the View panel lets you view any charts that were automatically created for your analysis and charts that you created. The X-axis of the charts is determined by the solver. For transient solutions it will be Time; for all other solutions it will be Iteration. To view a chart:

1. Click the **Solution Monitors** tab. The first chart that was created will be displayed.



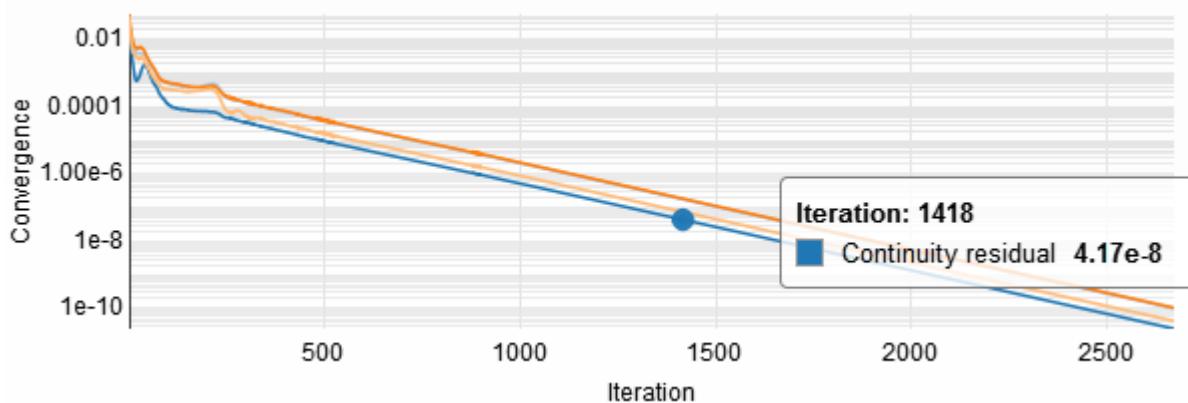
2. Select the chart that you want to view from the drop-down.



3. Set the **Y axis scale** to Logarithmic or Linear, whichever is more appropriate for the values in the chart.

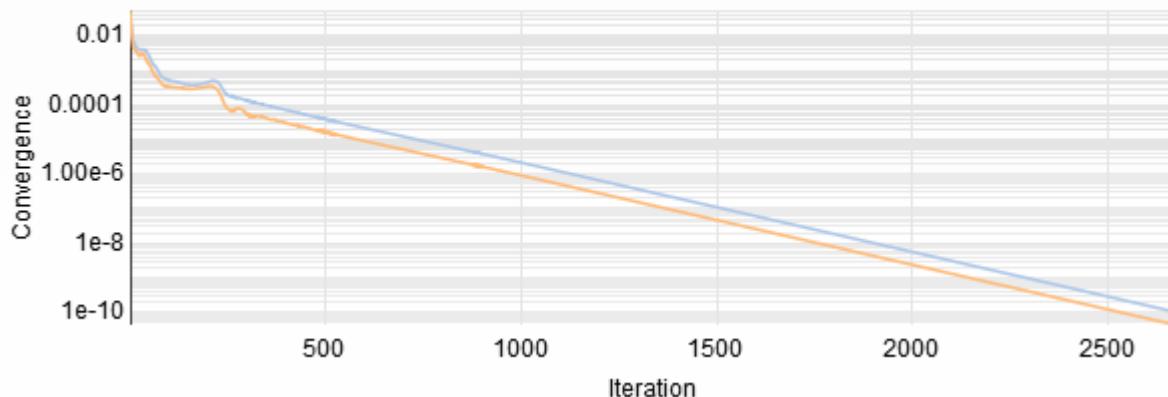
**Note:** When using a logarithmic scale, all negative values will be excluded from the residual plots.

4. Set the **X axis range** to Full if you want to see the entire set of results for the analysis. The default is Last 200 points. When changing between these two settings, it may take a few moments to display the points.
5. For time-dependent fluid flow analyses, you can set the **X axis** to Time or Iterations, whichever is more appropriate for the values in the chart. In some case X-axis of the charts is determined by the solver. For time-dependent solutions it will be Time; for all other solutions it will be Iteration.
6. Hover over a plot on the chart to see the value of that quantity at a particular iteration.



7. Click a quantity name above the chart to hide or show it in the chart.

○ Continuity residual    
 ● Velocity X residual    
 ○ Velocity Y residual  
● Velocity Z residual



You can hide all but one of the quantities. If you try to hide all, then all will be shown. If you double-click a quantity name, it will hide all but that one.

#### 4.11.3. Exporting Solution Monitor Data

You can export the data plotted on a solution monitor chart to a comma-separated values (CSV) file. The data can only be exported after the solution is completed.



1. While viewing a solution monitor chart, press .
2. Browse to the folder where you want to store the file and enter a file name.
3. Press **Save**

All of the points associated with your chart will be saved to the file, regardless of whether you are displaying only the last 200 points on your chart. Similarly, the exported data includes points for any plots that you have hidden on the chart.

## 4.12. Creating Additional Physics Workflows

Simulation [templates](#) in AIM offer additional physics options and methods to create or modify workflows:

[Create time-dependent fluid flow workflows](#)

- Create time-dependent thermal workflows
- Model compressible gas flows
- Model particles in fluid flow

### 4.12.1. Time-Dependent Fluid Flow

Time-dependent fluid flow analyses are used to simulate systems where the flow behavior varies over time. Examples of unsteady flow features include vortex shedding behind bluff bodies, transient heat transfer, and flows with time-dependent boundary conditions.

To run a time-dependent fluid flow simulation:

1. Set up a fluid flow simulation using the [fluid flow template](#). Select a **Calculation type** of **Time-dependent**. Import your geometry and set up meshing as you typically would. In the **Physics** panel, specify the time duration or the time step size to [solve the physics](#).
- Note:** If you happened to import your geometry without selecting the calculation type from the template, then go to the **Physics** panel and select a **Calculation type** of **Time-dependent**.
2. Set up the [fluid flow conditions](#) and any [physics specific options](#).
3. Enter flow specification for the static pressure and velocity as the [initial conditions](#).
4. Set up the solver controls and the various details of the [discretization](#). The default settings have been selected for you and provide good accuracy for most cases.
5. Add [solution monitors](#) for any quantity you are interested in plotting. (Residuals are plotted by default.)
6. [Solve the physics](#).

### 4.12.2. Time-Dependent Thermal Workflow

Time-dependent thermal analyses determine temperatures and other thermal quantities that can vary over time. The variation of temperature distribution over time is of interest in many applications such as cooling of electronic packages or a quenching analysis for heat treatment.

Many heat transfer applications such as heat treatment problems, electronic package design, nozzles, engine blocks, pressure vessels, fluid-structure interaction problems, etc. involve time dependent thermal analyses to capture transient effects.

A time-dependent thermal analysis can either be linear or nonlinear. Temperature dependent material properties (thermal conductivity, specific heat or density), or temperature dependent convection coefficients can result in nonlinear analyses that require an iterative procedure to achieve accurate solutions. The thermal properties of most materials do vary with temperature, so the analysis usually is nonlinear.

To run a time-dependent thermal simulation:

1. Set up a thermal simulation. Either:
  - Select the [thermal template](#) and choose the **Time-dependent** calculation type.
  - Import your geometry, set up meshing as you typically would, and, on the Physics panel set the **Calculation type** to **Time-dependent**.
2. On the **Physics** panel, for **Duration**, specify the end time for the calculation.
3. When you select a time-dependent simulation, the initial temperature is set by default to 22 degrees C. You can change this to an appropriate value for your analysis in **Physics > Initial Conditions**.
4. When defining your [thermal loads and constraints](#), you can use constants, a function of time, or a table of time. You can view a chart of the data by selecting **Show chart**.

For example, specify an expression of  $295.15[\text{K}] + 2[\text{K}]*\text{Time}^1[\text{s}^{-1}]$  for your Temperature or an expression of  $100 [\text{W m}^{-2}]*\sin(\text{Time}^1[\text{s}^{-1}]*1[\text{radian}])$  for your Heat Flux.

**Note:** You can also specify your temperature using Celsius. Relative temperatures are automatically converted to absolute temperatures for calculation

5. During the solve, you can monitor specific outputs during the solution by setting the X axis value on a [Solution Monitor chart](#).
6. After you solve, you can visualize your results at every point in the analysis by moving the slider or entering a time value. You can also animate the results over time.

Time history data is available for all result types. In the case of multiple calculated values, time history data is provided for each value. For example, a contour result will have calculated minimum and maximum values. For a calculated value, time history data is specific to that calculated value and is available on the Value property.

#### 4.12.3. Compressible Fluid Flows

Compressible flow occurs when the fluid density depends on pressure. Compressible flows can be characterized by the Mach number  $Ma \equiv u/c$ , where  $u$  is the flow velocity and  $c$  is the speed of sound, related to the derivative of density with respect to pressure at constant entropy:

$$c^2 = \left( \frac{\partial \rho}{\partial P} \right)_s$$

When  $Ma < 0.3$ , compressibility effects are typically unimportant and it is acceptable to model the fluid using a constant density assumption.

The Mach number also allows us to characterize the flow regime as follows:

Subsonic	flow has $Ma < 1$
Transonic	flow has regions with both $Ma < 1$ and $Ma > 1$
Supersonic	flow has $Ma > 1$

Transonic and supersonic flows may contain structures such as shocks, rarefaction waves, and contact discontinuities. In AIM you can model subsonic, transonic, and supersonic flows, although for supersonic flows, the presence of strong shocks may introduce stability problems during the calculation.

##### 4.12.3.1. Modeling Compressible Gas Flows

To set up a [compressible gas flow](#):

1. In the [Fluid Flow Template](#) on page 41, under **Additional flow physics**, select **Compressible flow (ideal gas)**.

This will in turn enable the energy equation and default to the [ideal gas law](#) material model in the **Material Assignments** panel. If your simulation involves multiple fluid volumes and one of them is incompressible, go to the **Material Assignments** panel for that region and select **Constant properties** as the **Material model**.

2. In the **Physics Options** panel, under **Flow Model > Operating Conditions**, [Set the Operating Pressure](#).

All gauge pressure inputs (such as those defined for boundary conditions and initial conditions) are with respect to this operating pressure. The gauge pressure is the absolute pressure minus the operating pressure.

Under **Turbulence Model > RANS Model > Production Settings**, enable **Compressible production** to include compressibility contributions in turbulence production. It is recommended for the prediction of free shear layers at high Mach numbers.

Under **Energy Model**, **Viscous heating** is enabled by default for compressible gases. This includes the viscous work term in the energy equation, which is often important for high speed flows.

### 3. Set appropriate [boundary conditions](#).

Due to the nature of the physics of compressible flow, additional options are available compared with incompressible flow. In some situations, such as a converging-diverging nozzle with supersonic flow, multiple solutions are possible, and it is important to set boundary conditions appropriately for the solver to find the correct solution. Select the flow **Regime** to be **Subsonic**, **Supersonic**, or **Mixed** depending on whether you expect the boundary flow to be predominantly subsonic, supersonic, or both.

At pressure inlets, velocity inlets, and mass flow inlets, if the flow **Regime** is set to **Subsonic**, the required inputs are the same as for incompressible flow. If the **Regime** is **Mixed** or **Supersonic**, the **Gauge static pressure** must also be specified. If the **Regime** is **Mixed**, the solver will use the specified value only for those boundary faces which are locally supersonic.

At outlets, if the **Regime** is set to **Subsonic**, the required inputs are the same as for incompressible flow. If the **Regime** is **Mixed**, the solver will use the specified gauge static pressure only for those boundary faces which are locally subsonic.

At supersonic outlets, no boundary inputs are required. Be aware, however, that this setting is very weak, and may not find the supersonic solution if multiple solutions exist (e.g., subsonic and supersonic exit flow in a converging-diverging nozzle), and the initial guess has subsonic flow. In this situation, it may be better to choose a **Mixed** regime with a pressure outlet and specify a gauge static pressure which forces the flow into the supersonic regime.

## 4.12.4. Particles in Fluid Flow

In AIM, you can simulate a wide range of discrete phase problems such as particle separation and classification, aerosol dispersion, and bubble stirring of liquids.

In addition to solving transport equations for the continuous phase, AIM allows you to simulate a discrete second phase in a Lagrangian frame of reference. This second phase consists of spherical particles (which may be taken to represent droplets or bubbles) dispersed in the continuous phase. The trajectories of these discrete phase entities are computed. The coupling between the phases and its impact on both the discrete phase trajectories and the continuous phase flow can be included.

The following discrete phase modeling options are available:

- Calculation of the discrete phase trajectory using a Lagrangian formulation that includes the discrete phase inertia, hydrodynamic drag, and the force of gravity.
- Prediction of the effects of turbulence on the dispersion of particles due to turbulent eddies present in the continuous phase.
- Heating/cooling of the discrete phase.
- Coupling of the continuous phase flow field prediction to the discrete phase calculations.

To track particles in your fluid flow simulation, follow the general steps outlined in [Particle Tracking Workflow](#) on page 504.

### 4.12.4.1. Particle Tracking Workflow

You can include a [discrete phase](#) in your simulation by defining the initial position, velocity, size, and temperature of individual particles. These initial conditions, along with your inputs defining the physical properties of the discrete phase, are used to initiate the trajectory and heat transfer calculation. The trajectory and heat transfer calculations are based on the force balance on the particle and on the convective heat transfer from the particle. These calculations use the local continuous phase conditions as the particle moves through the flow. The predicted trajectories and the associated heat transfer can be viewed graphically.

The procedure for setting up a fluid flow simulation with particle tracking is outlined below.

- In the **Fluid Flow template**, after importing or defining your geometry, select **Particles (including droplets and bubbles)** under **Additional flow physics**.

This will in turn create default materials, in the **Material Assignments** panel, which will need further attention.

- Assign a material for each phase under **Material Assignment > Material distribution**. There must be a **Continuous** (carrier) phase and a **Particles** (dispersed) phase defined to set up particle tracking.

**Note:** One-way coupling of phases will be modeled, where the continuous phase affects the dispersed phase, but not vice-versa.

- In the **Physics Options** panel, you can define the hydrodynamic drag and the force of gravity under **Flow Model > Flow Model Phase Interaction** (click  to show all properties).

- Drag force** is enabled by default and equates the particle inertia with the forces acting on the particle. It predicts the trajectory of a discrete phase particle (or droplet or bubble) by integrating the force balance on the particle, which is written in a Lagrangian reference frame.
- Drag coefficient** models are available to describe the interphase drag in particulate flows, or you can enter the drag function to specify the interphase exchange coefficient.
  - Automatic** is selected by default. It takes into consideration the state of the continuous phase and the particle diameter. If the injection can have submicron particles, **Slip regime** is enabled. The **Mean free path specification** is set to **Automatic** by default and will use the Kinetic Theory of Gases to calculate the mean free path. Otherwise, you can enter it as a **Constant value**.
  - Alexander Morsi** model is the most complete, adjusting the function definition frequently over a large range of Reynolds numbers, but calculations with this model may take longer than with the other models.
  - Schiller Naumann** is acceptable for general use.
  - User defined** allows you to specify the interphase exchange coefficient. If the exchange coefficient is equal to zero, there will be no drag forces applied.
  - Slip regime** applies the Stokes-Cunningham correction to the drag coefficient. This correction applies to small particles (relative to the mean free path of the gas phase). The slip regime is where the particle diameter and mean free path are comparable. The no-slip condition is not applicable on the gas/particle surface. The **Mean free path specification** is set to **Automatic** by default and will use the Kinetic Theory of Gases to calculate the mean free path. Otherwise, you can enter it as a **Constant value**.
- Gravity force** (accessed by selecting the **Particle buoyancy only** option under **Buoyancy**) models the buoyancy force on a particle immersed in a fluid. It is much more common to include gravity for particles than it is for the flow as gravity effects on the particle relative to the flow can be significant. Particle gravity can also be important for isothermal flows, but negligible for the flow field.

- For a simulation with turbulence modeling, you can include turbulent velocity fluctuations in the particle force balance by enabling **Particle dispersion** under **Physics Options > Turbulence Model > Turbulence Model Phase Interaction**.

**Note:** For any duplicated physics task containing particles, make sure you review the **Particle dispersion** setting as it will not be copied to the new task.

- For a simulation with heat transfer between the particle and continuous phases, you can set the **Particle heat transfer coefficient** under **Physics Options > Energy Model > Energy Model Phase Interaction** to either follow the **Ranz marshall** model (a correlation for flow past a spherical particle) or you can enter a value for the particle heat transfer coefficient.
- For any fluid flow boundary condition, **define the particle behavior** when a particle reaches the physical boundary to determine the fate of the trajectory at that boundary. Particles experience elastic collisions at symmetry boundaries and may be reflected or escape through all other boundaries.

7. **Include particle injections** to create the initial conditions that define the starting positions, velocities, and other parameters for each particle stream and the physical effects acting on the particle streams.
8. **Solve** and **visualize** the particle tracks.

#### 4.12.5. Pre-Stressed Modal Analysis

You can include the pre-stress effects of a static analysis in your modal analysis. A typical example is the large tensile stress induced in a turbine blade under a centrifugal load that can be captured by a static structural analysis. Because this causes significant stiffening of the blade, including this pre-stress effect will result in much higher, realistic natural frequencies in a modal analysis.

In pre-stressed modal analysis, you can apply the same support conditions used in the static analysis. AIM then deletes the structural loading conditions from the modal portion of the simulation.

To set up a pre-stress modal analysis:

1. Select the **Structural** template app.
2. On the structural template, select a **Calculation Type of Modal**. Then, under **Options**, select **Enable pre-stress**.

You can also select **Enable pre-stress** on the **Physics** panel when the calculation type is set to **Modal**.

3. Set physics conditions, connections, and other physics-related tasks for the pre-stressed modal analysis.
4. Solve.
5. View your eigenvalues and default results for your modal analysis.

**Note:** Results for the static structural simulation are not available in a pre-stressed modal analysis. If you would like to view results for a static structural analysis, then set it up as a separate and distinct simulation.

#### 4.12.6. Random Vibration Analysis

A random vibration analysis enables you to determine the response of structures to vibration loads that are random in nature. For example, you may want to determine the response of a sensitive electronic component mounted in a car when subjected to the vibration from the engine, pavement roughness, or acoustic pressure.

Loads such as the acceleration caused by the pavement roughness are not deterministic; that is, the time history of the load is unique every time the car runs over the same stretch of road. As a result, it is not possible to predict precisely the value of the load at a point in its time history. Such load histories, however, can be characterized statistically (mean, root mean square, standard deviation). Also, random loads are non-periodic and contain a multitude of frequencies. The frequency content of the time history (spectrum) is captured along with the statistics and then used as the load in the random vibration analysis. This spectrum, for historical reasons, is called Power Spectral Density, or PSD.

Since the input excitations in a random vibration analysis are statistical in nature, so are the output responses such as displacements, stresses, and so on.

Typical applications for random vibration analysis include aerospace and electronic packaging components subject to engine vibration, turbulence and acoustic pressures, tall buildings under wind load, structures subject to earthquakes, and ocean wave loading on offshore structures.

#### 4.12.6.1. Setting Up a Random Vibration Analysis

The base excitation for a random vibration analysis is applied in the form of Power Spectral Density (PSD). The PSD is a table of spectral values vs. frequency that captures the frequency content. The PSD captures the frequency and mean square amplitude content of the load's time history.

The base excitation could be an acceleration PSD (either in acceleration<sup>2</sup> units or in G<sup>2</sup> units), velocity PSD, or displacement PSD. In AIM, the base excitation, defined as a structural condition, is applied in the specified

direction to all entities that have the selected Fixed Support. Other support points in a structure, such as Frictionless Support, are not excited by the PSD.

You can apply multiple uncorrelated PSDs -- useful if different, simultaneous excitations occur in different directions.

Random vibration analysis is based on the mode-superposition method and thus requires a modal analysis. The modal analysis must extract enough modes to cover the PSD frequency range. A conservative rule of thumb is to extract enough modes to cover 1.5 times the maximum frequency in the PSD excitation.

See [Random Vibration Analysis Workflow](#) on page 507 for specific information on the workflow.

## 4.12.6.1.1. Random Vibration Analysis Workflow

To set up a random vibration analysis analysis:

1. Select the **Structural** template app.
2. On the structural template, select a **Calculation Type** of **Modal**. Then, under **Options**, select **Include random vibration**.

You can also select **Include random vibration** on the **Physics** panel when the calculation type is set to **Modal**.

3. In the **Structural Conditions**, define a table of spectral values vs. frequency that captures the frequency content as a **PSD Acceleration** (or **PSD G Acceleration** to specify the PSD table in  $G^2$  units), **PSD Velocity**, and/or **PSD Displacement**. In addition, specify any displacements, point masses, or supports.
4. In your **Solver Settings**, you can specify a constant damping value for your random vibration analysis. AIM sets this to a default value of 1%.

You can also choose to exclude insignificant modes from superimposing random vibration displacement responses, which improves performance but may reduce solution accuracy. As a result, use caution and carefully check your solution.

5. Solve.
6. View your results. By default, the template creates equivalent stress and 1 Sigma, 2 Sigma and 3 Sigma component displacements, but other result types are available.

## 4.12.6.2. Random Vibration Results

The results output by the solver are one sigma or one standard deviation values (with zero mean value). These results follow a Gaussian distribution. The interpretation is that 68.3% of the time the response will be less than the standard deviation value.

You can scale the result by 2 times to get the 2 sigma values. The response will be less than the 2 sigma values 95.45% of the time and 3 sigma values 99.73% of the time.

The Coordinate System setting for result objects is, by default, set to the solution reference frame and cannot be changed because the results only have meaning when viewed in the solution reference frame.

Since the directional results from the solver are statistical in nature they cannot be combined in the usual way. For example the X, Y, and Z displacements cannot be combined to get the magnitude of the total displacement. The same holds true for other derived quantities such as principal stresses.

Displacement results are relative to the base of the structure (the fixed supports). Velocity and acceleration results include base motion effects (absolute).

**Note:** The use of nodal averaging may not be appropriate in a random vibration analysis because the result values are not actual values but standard deviations. Moreover, the element coordinate system for the shell elements in a surface body may not all be aligned consistently when using the default coordinate system. Consider using unaveraged results for postprocessing instead.

#### 4.12.7. Eigenvalue Buckling Analysis

An Eigenvalue Buckling analysis predicts the theoretical buckling strength of an ideal elastic structure. This method corresponds to the textbook approach to an elastic buckling analysis: for instance, an eigenvalue buckling analysis of a column matches the classical Euler solution. However, imperfections and nonlinearities prevent most real-world structures from achieving their theoretical elastic buckling strength. Therefore, an Eigenvalue Buckling analysis often yields quick but non-conservative results.

In AIM, an eigenvalue buckling analysis is integrated with a structural simulation. This enables you to consider nonlinearities that are present in the pre-stressed environment, and therefore attain a more accurate real-world solution as compared to a traditional linear eigenvalue buckling analysis.

The nonlinearities present in the static analysis can be the result of nonlinear:

- Geometry (when large deformation effects are set on)
- Contact status (a contact type other than Bonded or No Separation with the MPC formulation)
- Material (for example, the definition of nonlinear material properties in Engineering Data, such as hyperelasticity, plasticity, etc.)
- Connection (for example, nonlinear joints and nonlinear springs)

A structure can have an infinite number of buckling load multiplier factors. Each load multiplier is associated with a different instability pattern. Typically the lowest load multiplier is of interest. Based upon how you apply loads to a structure, load multiplier can either be positive or negative. The application sorts load multiplier from the most negative values to the most positive values, although you can specify that negative multipliers are not included. The minimum buckling load multiplier may correspond to the smallest eigenvalue in absolute value.

For Pressure and Force per unit area, you could experience an additional stiffness contribution called the "pressure load stiffness" effect. In this case, the pressure acts as a follower load, which means that it continues to act in a direction normal to the specified geometry, even as the structure deforms. Force per unit area loads defined on a vector act in a constant direction even as the structure deforms. Therefore, a pressure applied to a face can result in a significantly different buckling load multiplier in the Eigenvalue Buckling analysis than a force per unit area applied to the same face with the same magnitude.

Buckling mode shapes do not represent actual displacements but help you to visualize how a part or an assembly deforms when buckling.

## Linear-based Eigenvalue Buckling Analysis

Note the following when you have created an Eigenvalue Buckling analysis when the structural analysis is linear:

- The buckling load multiplier is applied to all the loads defined in the simulation. If certain loads are constant (for example, self-weight gravity loads) while other loads are variable (for example, externally applied loads), you need to take special steps to ensure accurate results. For example, you can iterate on the eigenvalue buckling solution, adjusting the variable loads until the load factor becomes 1.0 (or nearly 1.0, within some convergence tolerance).
- If you receive all negative buckling load factor values for your Eigenvalue Buckling analysis and you wish to see them in the positive values, or vice versa, reverse the direction of all of the loads.
- You can apply a nonzero constraint in the static analysis. The load factors calculated in the buckling analysis should also be applied to these nonzero constraint values. However, the buckling mode shape associated with this load will show the constraint to have zero value.

# Non-linear Eigenvalue Buckling Analysis

In addition, in a nonlinear-based analysis, only the loads from the final static simulation step are considered in the eigenvalue buckling solution.

## 4.12.7.1. Eigenvalue Buckling Workflow

To set up an eigenvalue buckling workflow:

1. Select the **Eigenvalue buckling** calculation type, either in the **Structural** template or on the **Physics** panel.
2. Define your geometry and physics solution as you typically would, specifying your structural loads and constraints.
3. In the **Solver Settings**:
  - a) For **Max modes to find**, enter the number of buckling load factors (and therefore the corresponding buckling mode shapes of interest). AIM defaults to two buckling modes.
  - b) For **Include negative load multiplier**, select whether you want to extract both the negative and positive eigenvalues (load multipliers) or just the positive.
4. For an Eigenvalue buckling analysis, AIM creates a deformed shape contour result for every mode it was solved for. You can see the load multiplier factor on the contour result for that mode. The load multiplier property is also available on vector results, calculated values, and isosurfaces.

## 4.12.8. Topology Optimization

With a Topology Optimization analysis, you can compute an optimal structural design of your geometry for a selected region of your model with specific design goals and constraints. Topology optimization is a physics driven optimization that is based on a set of loads and boundary conditions. Currently, you can use optimization to either maximize the strength of your topology or minimize the response to free vibration.

To use topology optimization, you:

1. Select the [Topology Optimization template](#).
2. Define your design goal.
  - Select **Maximize strength** to maximize the stiffness and minimize compliance. This option reduces volume or mass but maintains the maximum stiffness possible for the remaining structure.
  - Select **Minimize response to free vibration** to maximize frequency. Then, specify a mode number (from 1 to 6) for the frequency to be optimized in **Mode to base frequency on** and specify the number of natural frequencies to solve for in **Total modes to extract**.
3. Specify whether you want AIM to smooth your shape before export.
4. Optionally, set [optimization options](#) such as your material removal strategy. By default, your optimization is set to maintain at least a percentage of the mass or volume of the model, and this is the only constraint required if you want to achieve convergence.
5. Optionally, specify [optimization conditions](#) such as stress limit, size limit, or symmetry. If you have chosen to deselect the option to limit material removal, you must set the **Stress Limit** in the **Optimization Conditions** if you want your solution to converge
6. Optionally, set loads and constraints.

Supported conditions when optimizing to maximize strength include Force, Moment, Pressure, Support, Displacement, and Bolt Loads defined via force.

Supported conditions when optimizing to minimize the response to free vibration include Support and Point Mass.

7. Solve your optimization. As the physics solution is progressing, you can see the evolution of the optimized shape.
8. After the solve is complete, [refine and confirm your geometry in the Shape task](#).
9. [Create a new model with optimized geometry](#), which can then be further refined in the Geometry Modeler or used in a physics simulation.

## 4.12.8.1. Refining the Optimized Shape

Once the shape has been optimized, you can refine your shape, setting how aggressively it is adjusted and specifying the density threshold.

To adjust your shape:

1. Adjust the **Shape adjustment** slider, either on the **Shape** panel or in the **Graphics** view, or enter a value in the **Shape** panel.  
The supported range is 0 to 1. With **Aggressive**, all marginal elements are removed, whereas with **Conservative** they are kept.
2. Enter a value for **Density threshold**, which controls elemental density.  
The supported range is 0.01 to 0.99 (greater than zero and less than 1). A value of 0 means you want the element removed; 1 means you want the element retained.
3. To create a smoother shape, select **Smooth shape**.

You can now export the new model to the Geometry Modeler.

## 4.12.8.2. Exporting the Optimized Geometry

Once you have finished optimizing your shape, you can export the new model to use when simulating and validating your design.

To export your new geometry:

On the **Shape** panel, click **Export New Model**, or right-click the **Shape** task and select **Export New Model**. The model is shown in a new **Geometry** task. You can then proceed with your physics simulation or open the Geometry Modeler to modify or export your model.

## 4.13. Adjust a Simulation by Adding Physics (Guide Me)

AIM has some built-in shortcuts that guide you in performing certain operations. These shortcuts are found as **Guide Me** in the context menu and appear when conditions are right.

- **Add Fluid-Solid Heat Transfer:** Extend a fluids-only simulation to include fluid-solid heat transfer. This is available for a fluid flow simulation where additional bodies without any solid regions are defined. This streamlines the creation of the solid-thermal physics on the additional bodies. [Adding a Fluid-Solid Heat Transfer \(Guide Me\)](#) on page 511
- **Add Structural Stress Computation:** Extends a fluids-only simulation, or a fluid and thermal solid heat transfer simulation to perform stress and fatigue evaluation of the solid bodies. This is available for a fluid flow simulation without a downstream structural physics task. This will add the additional task required for structural physics and set up physics coupling from the thermal solid bodies and/or the fluid bodies to enable the computation of stress and deformation. [Adding a Structural Stress Computation \(Guide Me\)](#) on page 511

### 4.13.1. Adding a Fluid-Solid Heat Transfer (Guide Me)

**Note:** When extending a fluid flow simulation, **Guide Me** will not appear unless additional bodies without solid regions are defined.

To add a fluid-solid heat transfer:

1. Go to the Physics task and right-click on the graphics background to open the context menu.
2. Select **Guide Me** and then **Add Fluid-Solid Heat Transfer**.
3. Define the material(s) and solid region(s) to include in the fluid-solid heat transfer simulation in the **Solid Physics Region** panel.
  - a) To assign only one material:
    - Keep 1 from the dropdown.
    - Define the solid region location
      - automatically by selecting **Use all unassigned bodies**, or
      - manually by selecting the location with **Select bodies forming the solid region**.
    - Select the **Material for solid region** from the dropdown list.
    - If defining heat transfer from external surfaces, select the type of **Heat transfer option**.
  - b) For more than one material, define the location manually then select the material and the type of **Heat transfer option**.
  - c) Click **Finish**.

### 4.13.2. Adding a Structural Stress Computation (Guide Me)

**Note:** When extending a fluid flow simulation, **Guide Me** will not appear unless additional bodies without solid regions are defined.

To add a structural stress computation:

1. Go to the Physics task and right-click on the graphics background to open the context menu.
2. Select **Guide Me** and then **Add Structural Computation**.
3. Specify **Additional Physics**.
  - a) Select the physics task to which you want to apply the additional physics.
  - b) Specify the additional physics that you want to evaluate. To model situations where:
    - Fluid-solid heat transfer occurs, including temperature variations within the fluid, select **Fluid-solid heat transfer**.
 

**Note:** This option will only be available if solid-thermal regions haven't already been created. If shown you'll also be asked to define the region and materials as per the fluid-solid heat transfer **Guide Me**.

      - The temperature field introduces thermal strains in the structural field, which, in turn result in thermal expansion and stresses, select **Structural stress due to solid thermal effects**.
      - Fluid forces are transferred to a structure without heat transfer, select **Structural stress due to fluid forces**.
    - c) To automatically calculate fatigue results for your static structural analysis based on the S-N curve values defined for the default material, select **Compute fatigue results**.
    - d) Click **Next**.
4. Define the material(s) and solid region(s) to include in the structural simulation (and optionally fluid-solid heat transfer simulation) in the **Solid Physics Region** panel.
  - a) To assign only one material:

- Keep 1 from the dropdown.
  - Define the solid region location
    - automatically by selecting **Use all unassigned bodies**, or
    - manually by selecting the location with **Select bodies forming the solid region**.
  - Select the **Material for solid region** from the dropdown list.
  - If defining heat transfer from external surfaces, select the type of **Heat transfer option**.
- b) For more than one material, define the location manually then select the material and the type of **Heat transfer option**.
- c) Click **Next**.
5. Specify **Structural Information**.
- a) Define the **Solid support locations** by selecting a face or selection set.
  - b) Click **Finish**.

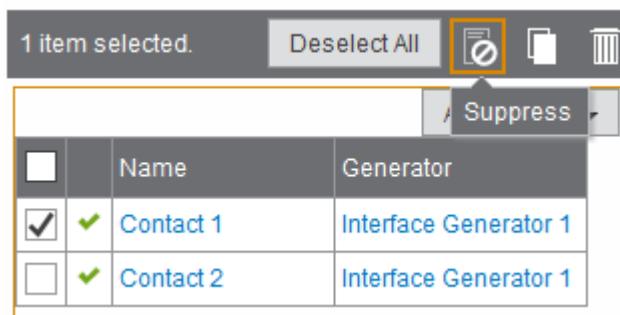
## 4.14. Suppressing a Structural Condition

If a condition is no longer required for your structural simulation, but may be needed in the future, you can suppress it rather than delete it entirely. Suppression is available for [structural loads and constraints](#) and interface conditions such as [contacts](#), [joints](#), and [springs](#).

You can suppress conditions from the summary panel for the condition type, or on the data panel for a specific interface condition, load, or constraint. Once a condition is suppressed, it is ignored by the simulation; however, it is still part of the project and can be unsuppressed for future use.

For example, to suppress one or more contacts:

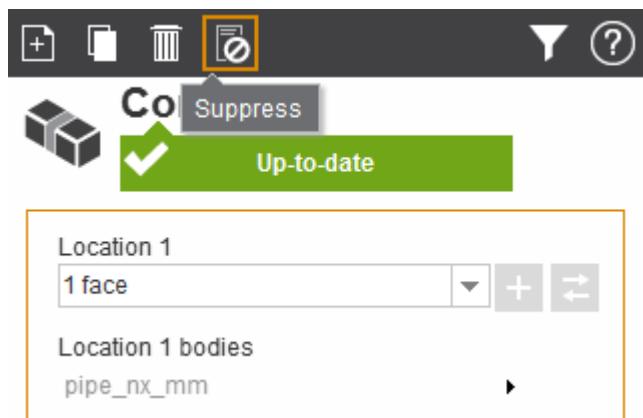
1. Click **Interface Conditions > Contacts** on the **Physics** panel, or select **Contacts** from the navigation bar.
2. Select one or more contacts.
3. Click the **Suppress**  icon.



The screenshot shows a software interface for managing contacts. At the top, a toolbar has a 'Deselect All' button and a 'Suppress' button, which is highlighted with a yellow border. Below the toolbar is a table with three columns: 'Name' and 'Generator' (both header), and 'Contact 1' and 'Contact 2' (data rows). Each contact row has a checkbox in the first column. The 'Contact 1' row has a checked checkbox, while the 'Contact 2' row has an unchecked checkbox. To the right of the table is a vertical scroll bar.

	Name	Generator
<input checked="" type="checkbox"/>	Contact 1	Interface Generator 1
<input type="checkbox"/>	Contact 2	Interface Generator 1

Or, view a specific contact's data panel and click **Suppress** .



# Chapter 5: Results

---

There are two categories of results in AIM:

- **Exploratory results:** a preview of some preliminary, lower fidelity results that is available immediately upon solving physics. These are highly interactive results that are available for a sub-set of typical variables and allow you to quickly identify and view areas of interest before creating higher fidelity results.
- **Evaluated results:** higher fidelity, conclusive results for all available result variables by setting up and evaluating specific results objects.

The **Exploring** preview is the default view when you first move to the **Results** task and is indicated by the **Exploring** toolbar. (See [Exploring the Result Preview](#) on page 551)

**Note:** For simulation studies created in AIM prior to R19.1, the physics must be re-solved to enable the **Exploring** preview. Some physics (springs, shells, modal) are not supported in **Exploring**.

Generating results to understand the behavior of an analyzed model is fundamental to any analysis. AIM supports a variety of result display types and tools to facilitate this process.

- [Contours](#) on page 515 allow you to display various solution quantities, such as displacement, stress, temperature, and electric field density over the entire, or a portion, of the model.
- [Vectors](#) on page 521 allow you to show the direction and magnitude of vector variables on a surface or volume.
- [Isosurface](#) on page 526 results allow you to display a surface of constant value for a specified variable.
- [Streamlines](#) on page 529, available for a fluid simulation, allow you to visualize the lines traveled by massless particles that are released from one or more surfaces in the physics region.
- [Particle Tracks](#) on page 532 available for a fluid flow simulation, allow you to visualize the path traveled by particles of a discrete phase that are injected at one or more surfaces in the physics region.
- [Line Chart](#) on page 534 results allow you to display selected variables against the position on a line.

In addition, you can perform a range of quantitative calculations to obtain single-value results using the [Calculated Value](#) on page 539. Quantities are values at particular location in the physics domain. You can use them to refer to:

- Solved and derived variables such as velocity, stress, total temperature etc.
- User-defined values based on expressions
- Location values such as coordinate values or mesh information

In an electromagnetic simulation, you can also use [Table](#) results objects to perform a variety of [Matrix](#) and [Loss](#) quantity calculations.

You can also modify the display of your result by:

- [Specifying a Variable Range](#) on page 575
- [Specifying a Color Distribution](#) on page 576
- [Specifying Deformation Scaling](#) on page 576
- Viewing a [Summary](#) on page 577 of the result

Finally, you can [animate your results](#).

## 5.1. Results Display Types

You can graphically analyze the distribution of solution variable values on a surface, volume or construction geometry (such as a line or plane) using [Vectors](#) on page 521, [Contours](#) on page 515, and [Isosurface](#) on page 526 objects. A [Line Chart](#) on page 534 can be used to plot the variables against the position on a line. In a

fluid simulation, you can use the [Streamlines](#) on page 529 object to visualize the lines traveled by massless particles that are released from one or more surfaces. You can also use [Particle Tracks](#) on page 532 to visualize the path traveled by particles of a discrete phase that are injected at one or more surfaces.

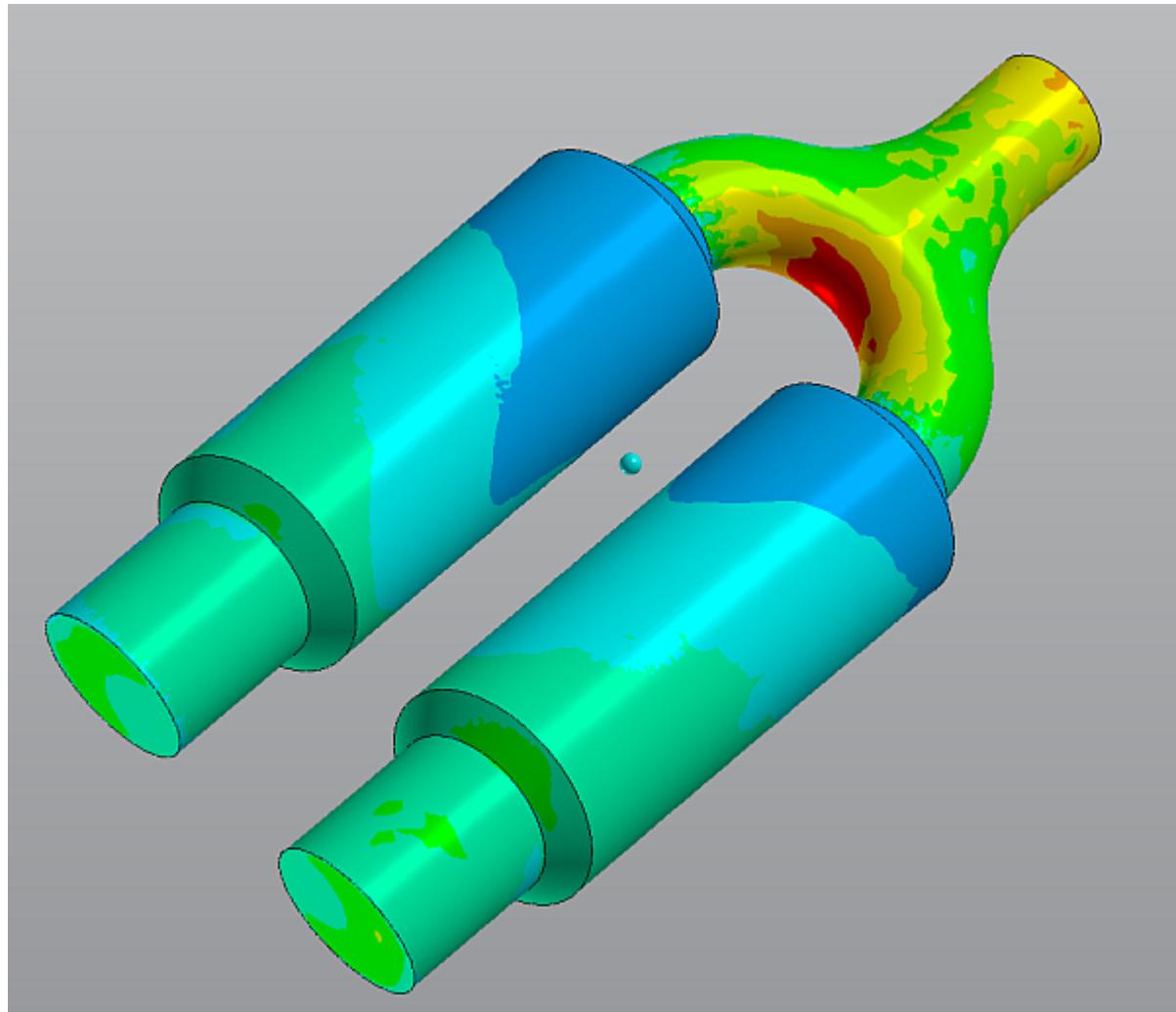
### 5.1.1. Contours

You can use a [Contour](#) to visualize and assess gradients for variables displayed on a surface or volume. Contour lines are lines of constant magnitude for a selected variable. The regions between the contour lines are filled with a color that denotes the value of the variable in that region.

On a given surface or volume, adjacent contour lines are:

- Widely spaced in regions where the variable is slowly changing.
- Closely spaced together in regions where the variable is rapidly changing.

The contour plot below indicates the variation of total pressure across a dual exhaust system.



For more information on the results variables available, see [Results Variables by Physics Type](#) on page 552. For information on the specialized contour results for linearized stresses, see [Linearized Stresses](#) on page 538.

## 5.1.1.1. Specifying a Contour

### Results > Object > Results > Contour

To create objects that display the results of your simulation, generate a physics solution.

Contours display regions containing a defined range of values of a given variable. To specify a contour result:

1. Open a **Contour** panel under the **Results** task by either:

- Selecting an existing **Contour** object.
- [Adding a New Contour](#) on page 520.

2. Under **Location**, you can specify a location by either:

- Selecting a location from the drop-down list.
- Selecting a location in the graphics viewer.
- Specifying an expression.
- You can also create new reference geometry or Results to be used as locations using the **Create** menu.

For details, see [Defining Locations](#) on page 71.

3. Specify the variable you want to display under **Variable**:

- Selecting an existing variable from the drop-down menu or entering a valid variable name.
- Entering an expression for the variable you want to plot. For details, see [Expressions](#).
- Defining your own variable by entering a solver variable after the text `UserDefined`, separated by a period. For example, `UserDefined.USUM`.

4. Execute any steps specific to your physics type.

If you want to...	For Physics Type	Then
Visualize variables by node values	Fluids	<p>Select the  <b>Use node values</b> option. Visualization of all variables, except <a href="#">extensive variables</a>, is always performed using node values. Visualizing by node value means that the colors are set on the nodes.</p> <p>For extensive variables, such as Face Area, Element Volume, Mass Flow, and Force, whose magnitudes depend directly on the mesh element size, visualization is performed by element data by default, as it is most appropriate. However, you can <b>Use node values</b> to perform node value based visualization for these extensive variables. Node values for such variables are calculated by interpolating the element data values.</p> <p><b>Note:</b> Some extensive variables like Element Volume, Orthogonal Quality, Element Aspect Ratio, and Face Area cannot be plotted when <b>Use node values</b> is selected.</p>
Display results for a particular natural frequency	Modal Structural	Use the <b>Mode</b> slider. You can review the frequency for the specified mode under <b>Summary &gt; Global &gt; Frequency</b> .

If you want to...	For Physics Type	Then
Control at which simulation step the result will be evaluated	Structural, Eigenvalue Buckling	Select a step from <b>Simulation step</b> . For more information on simulation steps, see <a href="#">Simulation Steps</a> on page 493.
Control at which sub-step the result will be evaluated and plotted in graphics	Structural, Eigenvalue Buckling, Polymer extrusion	Use the <b>Substep</b> slider. For more information on substeps for structural simulations, see <a href="#">Specifying Solution Progression for Static Structural</a> on page 440.
View the result at a specific time	Time-Dependent Thermal	Use the <b>Time</b> field or slider. You can enter the time for which you want to view the result, or use the slider to choose the desired time. You will need to evaluate the result at the newly selected time.
Visualize results on multiple frames of reference	Structural, Eigenvalue Buckling, Thermal, Electric Conduction, Electrostatic	Specify the reference frame under <b>Relative to</b> . Your options are: <ul style="list-style-type: none"> <li>Selecting an existing reference frame from the drop-down menu.</li> <li><a href="#">Creating Reference Frames</a> on page 77.</li> <li>For Joint results, selecting the <b>Solution Reference Frame</b> option. Frame-dependent variables are calculated relative to the selected Joint's local reference frame. When post-processing multiple joints, this option ensures each Joint's respective local reference frame is used to calculate the results.</li> </ul>
Display averaged contours for elemental quantities such as stresses and strains	Structural, Eigenvalue Buckling, Thermal, Electric Conduction, Electrostatic	<p>Set <b>Use nodal averaging</b>.</p> <p>Your options are:</p> <ul style="list-style-type: none"> <li>To display gradients across adjacent elements, disable <b>Use nodal averaging</b>. For this option, contours are evaluated by linear interpolation within each element, and the influence of the adjacent elements is ignored. The resulting un-averaged contours are discontinuous across element boundaries.</li> <li>To display continuous contours across elemental discontinuities such as element boundaries, enable <b>Use nodal averaging</b>.</li> </ul> <p><b>Note:</b> Contours for fluids simulations always use averaging.</p>
Display contours for the specified variable at particular frequency and phase angle	Frequency Response Electromagnetics	<p>Select the desired <b>Frequency</b> from the drop-down menu and specify a <b>Phase angle</b>.</p> <p>Because the computed field values are complex numbers, the <b>Phase angle</b> you specify determines the particular time or point in the period of the chosen <b>Frequency</b> used to create the plot.</p>

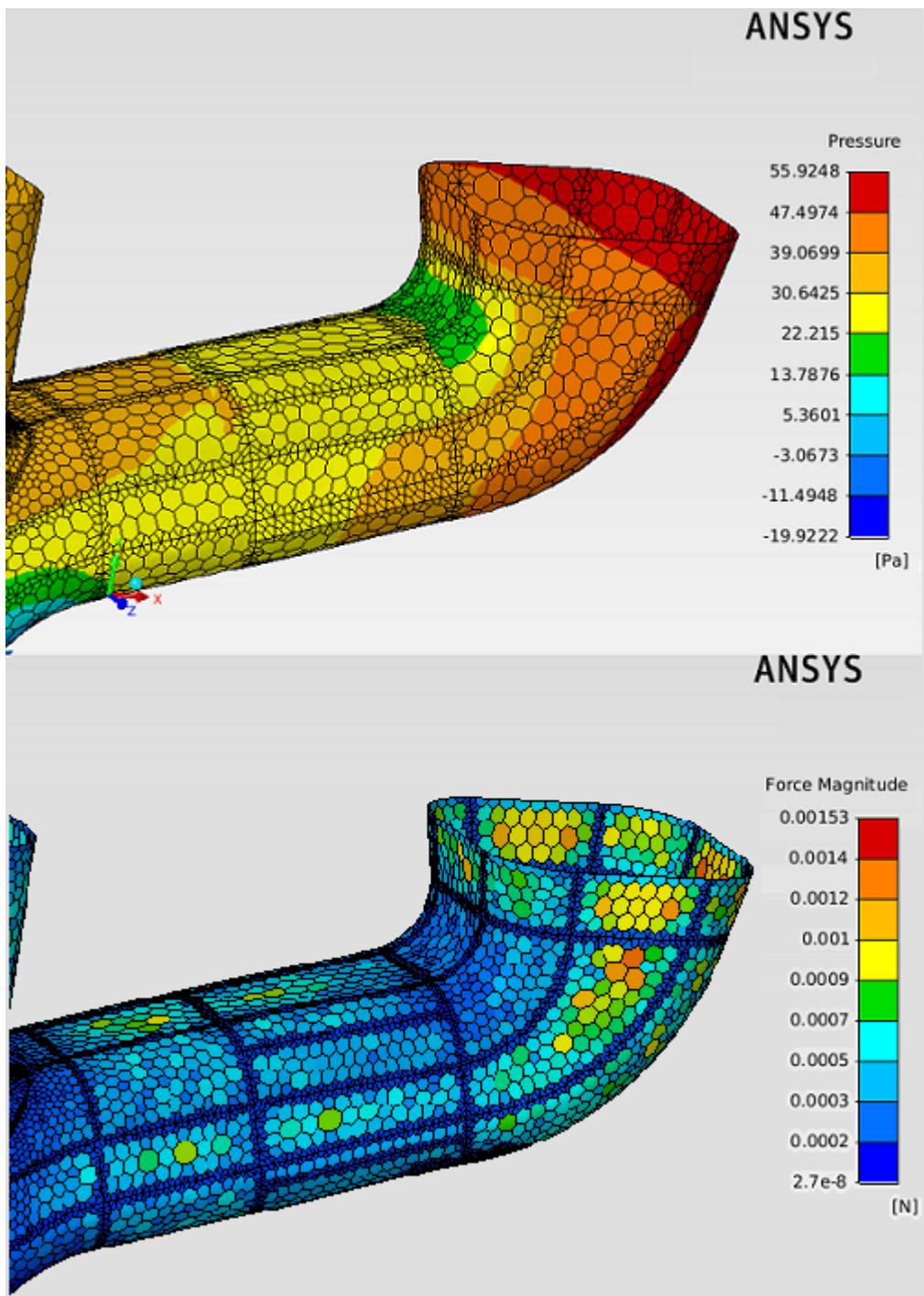
- To generate the resulting contours, click **Evaluate**.

In addition to the qualitative results in your graphics viewer, you can view quantitative data under [Summary](#) on page 577 information.

## 5.1.1.1. Extensive Variables

Contour plots of intensive variables are generally recommended because those values will not reflect the mesh element size. The magnitude of intensive variables does not scale with the mesh element size, whereas the magnitude of extensive variables does.

The images below are of results from the same simulation. It compares contour plots of pressure (intensive variable) to contours of force magnitude (extensive variable). Displaying pressure contours is recommended over force contours, because while the two variables have similar meaning, pressure contours do not depend directly on the mesh element size. The surface mesh is displayed to show that the force magnitude varies in response to the mesh element size.



The table below lists a number of extensive variables.

Table 5.1.1.1.1.1. List of Extensive Variables

Physics type	Extensive Variable
Fluid flow	Force Mass flow Pressure Force Viscous Force Element Wall Distance Element Volume Face Area Orthogonal Quality Element Aspect Ratio Non Overlap Fraction
Thermal (fluid flow)	Surface heat flow
Structural	Force vector Force reaction Contact reaction vector
Thermal (structural)	Heat flow reaction Heat flow
Electric conduction	Current reaction

## 5.1.1.2. Adding a New Contour

### Results > Object > Results > Contour

To create objects that display the results of your simulation, generate a physics solution.

To create a new contour object:

1. In the graphics viewer, right-click and select **Add > Results > Contour**.

For certain simulation types, you can also choose a predefined contour plot from the list to display:

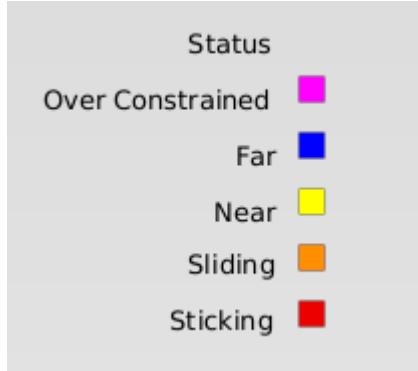
- For structural simulations, **Displacement Magnitude** or **Equivalent Stress**. And, if you selected fatigue analysis when setting up your simulation, **Fatigue Life**.
- For electric simulations, **Electric Potential**.
- For thermal simulations, **Temperature**.

Alternatively, you can create a new contour from the **Results** task page by selecting **Contour** from the drop-down list under **Objects > Results>Add**.

2. On the Contour panel specify the required variable and location information for the contour results.  
For details, see [Specifying a Contour](#) on page 516.
3. To display the contours, right-click in the graphics viewer and select **Evaluate Results**.

### 5.1.1.3. Contact Results

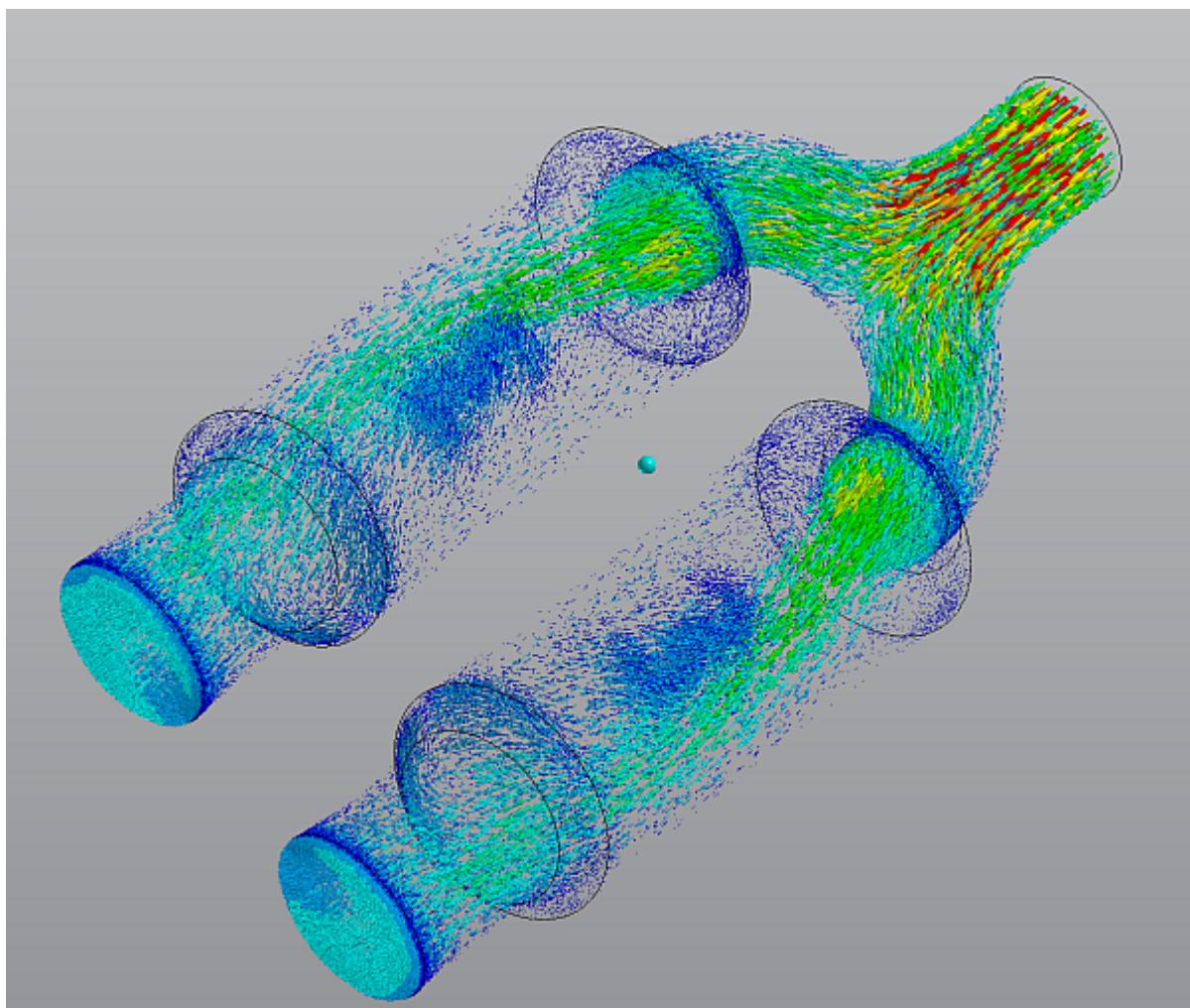
If your simulation contains **Contact** Interface Conditions, you can view the Contact Results by first [Specifying a Contour](#) on page 516 and then choosing one of the following variables to display.

Variable	Description
Pressure	The pressure created on the contact face.
Status	<p>Status codes include:</p> <ul style="list-style-type: none"> <li>• -3 - MPC bonded contact</li> <li>• -2 - MPC no-separation contact</li> <li>• 0 - open and not near contact</li> <li>• 1 - open but near contact</li> <li>• 2 - closed and sliding</li> <li>• 3 - closed and sticking</li> </ul> <p>The labels <b>Over Constrained</b>, <b>Far</b>, <b>Near</b>, <b>Sliding</b>, and <b>Sticking</b> are included in the legend for Status.</p> 
Frictional Stress	Stress created due to friction between the two sides of a contact.
Gap	The distance between the two sides of a contact.
Penetration	The amount of overlap that exists between the two sides of a contact.
Sliding Distance	The amplitude of the accumulated slip increments when the contact status is Sticking or Sliding.
Reaction	Returns the reaction result at contact.

### 5.1.2. Vectors

For both structural and fluids simulations, [you can use vector results](#) to display vector variables, showing the direction and magnitude, on a collection of seed points defined on a surface or volume.

The figure below shows velocity vectors plotted in the physics region of a dual exhaust system. The vectors help you visualize the velocity of particles in the different parts of the exhaust. The vectors in the figure below also help visualize recirculation zones near the walls of the pipes.



### 5.1.2.1. Specifying a Vector

#### Results > Object > Results > Vector

To create objects that display the results of your simulation, generate a physics solution.

Vectors display the direction and magnitude of a vector variable on a collection of points. To specify a vector:

1. Open a **Vector** panel under the **Results** task by either:

- Selecting an existing **Vector** object.
- [Adding a New Vector](#) on page 524.

2. Under **Location**, you can specify a location by either:

- Selecting a location from the drop-down list.
  - Selecting a location in the graphics viewer.
  - Specifying an expression.
- You can also create new reference geometry or Results to be used as locations using the **Create** menu.

For details, see [Defining Locations](#) on page 71.

3. Specify the variable you want to display under **Variable**:

- Selecting an existing variable from the drop-down menu or entering a valid variable name.

- Entering an expression for the variable you want to plot. For details, see [Expressions](#).
  - Defining your own variable by entering a solver variable after the text `UserDefined`, separated by a period. For example, `UserDefined.USUM`.
- 4.** Execute any steps specific to your physics type.

If you want to...	For Physics Type	Then
Display results for a particular natural frequency	Modal Structural	Use the <b>Mode</b> slider. You can review the frequency for the specified mode under <b>Summary &gt; Global &gt; Frequency</b> .
Control at which simulation step the result will be evaluated	Structural, Eigenvalue Buckling	Select a step from <b>Simulation step</b> . For more information on simulation steps, see <a href="#">Simulation Steps</a> on page 493.
Control at which sub-step the result will be evaluated and plotted in graphics	Structural, Eigenvalue Buckling, Polymer extrusion	Use the <b>Substep</b> slider. For more information on substeps for structural simulations, see <a href="#">Specifying Solution Progression for Static Structural</a> on page 440.
View the result at a specific time	Time-Dependent Thermal	Use the <b>Time</b> field or slider. You can enter the time for which you want to view the result, or use the slider to choose the desired time. You will need to evaluate the result at the newly selected time.
Control the distribution of sample points for vectors	Fluid	<p>For <b>Symbol distribution</b>, select one of the following:</p> <ul style="list-style-type: none"> <li>• Uniformly distributed</li> <li>• Volumetric grid sampling</li> <li>• Based on mesh</li> </ul> <p>For details, see <a href="#">Choosing a Vector Distribution Type</a> on page 524.</p>
Display averaged contours for elemental quantities such as stresses and strains	Structural, Thermal, Electric Conduction, Electrostatic	<p>Set <b>Use nodal averaging</b>.</p> <p>Your options are:</p> <ul style="list-style-type: none"> <li>• To display gradients across adjacent elements, disable <b>Use nodal averaging</b>. For this option, contours are evaluated by linear interpolation within each element, and the influence of the adjacent elements is ignored. The resulting un-averaged contours are discontinuous across element boundaries.</li> <li>• To display continuous contours across elemental discontinuities such as element boundaries, enable <b>Use nodal averaging</b>.</li> </ul> <p><b>Note:</b> Contours for fluids simulations always use averaging.</p>

If you want to...	For Physics Type	Then
Display vectors for the specified variable at particular frequency and phase angle	Frequency Response Electromagnetics	Select the desired <b>Frequency</b> from the drop-down menu and specify a <b>Phase angle</b> .  Because the computed field values are complex numbers, the <b>Phase angle</b> you specify determines the particular time or point in the period of the chosen <b>Frequency</b> used to create the plot.

- To generate the resulting vectors, click **Evaluate**.

In addition to the qualitative results in your graphics viewer, you can view quantitative data under [Summary](#) on page 577 information.

### 5.1.2.2. Adding a New Vector

#### Results > Object > Results > Vector

To create objects that display the results of your simulation, generate a physics solution.

To create a new vector:

- In the graphics viewer, right-click and select **Add > Results > Vector**.

For certain simulation types, you can also choose a pre-defined vector plots from the list to display:

- For structural simulations, **Force Reaction**.
- For electric simulations, **Current Density**.

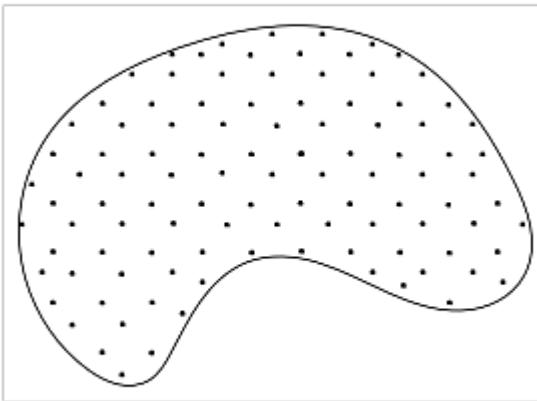
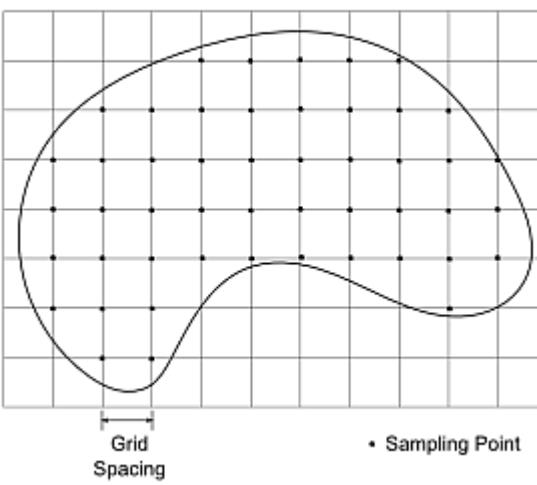
Alternatively, you can create a new vector from the **Results** task page by selecting **Vector** from the drop-down list under **Objects > Results>Add**.

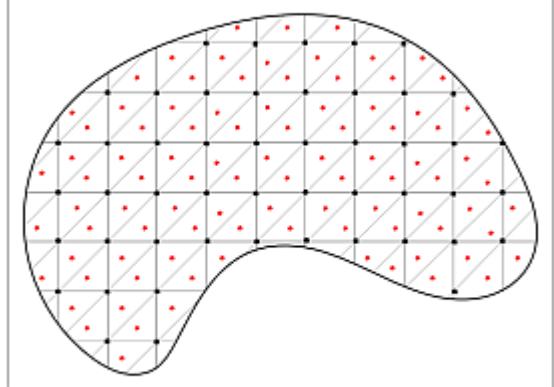
- On the Vector panel specify the required variable, location, and distribution information for the vector results.  
For details, see [Specifying a Vector](#) on page 522.
- To display the vectors, right-click in the graphics viewer and select **Evaluate Results**.

### 5.1.2.3. Choosing a Vector Distribution Type

In a fluid simulation, the primary goal of distributing sampling points is to capture flow patterns near critical regions in the flow field with the fewest number of points. You also want to make sure that there is sufficient coverage in non-critical regions and that you avoid sharp discontinuities across several sample points. You can control the distribution of sample points for vectors by modifying:

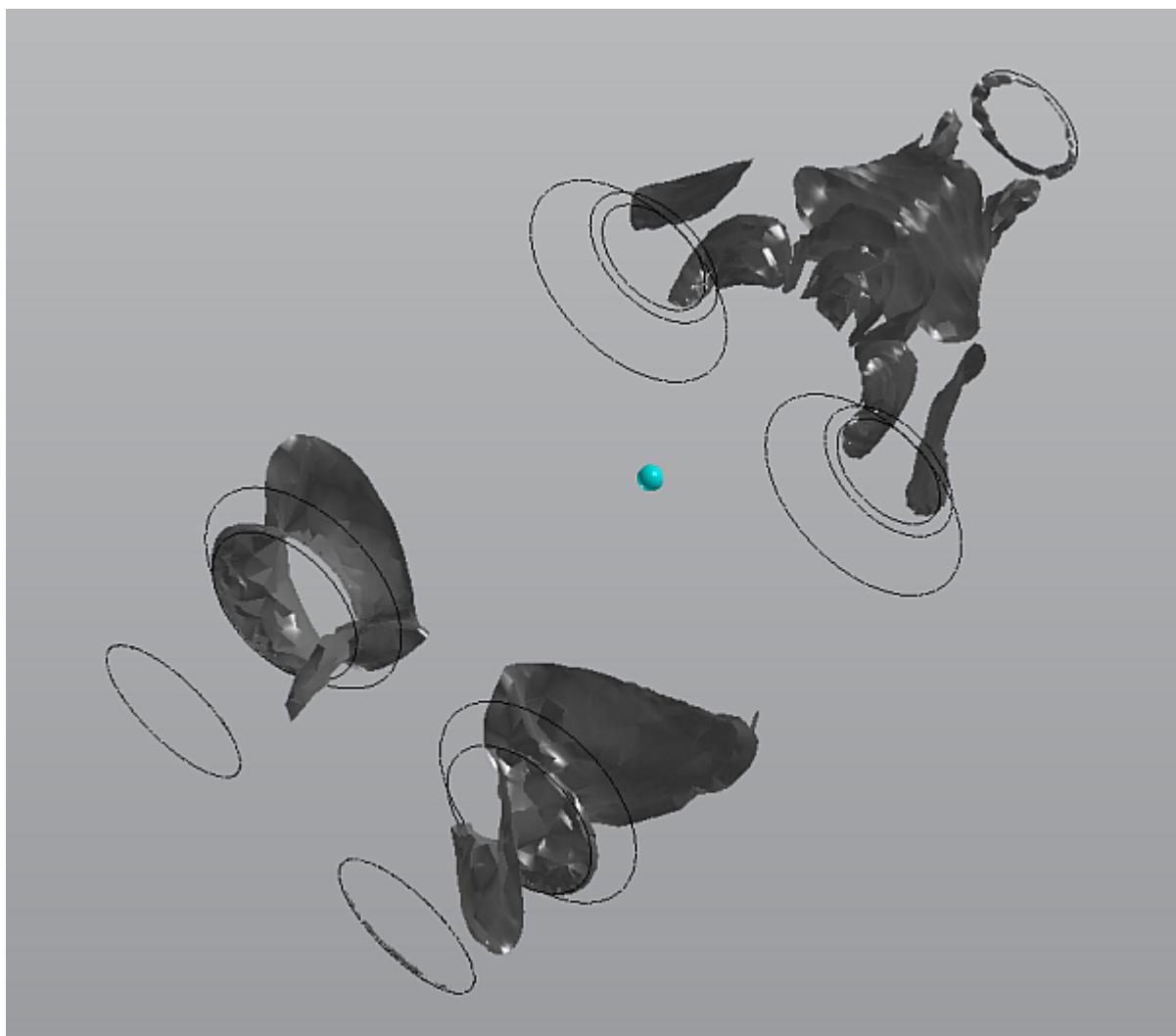
- Symbol distribution** for vectors

Condition	Setting	Example
<p>Create uniformly distributed sample points on a surface independent of mesh density.</p> <p>Control the number of sample points on the surface by specifying an <b>Approximate Number of Points</b>.</p> <p><b>Note:</b> You can specify this option only for surfaces such as planes, topological regions, and isosurfaces.</p>	<b>Uniformly Distributed</b>	
<p>Distribute sample points along a uniform rectangular grid within the volume extents.</p> <p>Specify <b>Grid Spacing</b> to modify the spacing of sample points the physics region.</p>	<b>Volumetric Grid Sampling</b>	

Condition	Setting	Example
<p>Create sample points based on mesh density. You can specify sample points on elements or vertices of the mesh by choosing either <b>At every Nth element</b> or <b>At every Nth vertex</b> under <b>Limit total number of symbols</b>.</p> <p>You can set a reduction factor to reduce the number of elements or vertices to be used as a sample point by specifying sample points <b>At every nth item</b>.</p>	<b>Based on</b> <b>Mesh</b>	 <ul style="list-style-type: none"> <li>• Vertex Sampling Point</li> <li>• Element Sampling Point</li> </ul>

### 5.1.3. Isosurface

You can use an [Isosurface](#) to display a surface of constant value for a specified variable.



### 5.1.3.1. Specifying an Isosurface

#### Results > Object > Results > Isosurface

To create objects that display the results of your simulation, generate a physics solution.

To specify an isosurface:

1. Open a **Isosurfaces** panel under the **Results** task by either:
    - Selecting an existing **Isosurface** object.
    - [Adding a New Isosurface](#) on page 528.
  2. Under **Location**, you can specify a location by either:
    - Selecting a location from the drop-down list.
    - Selecting a location in the graphics viewer.
    - Specifying an expression.
    - You can also create new reference geometry or Results to be used as locations using the **Create** menu.
- For details, see [Defining Locations](#) on page 71.
3. Specify the variable you want to display under **Variable**:

- Selecting an existing variable from the drop-down menu or entering a valid variable name.
- Entering an expression for the variable you want to plot. For details, see [Expressions](#).
- Defining your own variable by entering a solver variable after the text `UserDefined`, separated by a period. For example, `UserDefined.USUM`.

**4.** Execute any steps specific to your physics type.

If you want to...	For Physics Type	Then
Display results for a particular natural frequency	Modal Structural	Use the <b>Mode</b> slider. You can review the frequency for the specified mode under <b>Summary &gt; Global &gt; Frequency</b> .
Control at which simulation step the result will be evaluated	Structural, Eigenvalue Buckling	Select a step from <b>Simulation step</b> . For more information on simulation steps, see <a href="#">Simulation Steps</a> on page 493.
Control at which sub-step the result will be evaluated and plotted in graphics	Structural, Eigenvalue Buckling, Polymer extrusion	Use the <b>Substep</b> slider. For more information on substeps for structural simulations, see <a href="#">Specifying Solution Progression for Static Structural</a> on page 440.
View the result at a specific time	Time-Dependent Thermal	Use the <b>Time</b> field or slider. You can enter the time for which you want to view the result, or use the slider to choose the desired time. You will need to evaluate the result at the newly selected time.

**5.** You can specify the type of isosurface under **Isovalues > Isovalue specification** as either:

- Set Values
- Automatic

For details, see [Choosing an Isovalue Specification](#) on page 529.

**6.** To generate the resulting isosurface, click **Evaluate**.

In addition to the qualitative results in your graphics viewer, you can view quantitative data under **Summary** on page 577 information.

### 5.1.3.2. Adding a New Isosurface

#### Results > Object > Results > Isosurface

To create objects that display the results of your simulation, generate a physics solution.

To create a new isosurface:

**1.** In the graphics viewer, right-click and select **Add > Results > Isosurface**.

Alternatively, you can create a new isosurface from the **Results** task page by selecting **Isosurface** from the drop-down list under **Objects > Results>Add**.

**2.** On the Isosurface panel specify the required variable, location, and distribution information for the isosurfaces.

For details, see [Specifying an Isosurface](#) on page 527.

**3.** To display the isosurfaces, right-click in the graphics viewer and select **Evaluate Results**.

### 5.1.3.3. Choosing an Isovalue Specification

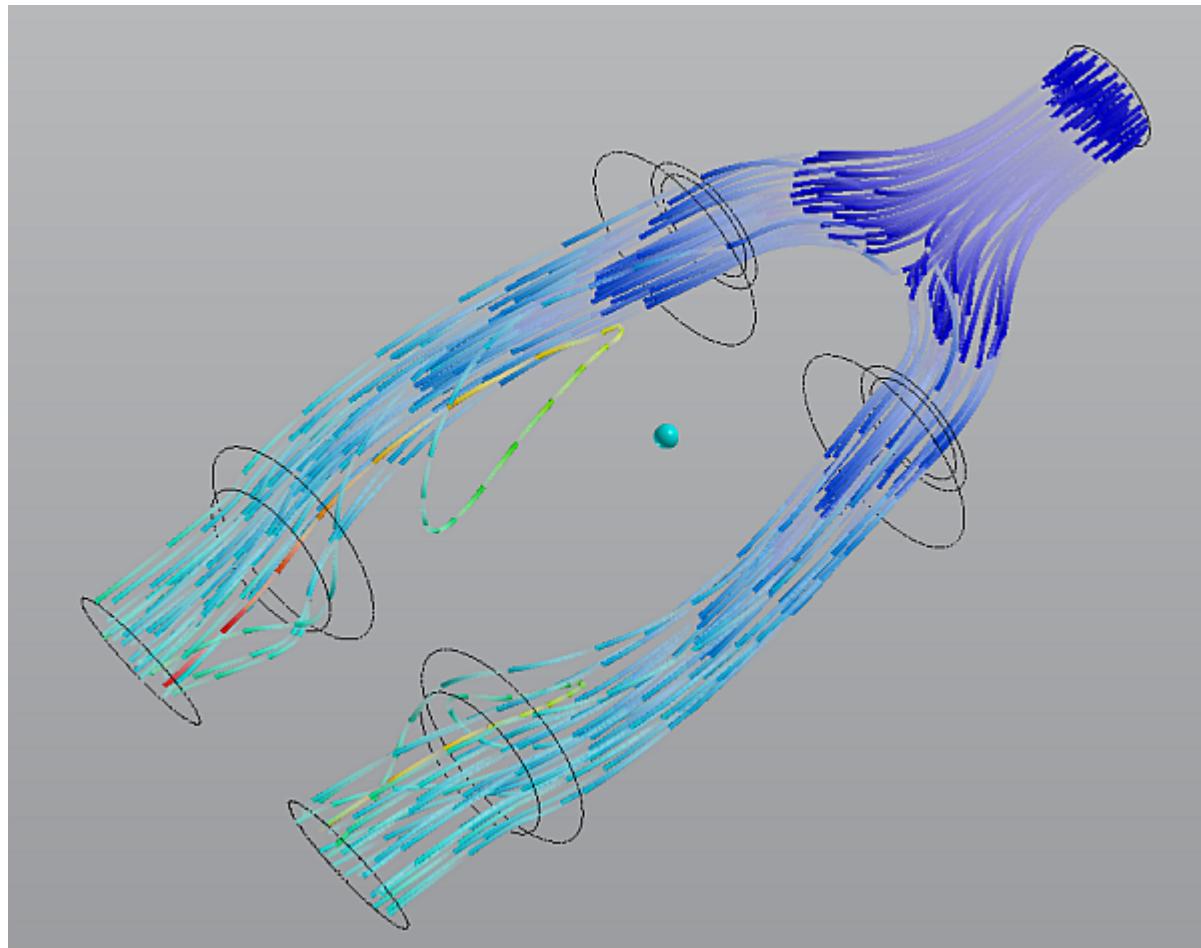
You can control the isosurfaces by modifying the **Isovalue specification** under **Isovalues**.

Condition	Option
Specify numerical values to plot for the given variable.	<b>Set Values</b>
Specify automatic generation of isosurfaces by requesting a particular <b>Number of isovalue</b> s evenly spaced between the minimum and maximum values for the specified variable.	<b>Automatic</b>
<b>Note:</b> The minimum and maximum variable values will not be included in the plotted isosurfaces.	

### 5.1.4. Streamlines

In a fluid flow simulation, [you can use streamlines](#) to visualize the lines traveled by massless particles that are released from one or more surfaces in the physics region. You can also use streamlines to identify the source of a particle for which you know the final destination.

The streamline below displays the path of air particles as they travel through a dual exhaust system. The streamlines help you visualize how the velocity of a particle changes as it passes through various stages of the exhaust.



## 5.1.4.1. Specifying a Streamline

### Results > Streamline

To create objects that display the results of your simulation, generate a fluid flow physics solution.

Streamlines display the path that a massless fluid particle would take through the fluid domain. To specify a streamline:

1. Open a **Streamline** panel under the **Results** task by either:
    - Selecting an existing **Streamline** object.
    - [Adding a New Streamline](#) on page 531.
  2. Specify the starting point or final destination for the streamlines under **Seed location** by either:
    - Selecting a location from the drop-down list.
    - Selecting a location in the graphics viewer.
    - Specifying an expression.
    - You can also create new reference geometry or Results to be used as locations using the **Create** menu.

For details, see [Defining Locations](#) on page 71.
  3. Modify the spacing and density of the seed points; set the **Distribution** by choosing either:
    - Uniformly distributed
    - Volumetric grid sampling
    - Based on mesh

For details, see [Choosing a Streamline Distribution Type](#) on page 531
  4. Set the **Direction** of the streamline as follows:
    - If you would like the streamlines to end up at a given location, set that location as the **Seed location** and set the **Direction** as **Backward**.
    - If you know the starting point of the streamline, set that location as the **Seed location** and set the **Direction** as **Forward**.
    - If you are interested in the streamlines passing through a given location (such as a small body in the middle of a region), or specifying an intermediate location, set that location as the **Seed location** and set the **Direction** as **Forward and Backward**.
  5. Optionally modify the **Appearance** of the Streamline.
  6. To create shorter streamlines, you can reduce the value under **Evaluation Limits > Maximum number of steps** to limit the calculation of the streamline, therefore shortening it. The default **Maximum number of steps** of 1000 will typically result in the maximum possible length for the streamline.
- Note:** Specifying the **Maximum number of steps** along with the **Approximate number of points** used, can potentially have a significant impact on the performance (time) of the calculation. If a large number of points are used (such as with **Volumetric grid sampling**), then you might want to start with a smaller number for the **Maximum number of steps** to prevent the calculation from taking a long time.
7. Optionally, you can control the speed of the streamline pulses using the **Animation > End time factor**. For values above 1, the streamline animation will continue after the evaluation limit or boundary is reached. The effects of this is best seen if you are using the segmented line style under **Appearance > Line Settings > Line style > Segments**.
  8. To generate the streamlines, click **Evaluate**.
  9. To [animate](#) the streamlines, click  in the graphics window.

If you want to modify the duration of the animation playback, go to the **Results** panel. Under **Graphics Control > Animation duration**, enter the length of time you would like the animation to take to complete all the animation loops specified by the **End time factor**.

In addition to the qualitative results in your graphics viewer, you can view quantitative data under [Summary](#) on page 577 information.

## 5.1.4.2. Adding a New Streamline

### Results > Object > Results > Streamline

To create objects that display the results of your simulation, generate a physics solution.

To create a new streamline:

1. In the graphics viewer, right-click and select **Add > Results > Streamline**.

Alternatively, you can create a new streamline from the **Results** task page by selecting **Streamline** from the drop-down list under **Objects > Results>Add**.

2. On the streamline panel specify the required variable, seed location, and distribution information for the streamlines.

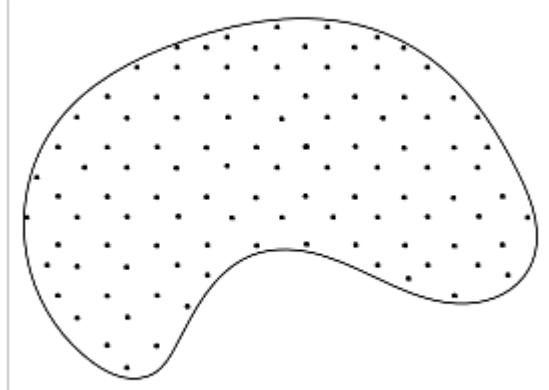
For details, see [Specifying a Streamline](#) on page 530.

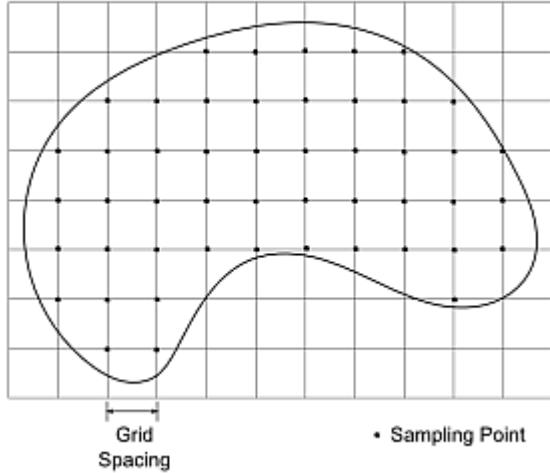
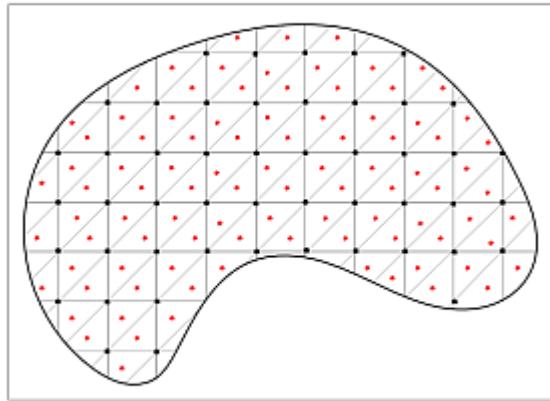
3. To display the streamlines, right-click in the graphics viewer and select **Evaluate Results**.

## 5.1.4.3. Choosing a Streamline Distribution Type

In a fluid simulation, the primary goal of distributing sampling points is to capture flow patterns near critical regions in the flow field with the fewest number of points. You also want to make sure that there is sufficient coverage in non-critical regions and that you avoid sharp discontinuities across several sample points. You can control the distribution of sample points for streamlines by modifying:

- **Seed points > Distribution** for streamlines

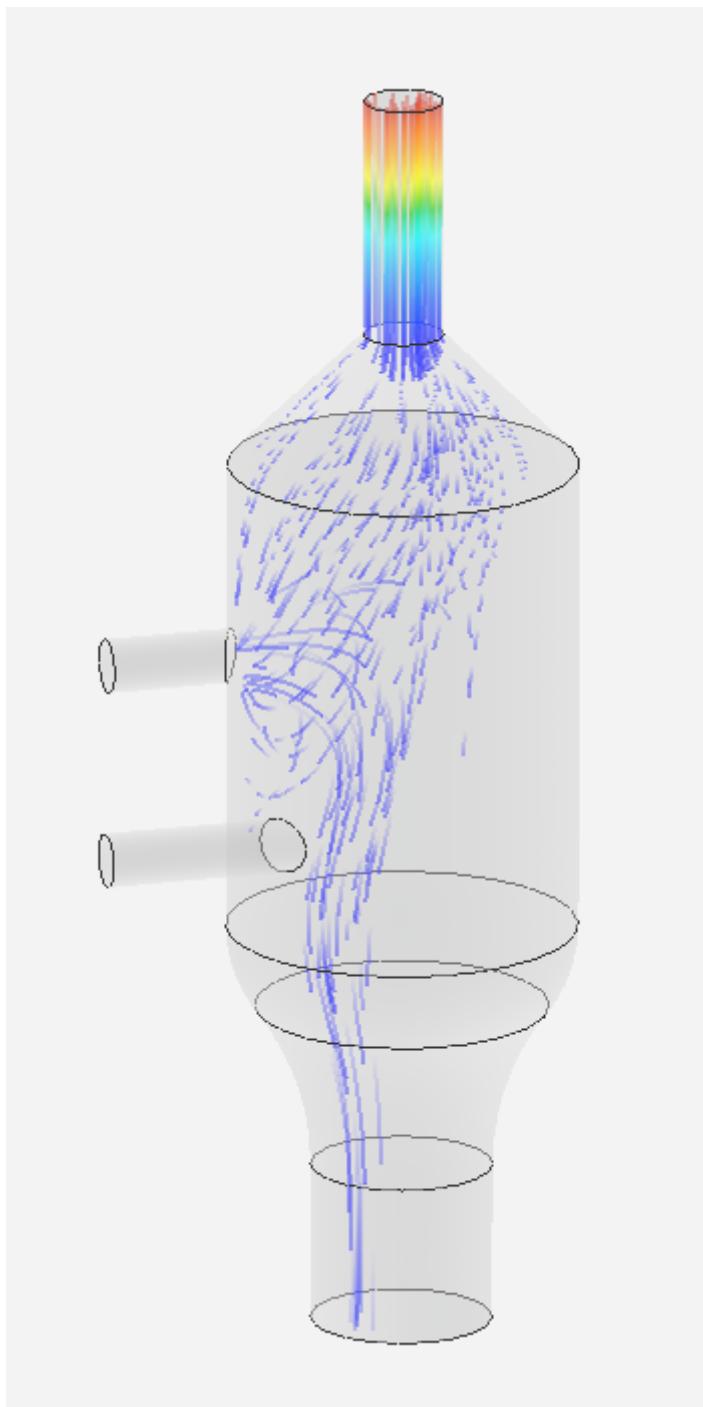
Condition	Setting	Example
Create uniformly distributed sample points on a surface independent of mesh density.	<b>Uniformly Distributed</b>	 <p>• Sampling Point</p>

Condition	Setting	Example
Distribute sample points along a uniform rectangular grid within the volume extents.  Specify <b>Grid Spacing</b> to modify the spacing of sample points the physics region.	<b>Volumetric Grid Sampling</b>	
Create sample points based on mesh <b>Based on density</b> . You can specify sample points <b>Mesh</b> on elements by choosing <b>At every Nth element</b> under <b>Limit total number of symbols</b> .  You can set a reduction factor to reduce the number of elements to be used as a sample point by specifying sample points <b>At every nth item</b> .		

### 5.1.5. Particle Tracks

In a fluid flow simulation, [you can use particle tracks](#) to visualize the path traveled by particles of a discrete phase that are injected at one or more surfaces in the physics region.

The particle tracks below display water droplets being injected at the top (along with a gentle airflow) and higher velocity air flowing in from the side to induce swirling in a mixer.



## 5.1.5.1. Specifying Particle Tracks

### Results > Particle Track

To create objects that display the results of your simulation, generate a fluid flow physics solution with [particle modeling](#).

To display particle tracks:

1. Open a **Particle Track** panel under the **Results** task.

To create a new particle track result object, right-click in the graphics viewer and select **Add > Results > Particle Track**.

2. Select the **Particle material** for which you want to display the particle tracks.
3. Specify the **Location** of the particle tracks by either:
  - Selecting a location from the drop-down list.
  - Selecting a location in the graphics viewer.
  - Specifying an expression.
4. Under **Appearance > Color**, determine if you want to color the displayed particle tracks by any of the different scalar fields. For example, if you select **By diameter** each particle's diameter will be a different color. Optionally, you can modify other settings under **Appearance**.
5. You can control the speed of the particle track pulses using the **Animation > End time factor**. For values above 1, the particle track animation will continue after the boundary is reached. The effects of this is best seen if you are using the segmented line style under **Appearance > Line Settings > Line style > Segments**.
6. To generate the particle tracks, click **Evaluate**.
7. To animate the particle tracks, click  in the graphics window.

If you want to modify the duration of the animation playback, go to the **Results** panel. Under **Graphics Control > Animation duration**, enter the length of time you would like the animation to take to complete all the animation loops specified by the **End time factor**.

In addition to the qualitative results in your graphics viewer, you can view quantitative data under **Summary**.

### 5.1.6. Line Chart

The **Line Chart** plots one or more variables against a position along a line for comparison; for example, moment and share force, velocity and pressure.

- [Specifying a Line Chart](#) on page 534
- [Adding a New Line Chart](#) on page 535
- [Viewing Line Charts](#) on page 535
- [Exporting Line Chart Data](#) on page 537

For more information on the results variables available, see [Results Variables by Physics Type](#) on page 552. For information on the specialized line chart results for linearized stresses, see [Linearized Stresses](#) on page 538.

## 5.1.6.1. Specifying a Line Chart

### Results > Object > Results > Line Chart

To create objects that display the results of your simulation, generate a physics solution.

Line charts display the direction of a variable on a line where the X axis is distance. To specify a line chart:

1. Open a **Line Chart** under the **Results** task by either:
  - Selecting an existing **Line Chart** object
  - [Adding a New Line Chart](#) on page 535
2. Under **Location**, specify a location by any of the following:
  - Selecting a line from the **Location** drop-down list.
  - Selecting a line from the **Location** drop-down list in the graphics viewer.

- Specifying an expression.
  - Using **Create** to create a new line. For details, see [Lines](#) on page 82
3. AIM picks a starting point and direction for the path based on the selected edges, shown graphically with an arrow. To reverse the path definition, enable **Reverse direction**.
  4. Enable **Display as percentage** to normalise axis values to a percentage of the maximum if variables of different units are selected.
  5. To add variables to your chart:
    - a) From the **Add or edit variables** drop-down list, select a variable.
    - b) Click **Add** to add it to the list.
    - c) Repeat for any additional variables.

To remove a selected variable from the **Variables** list, select the check box for it and click the **Delete** icon above. To replace a variable in the **Variables** list, select the check box for it, select another variable from the **Add or edit variables** drop-down list, and click **Replace**.

You can also enter an [expression](#) for the variable you want to plot, or define your own variable by entering a solver variable after the text `UserDefined`, separated by a period. For example, `UserDefined.USUM`.
  6. To generate the resulting line chart, click **Evaluate**.

## 5.1.6.2. Adding a New Line Chart

**Results > Object > Results > Line Chart**

To create objects that display the results of your simulation, generate a physics solution.

To create a new line chart:

1. In the graphics viewer, right-click and select **Add > Results > Line Chart**.

Alternatively, you can create a new line chart from the **Results** task panel by selecting **Line Chart** from the drop-down list under **Objects > Results > Add**.

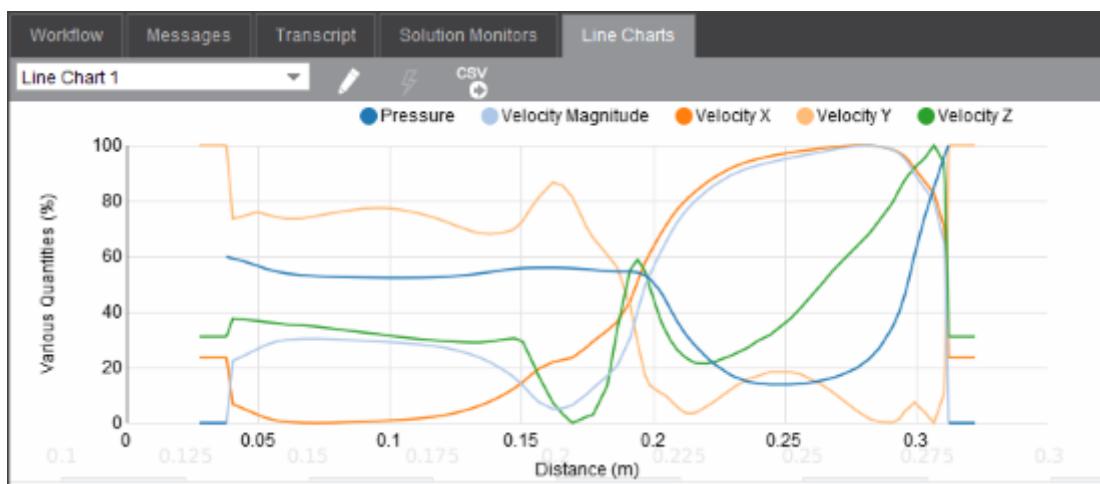
2. On the **Line Chart** panel, specify the required location and variables for the results.
3. To display the resulting line chart, click **Evaluate**.

## 5.1.6.3. Viewing Line Charts

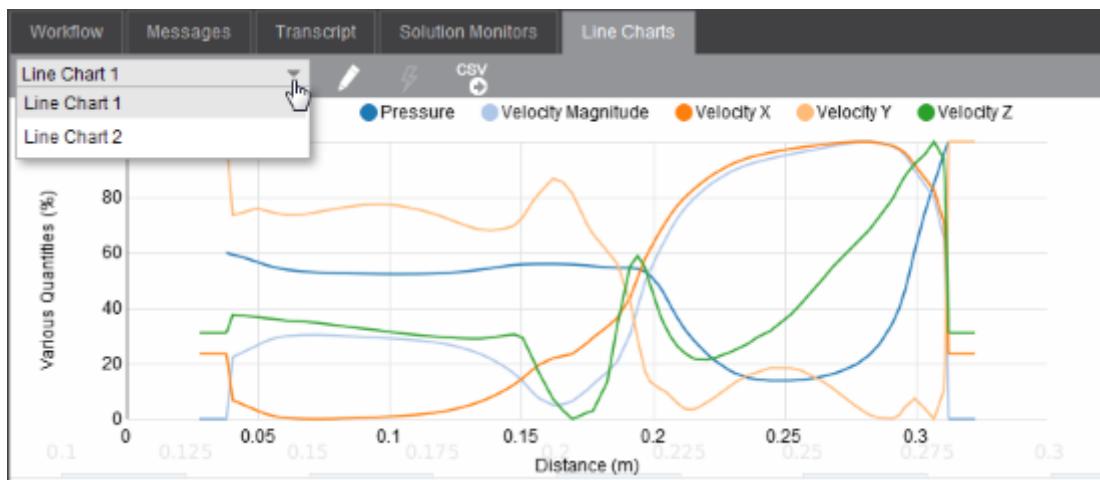
As soon as you define a line chart, a **Line Charts** tab appears in the **View** panel. The tab will display with **No Data Available** until the line chart results is evaluated.

To view a chart:

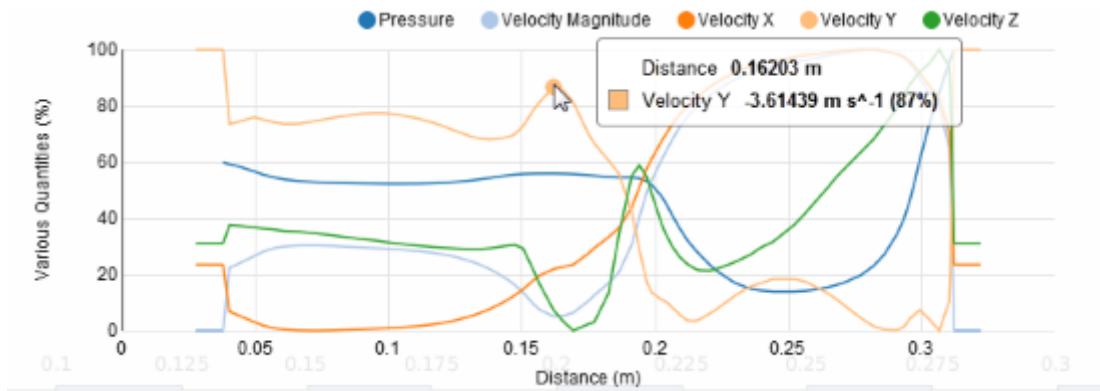
1. Click the **Line Charts** tab. The first chart that was created will be displayed. Expand the view panel to improve viewing.



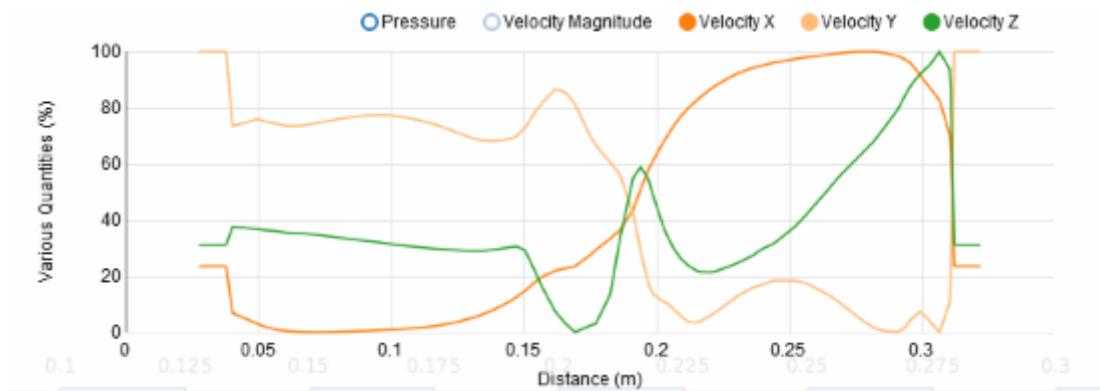
- Select the chart you want to view from the drop-down.



- Hover over a plot on the chart to see the value of that quantity at a particular distance.



- Click a quantity name above the chart to hide or show it in the chart. You can hide all but one of the quantities. If you try to hide all, then all will be shown. If you double-click a quantity name, it will hide all but that one.



## 5.1.6.4. Exporting Line Chart Data

You can export data plotted on a line chart to a comma-delimited CSV file. The data can only be exported after the solution is completed.



1. While viewing a line chart, press .
2. Browse to the folder where you want to store the file and enter a file name.
3. Press **Save**.

## 5.1.7. Shear Moment Diagrams

A shear-moment diagram is a beam result which simultaneously illustrates the distribution of shear forces, bending moments and displacements as a function of arc length along a path consisting of line bodies.

## 5.1.7.1. Specifying a Shear Moment Diagram

### Results > Object > Results > Shear-Moment Diagram

To create objects that display the results of your simulation, generate a physics solution.

To define a shear-moment diagram:

1. Select one or more contiguous beam edges to form a path. These edges should not intersect each other at any point.
2. AIM picks a starting point and direction for the path based on the selected edges, shown graphically with an arrow. To reverse the path definition, enable **Reverse direction**.
3. Enable **Display as percentage** to normalise axis values to a percentage of the maximum if variables of different units are selected.
4. For a multi-step simulation, you can control at which simulation step the result is evaluated by selecting a step from **Simulation step**. For more information on simulation steps, see [Simulation Steps](#) on page 493.
5. For a nonlinear simulation, you can control at which sub-step the result will be evaluated and plotted in graphics using the **Substep** slider. For more information on substeps for structural simulations, see [Specifying Solution Progression for Static Structural](#) on page 440.
6. To add variables to your chart:
  - a) From the **Add or edit variables** drop-down list, select a variable.
  - b) Click **Add** to add it to the list.
  - c) Repeat for any additional variables.

To remove a selected variable from the **Variables** list, select the check box for it and click the **Delete** icon above. To replace a variable in the **Variables** list, select the check box for it, select another variable from the **Add or edit variables** drop-down list, and click **Replace**.

### 5.1.8. Linearized Stresses

The Linearized Stress results calculate membrane, bending, peak, and total stress along a straight line. Evaluation generates 49 sample points along the line within the structural mesh. Component stress values at the points on the line are interpolated from the appropriate element's average corner nodal values. Stress components through the section are linearized by a line integral method and are separated into constant membrane stresses, bending stresses varying linearly between end points, and peak stresses (defined as the difference between the actual total stress and the membrane plus bending combination).

**Note:** Linearized stress evaluation is only supported on the global reference frame. Note that the line must be straight and entirely contained within the model's geometry.

Linearized stress results are available in two forms: as [contour results](#) and as [line charts](#). Contour results show the results at the beginning (inside), mid-length (center), and end (outside) of the line, while the line chart shows the linearized results along the sampling points on the line.

#### 5.1.8.1. Viewing Linearized Stresses as a Line Chart

##### Results > Object > Results > Linearized Stress Chart

Viewing linearized stresses as a line chart shows the linearized results along the sampling points on the line.

To view Linearized results along the line in chart form:

1. Create a **Linearized Stress Chart**.
2. [Set up the line chart](#) as you normally would.
3. Select a **Stress variable**.
4. **Evaluate**.

#### 5.1.8.2. Viewing Linearized Stresses as a Contour Result

##### Results > Object > Results > Contour

Contour results for linearized stress show the results at the beginning (inside), mid-length (center), and end (outside) of the line.

To view linearized stress results as a contour result:

1. Add a contour result with a stress variable and the location set to a line.
2. Select **Linearize**.
3. **Evaluate** your results.

The **Linearized Results** view shows Membrane, Bending, Membrane + Bending, Peak, and Total stress values. The bending stresses are calculated such that the neutral axis is at the midpoint of the line.

## 5.2. Calculated Results

You can perform a variety of exact quantitative calculations at a particular location in the physics domain. Calculated results are performed to obtain different types of information:

- Calculated values enable you to evaluate a single-value quantity within a physics region.
- A summary of the calculated minimum, maximum, and average values allow you to capture quantitative information for the global range and averages on Results objects.

### 5.2.1. Calculated Value

A Calculated Value result enables you to evaluate a single-value quantity within a physics region. [You can use these single value quantities](#), along with other results objects, to evaluate your design in more detail.

## 5.2.1.1. Specifying a Calculated Value

### Results > Object > Results > Calculated Value

To create objects that display the results of your simulation, generate a physics solution.

Calculated Value is used to evaluate single-value results for the physics. To create a single-value result:

- Open a **Calculated Value** panel under the **Results** task by either:

- Selecting an existing **Calculated Value** object.
- [Adding a New Calculated Value](#) on page 540.

- Under **Method**, set the type as either:

- Function calculator
- User-defined [expression](#) (not available for electromagnetic simulations)

For details, see [Choosing a Calculated Value Method](#) on page 541.

- Execute any steps specific to your physics type.

If you want to...	For Physics Type	Then
Calculate a value for a particular natural frequency	Modal Structural	Use the <b>Mode</b> slider.
Control at which simulation step the result will be evaluated	Structural, Eigenvalue Buckling	Select a step from <b>Simulation step</b> . For more information on simulation steps, see <a href="#">Simulation Steps</a> on page 493.
Control at which sub-step the result will be evaluated and plotted in graphics	Structural, Eigenvalue Buckling, Polymer extrusion	Use the <b>Substep</b> slider. For more information on substeps for structural simulations, see <a href="#">Specifying Solution Progression for Static Structural</a> on page 440.
View the result at a specific time	Time-Dependent Thermal	Use the <b>Time</b> field or slider. You can enter the time for which you want to view the result, or use the slider to choose the desired time. You will need to evaluate the result at the newly selected time.

If you want to...	For Physics Type	Then
Visualize results on multiple frames of reference	Structural, Eigenvalue Buckling, Thermal, Electric Conduction, Electrostatic	<p>Specify the reference frame under <b>Relative to</b>. Your options are:</p> <ul style="list-style-type: none"> <li>Selecting an existing reference frame from the drop-down menu.</li> <li><a href="#">Creating Reference Frames on page 77</a>.</li> <li>For Joint results, selecting the <b>Solution Reference Frame</b> option. Frame-dependent variables are calculated relative to the selected Joint's local reference frame. When post-processing multiple joints, this option ensures each Joint's respective local reference frame is used to calculate the results.</li> </ul>
Display averaged contours for elemental quantities such as stresses and strains	Structural, Eigenvalue Buckling, Thermal, Electric Conduction, Electrostatic	<p>Set <b>Use nodal averaging</b>.</p> <p>Your options are:</p> <ul style="list-style-type: none"> <li>To display gradients across adjacent elements, disable <b>Use nodal averaging</b>. For this option, contours are evaluated by linear interpolation within each element, and the influence of the adjacent elements is ignored. The resulting un-averaged contours are discontinuous across element boundaries.</li> <li>To display continuous contours across elemental discontinuities such as element boundaries, enable <b>Use nodal averaging</b>.</li> </ul> <p><b>Note:</b> Contours for fluids simulations always use averaging.</p>
Calculate a value at a particular frequency	Frequency response Electromagnetics	Select the desired <b>Frequency</b> from the drop-down menu.

- To generate the resulting single-value results, click **Evaluate**.

The calculated value result, whether generated by the **Function calculator** or a **User-defined expression**, is displayed on the panel under **Value**. It is also displayed in the graphics window at the specified location, but you can click and drag the value anywhere in the graphics window. Clicking the displayed value opens the **Calculated Value** property panel where you can modify your settings if needed.

For nonlinear and time-dependent simulations, you can view history data in chart form by clicking ▶ and



then the **Show chart** icon. You can monitor any of your calculated values by [creating charts](#) and [viewing](#) them in the **Solution Monitors** tab.

## 5.2.1.2. Adding a New Calculated Value

### Results > Object > Results > Calculated Value

To create objects that display the results of your simulation, generate a physics solution.

To create a new single-value result:

1. In the graphics viewer, right-click and select **Add > Results > Calculated Value**.

For certain simulation types, you can also choose a pre-defined calculated value objects from the list to display:

- For electric simulations, **Current Reaction**.
- For thermal simulations, **Heat Flow Reaction**.
- For static electromagnetics simulations, **Force Magnitude** and **Torque** are available if force and torque computations were added as Physics Options in the Physics task. All components of the computed force and torque are real numbers and can be evaluated directly.
- For frequency response electromagnetics simulations, **AC Force Magnitude**, **DC Force Magnitude**, and **Torque** are available if force and torque computations were added as **Physics Options** in the Physics task. When selecting a Variable to evaluate, note that all components of the computed DC Force (DCForce.x, DCForce.y, DCForce.z and DCForce.mag) are real numbers and can be evaluated directly. For AC Force, ACForce.mag is a real number, and can be evaluated directly; but ACForce.x, ACForce.y and ACForce.z are complex numbers and cannot be used as-is. To extract a real number for these, you can apply any of the following operators: real, imag, phase, or mag.

Alternatively, you can create a new single-value result from the **Results** task page by selecting **Calculated Value** from the drop-down list under **Objects > Results>Add**.

2. On the Calculated Value panel specify the required method, location, function, frequency (available for frequency response magnetics designs only), and variable information, or create a user-defined expression. For details, see [Specifying a Calculated Value](#) on page 539.
3. To display the single value results, click **Evaluate** on the Calculated Value panel.

To create and monitor a calculated value, follow the steps outlined in [Creating a Monitor Chart](#) on page 499. Any time you modify a calculated value, you will need to update both the physics solution and the results.

### 5.2.1.3. Choosing a Calculated Value Method

Calculated Value is used to evaluate single-value results for the physics. You can control your calculation type by modifying the **Method** in the Calculated Value panel.

Option	Description
<b>Function calculator</b>	You need to specify: <ul style="list-style-type: none"> <li>• Location on which to evaluate the function.</li> <li>• Function you want to evaluate. For details on the available functions, see <a href="#">Quantity Functions</a>.</li> <li>• Variable. For a list of available variables, see <a href="#">Variables</a>.</li> </ul>
<b>User-defined expression</b>	You can manually enter an expression using this method. For details on the expression syntax, see <a href="#">Expressions</a> . This method is not available for electromagnetic simulations.

### 5.2.1.4. Calculating a Pressure Drop

#### Results > Pressure Drop

To compute the drop in pressure between the two specified locations:

1. Select the location of high pressure. You can choose faces in your geometry, or a single point or plane. An expression can be used to select multiple planes.
2. Select the location of low pressure. You can choose faces in your geometry, or a single point or plane. An expression can be used to select multiple planes.
3. Click **Evaluate** to compute and display the pressure drop.

When faces or planes are selected the area weighted average pressure at that location will be computed. When a point is selected, a probe of pressure will be used. To compute the pressure drop, the value at the high pressure location, will have the low pressure subtracted from it.

## 5.2.1.5. Adding a Calculated Value to a Monitor Chart

**Results > Object > Results > Calculated Value**

To add a calculated value to a [monitor chart](#):



1. Click the arrow to the right of the **Value** field and select to open a pop-up box that allows you to assign the value to a monitor chart.
2. Select the chart(s) that you would like the calculated value to appear on, then click **OK**.

appears to the right of the **Value** field to indicate that the value is being tracked on a solution monitor chart. Click the icon to change which chart the value appears on.

You can only **Cancel** the pop-up box if no changes have been made to the selections.

3. To create a new chart that contains that calculated value, click **Add Chart**.

## 5.2.1.6. Examples of Calculated Value Monitors

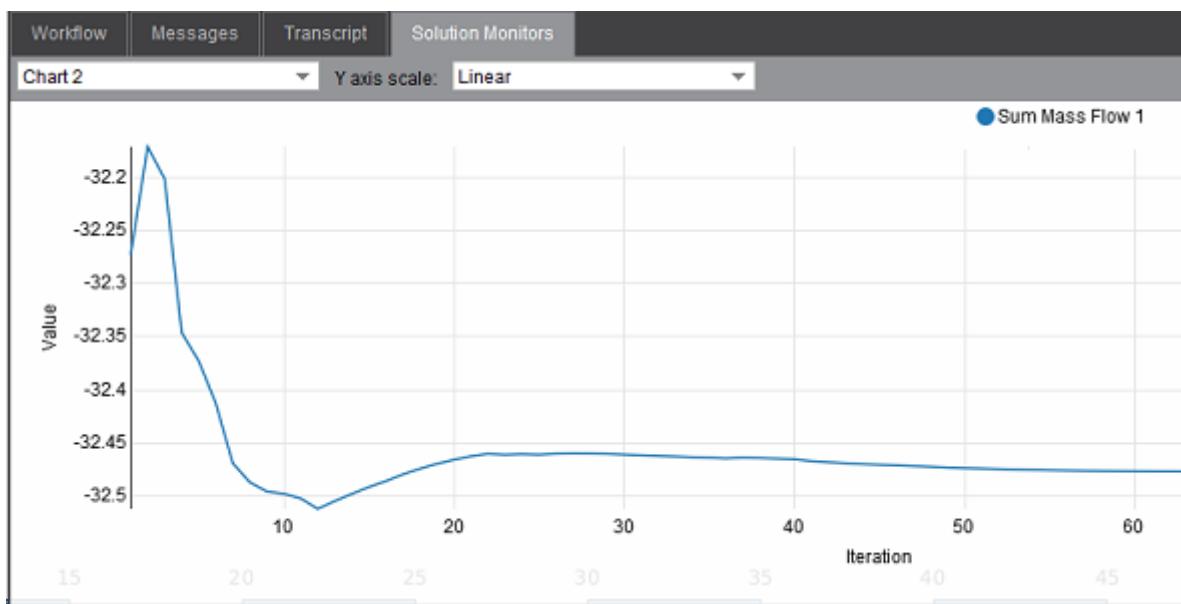
Calculated value monitor charts can be used to help [judge convergence](#), particularly in situations where residuals do not fully converge due to a localized instability. The solution may still be reliable provided relevant quantities of interest have stabilized.

The following examples demonstrate how to set up and create calculated values that are key to specific applications for monitoring purposes.

## Mass Flow Rate Through a Pressure-Specified Outflow Boundary

To create a monitor of mass flow rate through a pressure outlet:

1. [Add a calculated value](#) from the Results task.
2. Select an outlet for the **Location**.
3. Select **Sum** as the **Function** with a **Weight type** of **Simple**.
4. Set the **Variable** to **Mass flow**.
5. [Include a monitor](#) by expanding the arrow next to **Value** and selecting .
6. **Evaluate** the results and view the calculated value chart in the **Solution Monitors** tab (note that the Y axis was set to a linear scale).

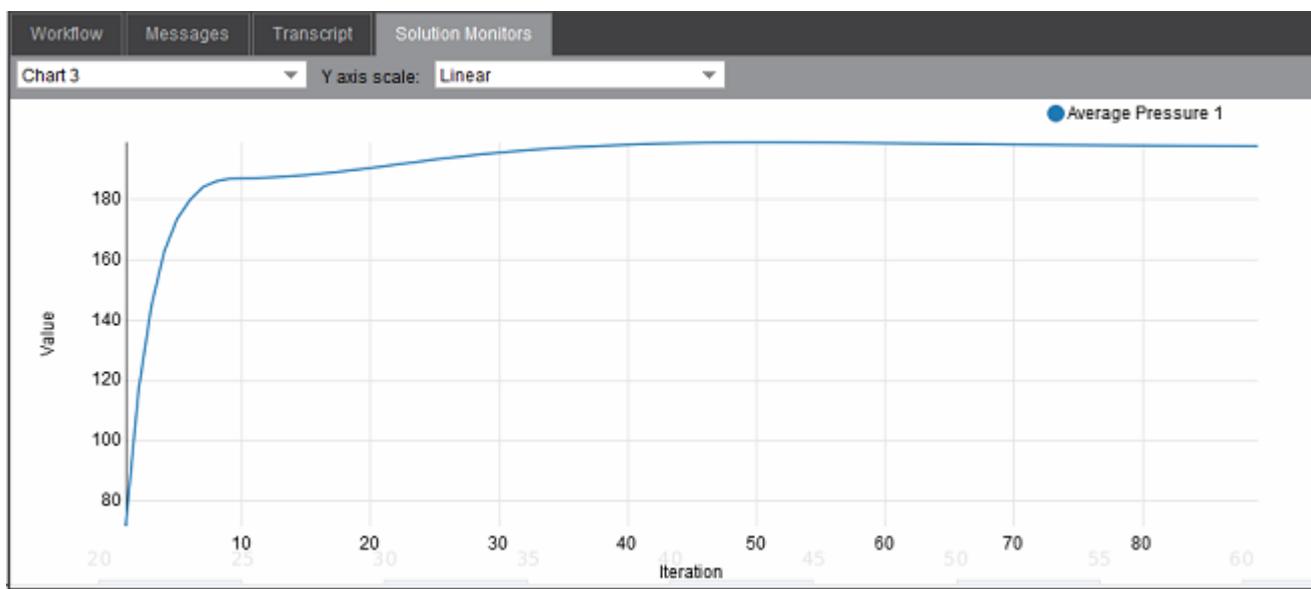


The mass flow rate levels off at -32.4776 kg/s and is no longer changing. You can therefore accept the solution as it is.

## Average Pressure at Velocity-Specified or Mass Flow-Specified Inflow Boundary

To create a monitor of average pressure through an inlet:

1. Add a calculated value from the Results task.
2. Select an inlet for the **Location**.
3. Select **Average** as the **Function** with a **Weight type** of **Simple**.
4. Set the **Variable** to **Pressure**.
5. **Include a monitor** by expanding the arrow next to **Value** and selecting .
6. **Evaluate** the results and view the calculated value chart in the **Solution Monitors** tab (note that the Y axis was set to a linear scale).

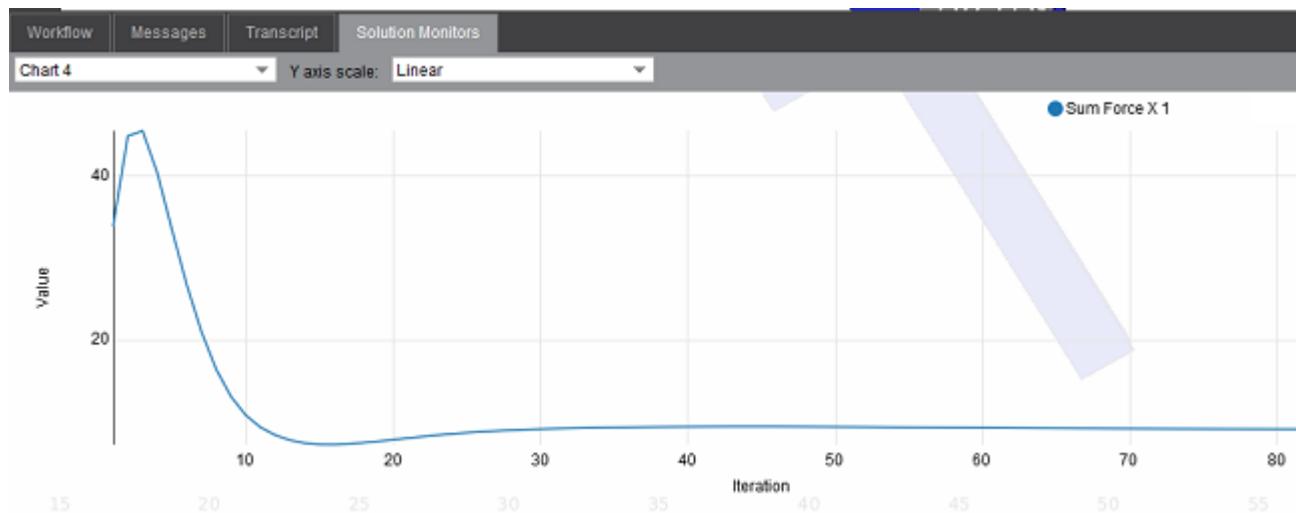


The average pressure levels off and the solution stabilizes at 197.59 Pa.

## Force Components on a Wall

To create a monitor of force on a wall:

1. Add a calculated value from the Results task.
2. Select a wall for the **Location**.
3. Select **Sum** as the **Function** with a **Weight type of Simple**.
4. Set the **Variable** to a force vector component such as **Force.x**.
5. Include a monitor by expanding the arrow next to **Value** and selecting .
6. Evaluate the results and view the calculated value chart in the **Solution Monitors** tab (note that the Y axis was set to a linear scale).

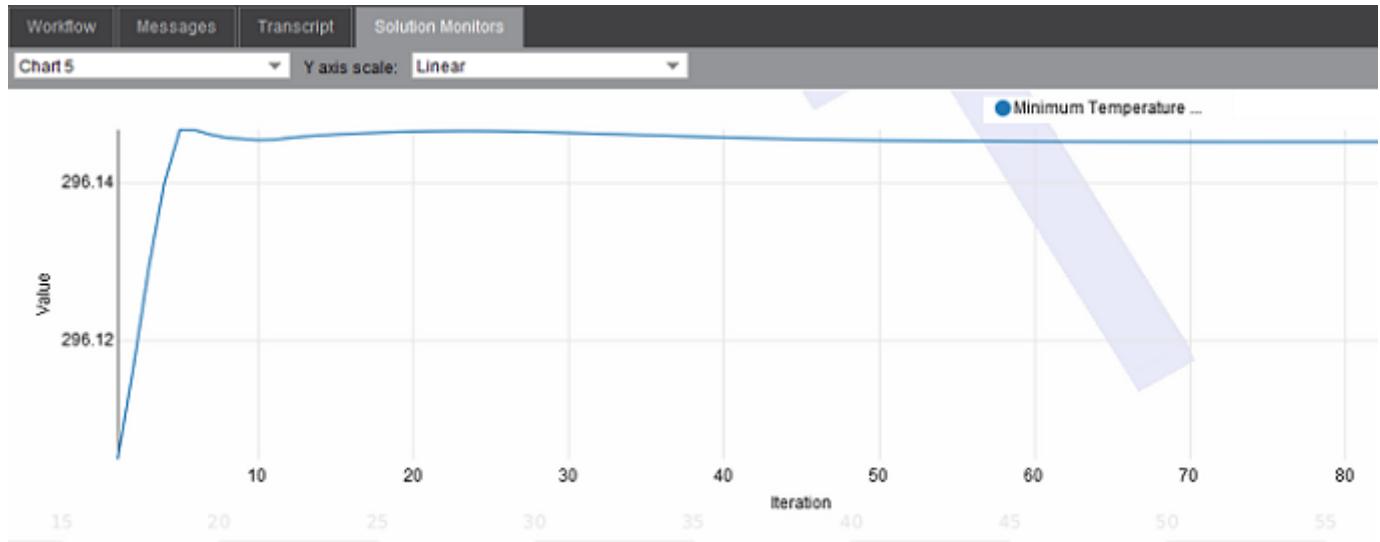


The force at the wall levels off at 9.0736 N and the solution is no longer changing.

# Minimum Temperature in a Body

To create a monitor of minimum temperature in a body:

1. Add a calculated value from the Results task.
2. Select a body or physics region for the **Location**.
3. Select **Minimum** as the **Function**.
4. Set the **Variable** to **Temperature**.
5. **Include a monitor** by expanding the arrow next to **Value** and selecting .
6. **Evaluate** the results and view the calculated value chart in the **Solution Monitors** tab (note that the Y axis was set to a linear scale).

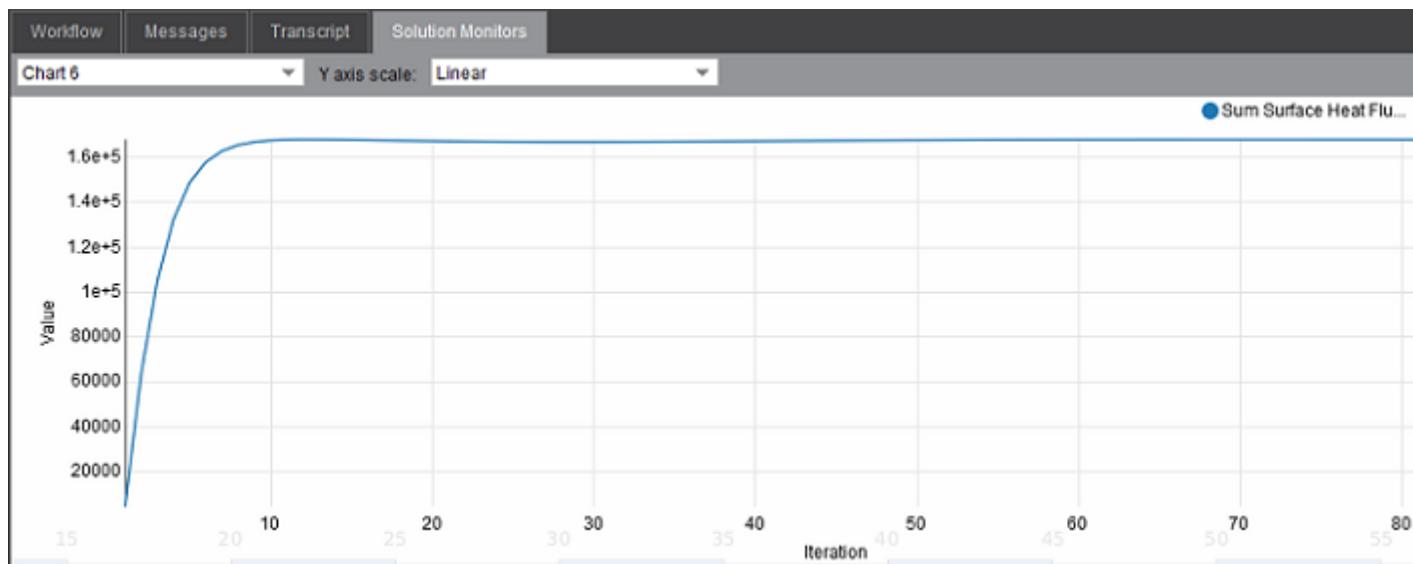


The minimum temperature in the body levels off at 296.15 K and the solution stabilizes.

# Heat Flow Through a Boundary Face

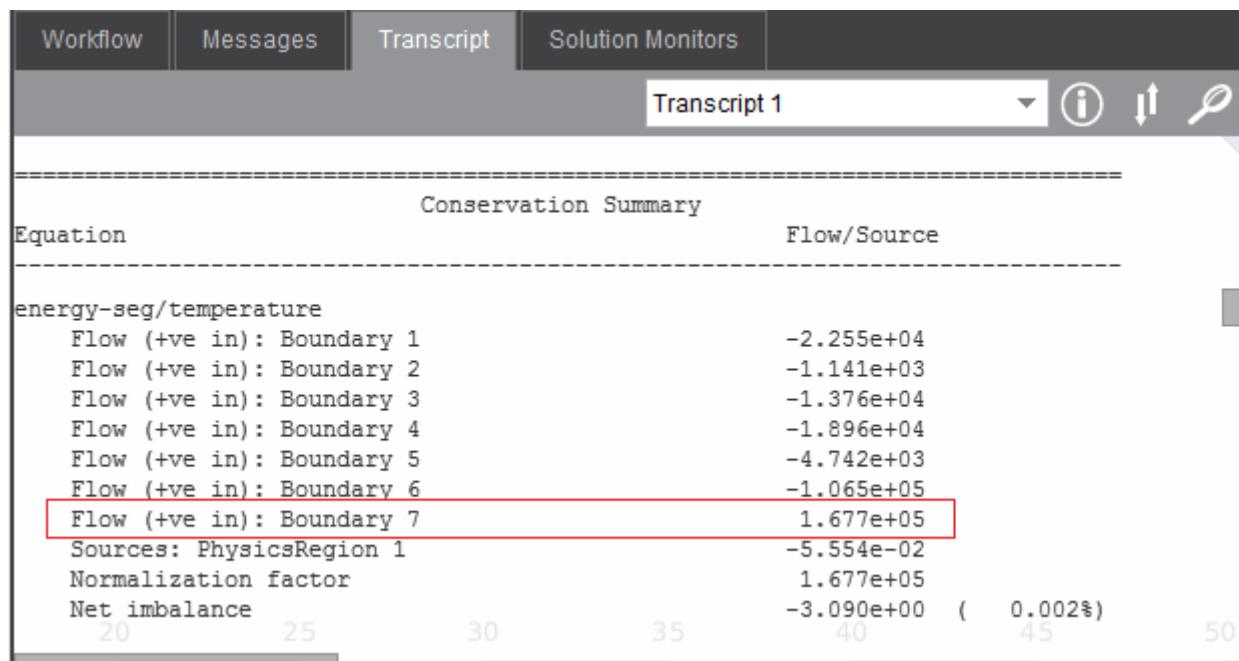
To create a monitor of heat flow through a boundary:

1. Add a calculated value from the Results task.
2. Select a face for the **Location**. For this example, a non insulated wall is used.
3. Select **Sum** as the **Function** with a **Weight type** of **Area**.
4. Set the **Variable** to **Surface Heat Flux**.
5. **Include a monitor** by expanding the arrow next to **Value** and selecting .
6. **Evaluate** the results and view the calculated value chart in the **Solution Monitors** tab (note that the Y axis was set to a linear scale).



The surface heat flow at the wall levels off at  $1.6765E+05$  W.

This boundary flow is also shown in the **Conservation Summary** section of the transcript, together with all other flows and sources, and the degree to which global conservation is achieved.



In the transcript, you will see that the value of the stabilized surface heat flow of the wall (Boundary 7) is represented in the calculated value monitor chart. Furthermore, the Net imbalance of the energy equation is very close to zero, signifying that the solution has converged.

### 5.2.2. Table Results

For both static and frequency response magnetics designs, you can use **Table** results objects to perform a variety of **Matrix** and **Loss** quantity calculations, and to display **Force components** results for any **Force Computations** present in the design.

## 5.2.2.1. Matrix Table Results

Matrix table results can be computed for both static and frequency response magnetics designs.

- For static magnetics designs, the solver uses the currents you specified when setting up **Current** conditions and **Terminals** to compute the various matrix values. By default, the **Table** panel is set to compute **Inductance** matrix values for the conductors in the design. The inductance matrix for a magnetics design represents quantities associated with the magnetic flux linkage between current loops. Each conductor in the matrix is assumed to be part of an independent current loop. The contributions of each loop to the inductance matrix are modeled using numerical matrix manipulation techniques. **Flux Linkage** between conductors, and the **Inductive Coupling Coefficient** between conductors can also be computed. The inductive coupling coefficient indicates how much flux in one coil is linked with another coil. This value will be 1 when coupling is 100%.
- For frequency response magnetics designs, in addition to the currents you specified when setting up **Current** conditions and **Terminals**, the solver also uses the frequency value (or range of values) you specified in the **Physics** panel **Settings** to compute the various matrix values for each frequency. For frequency response designs, the Table panel is set by default to compute **Resistance** matrix values (representing the ohmic losses in the conductors) for a selected **Frequency**. **Inductance** values and the **Inductive Coupling Coefficient** between conductors can also be computed for a selected **Frequency**.

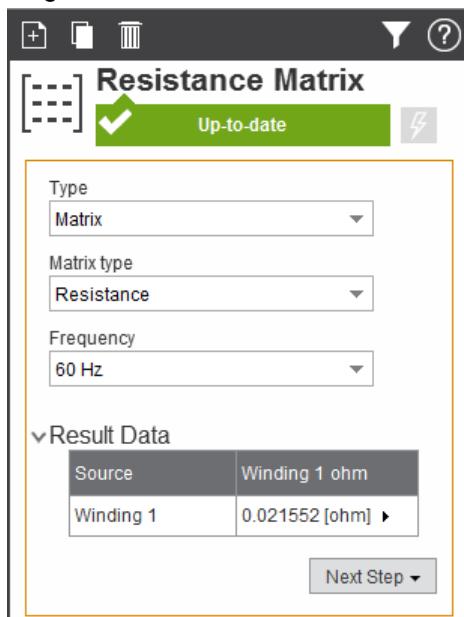
### 5.2.2.1.1. Specifying a Matrix Table

**Results > Object > Results > Table**

To create objects that display the results of your simulation, generate a physics solution.

To specify a matrix table:

- Open a matrix **Table** panel under the **Results** task by either:
  - Selecting an existing matrix **Table** object.
  - Add a new matrix table.
- Specify the **Matrix type**.
- For frequency response simulations only, specify the **Frequency**.
- To generate the results data, click **Evaluate**. An example is shown below.



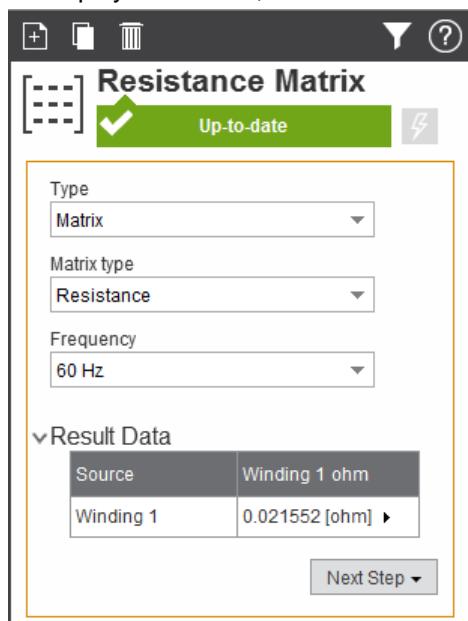
## 5.2.2.1.2. Adding a New Matrix Table

### Results > Object > Results > Table

To create objects that display the results of your simulation, generate a physics solution.

To create a new matrix table:

1. In the graphics viewer, right-click and select **Add > Results > Table**.
  - For static simulations, the **Type** is set to **Matrix** and the **Matrix type** is set to **Inductance** by default.
  - For frequency response simulations, the **Type** is set to **Matrix** and the **Matrix type** is set to **Resistance** by default.
- Alternatively, you can create a new matrix table from the **Results** task page by selecting **Table** from the drop-down list under **Objects > Results>Add**.
2. On the table panel, specify the required **Matrix type**.
3. For frequency response magnetics simulations only, specify the **Frequency**.
4. To display the results, click **Evaluate** on the matrix table panel. An example is shown below.



## 5.2.2.2. Loss Data Table Results

For both static and frequency response magnetics designs, **Total loss** results are associated with loss density fields. Total loss computations use the total integrated loss output from the solver averaged over two consecutive times. For frequency response designs only, Total loss is computed for a selected **Frequency**.

### 5.2.2.2.1. Specifying a Loss Data Table

#### Results > Object > Results > Table

To create objects that display the results of your simulation, generate a physics solution.

To specify a loss data table:

1. Open a matrix **Table** panel under the **Results** task by either:

- Selecting an existing matrix **Table** object.
  - Add a new loss data table.
2. The only selection for **Loss type** is **Total loss**.
  3. For frequency response simulations only, specify the **Frequency**.
  4. To generate the results data, click **Evaluate**.

## 5.2.2.2. Adding a New Loss Data Table

### Results > Object > Results > Table

To create objects that display the results of your simulation, generate a physics solution.

To create a new loss data table:

1. In the graphics viewer, right-click and select **Add > Results > Table**.
  - For static simulations, initially the **Type** is set to **Matrix** and the **Matrix type** is set to **Inductance** by default.
  - For frequency response simulations, initially the **Type** is set to **Matrix** and the **Matrix type** is set to **Resistance** by default.

Alternatively, you can create a new matrix table from the **Results** task page by selecting **Table** from the drop-down list under **Objects > Results>Add**.

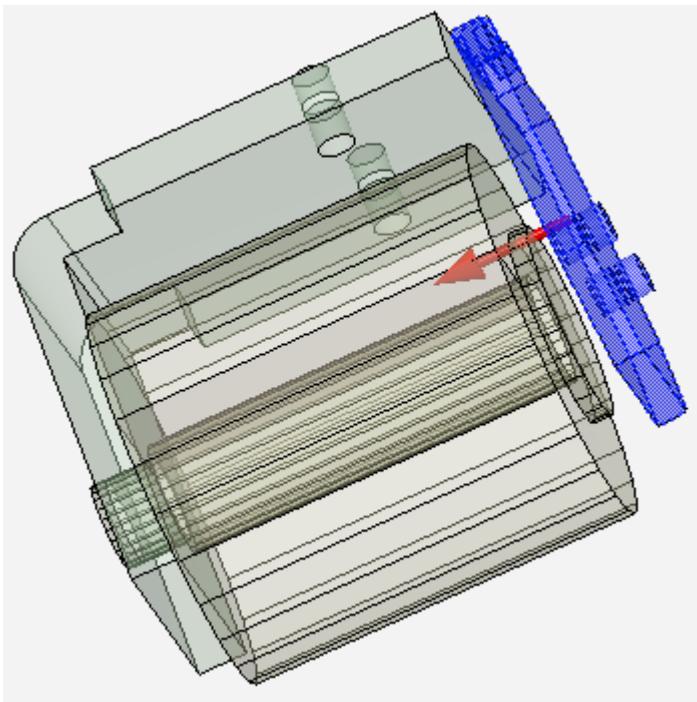
2. On the table panel, change the **Type** to **Loss data**.
3. The only selection for **Loss type** is **Total loss**.
4. For frequency response magnetics simulations only, specify the **Frequency**.
5. To display the results, click **Evaluate** on the matrix table panel.

## 5.2.2.3. Force Components Table Results

Tabular results of the X, Y, Z, and Magnitude components of [Force Computations](#) can be displayed in both static and frequency response magnetics designs.

- For static magnetics designs that include force computations, you can display results for the components of **Force** for a selected Force Computation **Location**.
- For frequency response designs, you can display results for the components of either **AC force** or **DC Force** for a selected Force Computation **Location**. Note that, for frequency response designs only, DC force results are displayed for a selected **Frequency**, and AC force results for a selected **Frequency** and **Phase angle**.

In the graphics view, a red arrow is shown to indicate the direction of the computed force.



### 5.2.2.3.1. Specifying a Force Components Table

#### Results > Object > Results > Table

To create objects that display the results of your simulation, generate a physics solution.

To specify a force components table:

1. Open a force components **Table** panel under the **Results** task by either:
  - Selecting an existing force components **Table** object.
  - [Adding a new force components table](#).
2. For **Location** select the desired Force Computation from the list.
3. For frequency response simulations only, specify either **DC force** or **AC force** as the **Force type**.
4. For frequency response simulations only, specify the **Frequency**.
5. If the **Force type** is **AC force**, specify a **Phase angle**.  
Because the computed force values are complex numbers, the **Phase angle** you specify determines the particular time or point in the period of the chosen **Frequency** used to compute AC force results.
6. To generate the results data, click **Evaluate**.

### 5.2.2.3.2. Adding a New Force Components Table

#### Results > Object > Results > Table

To create objects that display the results of your simulation, generate a physics solution.

To create a new force components table:

1. In the graphics viewer, right-click and select **Add > Results > Table**.
  - For static simulations, initially the **Type** is set to **Matrix** and the **Matrix type** is set to **Inductance** by default.

- For frequency response simulations, initially the **Type** is set to **Matrix** and the **Matrix type** is set to **Resistance** by default.

Alternatively, you can create a new force components data table from the **Results** task page by selecting **Table** from the drop-down list under **Objects > Results>Add**.

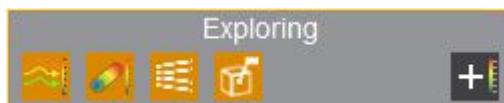
## 2. On the table panel, change the **Type** to **Force components**.

- For static designs, select the desired Force Computation from the **Location** list.
- For frequency response designs, select the desired Force Computation from the **Location** list, then specify the desired **Force type** (either AC force or DC force).
- For DC force, select the desired **Frequency** for which to calculate results.
- For AC force, select the desired **Frequency** and specify a **Phase angle** for which to calculate results. Because the computed force values are complex numbers, the Phase angle you specify determines the particular time or point in the period of the chosen Frequency used to compute AC force results.

## 3. To display the results, click **Evaluate** on the force components table panel.

## 5.3. Exploring the Result Preview

**Exploring** is an opportunity to preview and explore some preliminary, lower fidelity results after solving the physics but before setting up and evaluating result objects. This preview is the default view of your study when you first move to the **Results** task. The presence of the **Exploring** toolbar and plane controls are indications that you are in the **Exploring** preview.



You can explore the preview using:

- moveable cut planes that show results in real time as you move the plane controls,
- a drop-down list of variables to select what is displayed,
- your cursor to display the max value as you mouse over the preview while the markers in the legend give you an indication of the value range for that point, and
- toggles to show or hide streamlines, contours, field flow and labels.

As you're exploring, you may find something compelling with a particular position of the cut plane that you want to investigate further. You can create and save a high fidelity result on the cut plane location. See [Saving an Exploratory Contour Result as a High-fidelity Result Object](#) on page 551 to make use of this shortcut technique.

To exit **Exploring**:

- deselect both contour and streamline display,
- navigate to a defined result object, or
- add a new object by clicking **Add** in the **Results > Objects** data panel or right-clicking in the graphics viewer and selecting the appropriate object under **Add > Results**

**Note:** When you first expose the result preview, the plane controls for contours and streamlines are located in the same position; because they overlap, they may not be visible as two separate controls.

### 5.3.1. Saving an Exploratory Contour Result as a High-fidelity Result Object

Generate a physics solution.

1. Click on the **Results** task and confirm that you're in the **Exploring** preview (the **Exploring** toolbar will be present).
  2. Position the movable cut plane using the plane control.
  3. Specify the variable you want to display under **Variable** pulldown.
  4. Click . The new, up-to-date **Results** object definition will display in the data panel and a simulation result will appear in the graphics view.
- 

## 5.4. Exporting Results Data

After evaluating your [contour](#) and [vector](#) results, you can export your result values to a comma-separated values (CSV) file that can be used by other software tools for further analysis.

To export results data:

- 
1. Click  in the Contour or Vector panel, or right click the mouse button in the graphics window and select **Export to CSV**.
  - The exported CSV file will contain only [node value](#) based data for fluid contour results.
  2. Browse to the folder where you want to store the file and enter a file name.
  3. Click **Save**.

**Note:** Results data export is limited to steady-state simulations, where only the following results can be exported:

- Vectors on volumes
- Contours on volumes
- Contours on physics condition faces of fluid simulations

After exporting your results, make sure to review the data as there may be some cases where duplicate values appear in the CSV file. Search and remove any duplicates during any further post processing.

---

## 5.5. Results Variables by Physics Type

A number of variables are available for Results display for the different Physics types. For many of these variables there are components that can be viewed separately (Vector magnitude, x-direction, y-direction, and so on). A list of the general types of Results available per Physics is given here.

### 5.5.1. Fluid Variables

The following tables list the variables available for Fluids results.

Table 5.5.1.1. Fluid Pressure Variables

Variable Name	Variable Type	Field Variable Definition
Pressure	Scalar	The fluid gauge pressure.
AbsolutePressure	Scalar	The pressure field plus the operating pressure. If buoyancy is active, see <a href="#">buoyancy and pressure</a> for details.
TotalPressure	Scalar	The gauge pressure obtained by bringing the fluid isentropically to rest, sometimes referred to as stagnation pressure.

Table 5.5.1.2. Fluid Material Property Variables

Variable Name	Variable Type	Field Variable Definition
Density	Scalar	The fluid mass per unit volume.
DynamicViscosity	Scalar	The resistance of a fluid to shear deformation.
ThermalConductivity	Scalar	The ability of a fluid to transfer heat by conduction.
SpecificHeatCapacity	Scalar	The thermodynamic property of specific heat for incompressible materials.
SpecificHeatCapacityCp	Scalar	The thermodynamic property of specific heat at constant pressure.

Table 5.5.1.3. Fluid Velocity Variables

Variable Name	Variable Type	Field Variable Definition
MassFlow	Scalar	The mass flow rate through the surface. The sign convention is such that if the surface is a boundary, a positive mass flow rate represents inflow to the flow region.
qCriterion	Scalar	A variable derived from the velocity field to identify and visualize vortices.
StrainRate	Scalar	The second invariant of the strain rate tensor, which evaluates to the following:
$\text{StrainRate} = \sqrt{2 \left[ \left( \frac{\partial U_x}{\partial x} \right)^2 + \left( \frac{\partial U_y}{\partial y} \right)^2 + \left( \frac{\partial U_z}{\partial z} \right)^2 + \left( \frac{\partial U_x}{\partial y} + \frac{\partial U_y}{\partial x} \right)^2 + \left( \frac{\partial U_x}{\partial z} + \frac{\partial U_z}{\partial x} \right)^2 + \left( \frac{\partial U_y}{\partial z} + \frac{\partial U_z}{\partial y} \right)^2 \right]}$		
Velocity	Vector	The fluid velocity field (rate of change of fluid position vs. time at a fixed point, using an Eulerian formulation).
Velocity.x	Vector Component	The x, y, and z components of the velocity vector.
Velocity.y		
Velocity.z		
Velocity.mag	Vector Magnitude	The magnitude of the velocity vector (speed).
Vorticity	Vector	The curl of the velocity field, $\nabla \times U$ . This measures the rotation of a fluid element.
Vorticity.x	Vector Component	The x, y, and z components of the vorticity vector.
Vorticity.y		
Vorticity.z		

Variable Name	Variable Type	Field Variable Definition
Vorticity.mag	Vector Magnitude	The magnitude of the vorticity vector.

Table 5.5.1.4. Fluid Thermal Variables

Variable Name	Variable Type	Field Variable Definition
Temperature	Scalar	The fluid temperature.
SpecificInternalEnergy	Scalar	The energy associated with the total (translational, vibrational, and rotational) motion of molecules.
TotalTemperature	Scalar	The temperature obtained by bringing the fluid isentropically to rest, sometimes referred to as stagnation temperature. For incompressible flows, the total temperature is equal to the static temperature.
SpecificEnthalpy	Scalar	A measure of the energy in a system, including internal energy $e$ plus the flow work $p/\rho$ required to displace the other fluid: $h = e + p/\rho$
SpecificEntropy	Scalar	A measure of disorder in a system.
SpecificTotalEnthalpy	Scalar	The enthalpy obtained by bringing the fluid isentropically to rest, sometimes referred to as stagnation enthalpy: $h_{tot} = h + 1/2(U \cdot U)$ where $U$ is the velocity.

Table 5.5.1.5. Fluid Turbulence Variables

Variable Name	Variable Type	Field Variable Definition
EffectiveThermalConductivity	Scalar	The sum of the laminar and turbulent thermal conductivity.
EffectiveViscosity	Scalar	The sum of molecular and turbulence viscosity.
TurbulenceDissipationRate	Scalar	The dissipation rate of turbulence kinetic energy $\varepsilon$ .
TurbulenceEddyFrequency	Scalar	The turbulence eddy frequency $\omega$ is related to $k$ and $\varepsilon$ by $\omega = \varepsilon / (k C_\mu)$
TurbulenceIntensity	Scalar	The intensity of the turbulence kinetic energy relative to the mean kinetic energy.
TurbulenceKineticEnergy	Scalar	The kinetic energy associated with turbulence fluctuations, $k = 1/2 \bar{u}_i' \bar{u}_i'$
TurbulenceViscosity	Scalar	The viscosity due to turbulence mixing, as computed by the turbulence model.
TurbulenceViscosityRatio	Scalar	The ratio of turbulence viscosity to the molecular viscosity.

Variable Name	Variable Type	Field Variable Definition
YPlus	Scalar	The dimensionless distance from a wall face to the adjacent element center, nondimensionalized using the viscosity and friction velocity $y^* : y^+ = \rho u^* y / \mu$
ModifiedTurbulenceViscosity	Scalar	The transported quantity solved by the Spalart-Allmaras turbulence model.
TurbulenceIntermittency	Scalar	The probability that flow at a given point is turbulent, used by turbulence transition models.
ReynoldsTheta	Scalar	The Reynolds number based on boundary layer thickness, used by turbulence transition models.

Table 5.5.1.6. Fluid Wall Flux Variables

Variable Name	Variable Type	Field Variable Definition
SurfaceShearStress	Vector	The shear viscous force of the fluid on the surface per unit area.
SurfaceShearStress.x	Vector Component	The x, y, and z components of the SurfaceShearStress vector.
SurfaceShearStress.y		
SurfaceShearStress.z		
SurfaceShearStress.mag	Scalar	The magnitude of the SurfaceShearStress vector.
SurfaceHeatFlux	Scalar	The heat flow into the fluid per unit area.
SurfaceHeatFlow	Scalar	The heat flow into the fluid (SurfaceHeatFlux times area).

Table 5.5.1.7. Fluid Force Variables

Variable Name	Variable Type	Field Variable Definition
Force	Vector	The total force (pressure and viscous) of the fluid on the surface.
Force.x	Vector Component	The x, y, and z components of the Force vector.
Force.y		
Force.z		
Force.mag	Vector Magnitude	The magnitude of the Force vector.

Table 5.5.1.8. Fluid Pressure Force Variables

Variable Name	Variable Type	Field Variable Definition
PressureForce	Vector	The pressure force of the fluid on the surface.
PressureForce.x	Vector Component	The x, y, and z components of the PressureForce vector.
PressureForce.y		
PressureForce.z		
PressureForce.mag	Vector Magnitude	The magnitude of the PressureForce vector.

Table 5.5.1.9. Fluid Viscous Force Variables

Variable Name	Variable Type	Field Variable Definition
ViscousForce	Vector	The viscous force of the fluid on the surface.
ViscousForce.x	Vector Component	The x, y, and z components of the ViscousForce vector.
ViscousForce.y		
ViscousForce.z		
ViscousForce.mag	Vector Magnitude	The magnitude of the ViscousForce vector.

Table 5.5.1.10. Fluid Position Variables

Variable Name	Variable Type	Field Variable Definition
Position	Vector	The spatial coordinate vector.
Position.x	Vector Component	The x, y, and z components of the Position vector.
Position.y		
Position.z		
Position.mag	Vector Magnitude	The magnitude of the Position vector.

Table 5.5.1.11. Fluid Mesh Metrics Variables

Variable Name	Variable Type	Field Variable Definition
ElementAspectRatio	Scalar	A measure of the stretching of a cell. It is computed as the ratio of the maximum value to the minimum value of any of the following distances: the normal distances between the cell centroid and face centroids (computed as a dot product of the distance vector and the face normal), and the distances between the cell centroid and nodes.
ElementVolume	Scalar	The volume of the element.
ElementWallDistance	Scalar	The distance from the wall face centroid to the adjacent element centroid.
FaceArea.mag	Vector	Magnitude of the Face Area.
FaceArea.x	Vector Component	The x, y, and z components of the outward-directed Face Area vector.
FaceArea.y		
FaceArea.z		
OrthogonalQuality	Scalar	The range for <a href="#">orthogonal quality</a> is 0-1. The best (most orthogonal) cells have larger values, while lower quality (highly skewed) cells have smaller values.

Table 5.5.1.12. Fluid Time and Iteration Variables

Variable Name	Variable Type	Field Variable Definition
Iteration	Single Valued	A counter for the current iteration number.

Table 5.5.1.13. Fluid Residual Variables

Variable Name	Variable Type	Field Variable Definition
PressureResidual	Scalar	Residual of the continuity equation.
TemperatureResidual	Scalar	Residual of the energy equation.
TurbEddyFreqResid	Scalar	Residual of the turbulence eddy frequency equation.
TurbKinetEnergyResid	Scalar	Residual of the turbulence kinetic energy equation.
VelocityResidualX	Scalar	Residuals of the X, Y, and Z components of the momentum equation.
VelocityResidualY		
VelocityResidualZ		

### 5.5.2. Polymer Extrusion Variables

The following tables describe the variables available for polymer extrusion results.

Table 5.5.2.1. Polymer Material Properties Variables

Variable Name	Variable Type	Description
DynamicViscosity	Scalar	The resistance of a polymer to shear deformation.

Table 5.5.2.2. Polymer Fluid Fraction Variables

Variable Name	Variable Type	Description
Fluid Fraction $n$ ( $n=1-9$ )	Scalar	The ratio of Fluid $n$ to the entire flow field.

Table 5.5.2.3. Polymer Position Variables

Variable Name	Variable Type	Description
Displacement	Vector	Difference between the final calculated position and the initial position.
Displacement.mag	Vector Magnitude	The magnitude of the Displacement vector.
Displacement.x	Vector Component	The x, y, and z components of the Displacement vector.
Displacement.y		
Displacement.z		
Position	Vector	The spatial coordinate vector.
Position.mag	Vector Magnitude	The magnitude of the Position vector.
Position.x	Vector Component	The x, y, and z components of the Position vector.
Position.y		
Position.z		

Table 5.5.2.4. Polymer Pressure Variables

Variable Name	Variable Type	Description
Pressure	Scalar	The fluid gauge pressure.

Table 5.5.2.5. Polymer Stress Variables

Variable Name	Variable Type	Description
Pseudo Shear Rate	Scalar	Quantity deriving from the Shear Rate and used in the Simplified Viscoelastic model.
Shear Rate	Scalar	The rate of change of the fluid velocity.
Stress Along Velocity Direction	Scalar	Stress on a plane locally perpendicular to velocity vector.
Stress.xx	Tensor Component	A general three-dimensional strain state is calculated in terms of three normal (X, Y, Z) and three shear (XY, YZ, XZ) strain components aligned to the specified coordinate system.
Stress.xy	Tensor Component	
Stress.xz	Tensor Component	
Stress.yy	Tensor Component	
Stress.yz	Tensor Component	
Stress.zz	Tensor Component	
1st Principal Stress	Scalar	An infinitesimal volume of material at an arbitrary point on or inside the solid body can be rotated such that only normal stresses remain and all shear stresses are zero. The three normal stresses that remain are called the principal stresses .
2nd Principal Stress	Scalar	The principal stresses are always ordered such that $\sigma_1 > \sigma_2 > \sigma_3$ . The principal stresses and maximum shear stress are called invariants; that is, their value does not depend on the orientation of the part or assembly with respect to its specified coordinate system.
3rd Principal Stress	Scalar	

Table 5.5.2.6. Polymer Thermal Variables

Variable Name	Variable Type	Description
Temperature	Scalar	The polymer temperature.
ViscousHeating	Scalar	The heat generation by friction of fluid layers per unit time and per unit volume.

Table 5.5.2.7. Polymer Velocity Variables

Variable Name	Variable Type	Description
Velocity	Vector	The polymer velocity field (rate of change of polymer position vs. time at a fixed point, using an Eulerian formulation).
Velocity.mag	Vector Magnitude	The magnitude of the Velocity vector (speed).
Velocity.x	Vector Component	The x, y, and z components of the Velocity vector.
Velocity.y		
Velocity.z		

Table 5.5.2.8. Polymer Overlap Variables

Variable Name	Variable Type	Description
Inside n (n=1 to 5)	Scalar	The velocity node that is overlapped by a restrictor.

### 5.5.3. Polymer Blow Molding Variables

The following tables describe the variables available for polymer blow molding results.

Table 5.5.3.1. Polymer Contact Variables

Variable Name	Variable Type	Description
Contact Status	Scalar	The contact information for each node. It can have a value of either 0 or 1. If a node of the free surface is in contact, the field value is 1, otherwise the value is 0.
Contact Time	Scalar	The time during which a node has been in contact with the mold.
Thermal Transfer Status	Scalar	The information for each node stating if there is actually heat transfer in the case of a region interface.

Table 5.5.3.2. Polymer Position Variables

Variable Name	Variable Type	Description
Displacement Magnitude	Vector Magnitude	The magnitude of the Displacement vector.
Displacement X	Vector Component	The x, y, and z components of the Displacement vector.
Displacement Y		
Displacement Z		
Initial Position Magnitude	Vector Magnitude	The magnitude of the Initial Position vector.

<b>Variable Name</b>	<b>Variable Type</b>	<b>Description</b>
Initial Position X	Vector Component	The x, y, and z components of the Initial Position vector.
Initial Position Y		
Initial Position Z		
Position Magnitude	Vector Magnitude	The magnitude of the Position vector.
Position X	Vector Component	The x, y, and z components of the Position vector.
Position Y		
Position Z		

Table 5.5.3.3. Polymer Thickness Variables

<b>Variable Name</b>	<b>Variable Type</b>	<b>Description</b>
Thickness	Scalar	The thickness of the polymer sheet.

Table 5.5.3.4. Polymer Stress Variables

<b>Variable Name</b>	<b>Variable Type</b>	<b>Description</b>
Area Stretch Ratio	Scalar	The ratio of the area of an elementary material surface in the deformed configuration to that in the undeformed state.

Table 5.5.3.5. Polymer Thermal Variables

<b>Variable Name</b>	<b>Variable Type</b>	<b>Description</b>
Temperature	Scalar	The polymer temperature.

Table 5.5.3.6. Polymer Velocity Variables

<b>Variable Name</b>	<b>Variable Type</b>	<b>Description</b>
Velocity Magnitude	Vector Magnitude	The magnitude of the Velocity vector (speed).
Velocity X	Vector Component	The x, y, and z components of the Velocity vector.
Velocity Y		
Velocity Z		
Mold Velocity Magnitude	Vector Magnitude	The magnitude of the Mold Velocity vector (speed).
Mold Velocity X	Vector Component	The x, y, and z components of the Mold Velocity vector.
Mold Velocity Y		
Mold Velocity Z		

## 5.5.4. Structural Variables

The following tables list the variables available for structural results.

Table 5.5.4.1. Structural Strain Variables

Variable Name	Variable Type	Description
ElasticStrain.xx (.x)	Tensor Component	A general three-dimensional strain state is calculated in terms of three normal (X, Y, Z) and three shear (XY, YZ, XZ) strain components aligned to the specified coordinate system.
ElasticStrain.yy (.y)	Tensor Component	
ElasticStrain.zz (.z)	Tensor Component	
ElasticStrain.xy (.yx)	Tensor Component	
ElasticStrain.zy (.yz)	Tensor Component	
ElasticStrain zx (.xz)	Tensor Component	
ElasticStrain.p1	Scalar	From elasticity theory, an infinitesimal volume of material at an arbitrary point on or inside the solid body can be rotated such that only normal strains remain and all shear strains are zero. The three normal strains that remain are called the principal strains.
ElasticStrain.p2	Scalar	
ElasticStrain.p3	Scalar	
ElasticStrain.eqv	Scalar	
ElasticStrain.maxshear	Scalar	The principal strains are always ordered such that $\epsilon_1 > \epsilon_2 > \epsilon_3$ . The principal strains and maximum shear strain are called invariants; that is, their value does not depend on the orientation of the part or assembly with respect to its specified coordinate system.
ElasticStrain.intensity	Scalar	Elastic Strain intensity is defined as the largest of the absolute values of $\epsilon_1 - \epsilon_2$ , $\epsilon_2 - \epsilon_3$ , or $\epsilon_3 - \epsilon_1$ .
PlasticStrain.xx (x)	Tensor Component	Most common engineering materials exhibit a linear stress-strain relationship up to a stress level known as the proportional limit. Beyond this limit, the stress-strain relationship will become nonlinear, but will not necessarily become inelastic. Plastic behavior, characterized by nonrecoverable strain or plastic strain, begins when stresses exceed the material's yield point. Because there is usually little difference between the yield point and the proportional limit, AIM assumes that these two points are coincident in plasticity analyses.
PlasticStrain.yy (y)	Tensor Component	
PlasticStrain.zz (z)	Tensor Component	
PlasticStrain.xy (xy)	Tensor Component	
PlasticStrain.zy (zy)	Tensor Component	
PlasticStrain.zx (zx)	Tensor Component	
PlasticStrain.p1	Scalar	From elasticity theory, an infinitesimal volume of material at an arbitrary point on or inside the solid body can be rotated such that only normal strains remain and all shear strains are zero. The three normal strains that remain are called the principal strains.
PlasticStrain.p2	Scalar	
PlasticStrain.p3	Scalar	

PlasticStrain.eqv	Scalar	The equivalent plastic strain gives a measure of the amount of permanent strain in an engineering body. The equivalent plastic strain is calculated from the component plastic strain.
PlasticStrain.intensity	Scalar	Plastic Strain intensity is defined as the largest of the absolute values of $\epsilon_1 - \epsilon_2$ , $\epsilon_2 - \epsilon_3$ , or $\epsilon_3 - \epsilon_1$ .
TotalStrain.xx (x)	Tensor Component	The total strain is calculated by the addition of elastic, plastic, thermal, and creep strains.
TotalStrain.yy (y)	Tensor Component	A general three-dimensional strain state is calculated in terms of three normal (X, Y, Z) and three shear (XY, YZ, XZ) strain components aligned to the specified coordinate system.
TotalStrain.zz (z)	Tensor Component	
TotalStrain.xy (yx)	Tensor Component	
TotalStrain.yz (zy)	Tensor Component	
TotalStrain.zx (xz)	Tensor Component	
TotalStrain.p1	Scalar	From elasticity theory, an infinitesimal volume of material at an arbitrary point on or inside the solid body can be rotated such that only normal strains remain and all shear strains are zero.
TotalStrain.p2	Scalar	The three normal strains that remain are called the principal strains.
TotalStrain.p3	Scalar	The principal strains are always ordered such that $\epsilon_1 > \epsilon_2 > \epsilon_3$ . The principal strains are called invariants; that is, their value does not depend on the orientation of the part or assembly with respect to its specified coordinate system.
TotalStrain.eqv	Scalar	The equivalent total strain gives a total value of strain in any engineering body. Equivalent total strain is calculated from the total strain components.
TotalStrain.intensity	Scalar	Total strain intensity is defined as the largest of the absolute values of $\epsilon_1 - \epsilon_2$ , $\epsilon_2 - \epsilon_3$ , or $\epsilon_3 - \epsilon_1$ .
ThermalStrain.xx (.x)	Tensor Component	Thermal strain is computed when coefficient of thermal expansion is specified and a temperature load is applied in a structural analysis.
ThermalStrain.yy (.y)	Tensor Component	A general three-dimensional strain state is calculated in terms of three normal (X, Y, Z) and three shear (XY, YZ, XZ) strain components aligned to the specified coordinate system.
ThermalStrain.zz (.z)	Tensor Component	
ThermalStrain.xy (.yx)	Tensor Component	
ThermalStrain.yz (.yz)	Tensor Component	
ThermalStrain.zx (.xz)	Tensor Component	

ThermalStrain.p1	Scalar	From elasticity theory, an infinitesimal volume of material at an arbitrary point on or inside the solid body can be rotated such that only normal strains remain and all shear strains are zero. The three normal strains that remain are called the principal strains.
ThermalStrain.p2	Scalar	
ThermalStrain.p3	Scalar	
ThermalStrain.eqv	Scalar	The principal strains are always ordered such that $\epsilon_1 > \epsilon_2 > \epsilon_3$ . The principal strains are called invariants; that is, their value does not depend on the orientation of the part or assembly with respect to its specified coordinate system.
ThermalStrain.intensity	Scalar	Thermal Strain intensity is defined as the largest of the absolute values of $\epsilon_1 - \epsilon_2$ , $\epsilon_2 - \epsilon_3$ , or $\epsilon_3 - \epsilon_1$ .

Table 5.5.4.2. Structural Stress Variables

Variable Name	Variable Type	Description
Stress.xx (.x)	Tensor Component	A general three-dimensional stress state is calculated in terms of three normal (X, Y, Z) and three shear (XY, YZ, XZ) stress components aligned to the specified coordinate system.
Stress.yy (.y)	Tensor Component	
Stress.zz (.z)	Tensor Component	
Stress.xy (.yx)	Tensor Component	
Stress.zy (.yz)	Tensor Component	
Stress zx (.xz)	Tensor Component	
Stress.p1	Scalar	From elasticity theory, an infinitesimal volume of material at an arbitrary point on or inside the solid body can be rotated such that only normal stresses remain and all shear stresses are zero. The three normal stresses that remain are called the principal stresses .
Stress.p2	Scalar	
Stress.p3	Scalar	
Stress.eqv	Scalar	
Stress.maxshear	Scalar	The principal stresses are always ordered such that $\sigma_1 > \sigma_2 > \sigma_3$ . The principal stresses and maximum shear stress are called invariants; that is, their value does not depend on the orientation of the part or assembly with respect to its specified coordinate system.
Stress.intensity	Scalar	Stress intensity is defined as the largest of the absolute values of $\sigma_1 - \sigma_2$ , $\sigma_2 - \sigma_3$ , or $\sigma_3 - \sigma_1$ .

Table 5.5.4.3. Structural Displacement Variables

Variable Name	Variable Type	Description
Displacement	Vector	These are physical deformations that are calculated on and inside a model. Fixed supports prevent deformation; locations without a fixed support usually experience deformation relative to the original location. Deformations are calculated relative to the specified coordinate system.
Displacement.x	Vector Component	
Displacement.y	Vector Component	
Displacement.z	Vector Component	
Displacement.mag	Vector Magnitude	The three component deformations, Displacement.x, Displacement.y, and Displacement.z, and the resultant deformation, Displacement.mag, are available as individual results.

Table 5.5.4.4. Structural Force Reaction Variables

Variable Name	Variable Type	Description
ForceReaction	Vector	These are the reaction forces. These results are available when applied to constraints.  The three component force reactions, ForceReaction.x, ForceReaction.y, and ForceReaction.z, and the resultant force reaction, ForceReaction.mag, are available as individual results.
ForceReaction.x	Vector Component	
ForceReaction.y	Vector Component	
ForceReaction.z	Vector Component	
ForceReaction.mag	Vector Magnitude	

Table 5.5.4.5. Structural Moment Reaction Variables

Variable Name	Variable Type	Description
MomentReaction	Vector	These are the moment reactions. These results are available when applied to constraints.  The three component moment reactions, MomentReaction.x, MomentReaction.y, and MomentReaction.z, and the resultant moment reaction, MomentReaction.mag, are available as individual results.
MomentReaction.x	Vector Component	
MomentReaction.y	Vector Component	
MomentReaction.z	Vector Component	
MomentReaction.mag	Vector Magnitude	

Table 5.5.4.6. Structural Nodal Load Variables

Variable Name	Variable Type	Description
Force	Vector	These are the element nodal forces. These results are available when applied to geometry or loads.  The three component forces Force.x, Force.y, and Force.z, and the resultant force Force.mag are available as individual results.
Force.x	Vector Component	
Force.y	Vector Component	
Force.z	Vector Component	
Force.mag	Vector Magnitude	

Table 5.5.4.7. Bolt Pretension Variables

Variable Name	Variable Type	Description
Adjustment	Vector Magnitude	This represents the displacement that occurs from the applied pretension measured at the point where the bolt is sliced, or the displacement reported from the pretension node. This result is also available for reporting regardless of how the bolt is defined.
WorkingLoad	Vector Magnitude	This represents the constrained force reaction from the pretension load. It is the constrained reaction reported from the pretension node and reports a zero value during a step in which you have applied the preload (since there is no reaction at the bolt slice during preload step).  This is essentially the sum of all the forces acting through the pretension cut. This result is only applicable for load steps when the load is defined by either <b>Locked</b> or <b>Factor</b> .

Table 5.5.4.8. Structural Contact Variables

Variable Name	Variable Type	Description
ContactPressure	Scalar	This result shows the measured pressure between two surfaces in contact.
SlidingDistance	Scalar	The total sliding distance is the amplitude of total accumulated slip increments (a geometrical measurement) when the contact status is sticking or sliding. It contains contributions from the elastic slip and the frictional slip. Elastic slip due to sticking represents the reversible tangential motion from the point of zero tangential stresses. Ideally, the equivalent elastic slip does not exceed the user-defined absolute limit. The higher the tangent stiffness, the smaller the resulting elastic slip.
Penetration	Scalar	For most contact methods, the numerics allow for the two bodies to have a certain amount of penetration. This result is a measure of that numerical penetration.
Gap	Scalar	This result shows how close the two surfaces of the contact are.
ContactStatus	Scalar	A discrete result which gives an indication of the contact state over the model.

Table 5.5.4.9. Structural Contact Reaction Variables

Variable Name	Variable Type	Description
ContactReaction	Vector	These are the reaction forces on the contact.  The three component reactions and the resultant reaction are available as individual results.
ContactReaction.x	Vector Component	
ContactReaction.y	Vector Component	
ContactReaction.z	Vector Component	
ContactReaction.mag	Scalar	

Table 5.5.4.10. Structural Beam Variables

Variable Name	Variable Type	Description
AxialForce.mag	Scalar	The magnitude of the force along a beam element axis.
ShearForce.mag	Scalar	The magnitude of the force perpendicular to the beam element axis
ShearForce.x	Scalar	The global X, Y and Z components of the force perpendicular to the beam element axis.
ShearForce.y	Scalar	
ShearForce.z	Scalar	
BendingMoment.mag	Scalar	The magnitude of the moment in the plane perpendicular to the beam element axis.
BendingMoment.x	Scalar	The global X, Y and Z components of the moment in the plane perpendicular to the beam element axis
BendingMoment.y	Scalar	
BendingMoment.z	Scalar	
AxialStress	Scalar	The stress component due to the axial load encountered in a beam element.

Variable Name	Variable Type	Description
BendingStress.min	Scalar	From any bending loads, a bending moment in both the local Y and Z directions will arise. This leads to the following four bending stresses: Y bending stress on top/bottom and Z bending stress on the top/bottom. BendingStress.min is the minimum of these four bending stresses, and BendingStress.max the maximum.
CombinedStress.min	Scalar	The linear combination of the AxialStress and the BendingStress.min.
CombinedStress.max	Scalar	The linear combination of the AxialStress and the BendingStress.max.

Table 5.5.4.11. Structural Joint Variables

Variable Name	Variable Type	Description
JointTotalForce	Vector	Joint force and moment are by definition the action of the reference body on the moving body. For the ANSYS solver, the joint constraint forces and moments are reported in the joint reference coordinate system. The three component forces and the resultant force are available as individual results.
JointTotalForce.x	Vector Component	
JointTotalForce.y	Vector Component	
JointTotalForce.z	Vector Component	
JointTotalForce.mag	Scalar	
JointTotalMoment	Vector	Joint force and moment are by definition the action of the reference body on the moving body. For the ANSYS solver, the joint constraint forces and moments are reported in the joint reference coordinate system. The three component moments and the resultant moment are available as individual results.
JointTotalMoment.x	Vector Component	
JointTotalMoment.y	Vector Component	
JointTotalMoment.z	Vector Component	
JointTotalMoment.mag	Scalar	
RelativeDisplacement	Vector	Measures the relative displacement of the mobile body with respect to the reference body. The three component displacements and the resultant displacement are available as individual results.
RelativeDisplacement.x	Vector Component	
RelativeDisplacement.y	Vector Component	
RelativeDisplacement.z	Vector Component	
RelativeDisplacement.mag	Scalar	
RelativeRotation	Vector	Measures the relative rotation of the mobile body with respect to the reference body. The three component rotations and the resultant rotation are available as individual results.
RelativeRotation.x	Vector Component	
RelativeRotation.y	Vector Component	
RelativeRotation.z	Vector Component	
RelativeRotation.mag	Scalar	

Table 5.5.4.12. Structural Spring Variables for 2-D Springs

Variable Name	Variable Type	Description
SpringForce.mag	Scalar	The reaction force applied by the spring when it is stretched or compressed from its equilibrium position. It acts in the direction opposite to stretch or compress. Applicable to 1-D springs.
SpringMoment.mag	Scalar	The reaction moment applied by the spring when it is twisted from its equilibrium position. The moment acts in the direction opposite to twist direction. Applicable to 1-D springs.
SpringStretch.mag	Scalar	The stretch is the relative displacement between the two ends of the springs for a longitudinal spring. This elongation value can be positive (stretching the spring) or negative (compressing the spring). Applicable to 1-D springs.
SpringTwist.mag	Scalar	The twist is the change in the angle from the equilibrium position that two ends of the torsional spring subtends at its center. The twist value can be positive (increased subtended angle) or negative (decreased subtended angle). Applicable to 1-D springs.

Table 5.5.4.13. Structural Spring Variables for 3-D Springs

SpringTotalForce	Vector	The total reaction force applied by the spring when it is stretched or compressed from its equilibrium position. It acts in the direction opposite to stretch or compress. The three component forces and the resultant force are available as individual results. Applicable to 3-D springs.
SpringTotalForce.x	Vector Component	
SpringTotalForce.y	Vector Component	
SpringTotalForce.z	Vector Component	
SpringTotalForce.mag	Scalar	
SpringTotalMoment	Vector	The total reaction moment applied by the spring when it is twisted from its equilibrium position. The moment acts in the direction opposite to twist direction. The three component moments and the resultant moment are available as individual results. Applicable to 3-D springs.
SpringTotalMoment.x	Vector Component	
SpringTotalMoment.y	Vector Component	
SpringTotalMoment.z	Vector Component	
SpringTotalMoment.mag	Scalar	
SpringTotalStretch	Vector	The stretch is the relative displacement between the two ends of the springs for a longitudinal spring. This elongation value can be positive (stretching the spring) or negative (compressing the spring). The three component stretch vectors and the resultant stretch value are available as individual results. Applicable to 3-D springs.
SpringTotalStretch.x	Vector Component	
SpringTotalStretch.y	Vector Component	
SpringTotalStretch.z	Vector Component	
SpringTotalStretch.mag	Scalar	
SpringTotalTwist	Vector	The twist is the change in the angle from the equilibrium position that two ends of the torsional spring subtends at its center. The twist value can be positive (increased subtended angle) or negative (decreased subtended angle). The three component twist vectors and the resultant twist value are available as individual results. Applicable to 3-D springs.
SpringTotalTwist.x	Vector Component	
SpringTotalTwist.y	Vector Component	
SpringTotalTwist.z	Vector Component	
SpringTotalTwist.mag	Scalar	

Table 5.5.4.14. Fatigue Variables

Variable Name	Variable Type	Description
FatigueLife	Scalar	Fatigue Life represents the number of cycles of repeated loading until the part will fail due to fatigue.
FatigueDamage	Scalar	Fatigue Damage is defined as the design life divided by the fatigue life.
FatigueSafetyFactor	Scalar	Fatigue Safety Factor is defined as the ratio of alternating stress at design life to the equivalent alternating stress at a given point. The maximum safety factor reported is 15.
FatigueBiaxiality	Scalar	Biaxiality indication is defined as the principal stress smaller in magnitude divided by the larger principal stress with the principal stress nearest zero ignored. A biaxiality of zero corresponds to uniaxial stress, a value of -1 corresponds to pure shear, and a value of 1 corresponds to a pure biaxial state.
FatigueEquivalentStress	Scalar	In a Stress Life fatigue analysis, you must query an S-N curve to relate the fatigue life to the stress state. Thus the "equivalent alternating stress" is the stress used to query the fatigue S-N curve after accounting for fatigue loading type, mean stress effects, multiaxial effects, and any other factors in the fatigue analysis.

Table 5.5.4.15. Miscellaneous Mechanical Variables

Variable Name	Variable Type	Description
StructuralError	Scalar	You can insert an Error result based on stresses to help you identify regions of high error and therefore show where the model would benefit from a more refined mesh in order to get a more accurate answer.

## 5.5.5. Thermal Variables in Thermal/Structural Simulations

The following table describes the variables available for thermal results in a thermal/structural simulation.

	Variable Name	Variable Type	Description
<b>Thermal Variables</b>	Temperature	Scalar	In a steady-state or time-dependent thermal analysis, AIM calculates the temperature distribution throughout the structure. This result is available when applied to geometry or loads and constraints.
	HeatFlux	Vector	AIM calculates the heat flux ( $q/A$ , energy per unit time per unit area) throughout the body. The three component heat fluxes and the resultant heat flux value are available as individual results.
	HeatFlux.x HeatFlux.y HeatFlux.z	Vector Component	
	HeatFlux.mag	Vector Magnitude	These results are available when applied to geometry.
	TemperatureGradient	Vector	In addition to temperature, AIM also calculates the variation of the temperature results with time.  The three component temperature values and the resultant temperature value are available as individual results.
	TemperatureGradientx TemperatureGradienty TemperatureGradientz	Vector Component	
	TemperatureGradient.mag	Vector Magnitude	These results are available when applied to geometry.
<b>Thermal Nodal Load Variable</b>	HeatFlow	Scalar	AIM calculates the heat flow throughout the body. These results are available when applied to geometry or loads.
<b>Heat Flow Reaction Variable</b>	HeatFlowReaction	Scalar	You can obtain heat reaction ( $q$ , energy per unit time) at locations where a constraint is specified. Heat reaction is a scalar.
<b>Miscellaneous Thermal Variables</b>	ThermalError	Scalar	You can insert an Error result based on stresses to help you identify regions of high error and thus show where the model would benefit from a more refined mesh in order to get a more accurate answer.

### 5.5.6. Electric Conduction Variables

The following tables describe the variables available for electric conduction results.

Table 5.5.6.1. Electric Potential Variables

Variable Name	Variable Type	Description
ElectricPotential	Scalar	Represents contours of constant electric potential (voltage) in conductor bodies.

Table 5.5.6.2. Electric Field Variables

Variable Name	Variable Type	Description
ElectricField	Vector	A vector quantity representing the magnitude and direction of the electric field in the region of interest.
ElectricField.x	Vector Component	
ElectricField.y	Vector Component	
ElectricField.z	Vector Component	
ElectricField.mag	Vector Magnitude	The three component electric field variables, ElectricField.x, ElectricField.y, and ElectricField.z, and the resultant electric field, ElectricField.mag, are available as individual results.

Table 5.5.6.3. Current Density Variables

Variable Name	Variable Type	Description
CurrentDensity	Vector	A vector quantity representing the amount of electric current passing through a unit cross-sectional area.
CurrentDensity.x	Vector Component	
CurrentDensity.y	Vector Component	
CurrentDensity.z	Vector Component	
CurrentDensity.mag	Vector Magnitude	The three component current density variables, CurrentDensity.x, CurrentDensity.y, and CurrentDensity.z, and the resultant current density, CurrentDensity.mag, are available as individual results.

Table 5.5.6.4. Current Reaction Variables

Variable Name	Variable Type	Description
CurrentReaction	Scalar	The amount of current passing through a surface/voltage constraint. (This is analogous to heat reaction in a thermal analysis).

Table 5.5.6.5. Joule Heat Generation Variables

Variable Name	Variable Type	Description
HeatGeneration	Scalar	An electrical current passing through a conductor can generate heat (commonly called joule heating). This output is a measure of that quantity.

## 5.5.7. Electrostatic Variables

The following tables describe the variables available for electrostatic results.

Table 5.5.7.1. Electrostatic Variables

Variable Name	Variable Type	Description
ElectricPotential	Scalar	Represents contours of constant electric potential (voltage) in conductor bodies.
ChargeReaction	Scalar	A scalar quantity representing the reaction of charges on the voltage locations.

Variable Name	Variable Type	Description
ElectricField	Vector	A vector quantity representing the magnitude and direction of the electric field in the region of interest.
ElectricField.x	Vector Component	
ElectricField.y	Vector Component	
ElectricField.z	Vector Component	
ElectricField.mag	Vector Magnitude	The three component electric field variables, ElectricField.x, ElectricField.y, and ElectricField.z, and the resultant electric field, ElectricField.mag, are available as individual results.

### 5.5.8. Electromagnetic Variables

The following tables describe the variables available for electromagnetic results.

Table 5.5.8.1. Torque Variables

Variable Name	Variable Type	Field Variable Definition
Torque	Scalar	The magnitude of the torque.

Table 5.5.8.2. Force Variables

Variable Name	Variable Type	Field Variable Definition
Force.x	Scalar	The magnitude of the x component of the force vector.
Force.y	Scalar	The magnitude of the y component of the force vector.
Force.z	Scalar	The magnitude of the z component of the force vector.
Force.mag	Scalar	The magnitude of the force vector.

Table 5.5.8.3. Calculated Variables

Variable Name	Variable Type	Field Variable Definition
ACForce.x	Complex	The x component of the AC Force vector. Because complex numbers cannot be used in Calculated values directly, four methods are provided: ACForce.x.real, ACForce.x.imag, ACForce.x.mag, ACForce.x.phase.
ACForce.y	Complex	The y component of the AC Force vector. Because complex numbers cannot be used in Calculated values directly, four methods are provided: ACForce.y.real, ACForce.y.imag, ACForce.y.mag, ACForce.y.phase.
ACForce.z	Complex	The z component of the AC Force vector. Because complex numbers cannot be used in Calculated values directly, four methods are provided: ACForce.z.real, ACForce.z.imag, ACForce.z.mag, ACForce.z.phase.
ACForce.mag	Real	The magnitude of the AC Force vector.

Variable Name	Variable Type	Field Variable Definition
InducedVoltage	Complex	Induced voltage is an electrical potential created by and proportional to the rate of change of the flux linkage of the current defined by terminals.  Because complex numbers cannot be used in Calculated values directly, four methods are provided: InducedVoltage.ireal, InducedVoltage.imag, and InducedVoltage.phase.
InducedVoltage.mag	Real	The magnitude of the Induced Voltage vector.
WindingFluxLinkage	Complex	The flux linkage is the summation of the magnetic field linking with all coils in a current when the magnetic field passes through the loops of all coils.  Because complex numbers cannot be used in Calculated values directly, four methods are provided: WindingFluxLinkage.ireal, WindingFluxLinkage.imag, and WindingFluxLinkage.phase.
WindingFluxLinkage.mag	Real	The magnitude of the Winding Flux Linkage vector.

Table 5.5.8.4. DC Force Variables

Variable Name	Variable Type	Field Variable Definition
DCForce.x	Real	The magnitude of the x component of the DC Force vector.
DCForce.y	Real	The magnitude of the y component of the DC Force vector.
DCForce.z	Real	The magnitude of the z component of the DC Force vector.

Table 5.5.8.5. Loss Variables

Variable Name	Variable Type	Field Variable Definition
OhmicLossDensity	Real	Ohmic loss is always associated with conduction current distribution in conductors which are not perfect. Thus the resistivity of conductors is responsible for the ohmic power loss when current flows in such conductors. It is also called the Joule-Lenz effect. There is always a heating effect due to the ohmic loss, often called Joule heating.  <b>Note:</b> If an object has a fill factor specified, the conductivity value is multiplied by this fill factor. This is then the conductivity value used for post processing.

Variable Name	Variable Type	Field Variable Definition
CoreLossDensity	Real	Applicable to frequency response solutions, the core loss combines eddy current losses and hysteresis losses. It is applicable for the evaluation of core losses in steel (frequently used in applications such as electric machines, transformers), or in power ferrites.
DielectricLossDensity	Real	Dielectric loss is associated with loss density fields in frequency response solutions only. It is applicable when a high frequency electric field penetrates a dielectric material.  <b>Note:</b> If an object has a fill factor specified, the conductivity value is multiplied by this fill factor. This is then the conductivity value used for post processing.
HysteresisLossDensity	Real	Hysteresis loss is associated with loss density fields in frequency response solutions only. Hysteresis loss is short for magnetic hysteresis loss and represents the power loss in some magnetic materials (electric steels or ferrites) in alternating (sinusoidal) magnetic fields.
Total Loss	Real	Total loss is the sum of OhmicLossDensity, CoreLossDensity, DielectricLossDensity, and HysteresisLossDensity.

Table 5.5.8.6. Magnetic Variables

Variable Name	Variable Type	Field Variable Definition
CurrentDensity	Vector	A vector quantity representing the magnitude of current density ( $J$ ).
CurrentDensity.x	Vector Component	The three component current density variables, CurrentDensity.x, CurrentDensity.y, and CurrentDensity.z, and the resultant magnitude of the current density, CurrentDensity.mag, are available as individual results.
CurrentDensity.y	Vector Component	
CurrentDensity.z	Vector Component	
CurrentDensity.mag	Vector Magnitude	

Variable Name	Variable Type	Field Variable Definition
MagneticFieldIntensity	Vector	A vector quantity representing the magnetic field intensity (H).
MagneticFieldIntensity.x	Vector Component	The three component magnetic field intensity variables, MagneticFieldIntensity.x,
MagneticFieldIntensity.y	Vector Component	MagneticFieldIntensity.y, and
MagneticFieldIntensity.z	Vector Component	MagneticFieldIntensity.z, and the resultant magnitude of the magnetic field intensity, MagneticFieldIntensity.mag, are available as individual results.
MagneticFluxDensity	Vector	A vector quantity representing the magnetic flux density (B) vector.
MagneticFluxDensity.x	Vector Component	The three component magnetic flux density variables, MagneticFluxDensity.x,
MagneticFluxDensity.y	Vector Component	MagneticFluxDensity.y, and
MagneticFluxDensity.z	Vector Component	MagneticFluxDensity.z, and the resultant magnitude of the magnetic flux density, MagneticFluxDensity.mag, are available as individual results.
MagneticFluxDensity.mag	Vector Magnitude	
TotalMagneticFlux	Scalar	The surface integral of the normal component of the magnetic flux density passing through the surface.

Table 5.5.8.7. Current Density Variables

Variable Name	Variable Type	Field Variable Definition
CurrentDensity.mag	Scalar	The magnitude of current density, J.
CurrentDensity	Vector	The current density (J) vector.
TotalCurrent	Scalar	The surface integral of the normal component of the current density (J) over the selected faces.

Table 5.5.8.8. Thermal Variables

Variable Name	Variable Type	Field Variable Definition
Temperature	Scalar	The temperature distribution throughout the selected location.

## 5.6. Graphics Setting and Display

Each results object has appearance options that enable you to control how the object displays in the graphics viewer. Depending on which results object you are configuring, you may be able to set:

- Banded coloring so that you can control the number of colors used to display distinct variable ranges.
- Smooth coloring to display a variable without color bands.
- A user-specified variable range.
- A linear or logarithmic color distribution.

In addition to creating result objects for structural simulations, you can modify the rendering of these objects using **Graphics control**. For structural results with deformation, you can specify **Deformation scaling** to better visualize the magnitude of the deformation in your physics region.

Finally, the **Summary** displays quantitative values for the global range and averages on Results objects.

### 5.6.1. Specifying a Variable Range

#### Results > Object > Object #

To specify a variable range, you must create a supported results object such as a contour, vector, or streamline.

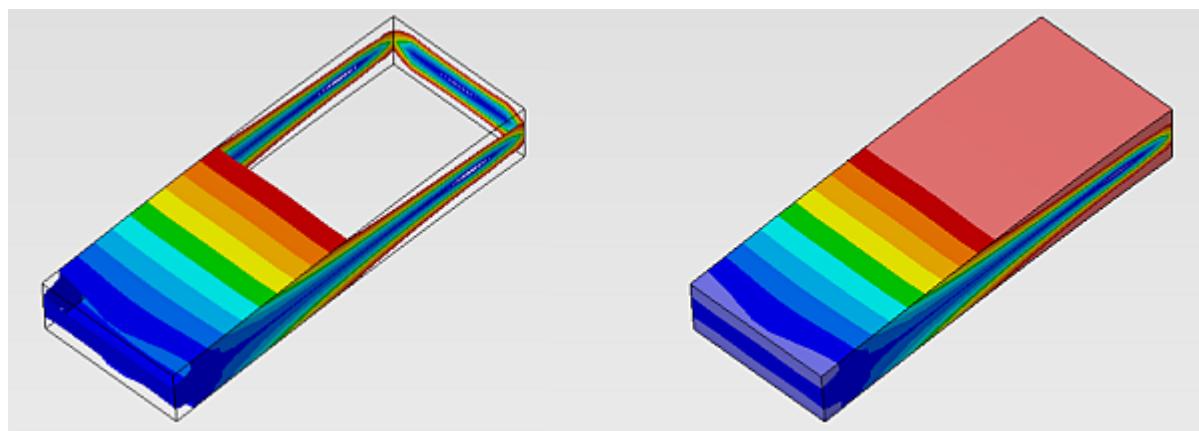
When viewing results objects, you can specify the range across which a variable will be displayed. Some cases where this is useful are:

- To view a narrower subset of the data in order to remove any outliers that skew the color ranges.
- To create plots with the same range in order to compare them more easily.

To specify a variable range:

1. Under **Appearance>Variable range**, select User-specified.
2. Enter the **Minimum** value for the range you want to display.
3. Enter the **Maximum** value for the range you want to display.

By default, **Clip to range** is selected. This hides any data that is outside of the specified range. If deselected, data that lies outside of the specified range will turn light red or light blue, depending on whether the variable is higher than the maximum or lower than the minimum (light red is higher, light blue is lower). The image below shows a user specified variable range with **Clip to range** selected (left) and deselected (right).



## 5.6.2. Specifying a Color Distribution

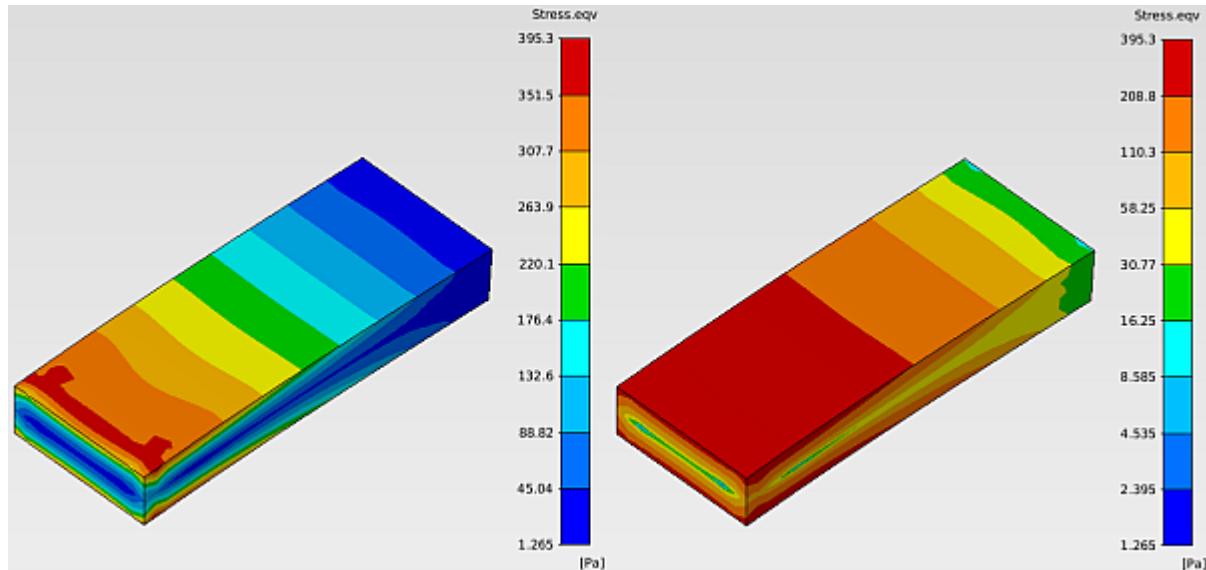
### Results > Contour > Contour #

To specify a colour distribution, you must create a contour results object.

Under **Appearance>Colour Distribution** choose either **Linear** or **Logarithmic**. The following table provides a description of the two options.

Option	Description	Useful for...
Linear (default)	The range of each contour band is constant. The range is determined by $\text{Max-Min} = K^*N$ , where Max and Min are the maximum and minimum values calculated in the current contour, N is the number of contour bands, and K is the range of each contour band.	Contours where the data varies evenly across the variable range.
Logarithmic	The ratio between adjacent contour bands is constant. The ratio is determined by $\text{Max}/\text{Min} = K^N$ , where Max and Min are the maximum and minimum values calculated in the current contour, N is the number of contour bands, and K is the ratio between adjacent contour bands.	Contours where the data is heavily skewed towards the lower end of the variable range.

The figure below shows the same contour with the colors distributed linearly (left) and logarithmically (right). You can see in the legend that the overall contour range is unchanged, while the individual contour band ranges are different.



## 5.6.3. Specifying Deformation Scaling

### Results > Graphics Control > Deformation scaling

To create objects that display the results of your simulation, generate a physics solution.

For a structural simulation with deformed regions, you can use **Deformation scaling** to display either the true or an exaggerated deformation of your model. This enables you to visualize the deflection more easily.

- The **Auto Scale** gives you the best scaling where the true deformation cannot be easily visualized. You can use **Custom**, **0.5x Auto**, **2x Auto**, and **5x Auto** to change the magnitude of the scaling.
- To display the true deformation of a physics region, set this option to **Actual**.
- To ignore the deformation, set this option to **None**.

#### 5.6.4. Summary

**Summary** displays quantitative values for the global range and averages on Results objects. On selected locations, **Summary** displays:

- **Calculated minimum** and **Calculated maximum**:

- When possible, the locations of minimum and maximum values are automatically displayed in the Graphics View. This can be hidden if required.
- For a structural simulation, these values are calculated as the nodal or elemental data for the lowest and highest value, depending on the selected variable.
- For a fluid simulation, these values are calculated as the nodal data for the lowest value and highest value for the selected variable.
- For a time-dependent simulation, you can view time history data in chart form by clicking  and then the **Show chart**  icon.
- For a non-linear simulation, you can view substep history data in chart form by clicking  and then the **Show chart**  icon.

**Note:** For a structural simulation with radiation loads, an extra node is created to account for ambient temperature. When you evaluate the result for this case, the minimum and maximum value of the result may include the value from this extra node.

- **Calculated average**:

- For a structural simulation, the average is calculated as the arithmetic average of the nodal or elemental value, depending on the selected variable.
- For a time-dependent simulation, time history is available in chart form by clicking the  icon.
- For a non-linear simulation, substep history is available in chart form by clicking the  icon.
- For a fluid simulation, the average is calculated as the geometric average for the variable value:

Location Type	Averaging Type
Point	Nodal data interpolated to the point location
Surface	Area-weighted average of face data
Volume	Volume-weighted average of element data
Streamline	Length-weighted average of nodal data interpolated to line segment centers
Plane, Isosurface	Area-weighted average of nodal data interpolated to face centers

For details on Averaging Type, see [Quantity Functions](#).

- **Calculated sum:**

For force reaction vectors in a structural simulation, **Calculated sum:x**, **Calculated sum:y** and **Calculated sum:z** are the components of the resultant vector. They are the arithmetic summation of the x, y, and z components of the vector variable for each node.

The magnitude of the resultant vector, **Calculated sum**, is calculated as the square root of the sum of the squares of sum:x, sum:y, and sum:z.

- **Frequency:**

For a modal structural simulation, you can display results for a particular natural frequency by specifying the mode associated with that frequency. You can review the frequency for the specified mode under **Global > Frequency**.

- For a time-dependent simulation, time history is available in chart form by clicking the  icon.
- For a non-linear simulation, substep history is available in chart form by clicking the  icon.

## 5.6.4.1. Displaying Locations of Minimum and Maximum Values

### Results > Object > Object # > Summary

To display locations of minimum and maximum values, you must create a results object.

The location of minimum and maximum values is shown by default and is represented by the  icon. You can click the icon and hide the location display; to re-display the locations:

1. Open the flyout menu by clicking the right arrow to the right of the property field.
2. Click the **Show location** icon.



**Note:**

The minimum and maximum values of some variables are computed at face centers. The location of the minimum and maximum values will still be annotated at the face center, even if the contour is plotted without using face centers which may appear out of sync with the contour plot. If a contour is plotted using face values, the minimum and maximum value locations will be in sync with the contour plot.

## 5.7. Animation Results

You can view and animate a single qualitative result object or multiple result objects. Multiple results are managed from any result summary grid.

- [Animating a Single Result](#) on page 579
- [Animating Multiple Results](#) on page 579
- [Animation Slider Behavior in Different Solution Types](#) on page 579

### 5.7.1. Animating a Single Result

Where available, you can animate your result using the animation controls available in the Graphics view.

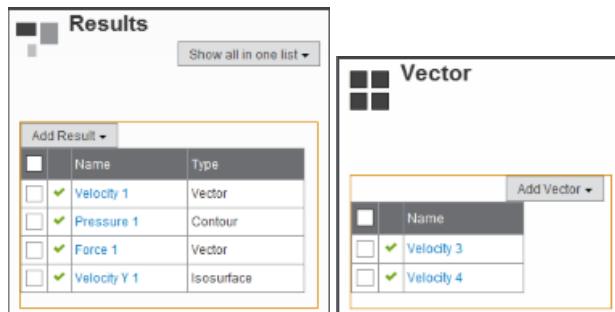
The animation slider in the Graphics view works in conjunction with these controls. How it works depends upon both the simulation type and if an animation is playing or paused.

Once **Play** is selected for an animation, the position of the slider's marker will represent that of the result being shown in the graphical representation. If the animation is paused, you may move the slider with the mouse to see the values at other frames of the animation.

Once the animation is stopped, it reverts to the position at which the result was evaluated. When the view is not animating, the behavior of the slider depends on the solution type. For more information, see [Animation Slider Behavior in Different Solutions](#).

### 5.7.2. Animating Multiple Results

You can display or animate multiple results at the same time from any results summary grid. Check the boxes to select the result objects you want to view together and click to play.



To preview a result, hover over the result name to display that result and legend; the previewed result displays regardless of the results selected.

**Note:**

- Multiple results animations do not display an animation slider.
- A result on a face location may obscure other results.
- Legends will not display when viewing multiple results. To view the legend of a particular result, hover over the desired result in the Graphics view.

### 5.7.3. Animation Slider Behavior in Different Solution Types

The slider available in the Graphics view (and sometimes the result type) will vary in behavior based on the solution type.

## Linear Static / Steady State and Modal Solutions

The animation is performed from the rest position to maximum magnitude of the selected result variable as calculated. In the case of modal solutions, this is the mode shape deformation. For streamlines, it is performed based on pseudo time. For these cases, you can only interact with the slider to change the frame while the animation is paused.

# Non-linear, Structural, Solid Thermal or DC Conduction Solutions

The animation is based on the values of the intermediary results sets. When stopped, you may select a different result set (based on percentage) to evaluate results.

## Linear or Non-linear Multi-Step Solutions

The animation is based on the values of the intermediary results sets for the simulation step selected in the data panel. When it's stopped, you may select a different result set (based on percentage for the selected step) to evaluate results.

## Time-Dependent Solutions

When animating results, such as contours and vectors, the animation is based on the duration of the simulation. When it's stopped, you may select a different time to evaluate results. In general, the displayed graphical results are interpolated from the nearest computed data sets. However, for fluid flow solutions, the calculated result values that are displayed under **Summary** are those results that are closest to the selected time point. You may choose to increase the density of the result sets in the **Solver Options > Output Control** panel for more frequent results.

For time dependent solutions, you may also type a time value or expression in the time display below the slider and then evaluate the result. If you define the time using an expression, you will no longer be able to adjust the value via the slider to ensure that the expression input is not lost.

For time-dependent fluid flow solutions, if you disable **Perform evaluation for full range** your result will update faster for the selected time, but if you move the slider to another time point you will need to evaluate with each time modification. In addition, you can only animate your result using the animation controls  available in the Graphics view if this is enabled.

**Note:** The calculated minimum, maximum, and average values that appear under **Summary** in, for example, the contour panel can only be evaluated for the currently selected time. However, you can view a chart displaying the variable over all time points by expanding the arrow to the right of the calculated value field

and selecting .



# Chapter 6: Using Alternative Workflows

---

In addition to [templates](#), AIM offers various other methods to create or modify workflows:

[Use shells in your simulation](#)

[Add fatigue analysis to your structural simulation](#)

[Model trusses and cables](#)

## 6.1. Additional Methods for Creating Simulations

The most efficient way to create an AIM simulation is to use a [template](#); however, you can also manually create a simulation.

- **Connect to CAD**

Imports a geometry into a Geometry task (through the Geometry Import Source object). You can change the **Source selection type** from **Active CAD Attach** to **Browse Source**, but you cannot edit the geometry in AIM.

- **Import Database**

To open an [archived](#) study from a local or networked file system, from the start page select **Browse**.

- **Add a Task**

Creates and adds the task you specify. Right-click in the **Workflow** tab, select **Add Task** and then the task you want to create.

Once you import or define a geometry, create the subsequent tasks in your simulation by right-clicking the Geometry task in the Workflow tab and select **Add Next**.

---

## 6.2. Efficient Modeling of Thin-Shell Structures

A surface body is an alternative way of representing a solid body, by using a surface and thickness. It offers computationally efficient solutions for simulating physics on thin-shell structures (typically defined when thickness < 1/20 times span). Surface bodies can be created from solids by applying mid-surface extraction (by selecting **Prepare** in the Geometry Modeler) to existing models. Surface bodies representing thin-shell structures are sometimes referred to as shells.

### 6.2.1. Workflow for Modeling Thin-Shell Structures

To model thin-shell structures using surface bodies:

1. Ensure the geometry import supports surface bodies.

- a) Select  > **Tools** > **Options**.  
 b) Select the **Geometry Import** category.  
 c) Under **Basic Options**, ensure **Surface Bodies** is selected.  
 d) Under **Advanced Options**, set the **Analysis Type** to **3D**.

2. Import your model. You can use the Structural template or manually import using the Geometry Modeling or Data Import tasks.

3. Shell Thickness must be defined in the Geometry task for any surface bodies with a positive non-zero thickness in the model. How it is defined depends on whether you are using integrated model editing.
  - [Defining Thickness in Geometry Modeler](#) on page 583
  - [Defining Thickness for Imported Geometry \(No Model Editing\)](#) on page 583
4. In the Physics task, AIM performs automatic contact generation as part of the template in the following cases.
  - When the model has surface bodies only, the edge-edge detection type is set.
  - If it has solid and surface bodies, the face-face, face-edge and edge-edge detection types are set.

For all other cases, you can [create the contacts for your surface bodies](#). You can also set your contact detection direction, include the effect of the thickness of the surface body during contact calculations for nonlinear contacts, and determine the DOF (degree of freedom) to be used for the MPC formulation.
5. Define your loads and constraints as you typically would. Surface bodies support all structural conditions applicable to solid bodies.
  - You can apply a [Pressure](#) to the face of a surface body, or you can apply a [Force](#) load to the edge of a surface body as a line pressure.
  - For an assembly of bodies that have different topologies (solid body, surface body), define a separate [Body Temperature](#) load for each topology.
6. View your [results](#). Note that shells have different results on top and bottom layers. Also, the Isosurface result is not applicable to shell locations.

### 6.2.2. Defining the Thickness for Surface Bodies Representing Shells

Shell thickness is defined automatically during geometry import if the thicknesses of the surface bodies are defined in the geometry. However, shell thickness must be defined in the Geometry task for any surface bodies without a defined thickness in the model. How it is defined depends on whether you are using integrated model editing.

#### 6.2.2.1. Defining Thickness in Geometry Modeler

If you defined a new geometry using the Geometry Modeler, or if you imported a geometry with the default **Allow geometry modeling** option enabled, edit the geometry to update or define thickness values. You must define a non-zero thickness for all surfaces that are to be included in the physics region. You can also parameterize your thickness values.

**Note:** If you want to define an offset, save your geometry and then import it without model editing enabled.

To define or modify thickness values in the Geometry Modeler:

1. In the Geometry task, click **Edit Geometry**.
2. In the Geometry Modeler, select surface bodies and define a non-zero thickness value in the properties view.

#### 6.2.2.2. Defining Thickness for Imported Geometry (No Model Editing)

##### Geometry > Section Definitions > Shell Thickness

If you imported a Geometry without allowing model editing, you can define thickness values directly in the Geometry task for any geometry that didn't have a thickness defined automatically upon import.

Thickness must be defined, either manually or automatically at import, for all faces of surface bodies included in your physics region. You may define thickness for a surface body location (i.e. all faces in the surface body), or individual faces within a surface body. If a thickness is defined for a surface body and individual faces within that body, the thickness defined for the faces will take precedence.

To define thickness values directly within the Geometry task:

1. In the Geometry task, define a new shell thickness by right-clicking and selecting **Add > Section Definitions > Shell Thickness**.
2. For the **Location**, select one or more bodies for which you want to define the thickness.
3. Enter a positive non-zero value for the **Thickness**.
4. For **Thickness distribution**, set the position of the surface body.

A surface body has a normal direction. By default, the surface body is aligned with the midsurface, but you can also select a one-sided offset or define an offset value.

5. When using an offset, you can also **Reverse offset direction**.
6. Repeat these steps for additional bodies.

### 6.2.3. Creating Contacts for Surface Bodies Representing Shells

To effectively set up [contacts](#) for surface bodies representing shells, you have several options available to you [when generating your contacts](#), on the contact setup itself, and in the [contact behavior](#). Contact setup is available in the Physics task.

## Generating Contacts

1. In the Physics task, create an [Interface Generator](#) as you normally would, selecting the relevant bodies for the location.
2. For **Detection types**, select the entity types for the connections you want to generate.
3. Generate your contacts.

## Setting Contact Detection Direction

The search for a contact detection of a contact pair involving a face is performed by the contact algorithm in the normal direction towards the contact. However, in some nonlinear cases you need to set the normal of the surface bodies at each side of the contact to ensure the contact is set up correctly. In this case, set the **Contact Detection Direction** property in the [Contacts panel](#) to **Manual** and select the location for which you want the normal reversed.

## Setting Shell Thickness

If you want to automatically include the effect of the surface body thickness during contact calculations, select **Include shell thickness where possible** in the [Contact Behavior panel](#).

The contact will be detected a distance of half the thickness away from the face.

**Note:** Shell thickness will be included only if the **Thickness Distribution** is set to **Equal from mid-surface**.

## Setting MPC Type for Bonded Contacts

You can also determine the DOF (degree of freedom) to be used for the MPC formulation by setting the **Constraint type** in the [Contact Behavior panel](#). This property is applicable for a bonded contact on a surface body with a contact formulation of MPC. You can set it to Target Normal, with the translation and rotation

coupled or uncoupled, or to coupled anywhere to be found inside the pinball region. For more information, see [Specifying Contact Behavior](#).

## 6.3. Fatigue Analysis

Fatigue in structural physics is defined as the weakening of a material subjected to repeated application of loads. Fatigue results in structures failing even before their ultimate tensile stress limit or yield stress limit is reached.

Fatigue analysis provides life, damage, and factor of safety information and uses a stress-life or strain-life approach, with several options for handling mean stress and specifying loading conditions. AIM also provides equivalent alternating stress and biaxiality indication information. Common uses for the strain-life approach are in notched areas where, although the nominal response is elastic, the local response may become plastic.

**Note:** Fatigue life has a large variation and is therefore represented on a logarithmic scale. However, you can turn logarithmic scale on or off in the **Appearance** settings for a fatigue Contour result.

For more information, see [Fatigue Analysis using Stress Life](#) on page 585 and [Fatigue Analysis using Strain Life](#) on page 589.

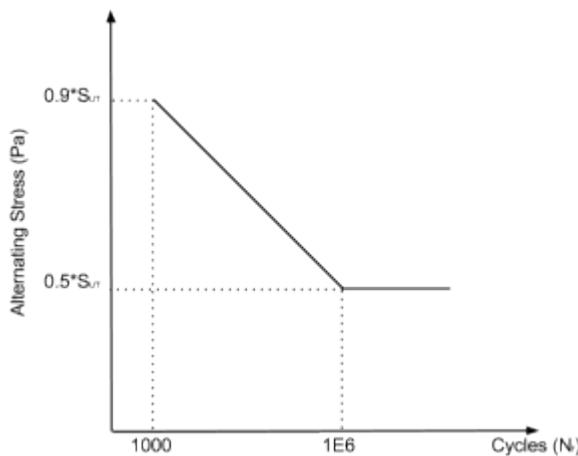


### Fatigue Analysis

#### 6.3.1. Fatigue Analysis using Stress Life

In structural simulations, you can use fatigue analysis in conjunction with typical stress analysis to decide the fatigue life of a structure when it is subjected to cycles of loads. For simple stress life based analysis, it is typically characterized by an S-N curve (cyclic stress ( $s$ ) against the logarithmic scale of cycles to failure ( $n$ )) as part of material definition of their structure.

To perform a stress-life analysis, you need the S-N curve data (alternating stress vs. number of cycles to failure). In the absence of S-N curve data, you can specify an ultimate strength value. Then AIM internally constructs a simplified S-N curve using just the ultimate strength, as illustrated below:



The default workflow for stress-life analysis is to direct AIM to compute the fatigue results as part of [using a structural template](#). Fatigue life is then calculated based on the S-N curve values defined for the default materials and using default fatigue settings. If you want to use fatigue settings other than the defaults, you can then [modify the fatigue settings](#) before viewing the fatigue contours.

If you want to assign other materials to your model, you can [specify the S-N curve and ultimate strength values when defining the materials in your model](#). You can then add fatigue life from the results and direct

AIM to use the values from your materials. If you want to use fatigue settings other than the defaults, you can [modify the options](#) before viewing the fatigue contours.

You can also add fatigue life from the results and [set your S-N curve values or ultimate strength value there](#); these values will apply to all materials in the model. You can then view the fatigue contours after [modifying any other desired fatigue settings](#).

## 6.3.1.1. Computing Fatigue Results for Stress Life Using a Template

### Results > Fatigue Settings

When you are using a default material such as Structural Steel or Aluminum, you can compute the fatigue results for stress life by setting an option in the template. AIM then uses the S-N curve data defined on these materials to calculate the fatigue results for your model. These results are available on the **Results** panel after you solve and evaluate your results. A Fatigue Life contour result is created by default, but you can also add contour results for [Fatigue Damage, Safety Factor, and other fatigue variables](#).

**Note:** If you want to define other materials for your model, you can [add the S-N curve or ultimate strength values on the materials](#) and assign them to your model, then add fatigue results, [modifying the fatigue settings as desired](#). You can also add fatigue results and define fatigue settings that [apply to all materials in a model](#).

To compute fatigue results as part of a template:

1. On the **Structural** template, select **Compute fatigue results**.
2. Define the rest of your mesh and physics solution tasks as you normally would, and then solve.
3. [Modify any of the fatigue settings](#), such as the loading type or design life.
4. Evaluate your Results task and view the Fatigue Life contour. You can also add contour results for [Fatigue Damage, Safety Factor, and other fatigue variables](#).

## 6.3.1.2. Analyzing Stress Life by Defining Materials and Properties

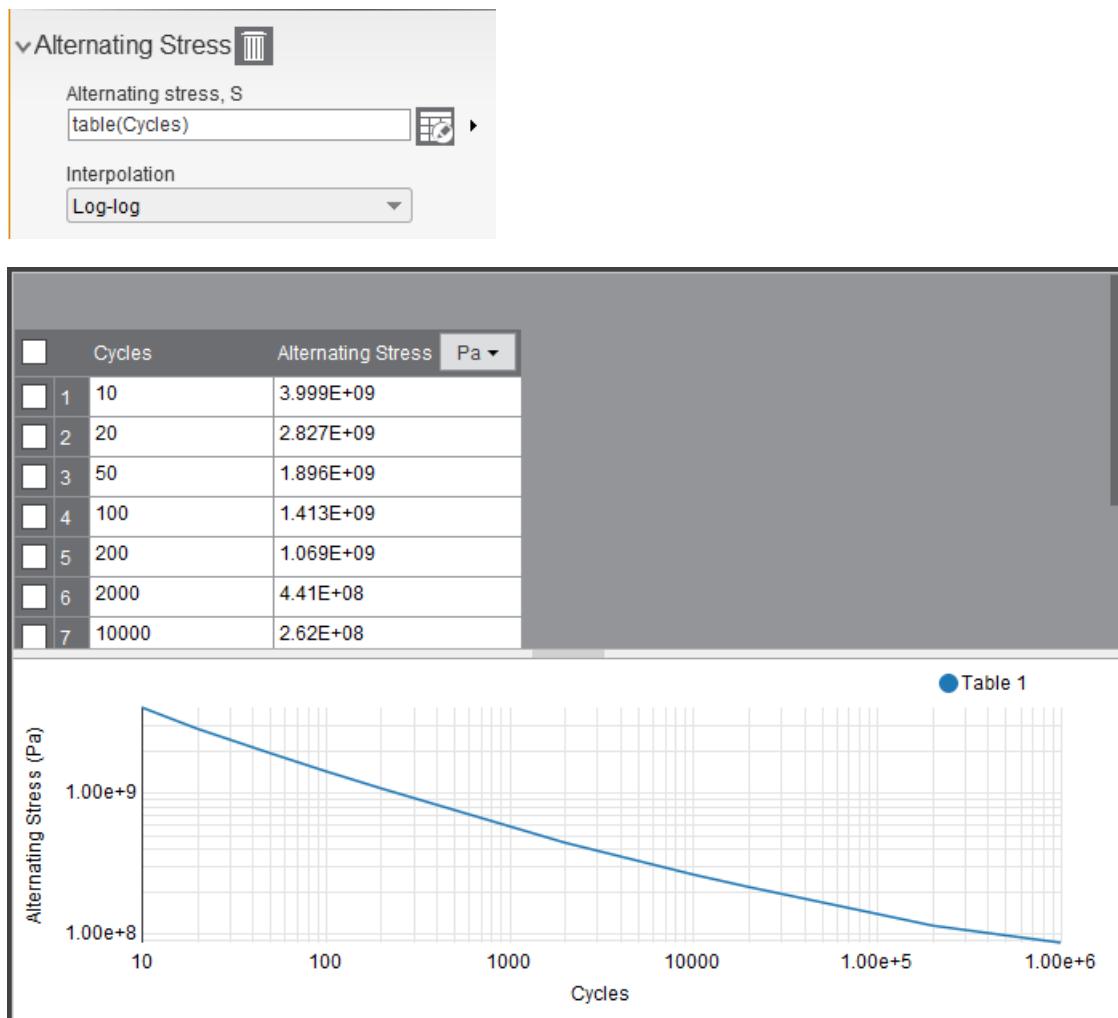
### Results > Fatigue Settings

If you are not using one of the default materials (Structural Steel, Aluminum), you can define S-N Curve and Ultimate Strength values as you define each material used in your model. When you then add a Fatigue contour result, your fatigue settings are fully defined, and you need only to modify the default values as desired. AIM automatically uses S-N curve values if they are defined. If S-N curve values are not defined but ultimate strength is, then AIM internally constructs a [simplified S-N curve using just the ultimate strength](#).

If you want to define S-N Curve and Ultimate Strength values that are used for *all* materials, you can do so in the [Fatigue Settings](#) on page 593.

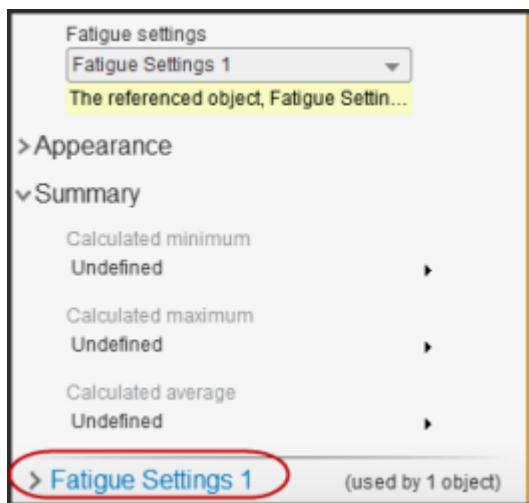
To analyze fatigue results using values defined on the materials:

1. In the **Physics** task, select **Material Assignments** to define your materials.
2. When defining the materials you want to use in this simulation, specify the S-N curve data as a table of alternating stresses vs. cycles in the **Alternating Stress, S** property.



**Important:** Currently, AIM only supports single S-N Curves at zero mean stress.

- a) Select the type of interpolation method for the S-N Curve. This interpolation method is used in both the chart display and the fatigue solver.
  - b) If you want to use Goodman or Gerber for your Mean stress theory, also define the **Tensile Ultimate Strength, S** property.
  - c) If you want to use Soderberg for your Mean stress theory, also define the **Tensile Yield Strength, S** property.
3. Define the rest of your mesh and physics solution tasks as you normally would.
  4. On the Results panel, select **Add > Fatigue Life**.
  5. Expand or click the **Fatigue Settings** for the result.



6. For **Analysis type**, ensure you have selected **Stress life**.
7. For **S-N curve data source**, ensure you have selected **Assigned materials**.
8. **Modify any of the fatigue settings**, such as the loading type or design life.
9. Evaluate your Results task and view the Fatigue Life contour. You can also add contour results for **Fatigue Damage, Safety Factor, and other fatigue variables**.

### 6.3.1.3. Analyzing Stress Life by Defining Stress Life Properties for All Materials

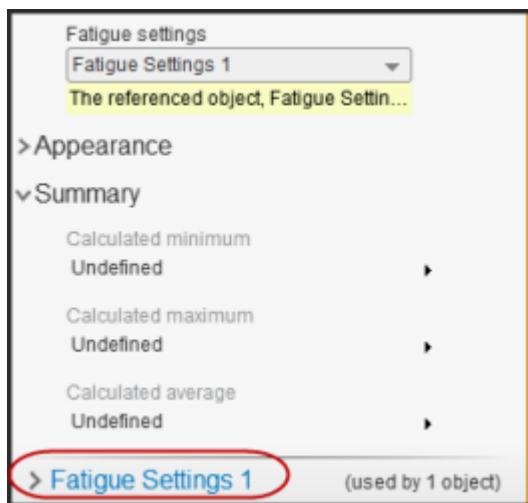
#### Results > Fatigue Settings

From the Fatigue Settings, you can define S-N Curve values that are used for all materials in your model. If you don't have S-N curve values, you can instead define an ultimate strength value, and AIM internally constructs a **simplified S-N curve using just the ultimate strength** and uses it when computing fatigue life.

You can also define S-N Curve and Ultimate Strength values as you define each material used in your model; see [Analyzing Stress Life by Defining Materials and Properties](#) on page 586 for more information.

To compute fatigue results using specified values:

1. In the **Physics** task, select **Material Assignments** to define your materials.
2. Define the rest of your mesh and physics solution tasks as you normally would and solve the physics.
3. On the Results panel, select **Add > Fatigue Life**.
4. Expand or click the **Fatigue Settings** for the result.



5. For **Analysis type**, ensure you have selected **Stress life**.
6. For **S-N curve data source**, select **S-N curve values**, and then specify the S-N curve data as a table of alternating stresses vs. cycles in the **Alternating Stress, S** property.

This S-N Curve data is then used for all materials. You can also [define different S-N Curve values for each material](#) used in the model.

**Important:** Currently, AIM only supports single S-N Curves at zero mean stress.

If you don't have S-N curve values, you can instead define an **Ultimate strength**, and AIM internally constructs a [simplified S-N curve using just the ultimate strength](#) and uses it when computing fatigue life.

- a) Select the type of interpolation method for the S-N Curve. This interpolation method is used in both the chart display and the fatigue solver.
- b) If you want to use Goodman or Gerber for your Mean stress theory, also define the **Tensile Ultimate Strength, S** property.
- c) If you want to use Soderberg for your Mean stress theory, also define the **Tensile Yield Strength, S** property.
7. [Modify any of the fatigue settings](#), such as the loading type or design life.
8. Evaluate your Results task and view the Fatigue Life contour. You can also add contour results for [Fatigue Damage, Safety Factor, and other fatigue variables](#).

### 6.3.2. Fatigue Analysis using Strain Life

You can also compute fatigue results using a strain-life approach. Common uses for the strain-life approach are in notched areas where, although the nominal response is elastic, the local response may become plastic.

While fatigue analysis can be computed with the structural template, this workflow defaults to stress-life. To compute strain life, define the strain life material properties when defining the materials for your model, then, in the fatigue settings, select strain life.

For more information, see [Strain Life](#) on page 589.

#### 6.3.2.1. Strain Life

While Stress life fatigue is based on empirical S-N curves, Strain life fatigue is based upon the Strain Life Relation Equation where the Strain Life Parameters are values for a particular material that best fit the equation to measured results.

The Strain Life Relation requires a total of six parameters to define the strain-life material properties; four strain-life parameter properties and the two cyclic stress-strain parameters.

The Strain Life Relation equation is shown below:

$$\frac{\Delta\varepsilon}{2} = \frac{\sigma'_f}{E} (2N_f)^b + \varepsilon'_f (2N_f)^c$$

The two cyclic stress-strain parameters are part of the equation below:

$$\Delta\varepsilon = \frac{\Delta\sigma}{E} + 2 \left( \frac{\Delta\sigma}{2K'} \right)^{\frac{1}{n'}}$$

Where:

$\frac{\Delta\varepsilon}{2}$  is the Total Strain Amplitude

$\Delta\sigma$  is 2 x Stress Amplitude

$E$  is the Modulus of Elasticity

$N_f$  is the Number of Cycles to Failure

$2N_f$  is the Number of Reversals to Failure

And the parameters required for a Strain Life analysis are:

$\sigma'_f$  is the Fatigue Strength Coefficient

$b$  is the Fatigue Strength Exponent (Basquin's Exponent)

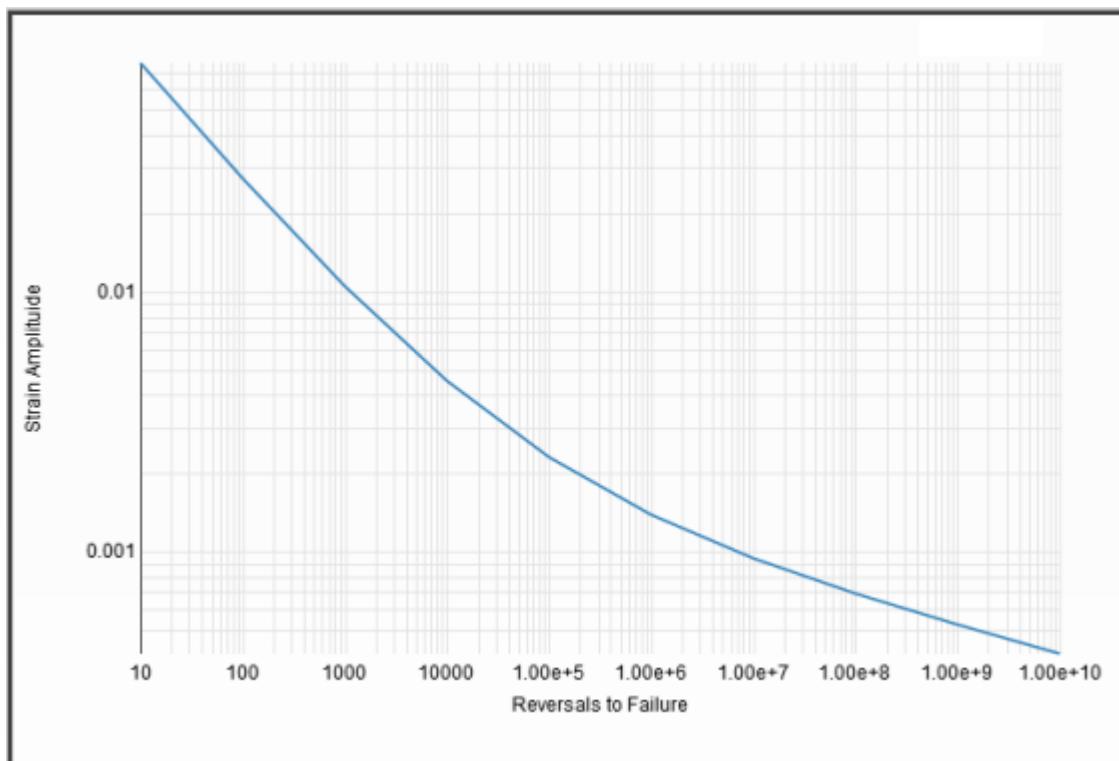
$\varepsilon'_f$  is the Fatigue Ductility Coefficient

$C$  is the Fatigue Ductility Exponent

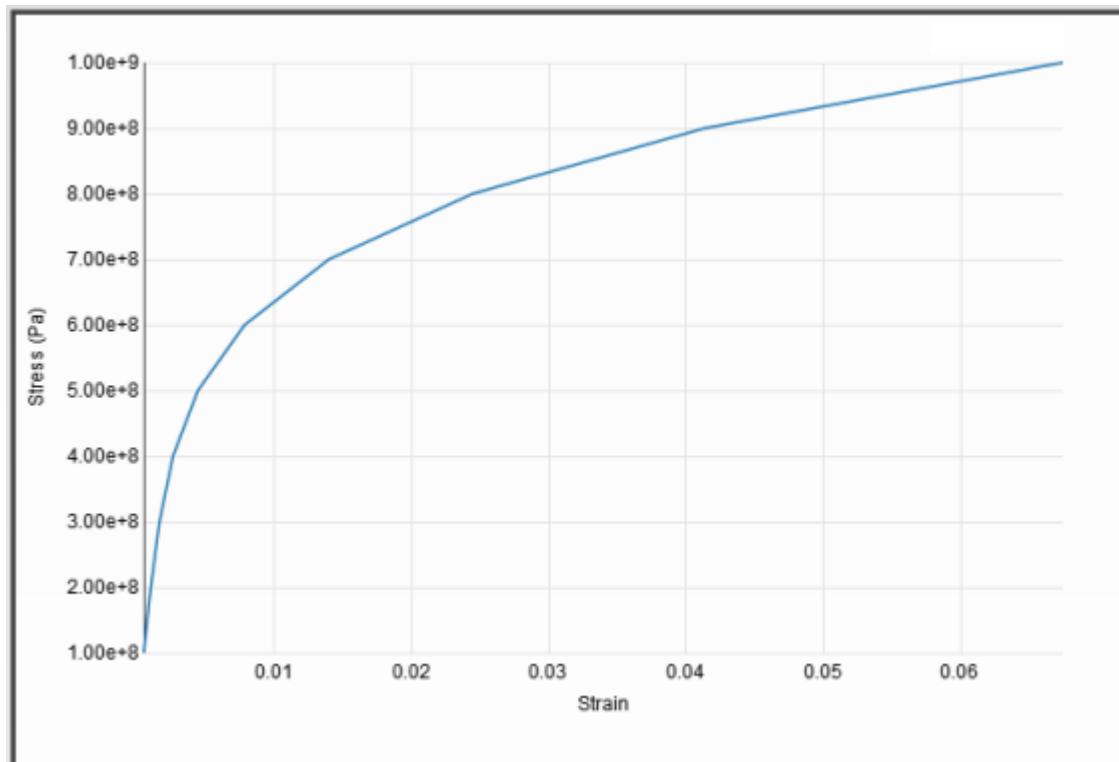
$K'$  is the Cyclic Strength Coefficient

$n'$  is the Cyclic Strain Hardening Exponent

The total Strain amplitude  $\frac{\Delta\varepsilon}{2}$  can be plotted as shown below in the Strain-life curve:



Similarly the cyclic stress strain equation can be plotted as shown below in the Cyclic Stress-Strain curve:



## 6.3.2.2. Computing Fatigue Results for Strain Life Using a Template

### Results > Fatigue Settings

When you are using a default material such as Structural Steel, you can compute the fatigue results for strain life by setting an option in the template. AIM then uses the strain material properties defined on this material to calculate the fatigue results for your model. These results are available on the **Results** panel after you solve and evaluate your results. A Fatigue Life contour result is created by default, but you can also add contour results for [Fatigue Damage](#), [Safety Factor](#), and other fatigue variables.

To compute fatigue results as part of a template:

1. On the **Structural** template, select **Compute fatigue results**.
2. Define the rest of your mesh and physics solution tasks as you normally would, and then solve.
3. In the [Fatigue Settings](#), select **Analysis type > Strain life**.
4. [Modify any of the other fatigue settings](#), such as the loading type or design life.
5. Evaluate your Results task and view the Fatigue Life contour. You can also add contour results for [Fatigue Damage](#), [Safety Factor](#), and other fatigue variables.

## 6.3.2.3. Analyzing Strain Life by Defining Strain Material Properties

### Results > Fatigue Settings

If you are not using the default Structural Steel material, you can specify your strain life material properties as you define each material used in your model. When you then add a Fatigue contour result, your fatigue settings are fully defined, and you need only to select strain life and modify the default values as desired.

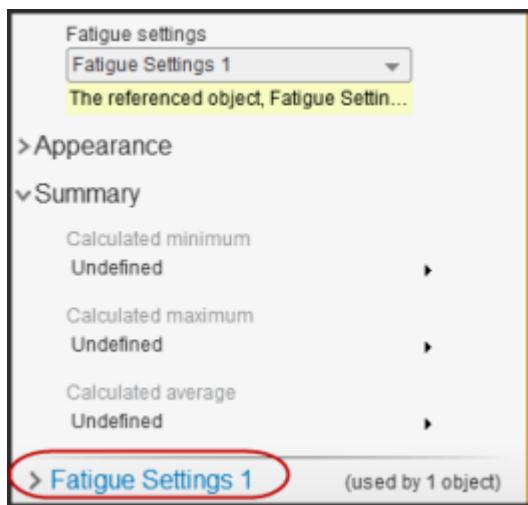
To perform a strain life fatigue analysis:

1. In the **Physics** task, select **Material Assignments** to define your materials.

The required material properties are:

- Strength Coefficient
- Strength Exponent
- Ductility Coefficient
- Ductility Exponent
- Cyclic Strength Coefficient
- Cyclic Strain Hardening Exponent

2. Define the rest of your mesh and physics solution tasks as you normally would.
3. On the Results panel, select **Add > Fatigue Life** or any other fatigue result.
4. Expand or click the **Fatigue Settings** for the result.



5. For **Analysis type**, select **Strain life**.
6. **Modify any of the fatigue settings**, such as the loading type or scale factor.
7. Evaluate your Results task and view the Fatigue Life contour. You can also add contour results for **Fatigue Damage**, **Safety Factor**, and other fatigue variables.

### 6.3.3. Fatigue Settings

The Fatigue Settings allow you to modify the default settings used to compute fatigue results.

Table 6.3.3.1. Typical Properties

If you want to...	then set...	More information:
Select the type of approach used for fatigue analysis.	<b>Analysis type</b>	Select Stress life or strain life. The default is stress life.
Accumulate the fatigue damage until the chosen simulation step.	<b>Loading &gt; Accumulate damage</b>	This is just a summation of the individual fatigue calculations; there is no cycle counting involved.
Select the type of applied constant amplitude loading based on the loading ratio.	<b>Loading &gt; Loading type</b>	<p>Select the loading type. Choose from the following:</p> <ul style="list-style-type: none"> <li>• Full Reversed (<math>r=-1</math>)</li> <li>• Zero (<math>r=0</math>)</li> <li>• Ratio</li> </ul> <p>If you select <b>Ratio</b>, specify the ratio in the <b>Loading ratio</b> field. For example, a loading ratio of -0.5 means the minimum stress (lower bound) is -0.5 times the stress in the body while the maximum stress (upper bound) remains 1 times the stress in the body.</p>
Specify design life for damage and safety factor calculations.	<b>Design Life &gt; Design life</b>	Enter the life that the system is being designed for, using the units defined in <b>Life units</b> . This option is only useful for damage and safety factor calculations and can be ignored while calculating Fatigue Life.

Table 6.3.3.2. Other Properties

If you want to...	then set... More information:
Scale the load magnitude.	<p><b>Loading &gt; Scale factor</b></p> <p>This setting scales the load magnitude by the specified value. This includes both alternating and mean stresses.</p> <p>For example, if you set this to 3, the amplitude (and mean) of a zero-based loading will be 1.5 times the stress in the body.</p> <p>This option is useful for seeing the effects of different finite element loading magnitudes without having to run the complete structural analysis repeatedly.</p> <p><b>Note:</b> This scale factor is applied after the stresses have been collapsed from a tensor into a scalar. Thus any multiaxial stress collapse methods that are sensitive to the sign (Von-Mises, Maximum Shear, Maximum Principal) may not give the same answer had the scale factor been applied to the environment load itself.</p>
Specify how mean stress effects are handled.	<p><b>Options &gt; Mean stress theory</b></p> <p>This setting specifies how the mean stress effects should be handled.</p> <p>If the analysis type is set to Stress Life, choose from None, Goodman, Soderberg, Gerber, and Mean Stress Curves. The Goodman, Soderberg, and Gerber options use static material properties along with S-N data to account for any mean stress while Mean Stress Curves uses experimental fatigue data to account for mean stress.</p> <p>If the analysis type is set to Strain Life, choose from None, Morrow, and SWT (Smith-Watson-Topper).</p> <p><b>Note:</b> For more information on these theories, see <i>Metal Fatigue In Engineering</i> by Ralph I. Stephens, et. al.</p>
Specify the stress component type.	<p><b>Options &gt; Stress component</b></p> <p>Because stresses are multiaxial but experimental fatigue data is usually uniaxial, the stress must be converted from a multiaxial stress state to a uniaxial one.</p> <p>You can choose from several types, including component stresses, von Mises, and a signed von Mises, which takes the sign of the absolute maximum principal stress. The signed von Mises is useful for accounting for any compressive mean stresses. In the case of Maximum Shear Stress, AIM uses a value of two times the maximum shear stress for calculations.</p>
Specify the maximum life for strain life to avoid skewed contour plots showing very high lives.	<p><b>Options &gt; Infinite life</b></p> <p>Since the strain-life method is equation-based, it has no built-in limit (unlike stress-life, for which AIM uses a maximum life equal to the last point on the SN curve).</p> <p>To avoid skewed contour plots showing very high lives, specify an Infinite Life value. For example, if you set a value of 1E9 cycles as the Infinite Life, the maximum life reported is 1E9.</p>
Change the life units used to display fatigue life results.	<p><b>Life units &gt; Unit type</b></p> <p>You can change the units used to display the fatigue life results. By default, fatigue results are shown in cycles.</p> <p>If you choose <b>Blocks</b>, specify the number of blocks per cycle.</p> <p>If you choose a unit of time, then specify the amount of time per cycle.</p>

## 6.4. Modeling Trusses and Cables

AIM includes support for analyzing uniaxial tension-compression scenarios and uniaxial tension-only scenarios like cables. To analyze trusses and/or cables, ensure that all bodies in your model include a line model type definition in the Geometry Modeler. AIM will then mesh each line body as a single element using automatic physics-aware meshing.

To define a line model type for the bodies in your model:

1. Click **Edit Geometry** in the **Geometry** task.
2. Set the line model type for each body to **Cable** or **Link/Truss**.
3. Close the Geometry Modeler and proceed with your analysis.

## 6.5. Using Flagship Products with AIM-based Physics Definition

AIM supports the transfer of geometry, mesh, and selected physics definition data to Mechanical.

If you are starting a new simulation and know that you want to combine AIM's flexible modeling capabilities without any physics definition to Mechanical or Fluent, you can use the **Connect to Mechanical** or **Connect to Fluent** templates to directly link AIM-based geometry and mesh definitions.

If you want to transfer meshing and physics definition data to Mechanical, follow the steps described in this section.

The transfer capabilities are summarized below:

Transfer Capability	Mechanical
Geometry (output)	X
Mesh (output)	X
Material Assignment	X
Selection Sets	X*
Local Coordinate System	X (see Details)
Structural Conditions	X (see Details)
Interface Conditions	Contacts and Joints

\* Selection sets created in the AIM **Meshing** task are transferred, but selection sets created in the AIM **Physics** task are not.

AIM expressions are not supported in a transfer to Mechanical. Parametric studies are not supported in a transfer to Mechanical.

### 6.5.1. Physics Transfer from AIM to Mechanical

This procedure describes transferring physics defined in AIM to a Mechanical system via the Workbench **Project** tab.

1. Use a **Structural** template to set up your structural study. Use the template as you normally would, but make sure to enable **Define mesh manually**.
2. Ensure that you have a valid mesh and physics configuration. Update your study.
3. Navigate to the **Project Schematic** using the **Project** tab, and drag a Mechanical system from the **Analysis Systems** list onto the **Project Schematic**.

4. Drag the **Physics Definition** cell onto the **Model** cell for a Mechanical system.
    - a) Physics definition data will be transferred if you connect the AIM **Physics Definition** cell to the flagship system.
    - b) Mesh data only will be transferred if you connect the AIM **Meshing** or AIM **Modeling** cell to the flagship system.
  5. All of your **Study** tasks will go out of date; update the **Physics Definition** cell in your **Study** to create the appropriate input files.
  6. Double-click the **Model** cell, or right click and select **Edit** to open Mechanical.
  7. You will see that your contacts and material data transferred into Mechanical. Note that not all physics conditions transfer; you should verify all of your settings before continuing. You can now continue with your analysis in Mechanical as you normally would.
- If you return to the AIM model to make changes after you've transferred it to Mechanical, you can then update the model to capture those changes from AIM. Geometry changes may require the rescoping of conditions applied within the flagship product. Physics definition changes may over-write input defined in the flagship product.

### 6.5.2. Details for AIM to Mechanical Transfer

#### Local Coordinate Systems

Location, type, and orientation are transferred; no other coordinate system settings are transferred from AIM. Only Cartesian and Cylindrical coordinate systems are supported. Spherical coordinate systems are not supported. If you have a reference frame with the Preferred Coordinate Type set to Spherical, it will not be transferred to Mechanical.

#### Structural Conditions

Structural conditions are transferred as named selections only; no values or settings are transferred from AIM. The named selection in Mechanical will follow the name as defined in the AIM Study. Once in Mechanical, you can redefine the structural conditions using the named selections.

#### General Contact

The **Scope Mode** setting in Mechanical will always be set to Manual, regardless of the **Contact > Location Definition Method** setting in AIM.

#### Contact Symmetry Settings

AIM Contact Setting	Mechanical Contact Setting
<b>Apply Symmetry</b> = No	<b>Behavior</b> = Asymmetric
<b>Apply Symmetry</b> = Yes	<b>Behavior</b> = Auto Asymmetric
<b>Detect Asymmetry Automatically</b> = Yes	
<b>Apply Symmetry</b> = Yes	<b>Behavior</b> = Symmetric
<b>Detect Asymmetry Automatically</b> = No	

#### Contact Behavior Settings

The **Program Controlled** option may have different implementations between AIM and Mechanical.

If the **Pinball > Define by** setting in AIM Contact Behavior is set to Auto Detection Value, Mechanical will convert it to Radius, but maintain the same value. The **Pinball > Define by > Factor** setting in AIM is only supported in Mechanical in Beta Options mode.

#### Constraint Settings

AIM Constraint Setting	Mechanical Constraint Type Setting
Target normal, uncouple translation to rotation	Projected, Uncoupled U to ROT
Target normal, couple translation to rotation	Distributed, All Directions
Inside pinball, couple translation to rotation	Distributed, Anywhere Inside Pinball

### Other Contact Behavior Settings

In AIM Contact Behavior, the **Elastic Slip Tolerance > Factor** and **Penetration Tolerance > Factor** must be larger than or equal to 0 and smaller than 1.

The following AIM contact behavior properties are not supported in Mechanical and are therefore not transferred:

- **Physics to transfer**
- **Normal Stiffness > Manual > Define by “Physics value”**
- **Tangential Stiffness**
- **Tangential stabilization damping factor**

# Chapter 7: Parameters and Design Points

AIM supports [parameters](#) and [design points](#).

- You can [define simulation properties as parameters](#) and use the **Design Points Dashboard** of AIM to [view the parameters](#). These parameters can also be used in other types of design exploration studies.
- A design point is a single set of parameter values representing one design alternative. Using the **Design Points Dashboard**, you can easily create new design points and perform [what-if](#) studies.

## 7.1. Parameters

A *parameter* is a characteristic of a model or a simulation that you [can quantify and vary](#) to determine its effect on the results of the simulation. This is known as [performing a what-if study](#).

A simulation can have input parameters and output parameters:

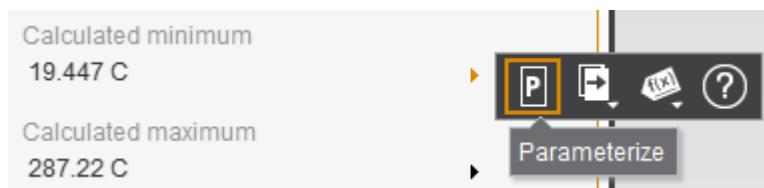
- *Input parameters* define the values to analyze for the model under investigation. They include CAD dimensions (such as length and radius), analysis parameters (such as pressure and material properties), and mesh parameters (such as mesh size).
- *Output parameters* are the response outputs from the analysis. They include volume, mass, frequency, stress, velocities, pressures, forces, and heat flux. The value of an output parameter is set by the application, based on the current results or state.

**Note:** Parametrizing CAD dimensions can cause the geometry to change the IDs used to reference certain faces and bodies. If these elements are used in downstream tasks, you may find that the tasks require attention when updating design points. To avoid this problem, in your CAD program, create named selections of all faces, bodies, and external flow boundaries that are used in tasks downstream from the geometry.

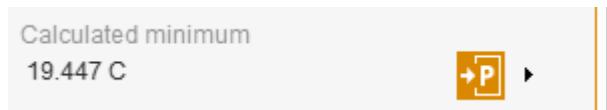
### 7.1.1. Defining Parameters

To parameterize a property:

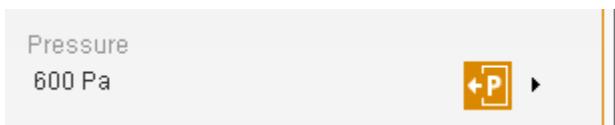
1. Open the flyout menu by clicking the right arrow to the right of the property field.
2. Click the **Parameterize** icon.



In this example, the property is an output parameter so the resulting parameter is an output parameter. The image beside the arrow indicates that this is an output parameter.



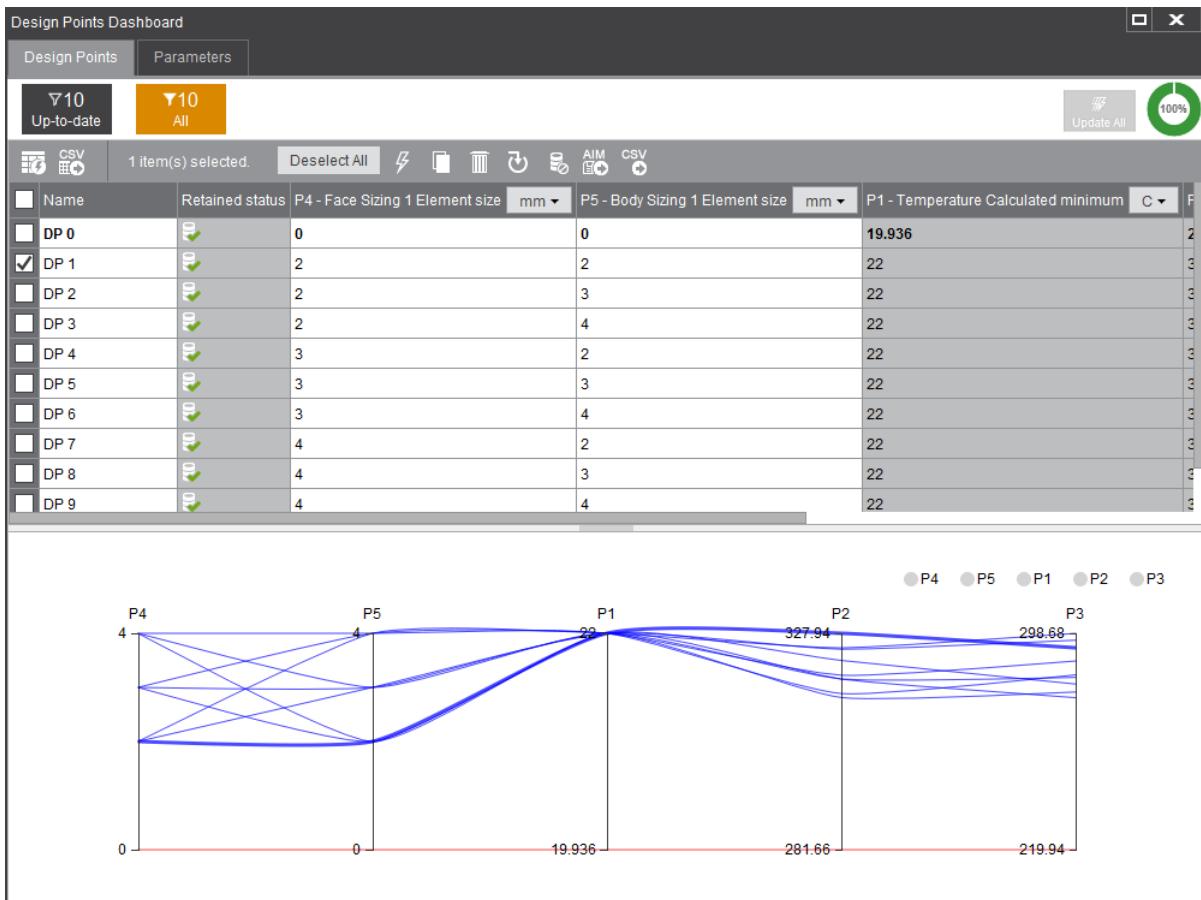
If the property is an input parameter, the resulting parameter is an input parameter, except when it is defined by an expression that makes it dependent on other properties.



On the **Parameters** tab of the **Design Points Dashboard**, the parameter appears.

Type	ID	Name	Value	Unit	Task	Object	Property
<b>Inputs</b>							
	P4	Face Sizing 1 Element size	0	mm	Mesh	Face Sizing 1	Element size
	P5	Body Sizing 1 Element size	0	mm	Mesh	Body Sizing 1	Element size
<b>Outputs</b>							
	P1	Temperature Calculated minimum	19.936	C	Results	Temperature	Calculated minimum
	P2	Temperature Calculated maximum	281.66	C	Results	Temperature	Calculated maximum
	P3	Calculated Value 1 Value	219.94	C	Results	Calculated Value 1	Value

On the **Design Points** tab of the **Design Points Dashboard**, the parameter appears in a table column.



## 7.1.2. Viewing Parameters

If you have a study that contains parameters, the **Parameters** tab of the **Design Points Dashboard** provides a convenient way to sort, view, and access these parameters.

To view the study's parameters:

1. Click the **Design Points Dashboard** icon.



2. Click the **Parameters** tab.

Type	ID	Name	Value	Unit	Task	Object	Property
<b>Inputs</b>							
	P4	Face Sizing 1 Element size	0	mm	Mesh	Face Sizing 1	Element size
	P5	Body Sizing 1 Element size	0	mm	Mesh	Body Sizing 1	Element size
<b>Outputs</b>							
	P1	Temperature Calculated minimum	19.936	C	Results	Temperature	Calculated minimum
	P2	Temperature Calculated maximum	281.66	C	Results	Temperature	Calculated maximum
	P3	Calculated Value 1 Value	219.94	C	Results	Calculated Value 1	Value

3. Use the dropdown menu above the table to organize parameters. You can organize parameters by:
  - Type (input or output)
  - Task (the workflow task in which the parameter was created, such as Mesh or Results)
  - Simulation (for studies that have multiple simulations)
  - Task type (for studies that have multiple parameter types per workflow task)
4. Click a link in the **Object** column to see the parameter in the data panel where the parameter is defined.

---

## 7.2. Design Points

A *design point* is a set of input parameter values and corresponding output parameter values associated with an individual parameterized project definition. Each design point is a single set of parameter values representing one design alternative.

Basically, you can think of a design point as a snapshot of your design given a set of parameter values, where output parameter values are calculated by an update of the project. Design points allow you to perform what-if studies or design exploration studies.

In AIM the design point shown in the graphics view is always the one that is set as current, which is DP0 by default. You cannot delete the current design point, but you can add new design points and perform other operations such as editing parameter input values.

For more information on performing basic design point operations, see:

- [Creating Design Points Manually](#) on page 601
- [Filtering Design Points](#) on page 603
- [Modifying Design Points](#) on page 607
- [Moving Parameters in the Dashboard](#) on page 608

For design point updates, you can select one or more design points for update or update all design points. For more information, see [Design Point Updates](#) on page 610.

Before generating data for a design point, you can mark the design point to retain its data within the project. For the current design point and all design points created in the **Design Points Dashboard**, data is retained by default.

When multiple design points have retained data, you can set a design point other than DP0 as current. Once design point data is generated, you can export it for use outside the project. For more information, see [Working with Design Point Update Data](#) on page 612.

**Note:** Non-parametric changes (any change other than a parameter value) to a current design point will cause other retained design points to go *out-of-date* starting with the first task that contains a parameter. By going out-of-date, any solve data associated with the task will be lost. Workarounds include:

- Finish the entire setup (for example, add all tasks and objects, and set all properties that won't be parameterized) before adding the parameters/design points.
- Set all design points (other than the current one) as non-retained since non-retained design points are not affected by this limitation.
- Save the project before making a non-parametric change, so you can access your old solve data if needed.

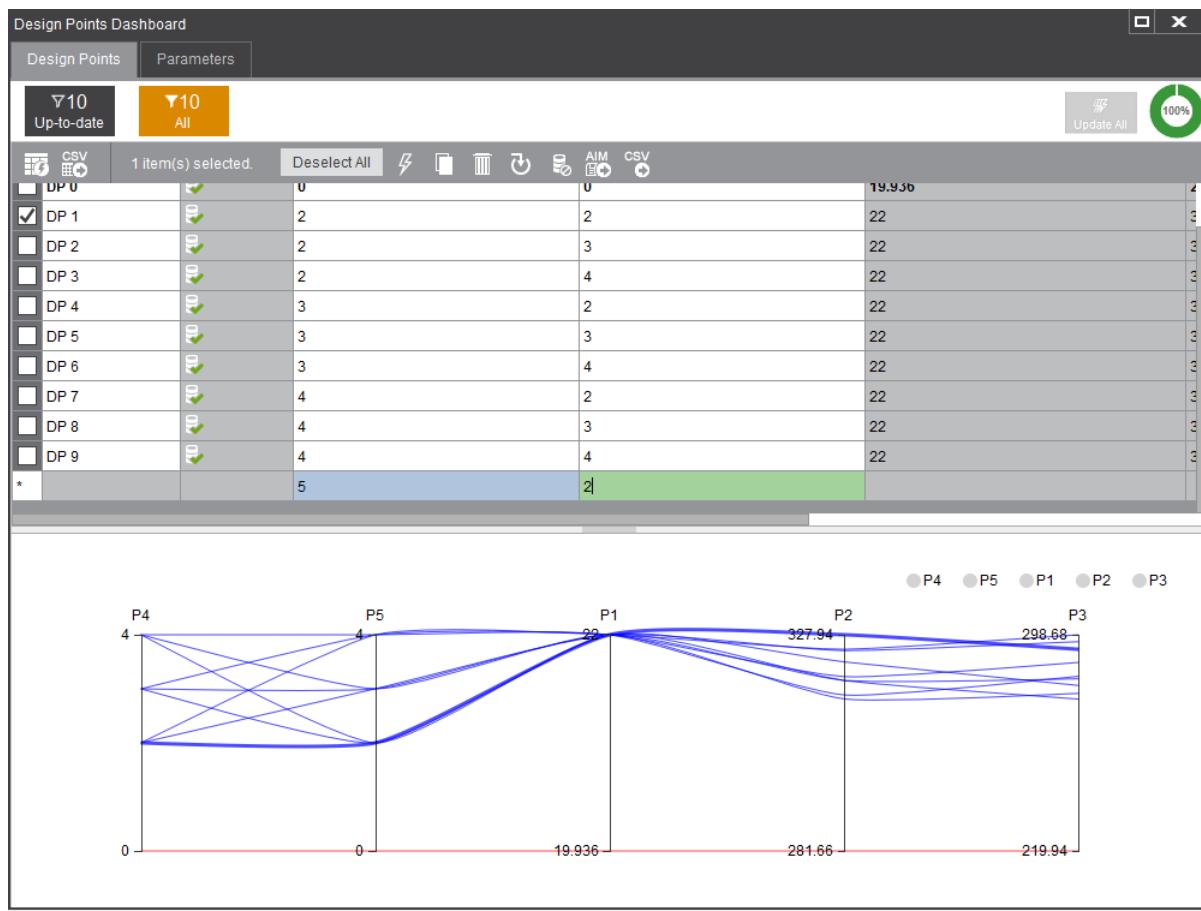
### 7.3. Creating Design Points Manually

To create a design point manually:

1. On the **Study** panel, click the **Design Points Dashboard** icon.



The **Design Points Dashboard** opens. The **Design Points** tab displays any existing design points, parameters, and parameter values in a table and chart.



2. In the new row at the end of the table, enter input parameter values and then press the **Enter** key

Your new design point is created. The default name for a design point is **DP *n***, where *n* increments the design point number by 1. You can rename the design point by clicking the **Name** field and typing a name that is meaningful to you.

The screenshot shows the 'Design Points Dashboard' interface. At the top, there are three buttons: 'Up-to-date' (10), 'Out-of-date' (1), and 'All' (11). Below this is a toolbar with CSV export and refresh icons. A table lists 10 design points (DP 1 to DP 10) with their retained status and a parameter value column.

Name	Retained status	P4 - Face Sizing 1
Basic Design	✓	0
DP 1	✓	2
DP 2	✓	2
DP 3	✓	2
DP 4	✓	3
DP 5	✓	3
DP 6	✓	3
DP 7	✓	4
DP 8	✓	4
DP 9	✓	4

**Note:** You can also create design points by duplicating existing design points. Use the check boxes to select one or more existing design points, click the **Duplicate selected** icon, and edit the input parameter values as needed.

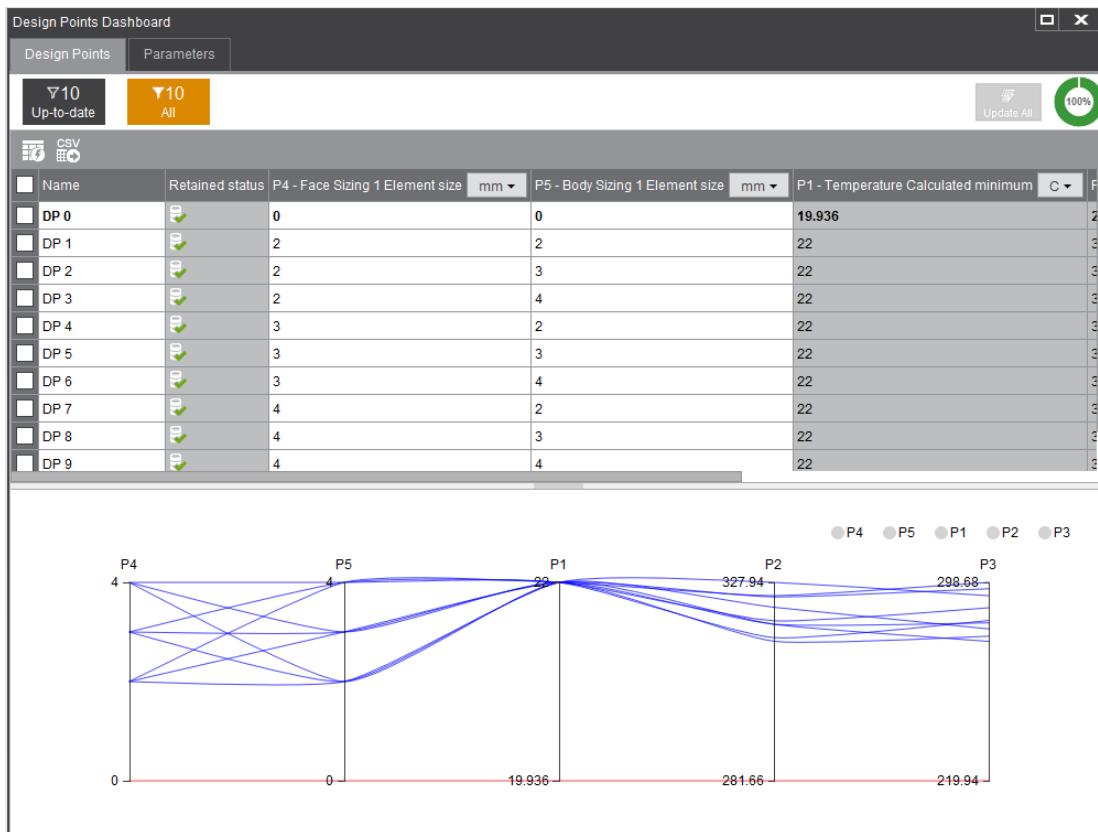
## 7.4. Filtering Design Points

You can filter design points based on their state and by ranges of their parameter values.

1. On the **Study** panel, click the **Design Points Dashboard** icon.



The **Design Points Dashboard** opens. The **Design Points** tab displays design points, parameters, and parameter values in a table and chart.



- To see only the parameters in a certain state, in the toolbar, click the button for the desired state. For descriptions, see [Design Point States](#).

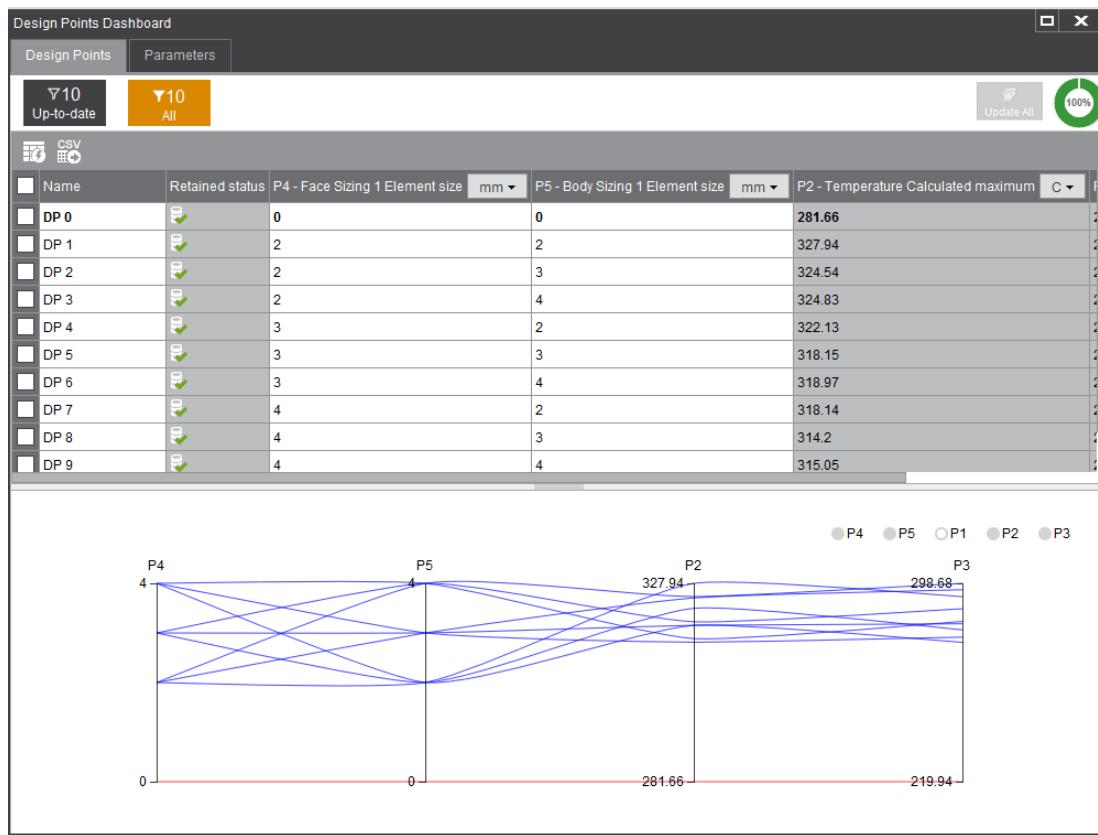
For example, if partially updated design points are present, the toolbar displays a **Partially Updated** button. Clicking it refreshes the table and chart so that they display only partially updated design points. The following figure shows all possible filter buttons.



To display all design points, click **All**.

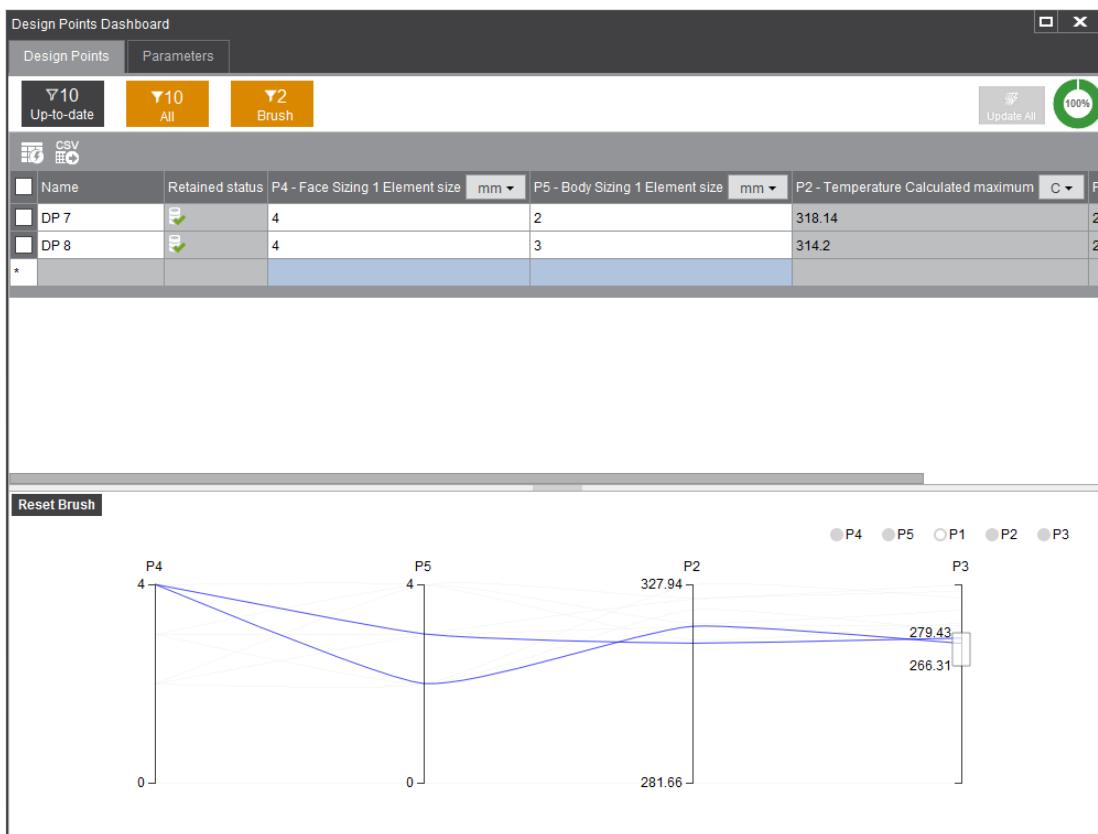
- To hide uninteresting parameters, in the upper right corner of the chart, click the control beside the parameter name to clear it.

In the following figure, the control beside P1 is cleared. Consequently, P1 is hidden.



- To filter by parameter values, along the Y axis for the parameter, click and drag to draw a box around the desired value range.

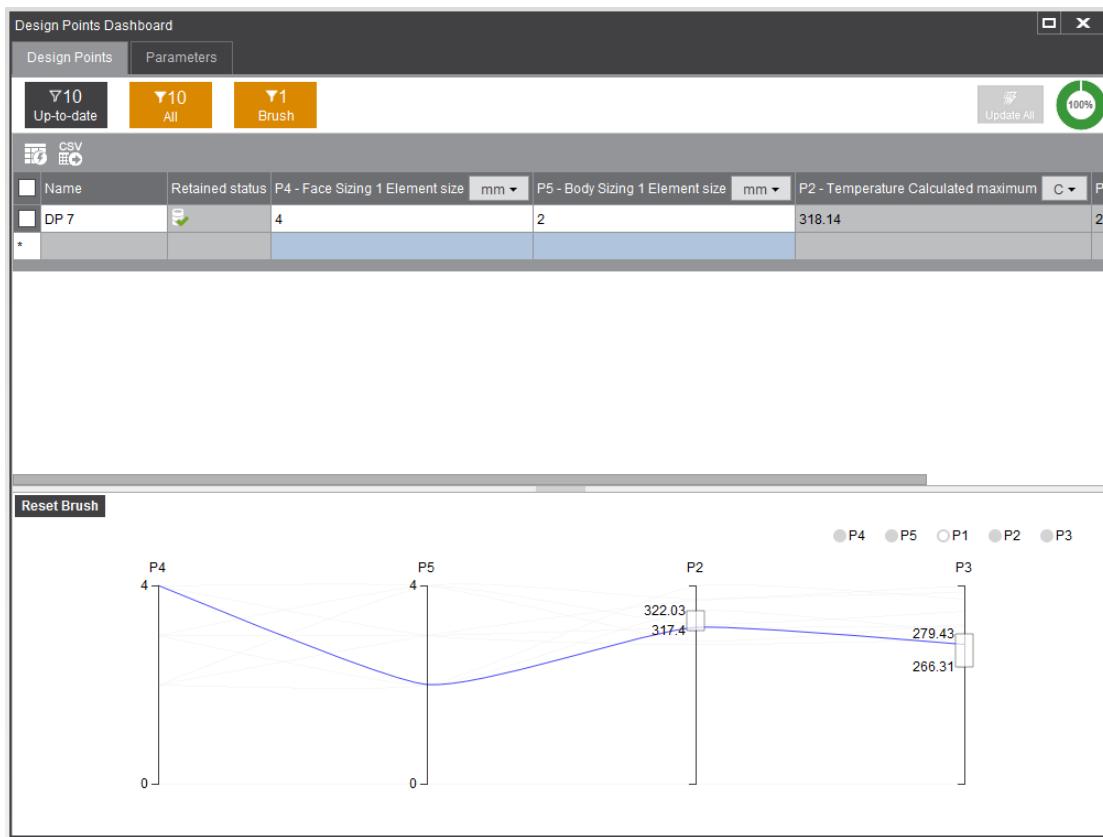
In the following figure, a box is drawn around Y-axis values for P3.



The design points that have values outside of the desired range for this parameter are hidden in both the chart and table. The limits of the range are displayed around the parameter-range box, which is known as a *brush*. You can resize the brush and slide it up and down to adjust the filtering at any time. In the toolbar, the **Brush** button indicates that two design points have passed the brush filtering.

##### 5. Optionally, repeat this operation for a second parameter.

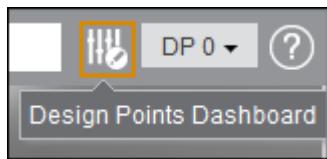
In the following figure, a box is also drawn around Y-axis values for P2. In the toolbar, the **Brush** button indicates that one design point has passed the brush filtering.



To toggle between the filtered results and the full results, in the toolbar, click the **Brush** button. To remove a single brush, click the axis on which the brush is drawn. To remove all brushes and restore the original display, in the upper right corner of the graph, click **Reset Brush**.

## 7.5. Modifying Design Points

1. On the **Study** panel, click the **Design Points Dashboard** icon.



The **Design Points Dashboard** opens. The **Design Points** tab displays any existing design points, parameters, and parameter values in a table and chart.

2. Modify values for input parameters as required.

As you change input parameter values, the corresponding output parameters become out-of-date:

The design point and any retained data that exists also become out-of-date:

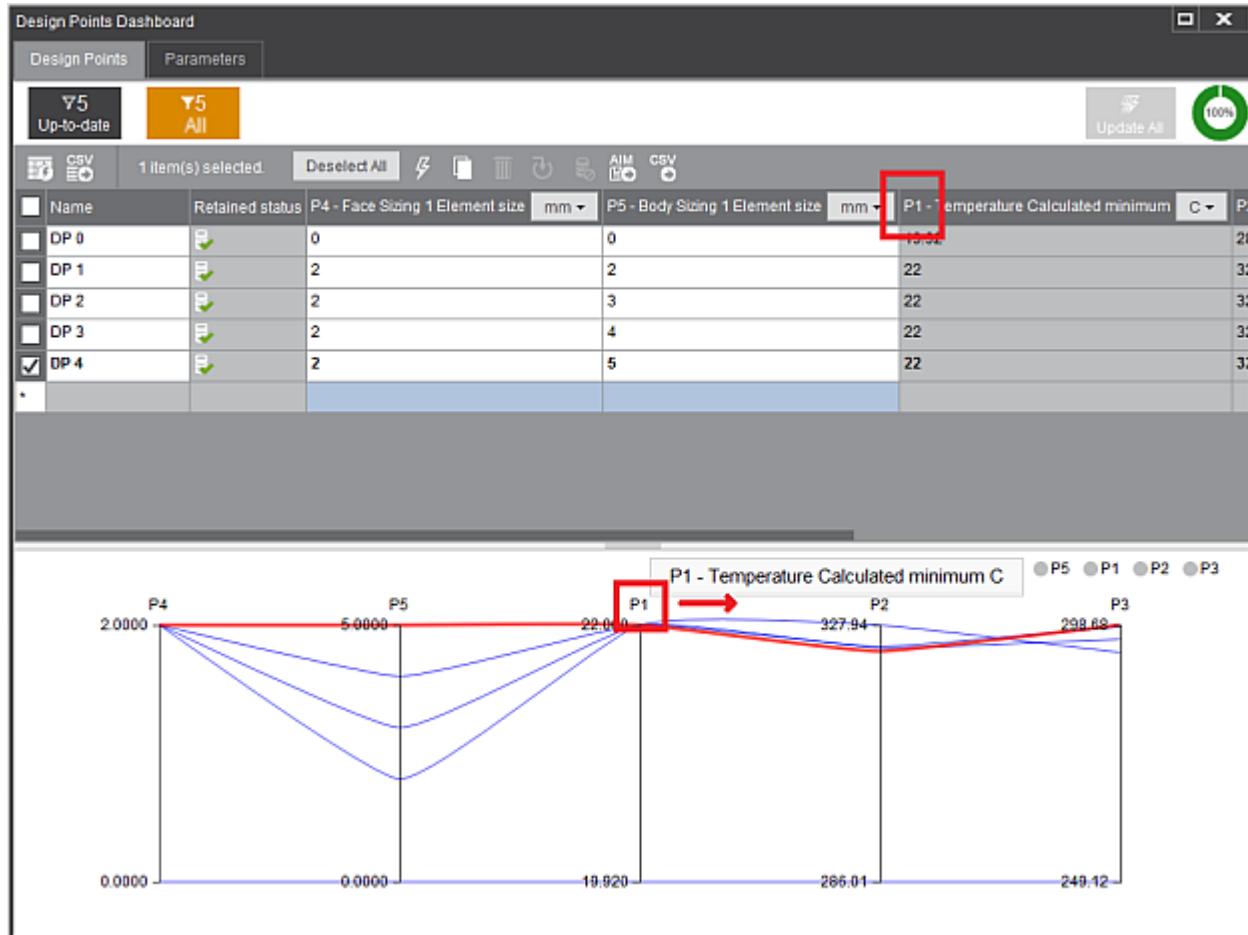
**Note:** It is possible for a design point to be up-to-date while its retained data is invalid, or vice versa. For more information, see [Retained Data Status](#) on page 613.

3. When finished, update the out-of-date design points. For more information, see [Updating Design Points](#).

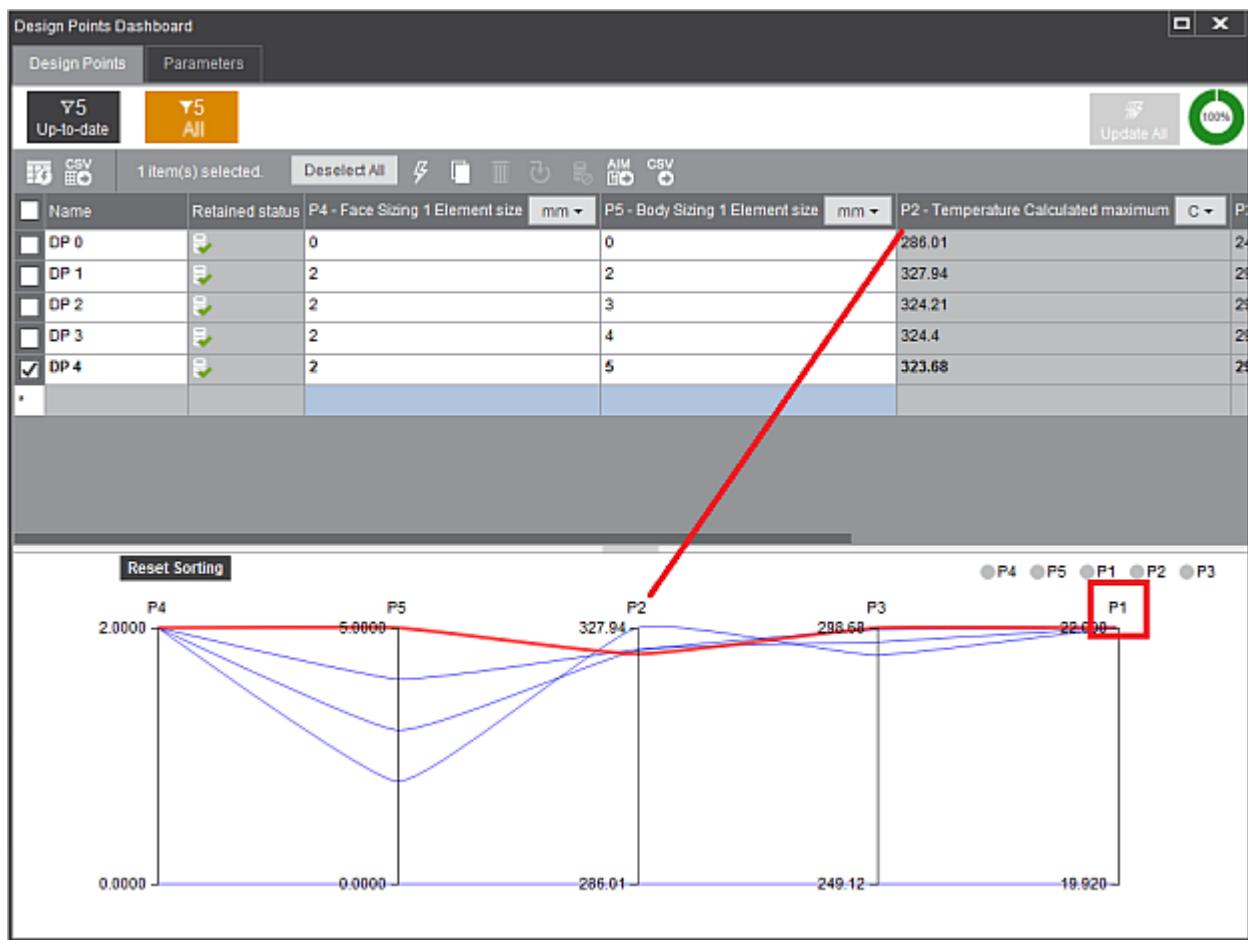
## 7.6. Moving Parameters in the Dashboard

You can change the order of the parameters in the **Design Points Dashboard**. For example, in the following figure, P1 has the same value in all design points. To move it to the far right of the dashboard:

- In the graph area, click the parameter ID and drag it to the right.



- Drop the parameter ID at the end of the graph.



The parameters are reordered in the graph and in the table view.

## 7.7. Design Point States

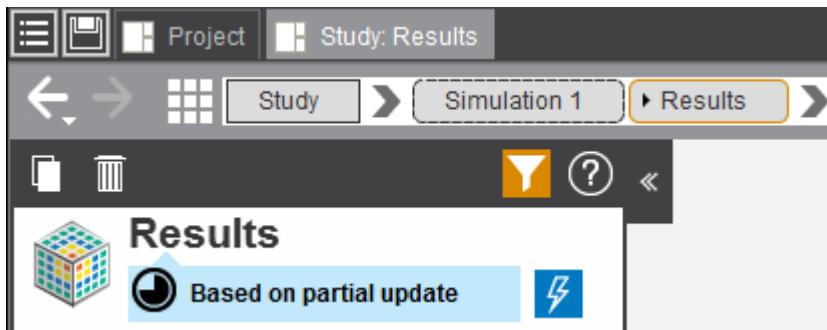
The state of an output parameter is indicated by an icon in the associated column of the design points table.

Changes elsewhere in the project can cause the state of some, but not all, output parameters to go out-of-date and require an update. Only the output parameters affected by the changes are shown as requiring an update. Output parameters that are not affected by the changes remain up-to-date.

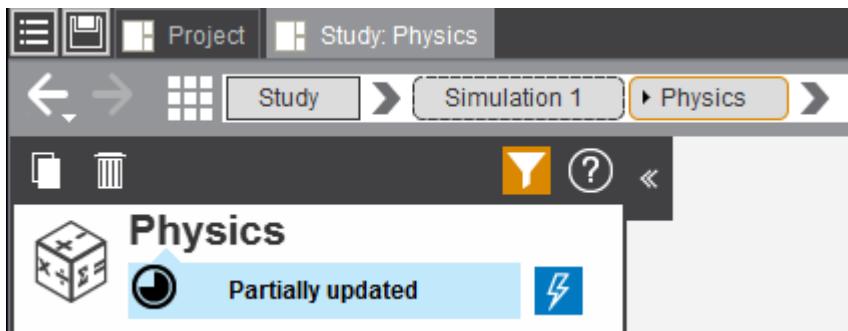
Icon	State	Description
No icon	Up-to-date	No icon displays when the output parameter is updated successfully.
	Out-of-date	The <b>Update Required</b> icon displays when the output parameter is not solved or requires an update because local data has changed.
	Failed	The <b>Update Failed, Update Required</b> icon displays when the update of the output parameter fails.

Icon	State	Description
	Partially updated	The <b>Partially Updated</b> icon displays when an output parameter is only partially updated because of an interruption to a project update or design point update.  <b>Note:</b> A black version of this icon  appears in the design points table for the <b>Parameter Set</b> bar and in status messages for AIM tasks based on partially updated design points.

In an AIM study, tasks based on partially updated design points display status messages. For example, the Results task displays **Based on partial update**.



The Physics task displays **Partially updated**.

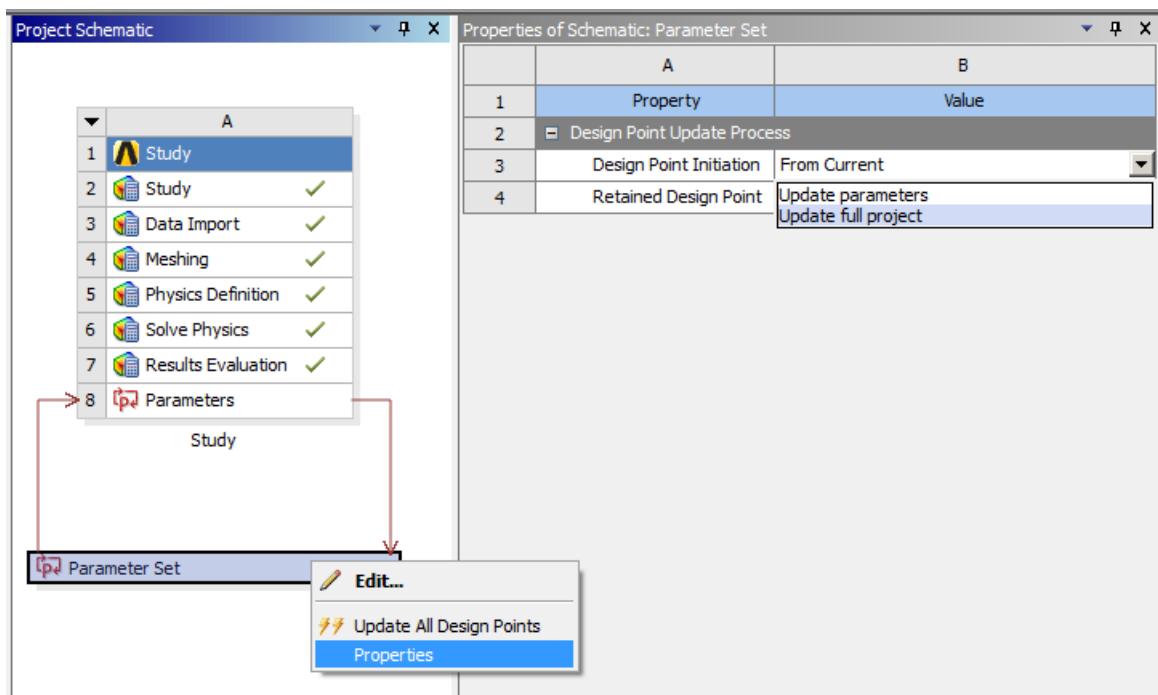


## 7.8. Design Point Updates

Output parameter values for a design point are calculated when you update the design point. [Updating design points](#) updates solution data only where output parameters have been defined.

In most cases, a design point update applies only to project components affected by parameter changes and their downstream components. Components without associated parameters or whose associated parameters did not change are not updated.

**Note:** In AIM, **Results** tasks that are not associated with output parameters are not updated and have a state of **Update Required** when the design point update completes. To ensure that all **Results** tasks are updated with a design point update, you can configure the **Retained Design Point** property for the **Parameter Set** bar to **Update full project**. To access this property, on the **Project** tab, right-click the **Parameter Set** bar and select **Properties**.



Non-parametric changes made to a project with up-to-date design points can cause all existing design points to go out-of-date. If changes invalidate your design points table, you must update the project, which can require significant time and computing resources. You should always save your project after updating all design points and before further modifying the project.

If you are updating multiple design points and the update for one design point fails, the failed design displays the **Update Failed, Update Required** icon. However, the update of the next design point begins immediately. Information about the failed design point update appears in the messages and in a failure summary dialog box.

### 7.8.1. Updating Design Points

In the design points table on the **Design Points** tab, you can select one or more design points for updating.

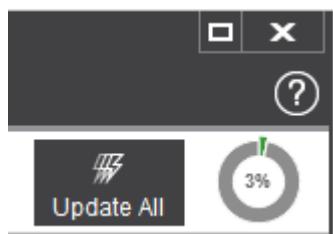
To update design points:

1. Click the check boxes to select the design points that you want to update.
2. Click the **Update selected** icon in the table toolbar.

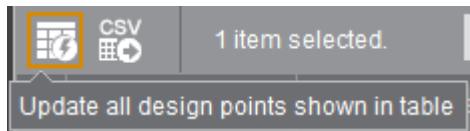


If you wanted to update all design points, you could do one of the following:

- Click the **Update All** button.



- Click the **Update all Design Points shown in table** icon.



If a previous update of a selected design point has been interrupted, the partially updated icon (⌚) may appear beside the values of the output parameters for the interrupted component. The next time that you select this design point for update, a dialog box opens. To continue, you must select one of the following buttons:

- Use Partially Updated:** Accept output parameters that are only partially updated as up-to-date, using the existing results to update design points.
- Update All.** Resume or restart the interrupted update, recalculating results for all output parameters that are partially updated and then updating all design points. The update resumes if the data point is retained and the analysis supports continuing the update. Otherwise, the update restarts. The dialog box notes that this can be a lengthy process.

To abandon the update, you would click the **Cancel** button.

You can track the progress of the update in the pie chart monitor. For more information, see [Monitoring Design Point Updates](#) on page 612.

### 7.8.2. Monitoring Design Point Updates

During a [design point update](#), the design points table, chart, filters, and pie chart monitor all refresh as the update progresses.

The pie chart monitor on the far right shows:

- Percentage of design points that have been updated
- Successful updates in green
- Failed updates in red
- Pending updates in light gray
- Partial updates in light blue

Hover over a portion of the pie chart to see the number of design points currently represented by that segment.

Updates of 20% of the table's 5 design points have been completed. Updates are pending for the remaining 80% of the design points.




---

## 7.9. Working with Design Point Update Data

When working with design points, you can retain and export data:

- [Retaining Design Point Data within the Project](#) on page 613
- [Exporting Design Points to New Projects](#) on page 614
- [Exporting Design Point Parameter Values to a CSV File](#) on page 615

### 7.9.1. Retaining Design Point Data within the Project

You can switch back and forth between multiple designs within the same project by setting different design points as current. To set a design point as current, it must have retained data.

By default, calculated data is always retained within the project for the current design point and for design points created via the **Design Points Dashboard**. Before updating design points, you can select additional non-current design points derived from parameterization and mark them [to retain data](#) or [not to retain data](#).

#### 7.9.1.1. Marking Design Points to Retain Data

To mark design points to retain their data:

1. On the **Design Points** tab, select the check boxes of the design points for which data is to be retained.
2. In the table toolbar, click the **Retain selected** icon.



An icon in the **Retained status** column displays, indicating both that the design point is marked for data retention and the status of its retained data. For more information, see [Retained Data Status](#) on page 613.

#### 7.9.1.2. Marking Design Points to Not Retain Data

To mark design points to not retain their data:

1. On the **Design Points** tab, select the check boxes of the design points for which data is not to be retained. Because data is always retained for the current design point, it is not be affected by this operation.
2. In the table toolbar, click the **Do not retain selected** icon.



The icon indicating retained data is removed from the **Retained status** column.

**Note:** After being clicked, the **Do not retain selected** icon becomes a **Retain selected** icon, which enables you to restore the retained state.

#### 7.9.1.3. Retained Data Status

You can verify that data has been retained by reviewing the **Retained status** column for each design point. The icon in this column indicates the availability, validity, and state of the retained data for the design point.

Different kinds of changes to the project have different impacts on retained data.

- Changes to the input parameter values for a design point cause the retained data to become out-of-date, but still valid.
- Non-parametric changes in the project cause the retained data for non-active design points to become invalid.

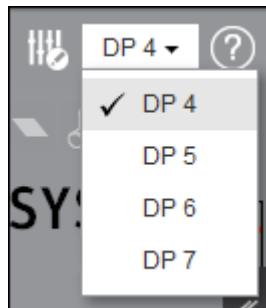
**Note:** The state of a design point and the status of its retained data do not necessarily match. It is possible for a design point to be up-to-date (all output parameters are up-to-date) while its retained data is invalid, or vice versa.

Icon	Status	Description
No icon	No retained data	No data is being retained for the design point.
	Up-to-date	The retained design point is successfully updated. Retained data exists and is up-to-date.
	Out-of-date	The existing retained data is out-of-date because input parameter values were modified. Update the design point to generate up-to-date retained data.
	Not available	Retained data either is not yet generated or has become invalid because of a non-parametric change in the project.

## 7.9.1.4. Setting a Design Point as Current

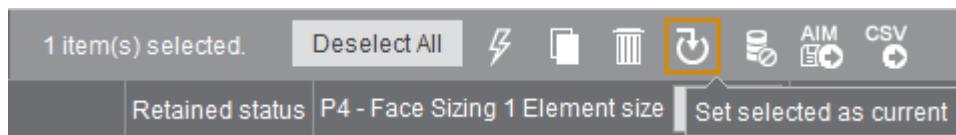
Only a design point with retained data can be set as current. You can set only one design point as current at a time.

On the **Study** panel, you can use the design point selection menu to set a design point as current. The list includes all design points with retained data.



In the **Design Points Dashboard**, you can also set a design point as current from the design points table.

1. Select the check box for the design point to set as current.
2. Click the **Set selected as current** icon in the table toolbar.



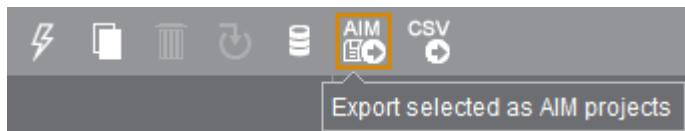
The design point becomes the current design point for the project and is displayed in bold font. In the chart, the current design point is displayed as a red line. Other design points display as blue lines. Setting a design point as current does not automatically update the design point.

## 7.9.2. Exporting Design Points to New Projects

Your project must be saved before you can export any design points to separate projects.

To export design points to separate projects:

1. On the **Design Points** tab, use the check boxes or buttons to select the design points that you want to export.
2. In the table toolbar, click the **Export selected as AIM projects** icon.



Fully independent projects named ***projectname\_dpn.wbpj***, are created as siblings in the same directory as the main project. If the design point has already been exported, exporting it again overwrites the existing exported files for this project. The content of the new project depends on the state of the design point's retained data at the time of the export.

- If valid retained data is available for an exported design point (retained data is either up-to-date or out-of-date), the retained data is used to create the new project.
- If no valid retained data is available for an exported design point (the design point is either not marked to retain data or is marked but retained data is not yet generated or has became invalid), the project exported is an out-of-date project based on the current design point with the parameter values for the design point applied. When you open the exported project, it will be out-of-date and require an update to solve for the exported design point.

If the design point fails to export, the files for that design point remain in the ***project\_files\dpn*** directory until you delete them manually or attempt to update the design point again.

### 7.9.3. Exporting Design Point Parameter Values to a CSV File

You can export design point parameter values to a comma-separated values (CSV) file that can be used by other software tools for further processing.

To export selected design points:

1. On the **Design Points** tab, select the check boxes of the design points that you want to export.
2. In the table toolbar, click the **Export parameter values to a CSV file** icon:



The parameter values of all existing design points are exported to a CSV file. The units, displayed in the units dropdown menu for each parameter in the header of the design points table, are not taken into account when the design point parameter values are exported to a CSV file. The values are always exported in units as defined in AIM.

**Note:** If the selected design point is out-of-date, its previous output values are not exported. The design point definition is exported to the CSV file, but the output cells in the file are empty.

The CSV file is created in an "[extended](#)" file format. For more information, see the next topic.

#### 7.9.3.1. Extended CSV File Format

Comma-separated values (CSV) files enable the transfer of data between software tools. In the standard CSV file format:

- Values are separated by commas.
- The optional header line indicates the name of each column.
- Each line is an independent record made of fields separated by commas.
- The format is not dependent on the locale, which means that the real number that is rendered as 12.345 in the US is written as "12.345" regardless of the regional settings of the computer.

AIM supports an extended CSV file format, which has the following characteristics:

- If a line starts with a "#" character, it is considered to be a comment line instead of a header or data line and is ignored.
- The header line is mandatory. It is the line where each parameter is identified by its ID (P1, P2, ..., Pn) to describe each column. The IDs of the parameters in the header line match the IDs of the parameters in the project.
- The first column is used to indicate a name for each row.
- A file can contain several blocks of data, with the beginning of each block being determined by a new header line.

```
# 10/1/2015 10:38:01 AM
# The parameters defined in the project are:
# P1 - WB_B [mm], P2 - WB_D [mm], P3 - WB_L [mm], P4 - WB_P [N],
# P5 - WB_E [MPa], P10 - WB_SIG [MPa], P8 - WB_DIS [mm], P9 - WB_BUCK [N]
#
# The following header line defines the name of the columns
# by reference to the parameters.
Name, P1, P2, P3, P4, P5, P10, P8, P9
DP 0, 2, 5, 100, 1000, 200000, 12000, 80, 1028.91145833333
DP 1, 3, 5, 150, 1000, 200000, ,
DP 2, 5, 5, 20, 1000, 200000, ,
```

---

## 7.10. Performing What-If Studies

Using the **Design Points Dashboard**, you can [perform what-if studies](#) to explore and evaluate design alternatives. A what-if study can help you to answer questions about your design, such as "How do variations on the load applied to this surface affect my stress and deformation outputs?"

By using a limited set of manually defined design points, you can use chart and results data to improve your understanding of how design variables influence the performance of your product.

### 7.10.1. Steps for Performing a What-If Study

In general, the steps for performing a what-if study are:

1. Manually create design points, varying the values of the input parameters that you want to explore.
2. Update the design points.
3. On the parameters parallel chart and design points table, review the results of your parametric variations and identify a design point of interest.
4. Set the design point as current.
5. Review the changes to the design.
6. Repeat steps 4 and 5 to review additional design alternatives.

Examples of performing what-if processes follow:

- [Performing a Simple What-if Study](#) on page 616
- [Performing a Large What-if Study](#) on page 618
- [Preparing to Troubleshoot a Failing Design Point Update](#) on page 620

### 7.10.2. Performing a Simple What-if Study

Begin with a study that is fully defined and successfully updated in AIM.

You will parameterize several dimensions and analyze how changing their sizes affects the performance of the model.

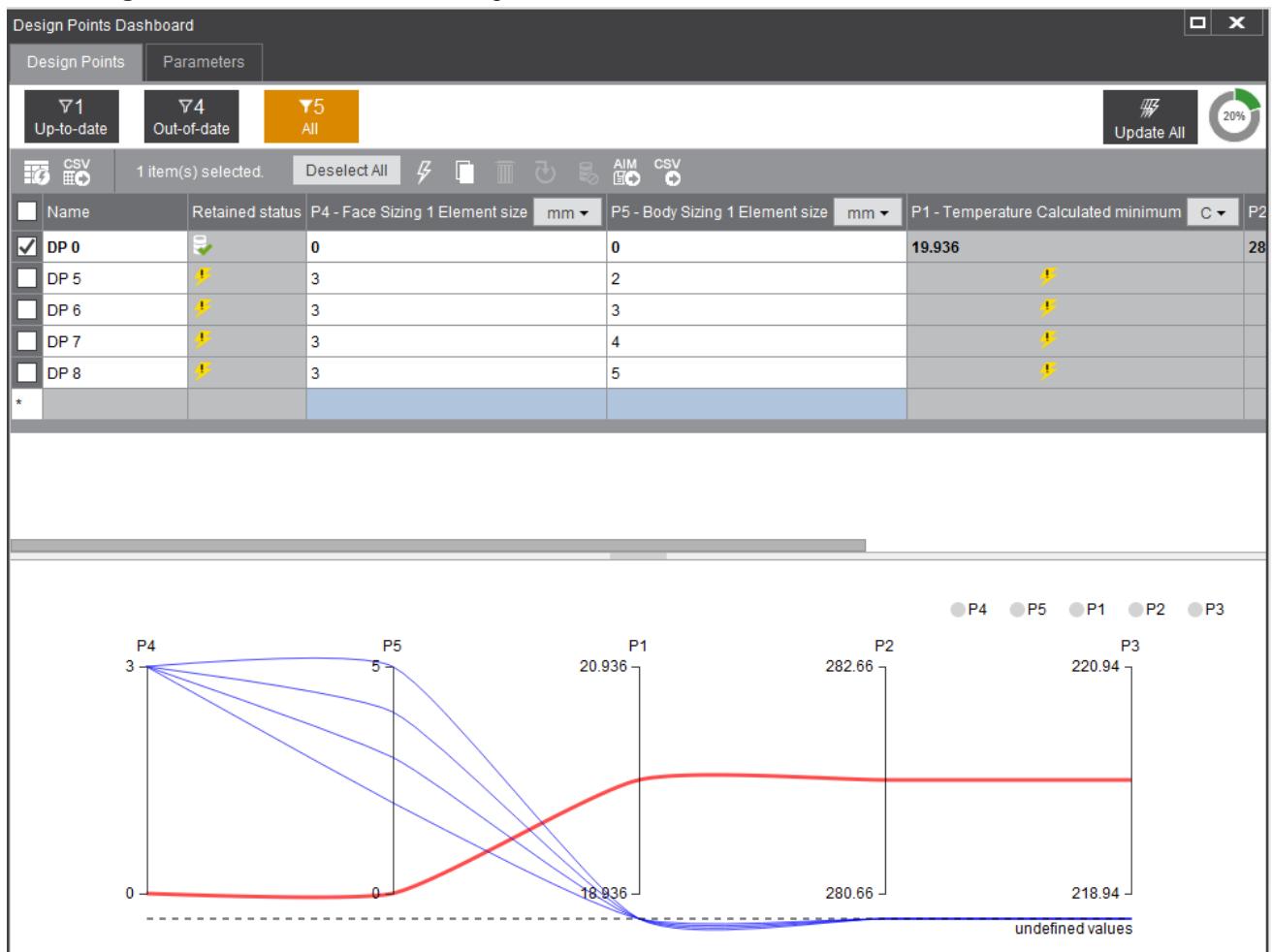
1. [Create input and output parameters.](#)

- Click the **Design Points Dashboard** icon.



The **Design Points Dashboard** opens. The **Design Points** tab displays the initial values of the parameters for DP0 in the table and parallel chart.

- In the table, create four additional design points that increase the value of two parameters of interest. Your **Design Points** tab will look something like this:



Notice that:

- The table contains five design points. The four new design points are out-of-date.
  - On update, the data from the new design points are retained by default because you created them with the **Design Points Dashboard**.
  - The parallel chart provides the graphical representation of the same data. All design points are visible. Output parameter values are undefined for the newly created design points.
  - Filter buttons show correct counts for up-to-date (1), out-of-date (4), and all (5) design points.
  - The monitor pie chart reflect the same state: 20% of the design points are updated.
- Update all design points. You can monitor the pie chart, design point filters, table of design points, and parallel chart refresh while the update progresses.

5. In the parallel chart, check the variation range obtained for the output parameters by observing the minimum and maximum of the corresponding axis. Note how the design points are distributed through the variation range of the outputs. You can mouse over design points in the chart to see corresponding values.
6. Choose the most interesting design point for your study and [set it as current](#).

When the data is loaded:

- In the table, the chosen design point becomes the current design point.
- In the chart, the chosen design point's values display as the red line.
- In the graphics view, the model is refreshed.

7. In the table header row, click the link for the studied output to open the object that owns the parameterized property. Close the **Design Points Dashboard** and continue postprocessing.

You saw how changes to an input parameter affected an output parameter and you were quickly able to choose the best design.

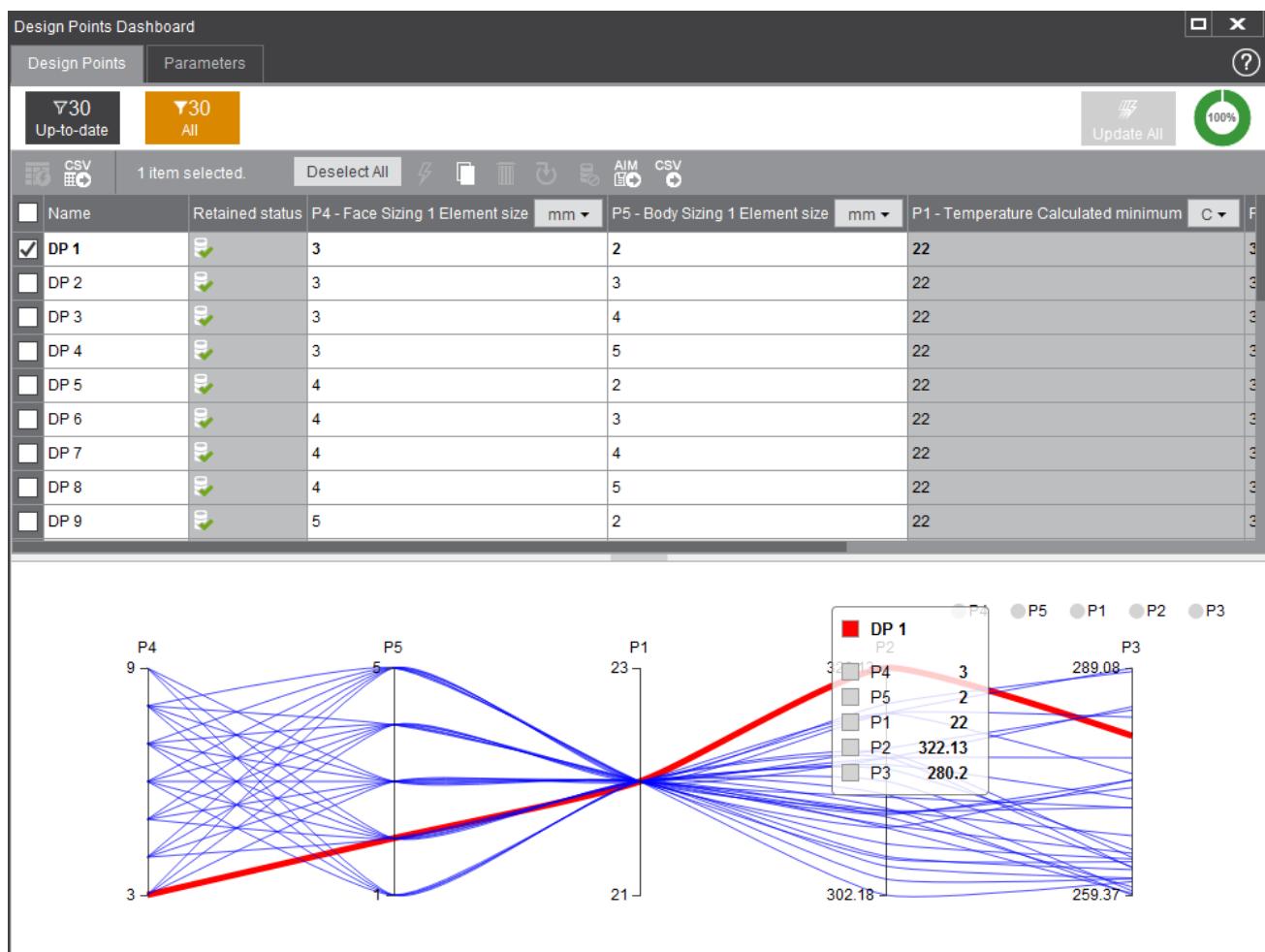
### 7.10.3. Performing a Large What-if Study

Consider an example in which you are starting with a study that has 30 design points and want to explore how variations of a few input parameters impact key results of the study. Your goal is to minimize an output parameter and keep another output parameter below a specified threshold value. The parallel chart and the table are used together to identify the best feasible designs and to verify the importance of input parameters.

1. Click the **Design Points Dashboard** icon.



The **Design Points Dashboard** opens. The **Design Points** table displays all design points, whatever their state. Because of the number of design points and parameters, only a part of the data is visible in the table.



2. In the parallel chart, click the red line representing the current design point.  
Notice that the current design point is checked and action buttons appear in the toolbar.
3. Check the axes of the parallel chart for two parameters to discover the obtained minimum and maximum values.
4. On the axis for one of those parameters, [use the mouse to filter out all values that are greater than a given threshold value.](#)  
Notice that:
  - All design points corresponding to the hidden values disappear from the chart.
  - The table is also filtered so that it contains the same design points as the chart.
5. Filter on another axis by decreasing the upper bound to keep about 10 design points.  
Again, all design points filtered out disappear from the chart and the table.
6. On the chart, drag one of the parameters to have the two parameters of interest displayed side by side.  
The order of parameters changes in the table as well.
7. Click the highlighted line to select the optimum design.  
The design point is selected in the table.
8. From the table, unselect the current design point.

The **Set as Current** button becomes available.

9. In the table's **Name** column, type over the name of the selected design point to rename it **Proposed Design**.
10. Set **Proposed Design** as current.

In the design point selection menu, the current design point is **Proposed Design**

11. Navigate to the **Result** object that owns the parameter and close the dashboard.

You saw how easy it was to minimize one output parameter and keep another output parameter below a specified threshold value to quickly choose the best design.

#### 7.10.4. Preparing to Troubleshoot a Failing Design Point Update

Begin with a study that is fully defined and successfully updated in AIM.

You have successfully set up and updated a study and now want to investigate parametric variations. You define several design points (retained by default) and update them all. However, the first one (DP1) fails. You want to stop the update of design points to investigate the problem.

1. View the error indicators that show that DP1 failed to update.
  - In the table, the error state of the failed output parameters is shown.
  - In the table, the design point remains undefined.
  - The filter buttons and the monitoring chart refresh to indicate that there is 1 failed design point.
- Note that DP2 starts to update.
2. Click the stop button to immediately abort the updating of all design points.
  - The DP2 update stops, the study switches back to the current design point, and the operation ends.
  - A message indicates that the **Update All Design Points** operation has been aborted.
  - Because AIM has switched back to the current design point (which is up-to-date), the message view does not show any additional message related to the failure of DP1.
3. Set DP1, the failed design point, as the current design point.
  - The study switches to DP1.
  - Failing tasks show an error state.
  - Messages produced during the update of DP1 are visible.
4. Close the **Design Points Dashboard** and start your investigation by reading the error messages and looking for failed tasks.



# Chapter 8: Expressions and Other Operations

---

AIM includes the ability to utilize expressions and other operations in your simulations.

---

## 8.1. Expressions

In its basic form, an *expression* is a string of characters used to define some quantity of interest that could be used for some purpose in an analysis. These quantities can be constants, expressions that evaluate to constants, expressions that evaluate to single values, or expressions that evaluate to fields that vary over physics regions.

You can define expressions using:

- Dimensional constants
- Scalar field variables
- Intrinsic mathematical functions
- Integrated quantity functions
- Any dimensionally consistent mathematical combination of the above

Expressions can be used to define many real properties in AIM, including:

- Boundary Conditions for Structural and Thermal Physics
- Boundary and Initial Conditions for Fluids Physics
- Postprocessing quantities for Fluids and Structural Results

**Note:** Define expressions only once all the contents of the expression are available; avoid referencing an object that has yet to be created, or using field variables before they are made available by finishing your physics setup.

There may be cases where an object has a property that is set by an expression, but the expression's dependencies are not automatically re-evaluated. In such a case, you can simply change the expression to something else, and then change it back to the expression again; that will force a re-evaluation of that specific expression. Alternatively, you can change another property of the object to cause the expression to be re-evaluated and the state recalculated.

For example, consider a case where you have duplicated a **Results** task that has a calculated value result generated using the function calculator. If the result's **Location** references a selection set, the calculated value result in the duplicated **Results** task will have an expression that references the original task's selection set. To correct this, in the duplicated **Results** task change the **Location** of the calculated value result to something else, then switch it back to the duplicated selection set. This causes AIM to re-evaluate the expression.

### 8.1.1. Creating Expressions

Enter an expression in the form of an operand followed optionally by operators and additional operands. Names used in expressions (variable names, function names, and so on) are case-sensitive.

For example, If **Pressure1** and **Pressure2** are properties of existing objects in the simulation, you can set a property field to:

```
Pressure1 + 3 * Pressure2
```

AIM expression and mathematical function evaluation is based on the Python programming language ([www.python.org](http://www.python.org)). The AIM expression parser supports:

- Standard mathematics functions and operators. For example:
  - $\cos(b)$
  - $2 * \text{Average}(\text{Pressure}, \text{GetBoundary}("@Inlet1"), "area")$
  - $\text{Pressure} / (\pi * \text{radius}^2)$
- Units for quantities. Dimensional quantities are defined in units that can be a combination of one or more base units. For example:
  - $10 [\text{m}]$
- Standard arithmetic operators including constants and trigonometric functions, among others. [Mathematical Operators and Functions](#) on page 644 has the complete list of arithmetic operators.
- Conditional statements. For example:
  - $500 [\text{N}] \text{ if } \text{diameter} \geq 15 [\text{mm}] \text{ else } 250 [\text{N}]$
  - $2 \text{ if } \text{count} \geq 8 \text{ else } 2.25 \text{ if } \text{count} == 7 \text{ else } 2.5 \text{ if } \text{count} == 6 \text{ else } 3$

An expression should be entered in the form of an operand followed optionally by operators and additional operands. For example, you can set a property field to `Pressure1 + 3 * Pressure2`, given that **Pressure1** and **Pressure2** are properties of existing objects in the simulation. Names used in expressions (variable names, function names, and so on) are case sensitive.

When entering expressions:

- Use the list and decimal separators defined in your locale settings. For example, if a comma is defined as a decimal separator and a semicolon defined as a list separator, use these when you type in an expression.
- If the expression has units, you must type in the units rather than relying on auto-completion.
- Expressions that involve quantities must be dimensionally consistent. The + and - operators require that the two operands have compatible units. For example, you cannot add an Area parameter to a Length parameter (both units must be `Length` or both units must be `Area`). The \* and / operators do not have this limitation; they allow one operand to be a quantity with a unit and the other operand to be a dimensionless factor or they allow both operands to be quantities with units where the result is a different quantity type. For example, Length/Time results in a quantity with a `Velocity` unit.

You can assign a name to an expression or expression value so it can be easily referenced from elsewhere in the Study. For more information, see [Reusing Expressions and Values](#) on page 638.

## 8.1.2. Mathematical Constants

These are the mathematical constants that you can use in AIM expressions:

Table 8.1.2.1. Constants

Description	Symbol	Value
pi	pi	3.1415927
Base of natural logarithm	e	2.7182818

$2 * \text{pi}$

## 8.1.3. Variables

The real power of expressions is to be able to perform operations with calculated data from a solver or other service in the system. Typical examples include performing quantitative calculations on solved field data or using a value from a solution as an input to some physics setup (this could include non-solver services where data from a solver may be used in mesh generation, or mesh information might be used to control physics setup).

ANSYS variables provide a way to reference calculated data in a consistent manner across all services, with the flexibility to access a wide range of variables in different contexts.

## 8.1.3.1. Variable Tensor Types

Variables can be defined as one of three tensor types:

<b>Scalar</b>	the variable is defined as a scalar (magnitude); for example, <i>Temperature</i>
<b>Vector</b>	the variable is defined as a vector (magnitude and direction); for example, <i>Velocity</i>
<b>Tensor</b>	the variable is defined as a tensor (symmetric or non-symmetric); for example, <i>Stress</i>

Access to scalar variables for use in expressions only requires you to know the name of the variable. However, for vectors and tensors, the tensor type is important because you need to know how to access the directional components of the variable. Expressions can contain scalars, and components of vectors and tensors.

### 8.1.3.1.1. Vector Variables

To access the vector components of a variable, enter the variable name followed by a '.' (dot/period) and the vector component name.

To refer to the Cartesian components of the vector field variable *Velocity*:

```
Velocity.x
Velocity.y
Velocity.z
```

### 8.1.3.1.2. Tensor Variables

To access the tensor components of a variable, enter the variable name followed by a '.' (dot/period) and the tensor component name.

To refer to the Cartesian components of the tensor field variable *Stress*:

```
Stress.xx
Stress.yy
Stress.zz
Stress.xy
Stress.yz
Stress.zx
```

### 8.1.3.1.3. The Position Vector

The position vector **Position** is provided to allow the referencing of coordinates in space:

```
Position = Position(x,y,z)
```

In the same way as for other field vectors, the components can be accessed as:

```
Position.x
Position.y
Position.z
```

## 8.1.4. Units

Units for numerical quantities are declared within square brackets after any number (with or without a space between the brackets and the number). Expressions that contain field variables will also have implied units dependent on the variable(s) being used and the unit system employed. Units can be specified after any number in an expression, but cannot be specified after a character (since any character string entered should have implied units).

Units can be specified in any valid unit system.

Table 8.1.4.1. Example Quantities and Units

Quantity	Dimensionality	Example Units
Acceleration	Length Time <sup>-2</sup>	m s <sup>-2</sup> ft s <sup>-2</sup> in s <sup>-2</sup> um ms <sup>-2</sup>
Current	Current	A mA pA
Density	Mass Length <sup>-3</sup>	kg m <sup>-3</sup> g cm <sup>-3</sup> lb ft <sup>-3</sup> slug in <sup>-3</sup> slinch in <sup>-3</sup>
Electric Charge	Current Time	A s coulomb pA s
Energy	Mass Length <sup>2</sup> Time <sup>-2</sup>	J BTU erg lbf ft slug in <sup>2</sup> s <sup>-2</sup>
Force	Mass Length Time <sup>-2</sup>	dyne N pdl lbf slug in s <sup>-2</sup>

<b>Quantity</b>	<b>Dimensionality</b>	<b>Example Units</b>
Length	Length	m cm foot in mm micron ft um yard
Pressure	Mass Length <sup>-1</sup> Time <sup>-2</sup>	Pa MPa N m <sup>-2</sup> bar torr mm Hg psi psf atm dyne cm <sup>-2</sup>
Power	Mass Length <sup>2</sup> Time <sup>-3</sup>	W BTU s <sup>-1</sup> HP erg s <sup>-1</sup> lbf ft s <sup>-1</sup>
Temperature	Temperature	K C R F
Temperature Difference	Temperature	K C R F

Quantity	Dimensionality	Example Units
Temperature Variance	Temperature^2	K^2
		C^2
		R^2
		F^2

You can access unit systems via the **Units > Unit Systems** menu option. For more information, see in the *Workbench User's Guide*.

You can also create custom unit systems by duplicating and editing predefined unit systems. For more information, see the "Custom Unit Systems" section of in the *Workbench User's Guide*.

## 8.1.4.1. Units in Expressions

Expressions that involve quantities must be dimensionally consistent. The + and – operators require that the two operands have compatible units. For example, you cannot add an Area parameter to a Length parameter; both units must be Length or both units must be Area. However, you can add or subtract two dissimilar units that are of the same dimensionality: 1 [m] + 1 [ft] is a valid expression because both units are Length.

The \* and / operators do not have this limitation. They allow one operand to be a quantity with a unit and the other operand to be a dimensionless factor. Or, they allow both operands to be quantities with units where the result is a different quantity type. For example, Length/Time results in a quantity with a Velocity unit.

The resulting units for arguments to a trigonometric function may be dimensionless or resolve to an angle.

- If the argument is a number or dimensionless, the value is interpreted as radians (in other words, 1 [] = 1 [rad]; a radian is a dimensionless quantity equal to unity (as defined by ISO 80000-1:2009, *Quantities and units -- Part 1: General*).
- If the argument is a quantity, the units must resolve to type Angle (for example, 1 [rev] = 360 [deg] = pi \* 2 [rad])

You can include units, assuming the unit makes sense in context of the expression. For example, P2+3 [mm] \*P3 is valid if mm is a valid unit expression for P2 and P3 (for example, if P2 has units of torque and P3 has units of force, or if P2 is in units of area and P3 is in units of length).

You can apply a unit to a whole expression by multiplying it by a dimensional constant. For example:

```
sin(0.1) * 1 [N]
```

Note that sin(0.1) [N] does not work because the syntax is ambiguous in Python.

The project unit system is used to evaluate the expressions. For temperatures, absolute temperature values are used in expression evaluation. All quantity values in an expression are converted to the project unit system.

## 8.1.4.2. Unit Multipliers

The general units syntax in ANSYS Workbench is defined as [multiplier|unit|^power], where multiplier is a multiplying quantity or its abbreviation (for example, mega (M), pico (P), and so on), unit is the unit string or abbreviation (for example, gram (g), pound (lb) foot (ft), meter (m), and so on), and power is the power to which the unit is raised.

Table 8.1.4.2.1. Unit Multipliers

Multiplier Name	Multiplier Value	Multiplier Abbreviation
exa	$10^{18}$	E
peta	$10^{15}$	P
tera	$10^{12}$	T
giga	$10^9$	G
mega	$10^6$	M
kilo	$10^3$	k
hecto	$10^2$	h
deca	$10^1$	da
deci	$10^{-1}$	d
centi	$10^{-2}$	c
milli	$10^{-3}$	m
micro	$10^{-6}$	u
nano	$10^{-9}$	n
pico	$10^{-12}$	p
femto	$10^{-15}$	f
atto	$10^{-18}$	a

### 8.1.4.3. Unit Declaration Syntax

When typing units in an expression, the units must be enclosed by square braces [ . . . ]. You will usually not see the braces when numbers are shown in a property field or when selecting units from a list of commonly used units. In general, units declarations must obey the following rules:

- A units string consists of one or more units quantities, each with an optional multiplier and optional power. Each separate units quantity is separated by one or more spaces.
- Abbreviations for multipliers and unit names are typically used, but full names are also supported.
- Powers are denoted by the ^ (caret) symbol. A power of 1 is assumed if no power is given. A negative power is typically used for unit division (for example, [kg m^-3] corresponds to kilograms per cubic meter).
- If you enter units that are inconsistent with the physical quantity being described, then an expression error will result.
- Units do not have to be given in terms of the fundamental units (mass, length, time, temperature, angle, and solid angle). For instance, Pa (Pascals) and J (Joules) are both acceptable as parts of unit strings.
- Units strings are case sensitive; for example, Kg and KG are both invalid as parts of units strings; kg is correct.

Examples of valid unit specifications are:

```
a * 2 [m s^-1] (where a is previously defined and has units of [m])
3 [m/s]
```

Note that in the above, the first expression has implied dimensions of [Length^2 Time^-1] and SI units of [m^2 s^-1] because a has the implied units of [m].

The following is not valid:

```
a = 5*Velocity [m/s]
```

## 8.1.4.4. Temperature Unit Conversions

When the specified project unit system uses the relative temperature units (C or F), the evaluation of expressions involving temperature, temperature differences, or temperature variances is a special case.

For the unit conversion of a specific temperature value,  $1 \text{ degC} = 274.15 \text{ K}$ . However, the unit conversion for a temperature interval (delta T) is  $1 \text{ degC} = 1 \text{ K}$ . The expression evaluator will take any temperature value and treat it as a specific temperature (not a temperature interval) by converting it to the absolute unit of the project unit system (either K or R). If the intent is to perform the evaluation in terms of temperature intervals, you need to start with temperatures in absolute units.

Similarly, any expressions to be evaluated in terms of temperatures or temperature differences will be converted to absolute units for calculation. In an expression with a temperature unit raised to a power other than 1 or a unit involving both temperatures and other units (e.g. a temperature gradient [ $\text{C}/\text{m}$ ]), the temperature will be assumed to be a temperature difference.

Table 8.1.4.4.1. Scenarios in which temperature-related units are converted before and during expression evaluation

Scenario	Example
Temperature unit conversion	$10 [\text{C}] * 2 = 566.3[\text{K}] = 293.15[\text{C}]$
Temperatures appearing as part of a mixed unit are converted as temperature intervals	$10 [\text{C}/\text{m}] * 50 [\text{m}] = 500[\text{K}]$
Temperatures raised to a power other than 1 are converted as temperature intervals	$10 [\text{C}^3] * 10 [\text{C}] = 2831.5[\text{K}^4]$

**Note:** If you create an expression that fits any of these scenarios, AIM will display an informational message indicating how the temperature units will be handled.

## 8.1.5. Single-Valued Expressions and Variables

The result of an expression can be a single value or an entire field of values. While many settings in a study could be set using a multi-valued expression (that is, one that changes over the specified location), some settings require that at a particular time/timestep or loadstep only a single-valued expression can be entered. Examples of this are the **Mass Flow** type boundary condition, **Angular Velocity/Rotation Speed** for rotating domains, and **Iteration** values.

These quantities could be set using expressions that:

- do not contain multi-valued field variables
- are the results of quantity functions
- are functions of single valued variables (see below)
- are combinations of the above

Some variables available for use in expressions only return a single value at each point in the calculation, and are not mesh or location dependent. An example of this type of variable is **Iteration**.

Other single valued variables may also be available depending on the physics of the case.

## 8.1.6. Solution-Dependent Expressions

Solution field dependent expressions can be used to set boundary conditions. Expressions can depend directly on the field variables as well as quantity functions (Average, Minimum, Maximum, etc.):

- Field variables are referenced using the standard naming syntax also used for postprocessing.

- Quantity functions are called using the syntax `<function_name>(<argument1>, <argument2>, ...)` where the arguments can be variables, locations, and other options depending on the specific function. Quantity functions return single values.

## Known Issues and Limitations

- Some properties such as **Mass Flow** or **Heat Flow** through a boundary are **single-valued**. An expression for these properties can depend on field values, but the result must be reduced to a single value—for example, by using a quantity function. An error message will appear if the expression does not evaluate to a single-valued result.
- Depending on what variables are used in your expression, the solver may not be able to evaluate the expression. This is due to the availability of data in the solver at run time. If this happens, then the physics task update will fail and you may see an error message in the fluids solver transcript.

### 8.1.7. Location References

You may need to [specify a location in an expression](#); for example, you might have an expression that calculates a quantity based on the value of a variable at a particular location.

A location can take the form of any object that resolves to a location in space. This could be a boundary, a physics region, a selection of geometric/topological entities in the graphical view (named or unnamed), or it could be an analytic or results-based object, such as a point, line, or isosurface.

You can also use location functions in expressions to specify the location of a group of objects such as those defined by a selection set. These functions return lists of entities that can be used as input to other functions. For more information, see [About Location Functions](#).

**Note:** If an expression in the **Variable** field of a **Contour** or **Vector** result contains a location, the location within the expression is ignored and the whole expression is evaluated on the location specified in the result object's **Location** property.

#### 8.1.7.1. Specifying Locations in Expressions

When creating an expression, you can use a function to retrieve the location of the object you want to use. The function names begin with `Get` and include the type of object you are retrieving (for example, a plane, a point, and so on). [Object Retrieval Functions](#) has a complete list of function names.

When specifying the object:

- You must enclose the object's name in quotation marks.
- The form of the object's name can be either:
  - The unique name (and type, for the `GetItem` function) of the object. You can find this by hovering over the icon to the left of the object's display name in its data panel.



- The display name of the object, either provided by default or entered by you, preceded by the @ symbol.

The following are examples of using object location references as function arguments.

- To get the area at a 2D location that has the display name "out", use:

```
Area(GetBoundary("@out"))
```

- To get the mass flow at a 2D location that has the display name "in1", use:

```
Sum(MassFlow, GetBoundary("@in1"), "Simple")
```

To reference lists of locations in expressions, enclose a comma-separated list in square brackets. For example:

```
Average(Pressure, [GetBoundary("@inlet1"), GetIsosurface("@Iso1")], "area")
```

## 8.1.7.2. Using Selection Sets in Expressions

A [selection set](#) is a named groups of objects in your model. You can use a selection set to specify locations in an expression. For example, in a fluid flow simulation of an exhaust manifold, you might specify all of the inlets as a selection set.

You use the function `GetSelectionSet` to get the location for the selection set. The syntax of the function can be either:

```
GetSelectionSet("<internal_name>")
```

or

```
GetSelectionSet("@<display_name>")
```

The selection set name must be enclosed in double quotes.

The following example shows the selection set `AllInlets` used as the location in a function:

```
Average(Pressure, GetSelectionSet("@AllInlets"), "area")
```

## 8.1.7.3. About Location Functions

You can use location functions in expressions to set the **Location** property of an object such as a boundary condition or a selection set. These functions return lists of entities that can be used as input to other functions. There are several different types of selection criteria functions:

- [Basic functions](#) retrieve the specified geometric entities from your model.
- [Set functions](#) add or remove entities from location lists.
- [Criteria functions](#) change the contents of an entity list based on entity criteria.

**Note:** If an expression in the **Variable** field of a **Contour** or **Vector** result contains a location, the location within the expression is ignored and the whole expression is evaluated on the location specified in the result's **Location** property.

### 8.1.7.3.1. Location Basic Functions

Basic functions return one particular type of entity. You can use these functions in other location functions as an input entity list in [other location functions](#).

Table 8.1.7.3.1.1. Basic Functions

Geometric Entity	Function Name	Returns
Bodies	AllBodies()	A list of all geometric body entities.
	AllBodiesIn(<Location>)	A list of all bodies to which <a href="#">Location</a> is scoped.

Geometric Entity	Function Name	Returns
Faces	AllFaces()	A list of all geometric surface entities.
	DefaultWalls()	All of the unassigned faces of the model in a fluid flow. Thus if you assign all of the non-Wall fluid flow condition types first, the faces that remain are the default location for the Wall boundary's <b>Location</b> field.
	DefaultConvectingSurfaces()	All surfaces except those to which another solid thermal boundary condition is applied, with these exceptions: For solid thermal simulations without fluid regions, use of DefaultConvectingSurfaces() in convection boundary conditions will not exclude surfaces to which radiation is applied, and vice versa. Otherwise, you can use it to define only one solid thermal boundary condition in the simulation.
Edges	AllEdges()	A list of all geometric edge entities.
Vertices	AllVertices()	A list of all geometric vertex entities.
All	Everywhere()	All internally generated nodes and elements participating in Structural simulations.

- AllBodies() returns a list of all bodies based on the current task's input model.
- AllBodiesIn(SelectionSet1) returns a list of all of the bodies in the specified selection set.

### 8.1.7.3.2. Location Set Operation Functions

The table below lists location functions that perform set operations on the input location lists.

Table 8.1.7.3.2.1. Functions that Perform Set Operations

Function Name	Returns...
Add (<LocationA>, <LocationB>)	The combined entities from <i>LocationA</i> and <i>LocationB</i> into a single unique entity list. Items that are in both <i>LocationA</i> and <i>LocationB</i> are listed only once in the final result.
Intersect (<LocationA>, <LocationB>)	The entities that are in both <i>LocationA</i> and <i>LocationB</i> .
Invert (<LocationA>)	All of the entities except for the entities contained in <i>LocationA</i> . This function respects the type(s) of entities contained in <i>LocationA</i> : <ul style="list-style-type: none"> <li>• If <i>LocationA</i> is a list of a portion of the vertices, the inverted list contains only vertices that are not in <i>LocationA</i>.</li> <li>• If <i>LocationA</i> is a list of vertices and edges, the inverted list contains all the vertices and edges not contained in <i>LocationA</i>.</li> </ul>

Function Name	Returns...
Subtract (<LocationA>, <LocationB>)	Returns the entities in <i>LocationA</i> that are not also in <i>LocationB</i> .

For information about specifying locations, see [Object Retrieval Functions](#) on page 648 and [Using Selection Sets in Expressions](#) on page 631.

To create list of all faces and edges:

```
Add (AllFaces(), AllEdges())
```

To find all entities that are not in the selection set "SelectionSet 1":

```
Invert (GetSelectionSet ("@SelectionSet 1"))
```

### 8.1.7.3.3. Location Criteria Functions

Each of the basic entity types (bodies, faces, edges, and vertices) from [Location Basic Functions](#) on page 631 and [Location Set Operation Functions](#) on page 632 have attributes that are available for use within location criteria functions.

Below is a list of attributes that are available for each entity type. When using these attributes and values in functions, case does not matter. See [Units](#) on page 625 for more information on how to write different quantity types.

Entity Type	Attribute	Value
Bodies	volume	Quantity
	type	Solid Surface Line
	x	Quantity
	y	Quantity
	z	Quantity
Faces	area	Quantity
	type	Plane Cylinder Cone Torus Sphere Spline
	radius	Quantity
	x	Quantity
	y	Quantity
	z	Quantity

Entity Type	Attribute	Value
Edges	length	Quantity
	type	Arc <sup>1</sup> Line Circle <sup>2</sup> Spline
	radius	Quantity
	x	Quantity
	y	Quantity
	z	Quantity
Vertices	x	Quantity
	y	Quantity
	z	Quantity

The following location criteria functions return lists of entities based on the criteria applied to the input entity list.

Table 8.1.7.3.3.1. Criteria Functions

Function Definition	Description
Convert (<Location>, "<ConvertToType>", "<ConvertOption>")	Returns all the entities of the entity type ConvertToType based on the selection found in Location.  ConvertToType can be any of "bodies", "faces", "edges", or "vertices".  The ConvertOption parameter can be "all" or "any" where "all" will return only the entities that are part of the entities in Location.  See <a href="#">Convert Options "All" and "Any" on page 635</a> for more information.
Largest (<Location>, "<Attribute>")	Returns all of the entities contained in Location that have Attribute with the largest value.
Match (<Location>, "<Attribute>", "<Value>")	Returns all of the entities contained in the Location that have Attribute equal to Value.
NotMatch (<Location>, "<Attribute>", "<Value>")	Returns all of the entities contained in the Location that have Attribute not equal to Value.
Range (<Location>, "<Attribute>", "<Lower>", "<Upper>")	Returns all of the entities contained in the Location that have Attribute equal to or within the range from Lower to Upper.

<sup>1</sup> Arc returns partial circles.

<sup>2</sup> Circle returns edges that make a complete circle.

Function Definition	Description
Smallest (<Location>, "<Attribute>")	Returns all of the entities contained in the Location that have Attribute with the smallest value.
Threshold (<Location>, "<Attribute>", "<Operator>", "<Value>")	Returns all of the entities contained in the Location that have Attribute with a value that satisfies the Operator. Operator can be any of the following: " >", "<", ">=", "<="

Find all bodies where x is less than or equal to 1.25 cm:

```
Threshold(AllBodies(), "x", "<=", "1.25 [cm]")
```

Find all faces with the smallest area:

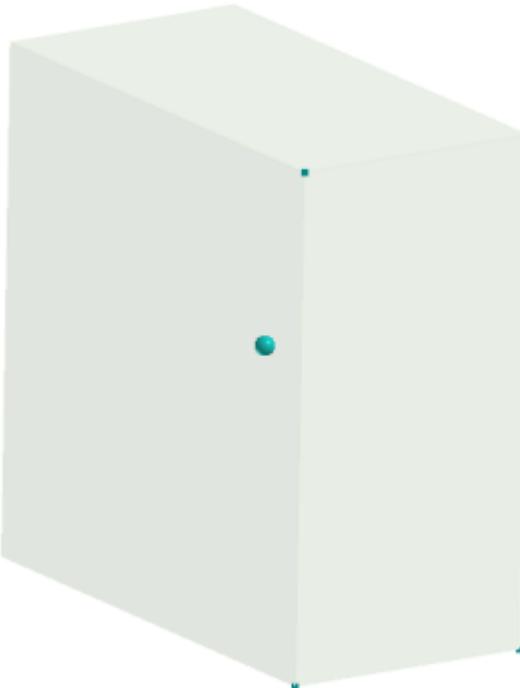
```
Smallest(AllFaces(), "area")
```

### 8.1.7.3.4. Convert Options "All" and "Any"

The Convert [location criteria function](#) enables you to convert "all" or "any" of the supplied entities to a new entity type.

To illustrate this, assume you have a rectangular body and you want to select all the vertices with the largest X coordinate and convert them to edges. The set of vertices that will be used during the conversion are shown below.

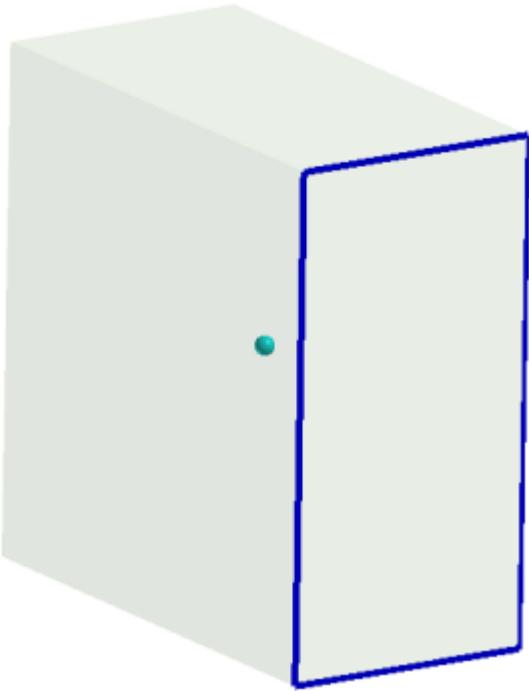
**Figure 8.1.7.3.4.1. Largest(AllVertices(),"x")**



When performing a conversion to edges, all of the edges of the model will be examined, noting the vertices that are associated with each edge.

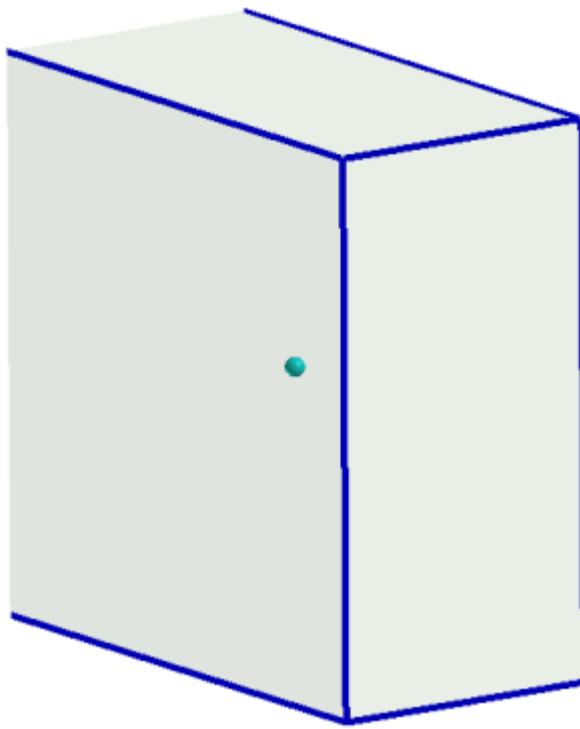
- If "all" is the selected option, the only edges included in the converted set are those for which all of the edges' vertices are included in the set of vertices input as the location. If an edge has only one or no vertex that is included in the input set of vertices, it will not be included.

**Figure 8.1.7.3.4.2. Convert(Largest(AllVertices(),"x"),"edges","all")**



- If "any" is the selected option, all edges that contain any vertex that is part of the set input as the location will be included.

**Figure 8.1.7.3.4.3. Convert(Largest(AllVertices(),"x"),"edges","any")**



### 8.1.8. Expression Composition

Expressions can be entered into any property field that requires direct user input. After you finish editing the property and press the Enter key, the expression is committed, and evaluated. The evaluated expression value is then displayed in the property box. If the expression cannot be evaluated, the expression text is shown instead. Where these properties are quantities, the units form part of the expression; for example, "10 [m s^-1]". These units are therefore used in the evaluation of the expression and are represented in the expression value.

The expression property boxes provide auto-completion of the expression components as they are entered.

## Auto-Completion of Parentheses

Parentheses are automatically added to the expression in order to assist in evaluation order when using binary operators (+-\* /). When you enter a binary operator, under circumstances where there is a left operand and it is appropriate to do so, parentheses will be added to the expression text. The added parentheses are initially ghosted, to signify a temporary state; you can decide to disregard these. Pressing the Delete key removes the parentheses from the expression text. Any other key entry or movement of the cursor will accept the parentheses and remove the ghosted effect.

## Auto-Completion of Units

Units are automatically added to the expression when entering a new expression. When you enter an expression that requires a quantity, there must be appropriate units declared at the end of the expression text. If you overwrite the entire expression text, then ghosted units are automatically appended to the end, in the form of " [m s^-1] ", or " \* 1 [m s^-1] " if the value is not a numeric literal.

The ghosted units that are appended to the end of the expression text are the default user units for the particular property. You may change the units manually. The ghosted highlight of the auto complete units signify that it is in a temporary state. You can accept these units by committing the expression in its current

state. If you press the Delete key while the cursor is to the left of the ghosted units, you remove the auto completed units and must enter them manually.

**Note:** In some instances the legend in the Graphics view will not display units associated with an expression being used with a variable. If this occurs, edit the expression and type the units in manually, rather than use auto-completion.

If you enter a unit string character (" [ "), then the ghosted completion disappears. A dropdown list will appear with an alphabetical list of units appropriate for this quantity type. Additionally if you enter a letter after the unit start, then not only will the units for this quantity type be suggested, but all units beginning with the partially entered string will be suggested.

The dropdown list will be scrollable if there are many suggestions available in the list. You can use the mouse to scroll through the list. Pressing the Esc key closes the dropdown list.

As you move your cursor down the list, the entry under the cursor is highlighted and becomes active. Click an active entry to select it. This will update the current expression input with the active suggestion, and close the dropdown list.

## Auto-Suggestion of Expression Components

As you enter a string in an expression, if you pause typing for a short time, a dropdown list is displayed containing a list of suggestions appropriate for any partial string prior to the current cursor position. The list of suggestions will be populated with those symbols available for the property being edited, including constants, variables, functions, parameters, named expressions, field variables, and field functions. Note that names of the expression components (variables, functions, and so on) are case sensitive.

The dropdown list will be scrollable if there are many suggestions available in the list. You can use the mouse to scroll through the list. Pressing the Esc key closes the dropdown list.

As you move your cursor down the list, the entry under the cursor is highlighted and becomes active. To select the active entry, click it or press the Enter key. This will update the current expression input with the active suggestion, and close the dropdown list. If the active suggestion selected is a function, then the cursor position is placed within the parentheses of the function ready for input again.

The suggestions listed in dropdown list and the expression properties have syntax highlighting applied to them as described in the following table:

Expression Component	Text Color
Numerals and mathematical operators	black
Variables	orange
Named Expressions	purple
Functions	blue
Constants	brown
Parameters and Named Values	pink

Suggested names appear in italic text if the item is scoped to the current analysis, or non-italic text if they have a general scope.

### 8.1.9. Reusing Expressions and Values

You can make an expression (or its value) reusable by giving it a name. You can then reference that named expression or named value when defining another expression.

## Named Value

If you assign a name to the value of a property, that named value becomes an independent object you can reference in the definition of another expression. For example, in a fluid flow simulation you need to create an inlet boundary condition. Select **Physics > Fluid Flow Conditions > Inlet**, then for the **Velocity magnitude** at **Inlet 1** you can enter:

5 [m s<sup>-1</sup>]

Using the fly-out menu you can create a named value (for example **v\_in1**). This named value can be used in a named expression.

To change **v\_in1**, go to the **Related Objects and Tasks > Related Objects > Named Expressions / Values > Named Expressions > v\_in1** panel.

## Named Expression

If you assign a name to an expression, that named expression becomes an independent object you can reference in the definition of another expression. For example, say that in your fluid flow simulation you need to create two inlet boundary conditions, and that the value of the first affects the value of the second.

1. Select **Physics > Fluid Flow Conditions > Inlet**, then for the **Velocity magnitude** at **Inlet 1** enter:

`v_in1 = 5 [m s-1]`

The **Velocity magnitude** displays as  $f = 5$  [m s<sup>-1</sup>]. To see the named value **v\_in1**, from any data panel select **Related Objects and Tasks > Related Objects > Named Expressions / Values > Named Expressions**.

2. Select **Physics > Fluid Flow Conditions > Inlet**, then for the **Velocity magnitude** at **Inlet 2** enter:

`v_in2 = 2 * v_in1`

To change **v\_in1**, go to the **Related Objects and Tasks > Related Objects > Named Expressions / Values > Named Expressions > v\_in1** panel.



[Video: Using Named Expressions and Named Values](#)

### 8.1.9.1. Named Expressions and Values

You can view named expression and named values in panels, their associated grid views, or at the property level.

## Named Expressions and Named Values in Panels

From the **Study** panel, you can view all of the named expressions and named values defined (though not necessarily used) in the study.

From the **Simulation**, task, or object panels, you can view all the named expressions and named values used in that part of the study.

If both named expressions and named values exist, the bottom of the panel shows the total number defined. If you expand **Named Expressions/Values**, you can see a breakdown of how many have been defined of each.

You can click the links to navigate to the associated grid.

# Named Expressions and Named Values Grids

You can view a grid containing named expressions, named values, or both. These grids can be accessed at the panel level, by clicking the associated links at the bottom.

- If accessed from the **Study** panel, the grid shows all named expressions and/or named valued defined (though not necessarily used) in the study.
- If accessed from the **Simulation**, task, or object panel, the grid shows all the named expressions and/or named values defined and in use for that part of the study.

The **Named Values** grid can also be accessed at the property level by clicking the icon next to a property that has been assigned a name.

The grids provide the following details and actions:

- For each named expression, you can view the name, the current expression, and the value of the expression. You can click **Name** to open the named expression object. When one or more named expressions are selected, toolbar icons enable you to duplicate or delete the selected expressions.
- For each named value, you can view the name, the property with which the named value is associated, and the current value of the property. You can click **Property** to navigate to the property being referenced. When one or more named values are selected, a toolbar icon enables you to delete the selected values.
- In the **Named Expressions/Values** grid, both named expressions and named values are shown. A dropdown menu enables you to display the named expressions and named expressions grouped separately, or all in a single list.

## Named Values at the Property Level

At the property level, a named values icon to the right of a property indicates that the property has been assigned a name. When you mouse over the named values icon, a tooltip shows the names assigned to the property.

### 8.1.9.2. Navigation to Named Expressions and Named Values

You can navigate to named expressions and named properties via the links on the associated grids.

From a named property, you can navigate to its named value.

#### 8.1.9.2.1. Navigating to a Named Expressions Object

To navigate to a named expressions object:

1. Open the **Named Expressions/Named Values** grid or the **Named Expressions** grid.
2. Click the **Name** for the named expression.

#### 8.1.9.2.2. Navigating to a Named Property

To navigate to a named property:

1. Open the **Named Expressions/Named Values** grid or the **Named Values** grid.
2. Click the **Property** for the named property.

## 8.1.9.2.3. Navigating to a Named Value

To navigate to a named property:

1. Navigate to the named property.
2. Click the named value icon to the right of the property.

## 8.1.9.3. Creation of Named Expressions and Named Values

Named values can be created at the property level via the flyout menu to the right of the field.

Named expressions can be defined:

- At the property level either via the flyout menu or by typing inline
- By using the **Add** and **Add Named Expressions** dropdown menus
- By using the “+” or **Duplicate** icon of a named expression object

### 8.1.9.3.1. Creating a Named Expression or Named Value via Menu at the Property Level

To create a named expression or named value via menu at the property level:

1. Open the property's flyout menu by clicking on the right arrow to the right of the field.
2. Click the **Create named expression/value** icon.
3. Select **Expression or Calculated Value**.
4. Enter the expression or value name.
5. Click **Create**.

### 8.1.9.3.2. Creating a Named Expression Inline at the Property Level

To create a named expression inline at the property level:

1. In a property field, enter the expression in the format **MyExpression=<expression>**. For example:  
**MyExpression= 10[C^3] \* 10[C]**
2. Press **Enter**.

### 8.1.9.3.3. Creating a Named Expression via Add Menus

The **Add** and **Add Named Expression** menus allow you to create a named expression either by adding a new named expression or duplicating an existing one.

- The **Add** menu displays next to the Named Expressions link at the bottom of the Study page and in the Named Expressions/Values grid when entries are organized by type.
- The **Add Named Expression** menu displays in the Named Expressions grid and the Named Expressions/Values grid.

To add a new named expression via either menu:

1. Click the menu.
2. Specify whether you want to create a new expression or duplicate an existing one.
  - To create new: Select **Named Expression**.
  - To duplicate existing: First select **Duplicate of**, and then select the named expression to be duplicated.
3. In the expression object that opens, you can click the name to edit it.
4. In the **Expression** field, enter the expression.

### 8.1.9.3.4. Naming Rules for Named Expressions and Named Values

The following rules apply to naming expressions and values:

- They must start with a letter (A-Z, or, a-z), or an underscore (\_), followed by zero or more letters, underscores, and digits (0-9).
- They cannot be the same as reserved keywords in Python.
- They cannot be the same as system variable names, system function names, or the names of solver variables.
- They cannot contain spaces.
- They are case-sensitive.

You will receive a warning if the name you enter is invalid.

### 8.1.9.4. Editing a Named Expression Object

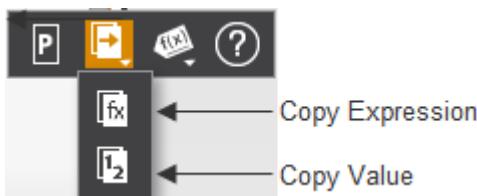
To edit a named expression object:

1. Navigate to the named expression object via its **Name** link in the **Named Expressions/Values** grid or the **Named Expressions** grid.
2. Click the name to enable editing.
3. Type changes into the **Expression** and **Description** fields.
4. The **Evaluated Value** field is non-editable. If evaluation is possible, it displays the current value of the expression.

### 8.1.9.5. Copying a Named Expression or its Value

To copy a named expression or its value at the property level:

1. Navigate to the property via its **Property** link in the **Named Expressions/Values** grid or the **Named Values** grid.
2. Open the flyout menu by clicking on the right arrow to the right of the property field.
3. Click the **Copy** button.
4. Specify whether you want to copy a named expression or the current value of the expression.



The name of the expression or the value of the expressions is now on the clipboard. You can use Ctrl+v to paste the name (and therefore the expression) or the value into another data field, or into any Windows application.

## 8.1.9.6. Duplication of Named Expressions

You can duplicate a named expression from its object, from **Named Expressions/Values** or the **Named Expressions** grid, or the **Study** panel.

The new named expression will have the default name **Copyof<namedexpression>**.

### 8.1.9.6.1. Duplicating a Named Expression from its Object

To duplicate a named expression from its object:

1. Open the Named Expressions object.
2. Click the **Duplicate** icon in the object toolbar.

### 8.1.9.6.2. Duplicating a Named Expression from a Grid

To duplicate a named expression from a grid:

1. Open the Named Expressions/Values grid or the Named Expressions grid.
2. Select one or more named expressions.
3. Click the **Duplicate** icon in the toolbar.

### 8.1.9.6.3. Duplicating a Named Expression from the Study Panel

To duplicate a named expression from the Study panel:

1. Open the Study panel.
2. Click the **Add** button to the right of Named Expressions.
3. Select **Duplicate of**.
4. Select the named expression to be duplicated.
5. Click **Create**.

## 8.1.9.7. Deleting a Named Expression or Named Value

To delete a named expression or named value:

1. Open the **Named Expressions/Values**, or the **Named Expressions** or the **Named Values** grid.
2. Select one or more named expressions or named values to be deleted.
3. Click the **Delete** icon in the toolbar.

## 8.1.9.8. Named Expressions and Named Values in the Construction of New Expressions

Once a named expression or named value has been created, you can use it to construct other expressions. As you type in a new expression into a property field, existing named expressions and named values become available as auto-completion options.

Prior to selection, auto-complete options for named expressions are displayed in purple text; auto-complete options named values are displayed in pink text.

For more information, see [Expression Composition](#) on page 637.

### 8.1.10. Mathematical Operators and Functions

## Built-In Mathematical Operators and Functions

The tables below list the mathematical operators and functions that can be used in expressions. Some of the functions may not be available depending on the task (**Physics**, **Results**, and so on), and physics region (Structural, Fluid Flow, and so on) in which they (or the location they reference) are being used. The syntax shown in the tables uses **x** and **y** to indicate operands, and **[a]** and **[b]** to indicate units for the operands.

Availability of the operators and functions is indicated in the tables using the following notation:

<b>UI</b>	Not physics-specific, available for all expressions
<b>FS</b>	Available for expressions relating to regions of Fluid Flow physics
<b>SS</b>	Available for expressions relating to regions of Structural, Thermal or Electric Conduction physics
<b>FP</b>	Available for expressions used when calculating results involving Fluid Flow physics
<b>SP</b>	Available for expressions used when calculating results involving Structural, Thermal or Electric Conduction physics
<b>MP</b>	Available for expressions used when calculating results involving Magnetics physics

See [Creating Expressions](#) on page 622 for examples.

Table 8.1.10.1. Basic Mathematical Operators

<b>Description</b>	<b>Availability</b>						<b>Syntax</b>	<b>X Units</b>	<b>Y Units</b>
	<b>UI</b>	<b>FS</b>	<b>SS</b>	<b>FP</b>	<b>SP</b>	<b>MP</b>			
Negation							-x [a]	any	
Addition							x [a] + y [b]	any	[a]
Subtraction							x [a] - y [b]	any	[a]
Multiplication							x [a] * y [b]	any	any
Division							x [a] / y [b]	any	any
Exponent							x [a] ** y	any	dimensionless

Table 8.1.10.2. Mathematical Functions

Description	Availability						Syntax	X Units	Y Units
	UI	FS	SS	FP	SP	MP			
Modulus					x % y			dimensionless	dimensionless
Floor					x[a] // y			any	[a]
Divide									
Integer					int(x)			dimensionless	
Nearest Integer					nint(x)			dimensionless	
Ceiling					ceil(x)			dimensionless	
Absolute Value					fabs(x[a])			any	
Absolute Value					abs(x[a])			any	
Floor					floor(x[a])			dimensionless	
$e^x$					exp(x)			dimensionless	
natural logarithm					log(x)			dimensionless	
Base 10 logarithm					log10(x)			dimensionless	
Power					pow(x[a], y)			any	dimensionless
Square root					sqrt(x[a])			any	
Arc cosine					acos(x) <sup>3</sup>			dimensionless	
Arc sine					asin(x) <sup>3</sup>			dimensionless	
Arc tangent					atan(x) <sup>3</sup>			dimensionless	
Arc tangent with quadrant					atan2(x[a], y[b]) <sup>3</sup>			any	[a]
Cosine					cos(x)			angle	
Sine					sin(x[a])			angle	
Tangent					tan(x[a])			angle	
Hyperbolic cosine					cosh(x)			dimensionless	
Hyperbolic sine					sinh(x)			dimensionless	

<sup>3</sup> The results of the inverse trigonometric functions on real numbers are dimensionless, and you must perform a mathematical operation on the result to add units if you need them (for example, multiply the result of the function by 1 [radian]).

Description	Availability						Syntax	X Units	Y Units
	UI	FS	SS	FP	SP	MP			
Hyperbolic tangent							tanh (x)		dimensionless
Minimum							min(x[a], y[a])	any	[a]
Maximum							max(x[a], y[a])	any	[a]
Conditional	<sup>4</sup>						x[a] if <log_expr> else y[b]	any	[a]

Table 8.1.10.3. Boolean Operators

Description	Availability						Syntax	X Units	Y Units
	UI	FS	SS	FP	SP	MP			
Not							not x		
Less Than or Equal to							x[a] <= y[b]	any	[a]
Less Than							x[a] < y[b]	any	[a]
Greater than or equal to							x[a] >= y[b]	any	[a]
Greater than							x[a] > y[b]	any	[a]
Not equal to							x[a] != y[b]	any	[a]
Equal to							x[a] == y[b]	any	[a]
AND							x and y		
OR							x or y		

## 8.1.11. Quantity Functions

*Quantity functions* return single values (with units) from specific locations. For example, you might get the average value of a variable at an inlet.

You can use quantity functions to define conditions, model parameters, sources, monitor points, and other aspects of the simulation or its results. The functions enable quantities evaluated on non-local regions to be applied elsewhere in the model.

Functions that have a weighting as an argument allow the other arguments to be weighted by another quantity. This is often required to get physically meaningful results from functions which operate across non-uniform grids and/or fields which vary in density, temperature, and so on.

The weight values have the following meaning:

Table 8.1.11.1. Function Weighting Values

<sup>4</sup> The Conditional function is supported for the following structural and thermal loads: Pressure; Moment; **Force** (when defined by **Force per unit area** and directional components); Heat Flux; Temperature; and Convection (on ambient temperature).

Weighting Value	Description
Simple	Arithmetic weighting (that is, a weighting factor of 1).
Length	Weight by the local length (for 1D objects).
Area	Weight by the local face area (for 2D objects).
Volume	Weight by the local volume (for 3D objects).
Mass	Weight by the local mass (for 3D objects).

**Note:** For functions used in structural analyses, only the Simple weighting type is allowed.

The table below lists the quantitative functions that can be used in expressions.

Some of the functions may not be available depending on the task (**Physics** or **Results**) and the physics region (Fluid Flow or Structural) in which they or the location they reference are used. The availability of the functions is indicated in the table by using the following notation:

<b>FS (Fluid Solve)</b>	Available for expressions relating to regions of Fluid Flow physics.
<b>FP (Fluid Postprocessing)</b>	Available for expressions used when calculating results involving Fluid Flow physics.
<b>SP (Structural Postprocessing)</b>	Available for expressions used when calculating results involving Structural, Thermal, or Electric-Conduction physics.

Table 8.1.11.2. Quantity Functions

Syntax	Returns	FS	SP	FP
Length(<locations>)	The length of <locations>			
Area(<locations>)	The area of <locations>			
Volume(<locations>)	The volume of <locations>			
Mass(<locations>)	The mass of <locations>			
Count(<locations>, "Node"   "Edge"   "Face"   "Element")	The number of mesh entities on <locations>			
CountIf(<logical_expr>, <locations>, "Node"   "Edge"   "Face"   "Element")	The number of mesh entities on <locations> if <logical_expr> is true			
Probe(<expr>, <locations>)	The value of <expr> at <locations>			
Minimum(<expr>, <locations>)	The minimum value of <expr> on <locations>			
Maximum(<expr>, <locations>)	The maximum value of <expr> on <locations>			
Average(<expr>, <locations>, "Simple"   "Length"   "Area"   "Volume"   "Mass")	The weighted average of <expr> over <locations>			
Sum(<expr>, <locations>, "Simple"   "Length"   "Area"   "Volume"   "Mass")	The weighted sum of <expr> over <locations>			

Syntax	Returns	FS FP	SP
<pre>SumIf(&lt;expr&gt;, &lt;logical_expr&gt;,&lt;locations&gt;, "Simple"   "Length"   "Area"   "Volume"   "Mass")  RMS(&lt;expr&gt;,&lt;locations&gt;, "Simple"   "Length"   "Area"   "Volume"   "Mass")</pre>	<p>The weighted sum of &lt;expr&gt; over &lt;locations&gt; if &lt;logical_expr&gt; is true</p> <p>The weighted root mean square of &lt;expr&gt; over &lt;locations&gt;</p>		

To set the named expression myExpr to be the area-weighted average of Pressure on the location defined by Inlet1, use:

```
myExpr = Average(Pressure,GetBoundary("@Inlet1"),"Area")
```

### 8.1.12. Object Retrieval Functions

The following functions are available to retrieve the specified objects. When specifying the object:

- You must enclose the object's name in quotation marks.
- The form of the object's name can be either:
  - The unique name (and type, for the GetItem function) of the object. You can find this by hovering over the icon to the left of the object's display name in its data panel.



- The display name of the object, either provided by default or entered by you, preceded by the @ symbol.

Table 8.1.12.1. Object Retrieval Functions

Function Syntax	Returns the Specified:
GetBoundary("<unique_name>")	Boundary condition
GetBoundary("@<display_name>")	
GetIsosurface("<unique_name>")	Isosurface
GetIsosurface("@<display_name>")	
GetItem("<type>","<unique_name>")	Item of the specified type
GetItem("<type>","@<display_name>")	
GetPlane("<unique_name>")	Plane
GetPlane("@<display_name>")	
GetPoint("<unique_name>")	Point
GetPoint("@<display_name>")	

Function Syntax	Returns the Specified:
GetPhysicsRegion("<unique_name>")	Physics region
GetPhysicsRegion("@<display_name>")	
GetSelectionSet("<unique_name>")	Selection set
GetSelectionSet("@<display_name>")	

For more information, see [Using Selection Sets in Expressions](#) on page 631.

### 8.1.13. Fluid Variables

The tables below lists the variable that can be accessed from the Fluid solver. These variables can be entered into data fields as part of an expression wherever access to the Fluid solver data is permitted. The variable names shown in the tables are the versions that are used in expressions. You may see more "user friendly" versions of the variable names used as labels or in dropdown lists (for example, Absolute Pressure vs. AbsolutePressure, or X Velocity vs. Velocity.x)

Variable availability can be model and model option dependent (for example, TurbulenceEddyFrequency is only available when using omega based models).

Table 8.1.13.1. Fluid Pressure Variables

Variable Name	Variable Type	Field Variable Definition
Pressure	Scalar	The fluid gauge pressure.
AbsolutePressure	Scalar	The pressure field plus the operating pressure. If buoyancy is active, see <a href="#">buoyancy and pressure</a> for details.
TotalPressure	Scalar	The gauge pressure obtained by bringing the fluid isentropically to rest, sometimes referred to as stagnation pressure.

Table 8.1.13.2. Fluid Material Property Variables

Variable Name	Variable Type	Field Variable Definition
Density	Scalar	The fluid mass per unit volume.
DynamicViscosity	Scalar	The resistance of a fluid to shear deformation.
ThermalConductivity	Scalar	The ability of a fluid to transfer heat by conduction.
SpecificHeatCapacity	Scalar	The thermodynamic property of specific heat for incompressible materials.
SpecificHeatCapacityCp	Scalar	The thermodynamic property of specific heat at constant pressure.

Table 8.1.13.3. Fluid Velocity Variables

Variable Name	Variable Type	Field Variable Definition
MassFlow	Scalar	The mass flow rate through the surface. The sign convention is such that if the surface is a boundary, a positive mass flow rate represents inflow to the flow region.

Variable Name	Variable Type	Field Variable Definition
qCriterion	Scalar	A variable derived from the velocity field to identify and visualize vortices.
StrainRate	Scalar	The second invariant of the strain rate tensor, which evaluates to the following:
		$\text{StrainRate} = \left[ 2 \left( \left( \frac{\partial U_x}{\partial x} \right)^2 + \left( \frac{\partial U_y}{\partial y} \right)^2 + \left( \frac{\partial U_z}{\partial z} \right)^2 \right) + \left( \frac{\partial U_x}{\partial y} + \frac{\partial U_y}{\partial x} \right)^2 + \left( \frac{\partial U_x}{\partial z} + \frac{\partial U_z}{\partial x} \right)^2 + \left( \frac{\partial U_y}{\partial z} + \frac{\partial U_z}{\partial y} \right)^2 \right]$
Velocity	Vector	The fluid velocity field (rate of change of fluid position vs. time at a fixed point, using an Eulerian formulation).
Velocity.x	Vector Component	The x, y, and z components of the velocity vector.
Velocity.y		
Velocity.z		
Velocity.mag	Vector Magnitude	The magnitude of the velocity vector (speed).
Vorticity	Vector	The curl of the velocity field, $\nabla \times U$ . This measures the rotation of a fluid element.
Vorticity.x	Vector Component	The x, y, and z components of the vorticity vector.
Vorticity.y		
Vorticity.z		
Vorticity.mag	Vector Magnitude	The magnitude of the vorticity vector.

Table 8.1.13.4. Fluid Thermal Variables

Variable Name	Variable Type	Field Variable Definition
Temperature	Scalar	The fluid temperature.
SpecificInternalEnergy	Scalar	The energy associated with the total (translational, vibrational, and rotational) motion of molecules.
TotalTemperature	Scalar	The temperature obtained by bringing the fluid isentropically to rest, sometimes referred to as stagnation temperature. For incompressible flows, the total temperature is equal to the static temperature.
SpecificEnthalpy	Scalar	A measure of the energy in a system, including internal energy $e$ plus the flow work $p/\rho$ required to displace the other fluid: $h = e + p/\rho$

Variable Name	Variable Type	Field Variable Definition
SpecificEntropy	Scalar	A measure of disorder in a system.
SpecificTotalEnthalpy	Scalar	The enthalpy obtained by bringing the fluid isentropically to rest, sometimes referred to as stagnation enthalpy: $h_{tot} = h + 1/2(U \cdot U)$ where $U$ is the velocity.

Table 8.1.13.5. Fluid Turbulence Variables

Variable Name	Variable Type	Field Variable Definition
EffectiveThermalConductivity	Scalar	The sum of the laminar and turbulent thermal conductivity.
EffectiveViscosity	Scalar	The sum of molecular and turbulence viscosity.
TurbulenceDissipationRate	Scalar	The dissipation rate of turbulence kinetic energy $\varepsilon$ .
TurbulenceEddyFrequency	Scalar	The turbulence eddy frequency $\omega$ is related to $k$ and $\varepsilon$ by $\omega = \varepsilon / (k C_\mu)$
TurbulenceIntensity	Scalar	The intensity of the turbulence kinetic energy relative to the mean kinetic energy.
TurbulenceKineticEnergy	Scalar	The kinetic energy associated with turbulence fluctuations, $k = 1/2 \bar{u}_i \bar{u}_i$
TurbulenceViscosity	Scalar	The viscosity due to turbulence mixing, as computed by the turbulence model.
TurbulenceViscosityRatio	Scalar	The ratio of turbulence viscosity to the molecular viscosity.
YPlus	Scalar	The dimensionless distance from a wall face to the adjacent element center, nondimensionalized using the viscosity and friction velocity $u^* : y^+ = \rho u^* y / \mu$
ModifiedTurbulenceViscosity	Scalar	The transported quantity solved by the Spalart-Allmaras turbulence model.
TurbulenceIntermittency	Scalar	The probability that flow at a given point is turbulent, used by turbulence transition models.
ReynoldsTheta	Scalar	The Reynolds number based on boundary layer thickness, used by turbulence transition models.

Table 8.1.13.6. Fluid Wall Flux Variables

Variable Name	Variable Type	Field Variable Definition
SurfaceShearStress	Vector	The shear viscous force of the fluid on the surface per unit area.

Variable Name	Variable Type	Field Variable Definition
SurfaceShearStress.x	Vector Component	The x, y, and z components of the SurfaceShearStress vector.
SurfaceShearStress.y		
SurfaceShearStress.z		
SurfaceShearStress.mag	Scalar	The magnitude of the SurfaceShearStress vector.
SurfaceHeatFlux	Scalar	The heat flow into the fluid per unit area.
SurfaceHeatFlow	Scalar	The heat flow into the fluid (SurfaceHeatFlux times area).

Table 8.1.13.7. Fluid Force Variables

Variable Name	Variable Type	Field Variable Definition
Force	Vector	The total force (pressure and viscous) of the fluid on the surface.
Force.x	Vector Component	The x, y, and z components of the Force vector.
Force.y		
Force.z		
Force.mag	Vector Magnitude	The magnitude of the Force vector.

Table 8.1.13.8. Fluid Pressure Force Variables

Variable Name	Variable Type	Field Variable Definition
PressureForce	Vector	The pressure force of the fluid on the surface.
PressureForce.x	Vector Component	The x, y, and z components of the PressureForce vector.
PressureForce.y		
PressureForce.z		
PressureForce.mag	Vector Magnitude	The magnitude of the PressureForce vector.

Table 8.1.13.9. Fluid Viscous Force Variables

Variable Name	Variable Type	Field Variable Definition
ViscousForce	Vector	The viscous force of the fluid on the surface.
ViscousForce.x	Vector Component	The x, y, and z components of the ViscousForce vector.
ViscousForce.y		
ViscousForce.z		
ViscousForce.mag	Vector Magnitude	The magnitude of the ViscousForce vector.

Table 8.1.13.10. Fluid Position Variables

Variable Name	Variable Type	Field Variable Definition
Position	Vector	The spatial coordinate vector.

Variable Name	Variable Type	Field Variable Definition
Position.x	Vector Component	The x, y, and z components of the Position vector.
Position.y		
Position.z		
Position.mag	Vector Magnitude	The magnitude of the Position vector.

Table 8.1.13.11. Fluid Mesh Metrics Variables

Variable Name	Variable Type	Field Variable Definition
ElementAspectRatio	Scalar	A measure of the stretching of a cell. It is computed as the ratio of the maximum value to the minimum value of any of the following distances: the normal distances between the cell centroid and face centroids (computed as a dot product of the distance vector and the face normal), and the distances between the cell centroid and nodes.
ElementVolume	Scalar	The volume of the element.
ElementWallDistance	Scalar	The distance from the wall face centroid to the adjacent element centroid.
FaceArea.mag	Vector	Magnitude of the Face Area.
FaceArea.x	Vector Component	The x, y, and z components of the outward-directed Face Area vector.
FaceArea.y		
FaceArea.z		
OrthogonalQuality	Scalar	The range for <a href="#">orthogonal quality</a> is 0-1. The best (most orthogonal) cells have larger values, while lower quality (highly skewed) cells have smaller values.

Table 8.1.13.12. Fluid Time and Iteration Variables

Variable Name	Variable Type	Field Variable Definition
Iteration	Single Valued	A counter for the current iteration number.

Table 8.1.13.13. Fluid Residual Variables

Variable Name	Variable Type	Field Variable Definition
PressureResidual	Scalar	Residual of the continuity equation.
TemperatureResidual	Scalar	Residual of the energy equation.
TurbEddyFreqResid	Scalar	Residual of the turbulence eddy frequency equation.
TurbKinetEnergyResid	Scalar	Residual of the turbulence kinetic energy equation.
VelocityResidualX	Scalar	Residuals of the X, Y, and Z components of the momentum equation.
VelocityResidualY		
VelocityResidualZ		

### 8.1.14. Polymer Extrusion Variables

The following tables describe the variables available for polymer extrusion results.

Table 8.1.14.1. Polymer Material Properties Variables

Variable Name	Variable Type	Description
DynamicViscosity	Scalar	The resistance of a polymer to shear deformation.

Table 8.1.14.2. Polymer Fluid Fraction Variables

Variable Name	Variable Type	Description
Fluid Fraction $n$ ( $n=1-9$ )	Scalar	The ratio of Fluid $n$ to the entire flow field.

Table 8.1.14.3. Polymer Position Variables

Variable Name	Variable Type	Description
Displacement	Vector	Difference between the final calculated position and the initial position.
Displacement.mag	Vector Magnitude	The magnitude of the Displacement vector.
Displacement.x Displacement.y Displacement.z	Vector Component	The x, y, and z components of the Displacement vector.
Position	Vector	The spatial coordinate vector.
Position.mag	Vector Magnitude	The magnitude of the Position vector.
Position.x Position.y Position.z	Vector Component	The x, y, and z components of the Position vector.

Table 8.1.14.4. Polymer Pressure Variables

Variable Name	Variable Type	Description
Pressure	Scalar	The fluid gauge pressure.

Table 8.1.14.5. Polymer Stress Variables

Variable Name	Variable Type	Description
Pseudo Shear Rate	Scalar	Quantity deriving from the Shear Rate and used in the Simplified Viscoelastic model.
Shear Rate	Scalar	The rate of change of the fluid velocity.
Stress Along Velocity Direction	Scalar	Stress on a plane locally perpendicular to velocity vector.

Variable Name	Variable Type	Description
Stress.xx	Tensor Component	A general three-dimensional strain state is calculated in terms of three normal (X, Y, Z) and three shear (XY, YZ, XZ) strain components aligned to the specified coordinate system.
Stress.xy	Tensor Component	
Stress.xz	Tensor Component	
Stress.yy	Tensor Component	
Stress.yz	Tensor Component	
Stress.zz	Tensor Component	
1st Principal Stress	Scalar	An infinitesimal volume of material at an arbitrary point on or inside the solid body can be rotated such that only normal stresses remain and all shear stresses are zero. The three normal stresses that remain are called the principal stresses .
2nd Principal Stress	Scalar	
3rd Principal Stress	Scalar	
		The principal stresses are always ordered such that $\sigma_1 > \sigma_2 > \sigma_3$ . The principal stresses and maximum shear stress are called invariants; that is, their value does not depend on the orientation of the part or assembly with respect to its specified coordinate system.

Table 8.1.14.6. Polymer Thermal Variables

Variable Name	Variable Type	Description
Temperature	Scalar	The polymer temperature.
ViscousHeating	Scalar	The heat generation by friction of fluid layers per unit time and per unit volume.

Table 8.1.14.7. Polymer Velocity Variables

Variable Name	Variable Type	Description
Velocity	Vector	The polymer velocity field (rate of change of polymer position vs. time at a fixed point, using an Eulerian formulation).
Velocity.mag	Vector Magnitude	The magnitude of the Velocity vector (speed).
Velocity.x	Vector Component	The x, y, and z components of the Velocity vector.
Velocity.y		
Velocity.z		

Table 8.1.14.8. Polymer Overlap Variables

Variable Name	Variable Type	Description
Inside n (n=1 to 5)	Scalar	The velocity node that is overlapped by a restrictor.

## 8.1.15. Polymer Blow Molding Variables

The following tables describe the variables available for polymer blow molding results.

Table 8.1.15.1. Polymer Contact Variables

Variable Name	Variable Type	Description
Contact Status	Scalar	The contact information for each node. It can have a value of either 0 or 1. If a node of the free surface is in contact, the field value is 1, otherwise the value is 0.
Contact Time	Scalar	The time during which a node has been in contact with the mold.
Thermal Transfer Status	Scalar	The information for each node stating if there is actually heat transfer in the case of a region interface.

Table 8.1.15.2. Polymer Position Variables

Variable Name	Variable Type	Description
Displacement Magnitude	Vector Magnitude	The magnitude of the Displacement vector.
Displacement X	Vector Component	The x, y, and z components of the Displacement vector.
Displacement Y		
Displacement Z		
Initial Position Magnitude	Vector Magnitude	The magnitude of the Initial Position vector.
Initial Position X	Vector Component	The x, y, and z components of the Initial Position vector.
Initial Position Y		
Initial Position Z		
Position Magnitude	Vector Magnitude	The magnitude of the Position vector.
Position X	Vector Component	The x, y, and z components of the Position vector.
Position Y		
Position Z		

Table 8.1.15.3. Polymer Thickness Variables

Variable Name	Variable Type	Description
Thickness	Scalar	The thickness of the polymer sheet.

Table 8.1.15.4. Polymer Stress Variables

Variable Name	Variable Type	Description
Area Stretch Ratio	Scalar	The ratio of the area of an elementary material surface in the deformed configuration to that in the undeformed state.

Table 8.1.15.5. Polymer Thermal Variables

Variable Name	Variable Type	Description
Temperature	Scalar	The polymer temperature.

Table 8.1.15.6. Polymer Velocity Variables

Variable Name	Variable Type	Description
Velocity Magnitude	Vector Magnitude	The magnitude of the Velocity vector (speed).
Velocity X Velocity Y Velocity Z	Vector Component	The x, y, and z components of the Velocity vector.
Mold Velocity Magnitude	Vector Magnitude	The magnitude of the Mold Velocity vector (speed).
Mold Velocity X Mold Velocity Y Mold Velocity Z	Vector Component	The x, y, and z components of the Mold Velocity vector.

### 8.1.16. Structural Variables

The table below lists the variables that can be accessed from the Structural solver. These variables can be entered into data fields as part of an expression wherever access to the Structural solver data is permitted. The variable names shown in the tables are the versions that are used in expressions. You may see more user-friendly versions of the variable names used as labels or in dropdown lists (for example, **Absolute Pressure** vs. **AbsolutePressure**, or **Velocity X** vs. **Velocity.x**).

Table 8.1.16.1. Structural Strain Variables

Variable Name	Variable Type	Description
ElasticStrain.xx (.x)	Tensor Component	A general three-dimensional strain state is calculated in terms of three normal (X, Y, Z) and three shear (XY, YZ, XZ) strain components aligned to the specified coordinate system.
ElasticStrain.yy (.y)	Tensor Component	
ElasticStrain.zz (.z)	Tensor Component	
ElasticStrain.xy (.yx)	Tensor Component	
ElasticStrain.zy (.yz)	Tensor Component	
ElasticStrain zx (.xz)	Tensor Component	
ElasticStrain.p1	Scalar	From elasticity theory, an infinitesimal volume of material at an arbitrary point on or inside the solid body can be rotated such that only normal strains remain and all shear strains are zero. The three normal strains that remain are called the principal strains.
ElasticStrain.p2	Scalar	
ElasticStrain.p3	Scalar	
ElasticStrain.eqv	Scalar	
ElasticStrain.maxshear	Scalar	The principal strains are always ordered such that $\epsilon_1 > \epsilon_2 > \epsilon_3$ . The principal strains and maximum shear strain are called invariants; that is, their value does not depend on the orientation of the part or assembly with respect to its specified coordinate system.
ElasticStrain.intensity	Scalar	Elastic Strain intensity is defined as the largest of the absolute values of $\epsilon_1 - \epsilon_2$ , $\epsilon_2 - \epsilon_3$ , or $\epsilon_3 - \epsilon_1$ .

PlasticStrain.xx (x)	Tensor Component	<p>Most common engineering materials exhibit a linear stress-strain relationship up to a stress level known as the proportional limit. Beyond this limit, the stress-strain relationship will become nonlinear, but will not necessarily become inelastic. Plastic behavior, characterized by nonrecoverable strain or plastic strain, begins when stresses exceed the material's yield point. Because there is usually little difference between the yield point and the proportional limit, AIM assumes that these two points are coincident in plasticity analyses.</p> <p>A general three-dimensional strain state is calculated in terms of three normal (X, Y, Z) and three shear (XY, YZ, XZ) strain components aligned to the specified coordinate system.</p>
PlasticStrain.yy (y)	Tensor Component	
PlasticStrain.zz (z)	Tensor Component	
PlasticStrain.xy (xy)	Tensor Component	
PlasticStrain.zy (zy)	Tensor Component	
PlasticStrain.zx (zx)	Tensor Component	
PlasticStrain.p1	Scalar	<p>From elasticity theory, an infinitesimal volume of material at an arbitrary point on or inside the solid body can be rotated such that only normal strains remain and all shear strains are zero. The three normal strains that remain are called the principal strains.</p> <p>The principal strains are always ordered such that <math>\epsilon_1 &gt; \epsilon_2 &gt; \epsilon_3</math>. The principal strains are called invariants; that is, their value does not depend on the orientation of the part or assembly with respect to its specified coordinate system.</p>
PlasticStrain.p2	Scalar	
PlasticStrain.p3	Scalar	
PlasticStrain.eqv	Scalar	The equivalent plastic strain gives a measure of the amount of permanent strain in an engineering body. The equivalent plastic strain is calculated from the component plastic strain.
PlasticStrain.intensity	Scalar	Plastic Strain intensity is defined as the largest of the absolute values of $\epsilon_1 - \epsilon_2$ , $\epsilon_2 - \epsilon_3$ , or $\epsilon_3 - \epsilon_1$ .
TotalStrain.xx (x)	Tensor Component	<p>The total strain is calculated by the addition of elastic, plastic, thermal, and creep strains.</p> <p>A general three-dimensional strain state is calculated in terms of three normal (X, Y, Z) and three shear (XY, YZ, XZ) strain components aligned to the specified coordinate system.</p>
TotalStrain.yy (y)	Tensor Component	
TotalStrain.zz (z)	Tensor Component	
TotalStrain.xy (yx)	Tensor Component	
TotalStrain.yz (zy)	Tensor Component	
TotalStrain.zx (xz)	Tensor Component	
TotalStrain.p1	Scalar	<p>From elasticity theory, an infinitesimal volume of material at an arbitrary point on or inside the solid body can be rotated such that only normal strains remain and all shear strains are zero. The three normal strains that remain are called the principal strains.</p> <p>The principal strains are always ordered such that <math>\epsilon_1 &gt; \epsilon_2 &gt; \epsilon_3</math>. The principal strains are called invariants; that is, their value does not depend on the orientation of the part or assembly with respect to its specified coordinate system.</p>
TotalStrain.p2	Scalar	
TotalStrain.p3	Scalar	
TotalStrain.eqv	Scalar	The equivalent total strain gives a total value of strain in any engineering body. Equivalent total strain is calculated from the total strain components.

TotalStrain.intensity	Scalar	Total strain intensity is defined as the largest of the absolute values of $\epsilon_1 - \epsilon_2$ , $\epsilon_2 - \epsilon_3$ , or $\epsilon_3 - \epsilon_1$ .
ThermalStrain.xx (.x)	Tensor Component	Thermal strain is computed when coefficient of thermal expansion is specified and a temperature load is applied in a structural analysis.  A general three-dimensional strain state is calculated in terms of three normal (X, Y, Z) and three shear (XY, YZ, XZ) strain components aligned to the specified coordinate system.
ThermalStrain.yy (.y)	Tensor Component	
ThermalStrain.zz (.z)	Tensor Component	
ThermalStrain.xy (.yx)	Tensor Component	
ThermalStrain.zy (.yz)	Tensor Component	
ThermalStrain.zx (.xz)	Tensor Component	
ThermalStrain.p1	Scalar	From elasticity theory, an infinitesimal volume of material at an arbitrary point on or inside the solid body can be rotated such that only normal strains remain and all shear strains are zero. The three normal strains that remain are called the principal strains.
ThermalStrain.p2	Scalar	
ThermalStrain.p3	Scalar	
ThermalStrain.eqv	Scalar	
ThermalStrain.intensity	Scalar	Thermal Strain intensity is defined as the largest of the absolute values of $\epsilon_1 - \epsilon_2$ , $\epsilon_2 - \epsilon_3$ , or $\epsilon_3 - \epsilon_1$ .

Table 8.1.16.2. Structural Stress Variables

Variable Name	Variable Type	Description
Stress.xx (.x)	Tensor Component	A general three-dimensional stress state is calculated in terms of three normal (X, Y, Z) and three shear (XY, YZ, XZ) stress components aligned to the specified coordinate system.
Stress.yy (.y)	Tensor Component	
Stress.zz (.z)	Tensor Component	
Stress.xy (.yx)	Tensor Component	
Stress.zy (.yz)	Tensor Component	
Stress.zx (.xz)	Tensor Component	

Variable Name	Variable Type	Description
Stress.p1	Scalar	
Stress.p2	Scalar	
Stress.p3	Scalar	
Stress.eqv	Scalar	
Stress.maxshear	Scalar	The principal stresses are always ordered such that $\sigma_1 > \sigma_2 > \sigma_3$ . The principal stresses and maximum shear stress are called invariants; that is, their value does not depend on the orientation of the part or assembly with respect to its specified coordinate system.
Stress.intensity	Scalar	Stress intensity is defined as the largest of the absolute values of $\sigma_1 - \sigma_2$ , $\sigma_2 - \sigma_3$ , or $\sigma_3 - \sigma_1$ .

Table 8.1.16.3. Structural Displacement Variables

Variable Name	Variable Type	Description
Displacement	Vector	
Displacement.x	Vector Component	
Displacement.y	Vector Component	
Displacement.z	Vector Component	
Displacement.mag	Vector Magnitude	The three component deformations, Displacement.x, Displacement.y, and Displacement.z, and the resultant deformation, Displacement.mag, are available as individual results.

Table 8.1.16.4. Structural Force Reaction Variables

Variable Name	Variable Type	Description
ForceReaction	Vector	
ForceReaction.x	Vector Component	
ForceReaction.y	Vector Component	
ForceReaction.z	Vector Component	
ForceReaction.mag	Vector Magnitude	The three component force reactions, ForceReaction.x, ForceReaction.y, and ForceReaction.z, and the resultant force reaction, ForceReaction.mag, are available as individual results.

Table 8.1.16.5. Structural Moment Reaction Variables

Variable Name	Variable Type	Description
MomentReaction	Vector	
MomentReaction.x	Vector Component	
MomentReaction.y	Vector Component	
MomentReaction.z	Vector Component	
MomentReaction.mag	Vector Magnitude	The three component moment reactions, MomentReaction.x, MomentReaction.y, and MomentReaction.z, and the resultant moment reaction, MomentReaction.mag, are available as individual results.

Table 8.1.16.6. Structural Nodal Load Variables

Variable Name	Variable Type	Description
Force	Vector	These are the element nodal forces. These results are available when applied to geometry or loads.
Force.x	Vector Component	
Force.y	Vector Component	
Force.z	Vector Component	
Force.mag	Vector Magnitude	

Table 8.1.16.7. Bolt Pretension Variables

Variable Name	Variable Type	Description
Adjustment	Vector Magnitude	This represents the displacement that occurs from the applied pretension measured at the point where the bolt is sliced, or the displacement reported from the pretension node. This result is also available for reporting regardless of how the bolt is defined.
WorkingLoad	Vector Magnitude	This represents the constrained force reaction from the pretension load. It is the constrained reaction reported from the pretension node and reports a zero value during a step in which you have applied the preload (since there is no reaction at the bolt slice during preload step).  This is essentially the sum of all the forces acting through the pretension cut. This result is only applicable for load steps when the load is defined by either <b>Locked</b> or <b>Factor</b> .

Table 8.1.16.8. Structural Contact Variables

Variable Name	Variable Type	Description
ContactPressure	Scalar	This result shows the measured pressure between two surfaces in contact.
SlidingDistance	Scalar	The total sliding distance is the amplitude of total accumulated slip increments (a geometrical measurement) when the contact status is sticking or sliding. It contains contributions from the elastic slip and the frictional slip. Elastic slip due to sticking represents the reversible tangential motion from the point of zero tangential stresses. Ideally, the equivalent elastic slip does not exceed the user-defined absolute limit. The higher the tangent stiffness, the smaller the resulting elastic slip.
Penetration	Scalar	For most contact methods, the numerics allow for the two bodies to have a certain amount of penetration. This result is a measure of that numerical penetration.
Gap	Scalar	This result shows how close the two surfaces of the contact are.
ContactStatus	Scalar	A discrete result which gives an indication of the contact state over the model.

Table 8.1.16.9. Structural Contact Reaction Variables

Variable Name	Variable Type	Description
ContactReaction	Vector	These are the reaction forces on the contact.  The three component reactions and the resultant reaction are available as individual results.
ContactReaction.x	Vector Component	
ContactReaction.y	Vector Component	
ContactReaction.z	Vector Component	
ContactReaction.mag	Scalar	

Table 8.1.16.10. Structural Beam Variables

Variable Name	Variable Type	Description
AxialForce.mag	Scalar	The magnitude of the force along a beam element axis.
ShearForce.mag	Scalar	The magnitude of the force perpendicular to the beam element axis
ShearForce.x	Scalar	The global X, Y and Z components of the force perpendicular to the beam element axis.
ShearForce.y	Scalar	
ShearForce.z	Scalar	
BendingMoment.mag	Scalar	The magnitude of the moment in the plane perpendicular to the beam element axis.
BendingMoment.x	Scalar	The global X, Y and Z components of the moment in the plane perpendicular to the beam element axis
BendingMoment.y	Scalar	
BendingMoment.z	Scalar	
AxialStress	Scalar	The stress component due to the axial load encountered in a beam element.
BendingStress.min	Scalar	From any bending loads, a bending moment in both the local Y and Z directions will arise. This leads to the following four bending stresses: Y bending stress on top/bottom and Z bending stress on the top/bottom. BendingStress.min is the minimum of these four bending stresses, and BendingStress.max the maximum.
BendingStress.max	Scalar	
CombinedStress.min	Scalar	The linear combination of the AxialStress and the BendingStress.min.
CombinedStress.max	Scalar	The linear combination of the AxialStress and the BendingStress.max.

Table 8.1.16.11. Structural Joint Variables

Variable Name	Variable Type	Description
JointTotalForce	Vector	Joint force and moment are by definition the action of the reference body on the moving body. For the ANSYS solver, the joint constraint forces and moments are reported in the joint reference coordinate system. The three component forces and the resultant force are available as individual results.
JointTotalForce.x	Vector Component	
JointTotalForce.y	Vector Component	
JointTotalForce.z	Vector Component	
JointTotalForce.mag	Scalar	
JointTotalMoment	Vector	Joint force and moment are by definition the action of the reference body on the moving body. For the ANSYS solver, the joint constraint forces and moments are reported in the joint reference coordinate system. The three component moments and the resultant moment are available as individual results.
JointTotalMoment.x	Vector Component	
JointTotalMoment.y	Vector Component	
JointTotalMoment.z	Vector Component	
JointTotalMoment.mag	Scalar	
RelativeDisplacement	Vector	Measures the relative displacement of the mobile body with respect to the reference body. The three component displacements and the resultant displacement are available as individual results.
RelativeDisplacement.x	Vector Component	
RelativeDisplacement.y	Vector Component	
RelativeDisplacement.z	Vector Component	
RelativeDisplacement.mag	Scalar	
RelativeRotation	Vector	Measures the relative rotation of the mobile body with respect to the reference body. The three component rotations and the resultant rotation are available as individual results.
RelativeRotation.x	Vector Component	
RelativeRotation.y	Vector Component	
RelativeRotation.z	Vector Component	
RelativeRotation.mag	Scalar	

Table 8.1.16.12. Structural Spring Variables for 2-D Springs

Variable Name	Variable Type	Description
SpringForce.mag	Scalar	The reaction force applied by the spring when it is stretched or compressed from its equilibrium position. It acts in the direction opposite to stretch or compress. Applicable to 1-D springs.
SpringMoment.mag	Scalar	The reaction moment applied by the spring when it is twisted from its equilibrium position. The moment acts in the direction opposite to twist direction. Applicable to 1-D springs.
SpringStretch.mag	Scalar	The stretch is the relative displacement between the two ends of the springs for a longitudinal spring. This elongation value can be positive (stretching the spring) or negative (compressing the spring). Applicable to 1-D springs.
SpringTwist.mag	Scalar	The twist is the change in the angle from the equilibrium position that two ends of the torsional spring subtends at its center. The twist value can be positive (increased subtended angle) or negative (decreased subtended angle). Applicable to 1-D springs.

Table 8.1.16.13. Structural Spring Variables for 3-D Springs

SpringTotalForce	Vector	The total reaction force applied by the spring when it is stretched or compressed from its equilibrium position. It acts in the direction opposite to stretch or compress. The three component forces and the resultant force are available as individual results. Applicable to 3-D springs.
SpringTotalForce.x	Vector Component	
SpringTotalForce.y	Vector Component	
SpringTotalForce.z	Vector Component	
SpringTotalForce.mag	Scalar	
SpringTotalMoment	Vector	The total reaction moment applied by the spring when it is twisted from its equilibrium position. The moment acts in the direction opposite to twist direction. The three component moments and the resultant moment are available as individual results. Applicable to 3-D springs.
SpringTotalMoment.x	Vector Component	
SpringTotalMoment.y	Vector Component	
SpringTotalMoment.z	Vector Component	
SpringTotalMoment.mag	Scalar	
SpringTotalStretch	Vector	The stretch is the relative displacement between the two ends of the springs for a longitudinal spring. This elongation value can be positive (stretching the spring) or negative (compressing the spring). The three component stretch vectors and the resultant stretch value are available as individual results. Applicable to 3-D springs.
SpringTotalStretch.x	Vector Component	
SpringTotalStretch.y	Vector Component	
SpringTotalStretch.z	Vector Component	
SpringTotalStretch.mag	Scalar	
SpringTotalTwist	Vector	The twist is the change in the angle from the equilibrium position that two ends of the torsional spring subtends at its center. The twist value can be positive (increased subtended angle) or negative (decreased subtended angle). The three component twist vectors and the resultant twist value are available as individual results. Applicable to 3-D springs.
SpringTotalTwist.x	Vector Component	
SpringTotalTwist.y	Vector Component	
SpringTotalTwist.z	Vector Component	
SpringTotalTwist.mag	Scalar	

Table 8.1.16.14. Fatigue Variables

Variable Name	Variable Type	Description
FatigueLife	Scalar	Fatigue Life represents the number of cycles of repeated loading until the part will fail due to fatigue.
FatigueDamage	Scalar	Fatigue Damage is defined as the design life divided by the fatigue life.
FatigueSafetyFactor	Scalar	Fatigue Safety Factor is defined as the ratio of alternating stress at design life to the equivalent alternating stress at a given point. The maximum safety factor reported is 15.
FatigueBiaxiality	Scalar	Biaxiality indication is defined as the principal stress smaller in magnitude divided by the larger principal stress with the principal stress nearest zero ignored. A biaxiality of zero corresponds to uniaxial stress, a value of -1 corresponds to pure shear, and a value of 1 corresponds to a pure biaxial state.

Variable Name	Variable Type	Description
FatigueEquivalentStress	Scalar	In a Stress Life fatigue analysis, you must query an S-N curve to relate the fatigue life to the stress state. Thus the “equivalent alternating stress” is the stress used to query the fatigue S-N curve after accounting for fatigue loading type, mean stress effects, multiaxial effects, and any other factors in the fatigue analysis.

Table 8.1.16.15. Miscellaneous Mechanical Variables

Variable Name	Variable Type	Description
StructuralError	Scalar	You can insert an Error result based on stresses to help you identify regions of high error and therefore show where the model would benefit from a more refined mesh in order to get a more accurate answer.

## 8.1.17. Electric Conduction Variables

The table below lists the electric conduction variables that can be accessed from the Structural solver. These variables can be entered into data fields as part of an expression wherever access to the electric conduction data is permitted. The variable names shown in the tables are the versions that are used in expressions. You may see more user-friendly versions of the variable names used as labels or in dropdown lists (for example, **Absolute Pressure** vs. **AbsolutePressure**, or **X Velocity** vs. **Velocity.x**).

Table 8.1.17.1. Electric Potential Variables

Variable Name	Variable Type	Description
ElectricPotential	Scalar	Represents contours of constant electric potential (voltage) in conductor bodies.

Table 8.1.17.2. Electric Field Variables

Variable Name	Variable Type	Description
ElectricField	Vector	A vector quantity representing the magnitude and direction of the electric field in the region of interest.
ElectricField.x	Vector Component	
ElectricField.y	Vector Component	
ElectricField.z	Vector Component	
ElectricField.mag	Vector Magnitude	The three component electric field variables, ElectricField.x, ElectricField.y, and ElectricField.z, and the resultant electric field, ElectricField.mag, are available as individual results.

Table 8.1.17.3. Current Density Variables

Variable Name	Variable Type	Description
CurrentDensity	Vector	A vector quantity representing the amount of electric current passing through a unit cross-sectional area.
CurrentDensity.x	Vector Component	
CurrentDensity.y	Vector Component	
CurrentDensity.z	Vector Component	
CurrentDensity.mag	Vector Magnitude	The three component current density variables, CurrentDensity.x, CurrentDensity.y, and CurrentDensity.z, and the resultant current density, CurrentDensity.mag, are available as individual results.

Table 8.1.17.4. Current Reaction Variables

Variable Name	Variable Type	Description
CurrentReaction	Scalar	The amount of current passing through a surface/voltage constraint. (This is analogous to heat reaction in a thermal analysis).

Table 8.1.17.5. Joule Heat Generation Variables

Variable Name	Variable Type	Description
HeatGeneration	Scalar	An electrical current passing through a conductor can generate heat (commonly called joule heating). This output is a measure of that quantity.

## 8.1.18. Electrostatic Variables

The table below lists the Electrostatic variables that can be accessed from the solver. These variables can be entered into data fields as part of an expression wherever access to the Electrostatic data is permitted. The variable names shown in the tables are the versions that are used in expressions. You may see more user-friendly versions of the variable names used as labels or in dropdown lists (for example, **Absolute Pressure** vs. **AbsolutePressure**, or **X Velocity** vs. **Velocity.x**).

Table 8.1.18.1. Electrostatic Variables

Variable Name	Variable Type	Description
ElectricPotential	Scalar	Represents contours of constant electric potential (voltage) in conductor bodies.
ChargeReaction	Scalar	A scalar quantity representing the reaction of charges on the voltage locations.
ElectricField	Vector	A vector quantity representing the magnitude and direction of the electric field in the region of interest.
ElectricField.x	Vector Component	The three component electric field variables, ElectricField.x, ElectricField.y, and ElectricField.z, and the resultant electric field, ElectricField.mag, are available as individual results.
ElectricField.y	Vector Component	
ElectricField.z	Vector Component	
ElectricField.mag	Vector Magnitude	

## 8.1.19. Thermal Variables in Thermal/Structural Simulations

The table below lists the thermal variables that you can use in Thermal and Thermal-Structural simulations. You can enter the variables into data fields as part of an expression wherever access to the thermal data is permitted.

The entries in the **Variable Name** column are used in expressions; however, you may see more user-friendly versions of those names in labels or dropdown lists. For example:

- Absolute Pressure rather than AbsolutePressure
- X Velocity rather than Velocity.x

	Variable Name	Variable Type	Description
<b>Thermal Variables</b>	Temperature	Scalar	In a steady-state or time-dependent thermal analysis, AIM calculates the temperature distribution throughout the structure. This result is available when applied to geometry or loads and constraints.
	HeatFlux	Vector	AIM calculates the heat flux ( $q/A$ , energy per unit time per unit area) throughout the body. The three component heat fluxes and the resultant heat flux value are available as individual results.
	HeatFlux.x HeatFlux.y HeatFlux.z	Vector Component	
	HeatFlux.mag	Vector Magnitude	These results are available when applied to geometry.
	TemperatureGradient	Vector	In addition to temperature, AIM also calculates the variation of the temperature results with time.  The three component temperature values and the resultant temperature value are available as individual results.
	TemperatureGradientx TemperatureGradienty TemperatureGradientz	Vector Component	
	TemperatureGradient.mag	Vector Magnitude	These results are available when applied to geometry.
<b>Thermal Nodal Load Variable</b>	HeatFlow	Scalar	AIM calculates the heat flow throughout the body. These results are available when applied to geometry or loads.
<b>Heat Flow Reaction Variable</b>	HeatFlowReaction	Scalar	You can obtain heat reaction ( $q$ , energy per unit time) at locations where a constraint is specified. Heat reaction is a scalar.
<b>Miscellaneous Thermal Variables</b>	ThermalError	Scalar	You can insert an Error result based on stresses to help you identify regions of high error and thus show where the model would benefit from a more refined mesh in order to get a more accurate answer.

### 8.1.20. Electromagnetics Variables

The table below lists the variables that can be accessed from the Magnetics solver. These variables can be entered into data fields as part of an expression wherever access to the electromagnetic data is permitted. The variable names shown in the tables are the versions that are used in expressions. You may see more user-friendly versions of the variable names used as labels or in dropdown lists (for example, **Force Magnitude** vs. Force.mag, or **Force X** vs. Force.x).

Table 8.1.20.1. Torque Variables

Variable Name	Variable Type	Field Variable Definition
Torque	Scalar	The magnitude of the torque.

Table 8.1.20.2. Force Variables

Variable Name	Variable Type	Field Variable Definition
Force.x	Scalar	The magnitude of the x component of the force vector.
Force.y	Scalar	The magnitude of the y component of the force vector.
Force.z	Scalar	The magnitude of the z component of the force vector.
Force.mag	Scalar	The magnitude of the force vector.

Table 8.1.20.3. Calculated Variables

Variable Name	Variable Type	Field Variable Definition
ACForce.x	Complex	The x component of the AC Force vector. Because complex numbers cannot be used in Calculated values directly, four methods are provided: ACForce.x.real, ACForce.x.imag, ACForce.x.mag, ACForce.x.phase.
ACForce.y	Complex	The y component of the AC Force vector. Because complex numbers cannot be used in Calculated values directly, four methods are provided: ACForce.y.real, ACForce.y.imag, ACForce.y.mag, ACForce.y.phase.
ACForce.z	Complex	The z component of the AC Force vector. Because complex numbers cannot be used in Calculated values directly, four methods are provided: ACForce.z.real, ACForce.z.imag, ACForce.z.mag, ACForce.z.phase.
ACForce.mag	Real	The magnitude of the AC Force vector.
InducedVoltage	Complex	Induced voltage is an electrical potential created by and proportional to the rate of change of the flux linkage of the current defined by terminals.  Because complex numbers cannot be used in Calculated values directly, four methods are provided: InducedVoltage.ireal, InducedVoltage.imag, and InducedVoltage.phase.
InducedVoltage.mag	Real	The magnitude of the Induced Voltage vector.
WindingFluxLinkage	Complex	The flux linkage is the summation of the magnetic field linking with all coils in a current when the magnetic field passes through the loops of all coils.  Because complex numbers cannot be used in Calculated values directly, four methods are provided: WindingFluxLinkage.ireal, WindingFluxLinkage.imag, and WindingFluxLinkage.phase.
WindingFluxLinkage.mag	Real	The magnitude of the Winding Flux Linkage vector.

Table 8.1.20.4. DC Force Variables

Variable Name	Variable Type	Field Variable Definition
DCForce.x	Real	The magnitude of the x component of the DC Force vector.
DCForce.y	Real	The magnitude of the y component of the DC Force vector.
DCForce.z	Real	The magnitude of the z component of the DC Force vector.

Table 8.1.20.5. Loss Variables

Variable Name	Variable Type	Field Variable Definition
OhmicLossDensity	Real	<p>Ohmic loss is always associated with conduction current distribution in conductors which are not perfect. Thus the resistivity of conductors is responsible for the ohmic power loss when current flows in such conductors. It is also called the Joule-Lenz effect. There is always a heating effect due to the ohmic loss, often called Joule heating.</p> <p><b>Note:</b> If an object has a fill factor specified, the conductivity value is multiplied by this fill factor. This is then the conductivity value used for post processing.</p>
CoreLossDensity	Real	<p>Applicable to frequency response solutions, the core loss combines eddy current losses and hysteresis losses. It is applicable for the evaluation of core losses in steel (frequently used in applications such as electric machines, transformers), or in power ferrites.</p>
DielectricLossDensity	Real	<p>Dielectric loss is associated with loss density fields in frequency response solutions only. It is applicable when a high frequency electric field penetrates a dielectric material.</p> <p><b>Note:</b> If an object has a fill factor specified, the conductivity value is multiplied by this fill factor. This is then the conductivity value used for post processing.</p>
HysteresisLossDensity	Real	<p>Hysteresis loss is associated with loss density fields in frequency response solutions only. Hysteresis loss is short for magnetic hysteresis loss and represents the power loss in some magnetic materials (electric steels or ferrites) in alternating (sinusoidal) magnetic fields.</p>
Total Loss	Real	<p>Total loss is the sum of OhmicLossDensity, CoreLossDensity, DielectricLossDensity, and HysteresisLossDensity.</p>

Table 8.1.20.6. Magnetic Variables

Variable Name	Variable Type	Field Variable Definition
CurrentDensity	Vector	A vector quantity representing the magnitude of current density (J).
CurrentDensity.x	Vector Component	The three component current density variables, CurrentDensity.x, CurrentDensity.y, and CurrentDensity.z, and the resultant magnitude of the current density, CurrentDensity.mag, are available as individual results.
CurrentDensity.y	Vector Component	
CurrentDensity.z	Vector Component	
CurrentDensity.mag	Vector Magnitude	
MagneticFieldIntensity	Vector	A vector quantity representing the magnetic field intensity (H).
MagneticFieldIntensity.x	Vector Component	The three component magnetic field intensity variables, MagneticFieldIntensity.x, MagneticFieldIntensity.y, and MagneticFieldIntensity.z, and the resultant magnitude of the magnetic field intensity, MagneticFieldIntensity.mag, are available as individual results.
MagneticFieldIntensity.y	Vector Component	
MagneticFieldIntensity.z	Vector Component	
MagneticFieldIntensity.mag	Vector Magnitude	
MagneticFluxDensity	Vector	A vector quantity representing the magnetic flux density (B) vector.
MagneticFluxDensity.x	Vector Component	The three component magnetic flux density variables, MagneticFluxDensity.x, MagneticFluxDensity.y, and MagneticFluxDensity.z, and the resultant magnitude of the magnetic flux density, MagneticFluxDensity.mag, are available as individual results.
MagneticFluxDensity.y	Vector Component	
MagneticFluxDensity.z	Vector Component	
MagneticFluxDensity.mag	Vector Magnitude	
TotalMagneticFlux	Scalar	The surface integral of the normal component of the magnetic flux density passing through the surface.

Table 8.1.20.7. Current Density Variables

Variable Name	Variable Type	Field Variable Definition
CurrentDensity.mag	Scalar	The magnitude of current density, J.
CurrentDensity	Vector	The current density (J) vector.
TotalCurrent	Scalar	The surface integral of the normal component of the current density (J) over the selected faces.

Table 8.1.20.8. Thermal Variables

Variable Name	Variable Type	Field Variable Definition
Temperature	Scalar	The temperature distribution throughout the selected location.

## 8.2. ANSYS Discovery AIM Journaling and Scripting Overview

ANSYS Discovery AIM enables you to record your actions (journaling) and modify those actions programmatically (scripting), much as you can in Workbench.

AIM generally uses the Workbench project and data model concepts:

- The methods that are available are detailed in the data containers.
- The study-based commands that are available are described in the namespaced commands.
- There are the following additions:

### Project Elements

- Study
- Task
- Task Group

### Scripting Interface Concepts

- AssociatedObject

### Apps and Template Apps

- Simulation Template

- There is the following exception:

- Expressions must evaluate to a “quantity” type, which requires the units to be included; therefore none of the examples in the section apply.

You can find detailed descriptions of the entire project and data model concepts, including these AIM-specific ones, in the Discovery AIM and Workbench Scripting Guide.