

# Simcenter™ Flotherm™ User Guide

Software Version 2021.1

**SIEMENS**

Unpublished work. © 2021 Siemens

This material contains trade secrets or otherwise confidential information owned by Siemens Industry Software, Inc., its subsidiaries or its affiliates (collectively, "Siemens"), or its licensors. Access to and use of this information is strictly limited as set forth in Customer's applicable agreement with Siemens. This material may not be copied, distributed, or otherwise disclosed outside of Customer's facilities without the express written permission of Siemens, and may not be used in any way not expressly authorized by Siemens.

This document is for information and instruction purposes. Siemens reserves the right to make changes in specifications and other information contained in this publication without prior notice, and the reader should, in all cases, consult Siemens to determine whether any changes have been made. Siemens disclaims all warranties with respect to this document including, without limitation, the implied warranties of merchantability, fitness for a particular purpose, and non-infringement of intellectual property.

The terms and conditions governing the sale and licensing of Siemens products are set forth in written agreements between Siemens and its customers. Siemens' **End User License Agreement** may be viewed at: [www.plm.automation.siemens.com/global/en/legal/online-terms/index.html](http://www.plm.automation.siemens.com/global/en/legal/online-terms/index.html).

No representation or other affirmation of fact contained in this publication shall be deemed to be a warranty or give rise to any liability of Siemens whatsoever.

**TRADEMARKS:** The trademarks, logos, and service marks ("Marks") used herein are the property of Siemens or other parties. No one is permitted to use these Marks without the prior written consent of Siemens or the owner of the Marks, as applicable. The use herein of third party Marks is not an attempt to indicate Siemens as a source of a product, but is intended to indicate a product from, or associated with, a particular third party. A list of Siemens' trademarks may be viewed at: [www.plm.automation.siemens.com/global/en/legal/trademarks.html](http://www.plm.automation.siemens.com/global/en/legal/trademarks.html). The registered trademark Linux® is used pursuant to a sublicense from LMI, the exclusive licensee of Linus Torvalds, owner of the mark on a world-wide basis.

Support Center: [support.sw.siemens.com](http://support.sw.siemens.com)

Send Feedback on Documentation: [support.sw.siemens.com/doc\\_feedback\\_form](http://support.sw.siemens.com/doc_feedback_form)

# Table of Contents

---

<b>Chapter 1</b>	
<b>Product Overview.....</b>	<b>23</b>
Functional Overview .....	23
Product Workflow .....	25
Define the Requirements .....	25
Add a Solution Grid .....	26
Solve the Model .....	27
Analyze the Results .....	28
Product GUI.....	31
<b>Chapter 2</b>	
<b>Defining the Requirements .....</b>	<b>33</b>
Establishing the Requirements.....	33
Start Simple .....	33
Main Features to Consider When Modeling .....	33
<b>Chapter 3</b>	
<b>Working on a Project .....</b>	<b>35</b>
Project Overview .....	36
Starting and Running the Product .....	38
Starting and Running in Interactive Mode .....	38
Starting From a Command Window.....	38
Exiting.....	38
FloSCRIPT.....	40
About FloSCRIPT.....	40
Recording a Sequence of Operations to a FloSCRIPT File .....	42
Running a FloSCRIPT File.....	43
Project Manager Window .....	44
Window Layout .....	45
Property Sheets .....	46
Data Tree.....	47
New Object Palette .....	49
Project Attributes/Library Trees.....	53
Project Manager Status Bar .....	54
Workplane.....	56
Snap Grid .....	57
Project Operations .....	59
Creating a New Project.....	59
Setting the Default Project .....	60
Installing the Application Examples.....	60
Loading Projects .....	62
Loading an Existing Project .....	62

---

Owning a Project . . . . .	62
Changing the Solution Directory . . . . .	63
Cataloging Projects . . . . .	64
Importing External Projects . . . . .	65
Importing a FloXML Project . . . . .	65
Importing a PDML Project . . . . .	65
Importing a PACK Project . . . . .	66
Importing a V1.4 Project . . . . .	66
Importing a V2/V3 Project . . . . .	67
Saving . . . . .	68
Saving a GDA Image to a Graphics File . . . . .	68
Saving a Project . . . . .	68
Saving a Project as a Template . . . . .	69
Exporting Projects in SAT or IGS Files . . . . .	70
Exporting Thermal Netlists and BCI-ROM Files . . . . .	70
Archiving . . . . .	76
Archiving Project Data in PDML Format . . . . .	76
Archiving Project Data in PACK Format . . . . .	77
Deleting Projects . . . . .	78
Project Manager Operations . . . . .	80
Showing and Hiding Toolbars . . . . .	80
Detaching/Reattaching the Drawing Board . . . . .	81
Detaching/Reattaching the Profiles Window . . . . .	82
Changing Global Units . . . . .	83
Changing Local Units . . . . .	83
Undoing and Redoing the Last Operation . . . . .	84
Changing the Coordinate System . . . . .	85
Using Parallel Processing . . . . .	85
Data Tree Operations . . . . .	87
Viewing Summary Information . . . . .	87
Adding Assemblies . . . . .	89
Adding Parts . . . . .	90
Colocating Objects . . . . .	90
Selecting Objects in the Data Tree . . . . .	91
Promoting and Demoting Geometry in the Data Tree . . . . .	92
Topping Assemblies or Parts . . . . .	92
Hiding Objects From the Data Tree (Lightweight View of Assemblies) . . . . .	93
Simple Searches for Objects or Attributes . . . . .	93
Complex Searches for Objects or Attributes . . . . .	94
Applying Simultaneous Changes to More Than One Object . . . . .	96
Creating Unique Names for Objects . . . . .	97
Zoom-In Projects . . . . .	98
Zoom-in Project Example . . . . .	98
Creating a Zoom-In Project . . . . .	98
Determining the Coupling Metric . . . . .	103
Project Manager and General Dialog Boxes . . . . .	104
Base Project Changed Dialog Box . . . . .	105
BCI-ROM Preferences Dialog Box . . . . .	106
Create Category Dialog Box . . . . .	108

## Table of Contents

---

Find Dialog Box . . . . .	109
Find Dialog Box - Quick Criteria Tab . . . . .	110
Find Dialog Box - Extended Criteria Tab . . . . .	112
Global Units Dialog Box . . . . .	115
License Timeout Dialog Box . . . . .	116
Load Project Dialog Box . . . . .	117
Message Window . . . . .	119
New Project Dialog Box . . . . .	120
Notes Dialog Box . . . . .	122
Save Project Dialog Box . . . . .	123
Thermal Netlist Preferences Dialog Box . . . . .	125
Thermal Netlist/BCI-ROM Export Progress Dialog Box . . . . .	126
User Preferences Dialog Box . . . . .	127
User Preferences Dialog Box - Project Manager Tab . . . . .	128
User Preferences Dialog Box - Drawing Board Tab . . . . .	130
User Preferences Dialog Box - Summary Tab . . . . .	132
User Preferences Dialog Box - Analyze Mode Tab . . . . .	133
Zoom-In Creation Dialog Box . . . . .	136
 <b>Chapter 4</b>	
<b>Defining the Mathematical Model. . . . .</b>	<b>137</b>
The Modeling Method . . . . .	138
The Solution Domain . . . . .	140
Solution Domain Overall Definitions . . . . .	140
Moving and Resizing the Solution Domain in the Drawing Board . . . . .	140
Boundary Types . . . . .	141
Non-Cuboid Solution Domains . . . . .	141
Examples of Open and Symmetry Boundaries . . . . .	141
Symmetry and Radiation . . . . .	142
Initial Values . . . . .	143
Initial Values and Solution Type . . . . .	143
The Solution Variables . . . . .	143
Starting From a Previous Solution Set . . . . .	144
Viewing Initial Fields . . . . .	145
Setting Initial Field Values Over Subdomains . . . . .	146
Solution Variable Types . . . . .	147
Base and Standard Derived Variables . . . . .	147
Auxiliary Variables . . . . .	147
Project Solution Variables Procedures . . . . .	148
Determining Base Solution Variables . . . . .	148
Determining Additional Derived Variables . . . . .	148
Modeling Selections . . . . .	150
Types of Solution . . . . .	150
Mass Flux . . . . .	150
Heat Flux . . . . .	151
Surface Temperature . . . . .	152
Temperature Gradient . . . . .	153
Power Density . . . . .	153

Bn and Sc . . . . .	155
Modeling Dimensionality . . . . .	156
Capture Index . . . . .	159
Concept of the Capture Index . . . . .	159
Data Center Aisle Groups and Sub-Groups . . . . .	160
Aisle Group CI Calculation . . . . .	161
Assessing CI Values for Data Center Design . . . . .	162
Capture Index Procedures . . . . .	164
Attaching an Aisle Group or Sub-Group Name to an Object . . . . .	164
Checking Aisle Group Attachments in Data Tree Summary Information . . . . .	165
Viewing Capture Index Results . . . . .	166
Adding Capture Index to Command Center Scenarios . . . . .	167
Solar Radiation . . . . .	168
Solar Radiation Model . . . . .	168
Including Solar Radiation . . . . .	168
Airflow . . . . .	170
Laminar Airflow . . . . .	170
Turbulent Airflow and Models . . . . .	170
Revised Algebraic Model . . . . .	171
Automatic Algebraic Model . . . . .	172
LVEL K-Epsilon Model . . . . .	172
Gravity . . . . .	175
Modeling Gravity . . . . .	175
Calculation of Gravity . . . . .	176
Changing the Fluid Properties . . . . .	178
Reference Temperature and Pressure . . . . .	180
Datum Pressure . . . . .	180
Modeling a Remote Radiation Source . . . . .	180
Ambient Temperature When Modeling Buoyancy Force . . . . .	182
Default Initial Value for Temperature . . . . .	182
Transient Variation of Temperature . . . . .	182
Joule Heating Analysis . . . . .	183
Joule Heating Modeled by Source SmartParts . . . . .	183
Joule Heating and Power Map SmartParts . . . . .	183
Solution Domain Dialog Boxes and Property Sheets . . . . .	184
System Property Sheet . . . . .	185
System Property Sheet Boundaries Tab . . . . .	186
Subdomain Property Sheet . . . . .	187
Subdomain Property Sheet Initial Values Tab . . . . .	188
Solar Configuration Dialog Box . . . . .	189
Model Setup Tab . . . . .	191
Model Setup Tab - Gravity Section . . . . .	193
Model Setup Tab - Turbulence Section . . . . .	194
Model Setup Tab - Global System Settings Section . . . . .	195
Model Setup Tab - Stored Variables Section . . . . .	197
Model Setup Tab - Auxiliary Variables Section . . . . .	199
Solver Control Tab . . . . .	200
Solver Control Tab - Monitor Point Solution Control Section . . . . .	204
Solver Control Tab - Variable Solution Control Section . . . . .	205

---

## Table of Contents

---

Solution Set Dialog Box .....	208
Solution Type Changed Dialog Box .....	209
<b>Chapter 5</b>	
<b>Creating, Importing, and Exporting Geometry.....</b>	<b>211</b>
Overview of Geometry .....	212
Coordinate Systems .....	215
Construction Precedence Rules .....	218
Construction Precedence Rules Overview .....	218
Context Rules for Primitives.....	219
Blocks or Prisms in Surface Contact .....	219
Overlapping Intersecting Blocks or Prisms .....	220
Block Placed on the Surface of a Collapsed Cuboid .....	222
Cuboid Block Intersected by a Collapsed Cuboid .....	222
Cuboids in Contact With Sloping Blocks.....	223
Cuboids in Contact With Domain Edge .....	224
Context Rules for SmartParts .....	225
Context Rules for Flow Devices.....	225
Context Rules for Overlapping Fixed Flows .....	225
Context Rules for Overlapping Supplies and Extracts of Recirculation Devices .....	226
Context Rules for Overlapping Sources .....	226
Context Rules for Overlapping Components .....	226
Context Rules for Network Assemblies and Heat Pipes .....	226
Context Rules for Plate Conduction.....	227
Geometry Files.....	229
PDML Files.....	229
FloXML Files .....	229
FloFEA Files.....	232
ECAD File Formats .....	232
ECXML Files .....	233
JEDEC Part Model XML Files (ThermalSection) .....	234
Files Imported via MCAD Bridge .....	235
Files Imported via EDA Bridge .....	235
Power Map Files .....	235
Geometry Create and Import Operations.....	236
Creating Complex Shapes.....	236
Promoting and Demoting Objects.....	237
Copying Objects .....	237
Creating a Pattern of Objects .....	238
Mirroring an Object .....	239
Importing FloXML Geometry Files .....	240
Importing IDF Files .....	241
Importing PDML Geometry Files.....	242
Importing ECXML Files .....	242
Importing Network Assemblies From JEDEC Part Model XML Files .....	243
Importing T3Ster Files .....	243
Importing V1.4 Library Files .....	244
Importing V2/V3 Project Assembly Files .....	244

Export Operations .....	245
Exporting Geometry .....	245
Exporting an Assembly to FloTHERM PACK .....	246
Exporting Network Assemblies to JEDEC Part Model XML Files .....	247
Converting Files to PDML Format .....	248
Exporting ECXML Files .....	248
Creating FloXML From Spreadsheets .....	249
Pattern Creation Dialog Box .....	251
<b>Chapter 6</b>	
<b>Viewing and Manipulating Geometry.....</b>	<b>253</b>
The Drawing Board .....	254
Interaction of the Drawing Board With the Data Tree .....	254
Drawing Board Views .....	255
Drawing Board Color Conventions .....	255
The Active Pane .....	256
Snap Modes .....	256
Viewing Geometry .....	258
Changing Geometry Rendering .....	258
Changing Mouse Mode in Create Mode .....	259
Tabbing Between Views .....	259
Controls for Viewing the Spatial Solution Grid .....	260
Changing the Orientation of a Drawing Board Pane .....	262
Aligning the View .....	262
Rotating the View Incrementally .....	263
Changing the View Using the Mouse .....	263
Resizing the View .....	264
Hiding Geometry .....	264
Changing Geometry Transparency .....	265
Changing the GDA Background .....	266
Manipulating Geometry .....	268
Selecting Objects in the Drawing Board .....	268
Moving Objects Using the Mouse .....	269
Aligning an Object Using the Keyboard .....	271
Aligning One or More Objects to Another Object .....	271
Moving Objects by Specified Coordinate Distances .....	271
Resizing Objects Using the Mouse .....	272
Copying Objects in the Drawing Board .....	274
Measuring the Distance Between Geometry Vertices or Edges .....	275
Rotating Objects .....	276
Drawing Board Dialog Boxes .....	278
Align Dialog Box .....	279
Measure Dialog Box .....	280
Move Dialog Box .....	281
Rotate View Dialog Box .....	282
Drawing Board Icons .....	283

---

## Table of Contents

---

<b>Chapter 7</b>	
<b>Libraries .....</b>	<b>285</b>
Library Overview .....	285
Library Folder Operations .....	287
Adding a New Library .....	287
Adding Items to a Library From the Current Project .....	287
Importing Library Files .....	288
Copying Libraries .....	288
Refreshing a Library Folder .....	289
Deleting a Library Folder .....	289
Exporting a Library Folder .....	289
Library Item Operations .....	290
Copying Geometry From Libraries to the Data Tree .....	290
Copying an Attribute From a Library Folder to the Data Tree .....	291
Moving and Copying Library Items Between Library Folders .....	291
Saving Project Geometry to a Library Folder .....	292
Starter Libraries .....	292
Generic Data Center Library .....	295
Generic CRAC Units .....	295
Generic Cabinets .....	296
Equipment for High Detail Cabinet .....	297
Generic Floor Grille .....	297
Generic Floor Support .....	297
Gilberts Data Center Library .....	297
Library Property Sheets and Dialog Boxes .....	299
Library Data Property Sheet .....	300
Create Library Dialog Box .....	301
<b>Chapter 8</b>	
<b>Spatial Solution Grid .....</b>	<b>303</b>
Spatial Solution Grid Overview .....	304
Overview of Grid Modification .....	304
Grid Calculation Order .....	305
Grid Suitability .....	305
Grid Constraints .....	307
Grid Localization .....	309
General Effect of Localizing a Grid .....	309
Nested Localized Grid Spaces .....	310
Abutment of Localized Grid Spaces .....	310
Keypoints and Localized Grids .....	311
Localization of Inflated Grids .....	311
Grid Space Overlap When Localizing .....	312
Localization Near the Edge of the Solution Domain .....	313
Objects Located Inside a Localized Grid .....	313
Object Location Restrictions When Using Localized Grids .....	314
Grid Smoothing and Localized Cells .....	315
Minimum Cell Size and Localized Cells .....	316
Grid Inflation .....	317

Grid Smoothing . . . . .	319
Keypoint Deactivation (De-Keypointing) . . . . .	320
Grid Operations . . . . .	322
Debugging the Grid of Large Aspect Ratio Cells . . . . .	322
Setting System Grid Constraints . . . . .	323
Setting an Object Grid Constraint . . . . .	323
Localizing the Grid Around an Object . . . . .	324
Inflating the Grid Around an Object . . . . .	325
Deflating the Grid Around an Object . . . . .	325
Smoothing the Grid . . . . .	325
Solution Grid Property Sheets and Dialog Boxes . . . . .	327
System Grid Property Sheet . . . . .	328
Grid Summary Dialog . . . . .	330
Grid Changed Dialog Box . . . . .	332
Previous Software Version Dialog Box . . . . .	333
<b>Chapter 9</b>	
<b>Transient Analysis . . . . .</b>	<b>335</b>
Transient Analysis Introduction . . . . .	335
Total Transient Time and the Transient Solution Period . . . . .	336
Time Steps and the Time Distribution Plot . . . . .	337
Transient Analysis Residuals . . . . .	339
Transient Analysis Resolution . . . . .	340
Time Patches . . . . .	342
Time Step Distribution Types . . . . .	343
Transient Keypoints and Keypoint Tolerance . . . . .	344
Transient Functions (Time-Varying Attributes) . . . . .	344
Save Times . . . . .	345
Transient Analysis Procedures . . . . .	346
Obtaining Initial Data From Steady-State Analysis . . . . .	346
Setting Up a Transient Analysis Time Grid . . . . .	346
Creating Time Patches . . . . .	347
Setting Save Times . . . . .	348
Solving a Transient Analysis . . . . .	349
Viewing Plots of Transient Save Time Data . . . . .	350
Viewing Transient Save Time Data in Tables . . . . .	350
Transient Solution Dialog Box . . . . .	352
<b>Chapter 10</b>	
<b>Solving the Project . . . . .</b>	<b>355</b>
Solver Overview . . . . .	356
Monitor Points . . . . .	356
Pre-Solve Checks . . . . .	357
Exchange Factors . . . . .	358
Interactive Solve Procedures . . . . .	359
Obtaining Maximum Solver Performance . . . . .	359
Sanity Checking . . . . .	359
Solving in Interactive Mode . . . . .	360

## Table of Contents

---

Re-Initializing Variable Fields . . . . .	361
Interrupting and Resuming an Interactive Solution . . . . .	361
Aborting an Interactive Solution. . . . .	362
Setting Solution Process Priority Using the Task Manager . . . . .	362
Batch Mode Solve . . . . .	363
Files Used Before, During, and After Batch Solves . . . . .	364
Batch Files . . . . .	364
logit File . . . . .	364
floerror.log File . . . . .	365
Output CSV Files . . . . .	366
Batch Solve Procedures . . . . .	367
Solving in Batch Mode . . . . .	367
Using batchSolve to Solve in Batch Mode on Linux Systems . . . . .	368
Environment Setup on Linux Systems . . . . .	368
Monitoring the Solution in Batch Mode . . . . .	368
Interrupting a Batch Solution . . . . .	369
Setting up a Batch Mode Completion Email Notification . . . . .	369
Setting Solution Process Priority Using an Environment Variable . . . . .	370
Transient Analysis Cases . . . . .	371
Solve Dialog Boxes . . . . .	373
Solver Progress Dialog Box . . . . .	374
Exchange Factors Calculation Progress Dialog Box . . . . .	375
<b>Chapter 11</b>	
<b>Solution Monitoring and Profile Plots . . . . .</b>	<b>377</b>
Solution Monitoring. . . . .	377
Profiles Window . . . . .	379
Profile Plot Features. . . . .	380
Available Plots. . . . .	384
Residuals v Iteration Plot (Steady State) . . . . .	385
Residuals v Iteration Plot (Transient) . . . . .	387
Residual Iterations v Time Plot (Transient) . . . . .	388
Residuals v Time Plot (Transient) . . . . .	389
Monitor Points v Iteration Plot (Steady State) . . . . .	390
Monitor Points v Time Plot (Transient) . . . . .	392
Results v Distance Plot . . . . .	394
Plot Procedures . . . . .	396
Creating New Plots . . . . .	396
Configuring Monitor Point Plots . . . . .	396
Tiling Plots . . . . .	397
Exporting a Plot Profile . . . . .	398
Plot Property Sheet . . . . .	400
Plot Property Sheet Settings Tab . . . . .	401
<b>Chapter 12</b>	
<b>Advanced Solution Controls . . . . .</b>	<b>403</b>
The Solution Process . . . . .	403
Program-Calculated Termination Residuals . . . . .	405

Possible Solution Scenarios . . . . .	407
Controlling the Solution . . . . .	408
Techniques for Controlling the Solution . . . . .	410
False Time Step Relaxation . . . . .	410
Multi Grid Damping . . . . .	412
Inner Iterations . . . . .	413
Fan Relaxation . . . . .	414
Monitor Point Convergence for Temperature . . . . .	414
Error Field . . . . .	416
User-Specified Termination Residuals . . . . .	417
Double-Precision Solver . . . . .	418

**Chapter 13**

<b>Viewing Results . . . . .</b>	<b>419</b>
Analyze Mode . . . . .	419
Results Tree . . . . .	422
Transient Time Step Selector . . . . .	423
Results Tree Property Sheet Functionality . . . . .	424
Analyze Mode Mouse Pointer . . . . .	426
Analyze Mode Operations . . . . .	428
Changing Mouse Mode in Analyze Mode . . . . .	428
Changing the Appearance of the GDA in Analyze Mode . . . . .	428
Manipulating the View in Analyze Mode . . . . .	430
Saving the Current Analyze State . . . . .	430
Loading a Previously Saved Analyze State . . . . .	430
Reinitializing the Analyze State . . . . .	431
Working With Plots . . . . .	432
Creating a Plane Plot . . . . .	432
Associating an Existing Plane Plot with Geometry . . . . .	434
Associating an Existing Plane Plot with Results . . . . .	434
Moving a Plane Plot . . . . .	434
Cutting a Plane Plot . . . . .	435
Creating a Surface Plot . . . . .	436
Creating an Isosurface . . . . .	437
Creating and Moving Particle Sources . . . . .	438
Moving a Plot Legend . . . . .	440
Working With Annotations . . . . .	442
Adding an Annotation to Geometry or a Plot . . . . .	442
Adding a Global Maximum or Minimum Annotation . . . . .	443
Adding an Annotation to the Center of a Geometry Object . . . . .	443
Adding a Maximum or Minimum Annotation to a Plot . . . . .	444
Adding a Note to an Annotation . . . . .	444
Moving an Annotation . . . . .	445
Working with Viewpoints . . . . .	446
Creating a Viewpoint . . . . .	446
Jumping to a Viewpoint . . . . .	446
Working With Animations . . . . .	448
Animating Plots . . . . .	448

## Table of Contents

---

Saving Animations . . . . .	449
Selecting Geometry Associated With a Plot or Annotation . . . . .	450
Exporting Cell by Cell Results Data . . . . .	450
Results Property Sheets . . . . .	454
Scalar Field Variable Property Sheet . . . . .	455
Vector Field Property Sheet . . . . .	457
Isosurface Property Sheet . . . . .	459
Plane Plot Property Sheet . . . . .	460
Surface Plot Property Sheet . . . . .	463
Particle Source Property Sheet . . . . .	464
Particle Source Property Sheet - Definition Tab . . . . .	465
Particle Source Property Sheet - Appearance Tab . . . . .	467
Annotation Property Sheet . . . . .	469
Annotation Property Sheet - Definition Tab . . . . .	470
Annotation Property Sheet - Location Tab . . . . .	472
Animation Property Sheet . . . . .	473
Saved Tables Property Sheet . . . . .	474
Viewpoint Property Sheet . . . . .	475
<b>Chapter 14</b>	
<b>Reporting Project Data and Results in Tables . . . . .</b>	<b>477</b>
Results Tables . . . . .	477
Temperature and Heat Flow Values in Solid Conductor Fluxes Results Tables . . . . .	478
Heat Transfer Coefficient Values in Solid Conductor Fluxes Results Tables . . . . .	480
Reported Junction and Case Temperatures of EDA Components . . . . .	480
Tables Operations . . . . .	482
Viewing Tables . . . . .	482
Searching Results Tables . . . . .	483
Expanding Results Tables to Show SmartPart Primitive Data . . . . .	485
Selecting Table Cell Data . . . . .	486
Resizing Table Columns . . . . .	486
Hiding Results Table Columns . . . . .	487
Sorting Tables . . . . .	487
Copying Selected Results Table Cell Data . . . . .	488
Saving the Currently Opened Tables for Re-Display . . . . .	488
Exporting a Results Table . . . . .	490
Exporting a Legacy Results Table . . . . .	490
Export Legacy Tables Dialog Box . . . . .	492
Results Tables Data . . . . .	493
Geometry Results Tables . . . . .	494
Assemblies Results Table . . . . .	495
Collapsed Resistances Results Table . . . . .	497
Compact Component Results Table . . . . .	498
Component Fluxes Results Table . . . . .	500
Controllers Results Table . . . . .	502
Coolers Results Table . . . . .	503
Cutouts/Overall Results Table . . . . .	504
Cutouts/Overall Results Summary Results Table . . . . .	505

Dies Results Table . . . . .	506
EDA Boards Results Table . . . . .	507
EDA Components Results Table . . . . .	508
EDA Heat Sinks Results Table . . . . .	510
Enclosures Results Table . . . . .	511
Fans Results Table . . . . .	512
Fixed Flows Results Table . . . . .	513
Heat Pipes Results Table . . . . .	514
Heat Sinks Results Table . . . . .	515
Monitor Points Results Table . . . . .	516
Network Assemblies Results Table . . . . .	517
Perforated Plates Results Table . . . . .	518
Power Maps Results Table . . . . .	519
Racks Results Table . . . . .	520
Recirculation Devices Results Table . . . . .	521
Region Mean Flows Results Table . . . . .	522
Regions Summary Results Table . . . . .	523
Region Variables Values Results Tables . . . . .	524
Solid Conductor Fluxes Results Table . . . . .	525
Solid Conductor Temperatures Results Table . . . . .	527
Solid Conductors Summary Results Table . . . . .	528
Sources Results Table . . . . .	530
TEC Results Table . . . . .	531
Legacy Tables . . . . .	532
Geometry Model Results Table CSV File . . . . .	533
Base Grid and Localized Grid Results Tables CSV Files . . . . .	535
Transient Grid Results Table CSV File . . . . .	536
Object/Attributes Results Table CSV File . . . . .	537

## Chapter 15

### Additional Auxiliary Variables . . . . . **539**

Available Auxiliary Variables . . . . .	540
Flow Angle . . . . .	540
Total Pressure . . . . .	540
Calculating and Displaying Additional Auxiliary Variables . . . . .	542
Calculating Flow Angle . . . . .	542
Calculating Total Pressure . . . . .	542
Displaying Auxiliary Variables Values . . . . .	543
Flow Angle Dialog Box . . . . .	544

## Chapter 16

### Frequently Asked Questions . . . . . **545**

Project Manager FAQs . . . . .	545
Drawing Board FAQs . . . . .	547
Solution Grid FAQs . . . . .	547
Solution Variables FAQs . . . . .	547
Building Geometry FAQs . . . . .	549
Fan FAQs . . . . .	549

## Table of Contents

---

Heat Sink FAQs .....	550
Recirculation Device FAQs .....	550
Support Site FAQs .....	551
Solver FAQs .....	551
Modeling FAQs .....	552
FloXML FAQs .....	553
<b>Chapter 17</b>	
<b>Troubleshooting .....</b>	<b>559</b>
Problems When Loading, Saving, Importing, Exporting, Packing, and Unpacking .....	559
Floating Point (FP) Errors .....	560
Convergence Problems .....	561
Display Problems .....	563
Unexpected Results .....	564
Miscellaneous Problems .....	566
<b>Chapter 18</b>	
<b>Messages .....</b>	<b>569</b>
Message Types .....	569
Message Format .....	570
Error Window .....	571
Generated Messages .....	573
System Error Messages .....	573
User System Messages (1000 Series) .....	573
Drawing Board Messages (3000 Series) .....	576
Project Manager Messages (4000 Series) .....	577
Profiles Messages (5000 Series) .....	580
Tables Messages (6000 Series) .....	582
Converter Messages (7000 Series) .....	583
Solver Messages (8000 Series) .....	586
Translator Messages (9000 Series) .....	587
EFG Messages (10000 Series) .....	595
Data Storage Messages (11000 Series) .....	596
MCAD Messages (12000 Series) .....	597
FloSCRIPT Messages (15000 Series) .....	600
Interface Messages (17000 Series) .....	601
Logic Error Messages (19000 Series) .....	601
Command Center Messages (20000 Series) .....	601
<b>Chapter 19</b>	
<b>Technical Support .....</b>	<b>603</b>
Global Customer Support and Success .....	603
Training .....	603
<b>Appendix A</b>	
<b>Command Reference .....</b>	<b>605</b>
flotherm .....	606
batchSolve .....	612

flogate\_cl . . . . . 613

**Appendix B**  
**Advanced Operations** . . . . . 615

Environment Variables . . . . . 615

**Glossary**

**Index**

**Third-Party Information**

# List of Figures

---

Figure 1-1. Problem Specification .....	25
Figure 1-2. Modeling an Electronics Cabinet .....	26
Figure 1-3. Computational Method .....	26
Figure 1-4. Electronics Cabinet With Superimposed Grid .....	27
Figure 1-5. Solving .....	27
Figure 1-6. Circulation Patterns in Electronics Cabinet .....	28
Figure 1-7. Increased Size Inlet Vent .....	29
Figure 1-8. Stagnant Region Near the Front of the Lower Rack of Boards .....	29
Figure 3-1. Window Layout .....	45
Figure 3-2. Data Tree Object Name in Edit Mode .....	47
Figure 3-3. Dimension Units Dropdown List .....	47
Figure 3-4. New Object Palette .....	49
Figure 3-5. Grid Cell High Face .....	55
Figure 3-6. Isometric View Showing Solution Domain and Workplane .....	56
Figure 3-7. The Snap Grid .....	58
Figure 3-8. Example Toolbar Visibility Dialog Box .....	81
Figure 3-9. Summary Information Column Headers .....	88
Figure 3-10. Adding Assemblies .....	89
Figure 3-11. A Cuboid Added to an Assembly .....	90
Figure 3-12. Adding Siblings .....	91
Figure 3-13. Topping .....	93
Figure 3-14. Creating Unique Object Names .....	97
Figure 3-15. Example of Parent Solution Data Appended to Zoom-In Project Notes .....	101
Figure 3-16. Representation of Boundary Conditions .....	102
Figure 3-17. Cutouts on Data Tree .....	102
Figure 4-1. Open and Symmetry Boundaries .....	142
Figure 4-2. Cutouts on the Solution Domain Boundary .....	142
Figure 4-3. Initial Values in Subdomain .....	146
Figure 4-4. Storage Mass Fluxes .....	151
Figure 4-5. Grid Layout .....	156
Figure 4-6. Fluid Speed .....	157
Figure 4-7. Velocity Vector .....	158
Figure 4-8. Cold Aisle CI Examples .....	159
Figure 4-9. Hot Aisle CI Examples .....	160
Figure 4-10. Example Cold Aisle Group .....	161
Figure 4-11. Example Hot Aisle Group .....	162
Figure 4-12. Accessing CI Values .....	163
Figure 4-13. Display of Aisle Groups in the Data Tree Summary .....	165
Figure 4-14. One Vector Component is Zero .....	176
Figure 4-15. Non-Zero Vector Components .....	176

---

Figure 4-16. Angled Gravity . . . . .	176
Figure 4-17. Application of Gravity Force . . . . .	178
Figure 4-18. Modeling a Remote Radiation Source . . . . .	181
Figure 4-19. Example Alignment . . . . .	190
Figure 5-1. Power Map Composite Cuboid Geometry . . . . .	215
Figure 5-2. Construction Precedence Rules . . . . .	218
Figure 5-3. Cuboid Blocks in Surface Contact . . . . .	219
Figure 5-4. No Heat Flux Across Common Surfaces . . . . .	220
Figure 5-5. Overlapping Blocks . . . . .	221
Figure 5-6. Overlapping Blocks — Block B Specified Second . . . . .	221
Figure 5-7. Overlapping Blocks — Block A Specified Second . . . . .	222
Figure 5-8. Block Resting on a Collapsed Cuboid . . . . .	222
Figure 5-9. Block With a Penetrating Collapsed Cuboid . . . . .	222
Figure 5-10. Cuboid in Contact With Sloping Block . . . . .	223
Figure 5-11. Cuboids in Contact With Domain Edge . . . . .	224
Figure 5-12. Context Rules for Recirculation Devices and Fixed Flows . . . . .	225
Figure 5-13. Context Rules for Fans and Resistances . . . . .	225
Figure 5-14. Example Layouts Where Conduction Between Collapsed Cuboids Will Occur . . . . .	227
Figure 5-15. Example Layout Where Conduction Between Collapsed Cuboids Will Not Occur . . . . .	227
Figure 5-16. Plate Conduction Example Showing Break in Conduction . . . . .	228
Figure 5-17. Objects Before Mirroring . . . . .	239
Figure 5-18. Objects After Mirroring . . . . .	240
Figure 6-1. Collapsed and Expanded Assemblies in the Data Tree and Drawing Board . . . . .	255
Figure 6-2. Grid Viewing Icons . . . . .	260
Figure 6-3. Solid and Wireframe Views and Grid Visibility . . . . .	261
Figure 6-4. Unprojected and Projected Grid Views . . . . .	261
Figure 6-5. Selected Object . . . . .	269
Figure 6-6. Mouse Icon When Moving an Object . . . . .	270
Figure 6-7. Moving a Cuboid . . . . .	270
Figure 6-8. Mouse Icons When Resizing an Object . . . . .	273
Figure 6-9. Resizing a Cuboid . . . . .	274
Figure 6-10. Copy Cursor . . . . .	275
Figure 7-1. The Libraries . . . . .	286
Figure 7-2. Generic CRAC Units . . . . .	295
Figure 8-1. Maximum Aspect Ratios . . . . .	306
Figure 8-2. Example Grid Constraints Attached to Two Cuboids . . . . .	308
Figure 8-3. Localized Grid . . . . .	309
Figure 8-4. Nesting Localized Grid Spaces . . . . .	310
Figure 8-5. Close and Abutting Localized Grid Spaces . . . . .	311
Figure 8-6. Unlocalized and Localized Grid Inflation . . . . .	312
Figure 8-7. Localized Overlapping Localized Grids . . . . .	312
Figure 8-8. Localization Near the Solution Domain Edge . . . . .	313
Figure 8-9. Object Internally Abutting a Localized Grid . . . . .	314
Figure 8-10. Object Lying More Than One Grid Cell Inside a Localized Grid . . . . .	314

## List of Figures

---

Figure 8-11. Localized Grid Before Smoothing . . . . .	315
Figure 8-12. Localized Grid After Smoothing . . . . .	316
Figure 8-13. Closely-Spaced Objects in Different Grid Spaces . . . . .	316
Figure 8-14. Default Grid Inflation . . . . .	317
Figure 8-15. Grid With Inflation Layers . . . . .	318
Figure 8-16. Grid Inflation and Localization . . . . .	319
Figure 8-17. Example of Keypoint Deactivation . . . . .	321
Figure 8-18. Cuboid With Localized Grid Switched On . . . . .	324
Figure 9-1. Transient Analysis of Temperature Versus Time for Two Monitor Points . . . . .	336
Figure 9-2. Sequence of Short Solution Periods . . . . .	336
Figure 9-3. Period Start Time Updating in Transient Solution Dialog Box . . . . .	337
Figure 9-4. Time Step Distribution Plot . . . . .	338
Figure 9-5. Time Steps Along a 1D Time Axis . . . . .	338
Figure 9-6. Monitor Point Values at Each Time Step . . . . .	339
Figure 9-7. Example of Non-Accumulation of Residuals . . . . .	340
Figure 9-8. Transient Analysis Resolution With 15, 30 and 60 Time Steps . . . . .	341
Figure 9-9. Additional Time Steps . . . . .	342
Figure 9-10. Local Grid Refinement . . . . .	342
Figure 9-11. Time Step Distribution Types . . . . .	343
Figure 9-12. Transient Function Overlay . . . . .	344
Figure 9-13. Save Times at Every Fourth Time Step . . . . .	345
Figure 10-1. Approaching Convergence . . . . .	356
Figure 10-2. Transient Solutions . . . . .	357
Figure 10-3. Extract From logit File . . . . .	365
Figure 11-1. Detached Profiles Window . . . . .	379
Figure 11-2. Reattached (Docked) Profiles Window . . . . .	380
Figure 11-3. Scrolling Through Large Numbers of Monitor Points in a Profiles Plot . . . . .	382
Figure 11-4. Profile Plot Annotations and Context-Sensitive Menu . . . . .	383
Figure 11-5. Steady State Solution Plot . . . . .	385
Figure 11-6. Docked Transient Solution Plot of Residuals v Iteration for a Time Step . . . . .	387
Figure 11-7. Residuals Iterations v Time Plot . . . . .	388
Figure 11-8. Residuals v Time Plot . . . . .	389
Figure 11-9. Monitor Points v Iteration Plot . . . . .	390
Figure 11-10. Monitor Points v Time Plot . . . . .	392
Figure 11-11. Results v Distance Plot . . . . .	394
Figure 11-12. Example Results v Distance Plot Data Processed in Excel . . . . .	399
Figure 12-1. Convergent Solution . . . . .	407
Figure 12-2. Divergent Solution . . . . .	407
Figure 12-3. High-Level Stable Residual Error . . . . .	407
Figure 12-4. Low-Level Stable Residual Error . . . . .	408
Figure 12-5. High-Level Oscillation Residual Error . . . . .	408
Figure 12-6. Low-Level Oscillation Residual Error . . . . .	408
Figure 12-7. Required Accuracy . . . . .	415
Figure 12-8. Number of Iterations . . . . .	415
Figure 12-9. Residual Error . . . . .	417

---

Figure 13-1. Results Transient Time Step Selector . . . . .	424
Figure 13-2. Plane Plot Property Sheet . . . . .	425
Figure 13-3. Changes Applied to Multiple Plots . . . . .	426
Figure 13-4. Plane Temperature Plot Through Selected Geometry . . . . .	433
Figure 13-5. Cut Plane Plots . . . . .	436
Figure 13-6. Isosurface Example . . . . .	438
Figure 13-7. Particle Smear and Width . . . . .	439
Figure 13-8. Track, Particle, Ribbon and Pipe Streamlines . . . . .	440
Figure 13-9. Example of a Plane or 2D Object Aligned in the X Direction . . . . .	451
Figure 13-10. Example csv Results File for a Plane Plot in the X Direction . . . . .	452
Figure 13-11. Example csv Results File for a Volume Object - Cell Results for Temperature	
452	
Figure 13-12. View Stagger . . . . .	453
Figure 14-1. Two Cuboids in Contact With Each Other . . . . .	479
Figure 14-2. Solid Conductors Summary Results Table Expanded to Show SmartPart Details	
486	
Figure 14-3. Compact Component Temperatures for an Area Array Package Type Modeled	
Using a General Network of Resistors . . . . .	498
Figure 14-4. Compact Component Temperatures for a Peripheral Package Type Modeled Using	
a General Network of Resistors . . . . .	499
Figure 14-5. Compact Component Temperatures for an Area Array Package Type Modeled With	
a Two-Resistor Representation . . . . .	499
Figure 14-6. Compact Component Heat Flow for an Area Array Package Type Modeled Using a	
General Network of Resistors . . . . .	500
Figure 14-7. Compact Component Heat Flow for a Peripheral Package Type Modeled Using a	
General Network of Resistors . . . . .	500
Figure 14-8. Compact Component Heat Flow for an Area Array Package Type Modeled With a	
Two-Resistor Representation . . . . .	501
Figure 16-1. Root Assembly and Sub-Assemblies . . . . .	549
Figure 16-2. Mirroring Around the Workplane . . . . .	549
Figure 16-3. Example Usage of FloXML Orientation Tag . . . . .	556
Figure 18-1. Message Hyperlink to Data Tree . . . . .	571

# List of Tables

---

Table 3-1. Summary Information Column Content .....	88
Table 3-2. Allowed Geometry Criteria .....	99
Table 4-1. Power Density Support for Objects .....	153
Table 5-1. Primitives .....	212
Table 5-2. SmartParts .....	214
Table 5-3. Assembly FloXML Examples .....	231
Table 5-4. Project FloXML Examples .....	231
Table 5-5. Supplied XML Spreadsheet Examples .....	249
Table 6-1. Drawing Board Color Conventions .....	256
Table 6-2. Object Axes Directions .....	272
Table 7-1. Starter Libraries .....	292
Table 8-1. Guidelines on Aspect Ratio Levels .....	307
Table 12-1. False Time Step Relaxation .....	411
Table 13-1. Analyze Mode Mouse Pointers .....	426
Table 16-1. Base Variables Required by the Auxiliary Variables .....	548
Table 18-1. Message Numbering .....	569
Table A-1. Results Tables Output CSV Files (-o Option) .....	607
Table A-2. All Tables Output CSV Files (-O Option) .....	609
Table B-1. Environment Variables .....	615



# Chapter 1

## Product Overview

---

Simcenter™ Flotherm™ software is a thermal modeling analysis tool designed to simulate the airflow and heat transfer in electronic systems, subsystems, and packages. Using Simcenter Flotherm shortens design cycles and yields better, more reliable products, being “correct by design”.

You can use simulation during the earliest stage of electronics systems development to observe the effects of design changes on the thermal behavior, prior to building and testing the prototype.

<b>Functional Overview .....</b>	<b>23</b>
<b>Product Workflow .....</b>	<b>25</b>
Define the Requirements .....	25
Add a Solution Grid .....	26
Solve the Model .....	27
Analyze the Results .....	28
<b>Product GUI.....</b>	<b>31</b>

## Functional Overview

Simcenter Flotherm uses Computational Fluid Dynamics (CFD) to analyze airflow and heat transfer and is specifically designed to investigate these phenomena within electronics equipment.

Airflow can be caused by natural (hot air rises) or mechanical (fans) means. There are three modes of heat transfer:

- **Conduction** — Heat transfer through a solid or stationary fluid.
- **Convection** — Heat transfer from a surface into a fluid.
- **Radiation** — Heat transfer from one surface to another.

Airflow and heat transfer are governed by conservation laws that can be expressed in a partial differential form (Navier Stokes Equations).

In Simcenter Flotherm, the conservation equations are converted to a finite volume form. As the name implies, the converted equations require a volume in which values of temperature, pressure, and velocity are calculated. Hence the space (called the solution domain) represented in the model is split into a number of small volumes or grid cells.

The more grid cells in the model, the more points are calculated and the better the resolution of the case, but at the expense of additional computer overheads to calculate the results.

The conservation equations are coupled (values for a variable depend on surrounding values of that variable and also other variables) and non-linear. Hence they are solved in an iterative manner until the errors in the conservation equations are at an acceptable level.

Working with Simcenter Flotherm, you define the requirements, set up the mathematical modeling parameters, construct the geometry, add solution grid, solve the solution, and display the results.

# Product Workflow

A high-level overview of the process.

<b>Define the Requirements</b>	.....	25
<b>Add a Solution Grid</b>	.....	26
<b>Solve the Model</b>	.....	27
<b>Analyze the Results</b>	.....	28

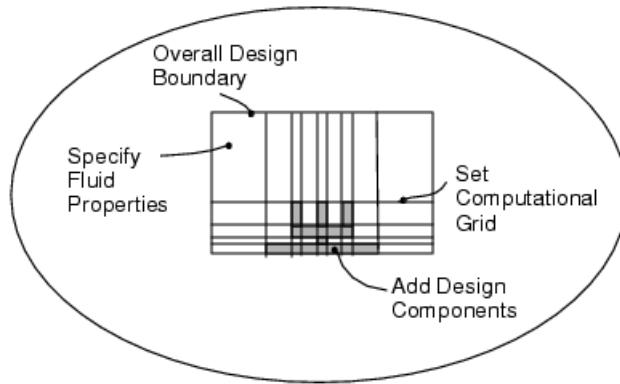
## Define the Requirements

You define the overall dimensions of the volume or enclosure to be investigated, including the individual components that make up the design structure, and superimpose the computational grid.

Each grid cell denotes a region for which Simcenter Flotherm calculates flow values (for example, velocities, temperature, and pressure) for the model.

The properties of the air flow (for example, density, viscosity, specific heat) set at the default values for dry air at 30°C can be changed, if necessary, to reflect the type of coolant used. Appropriate boundary conditions (for example, ambient temperature, known mass flow rate, and heat sources) can also be added.

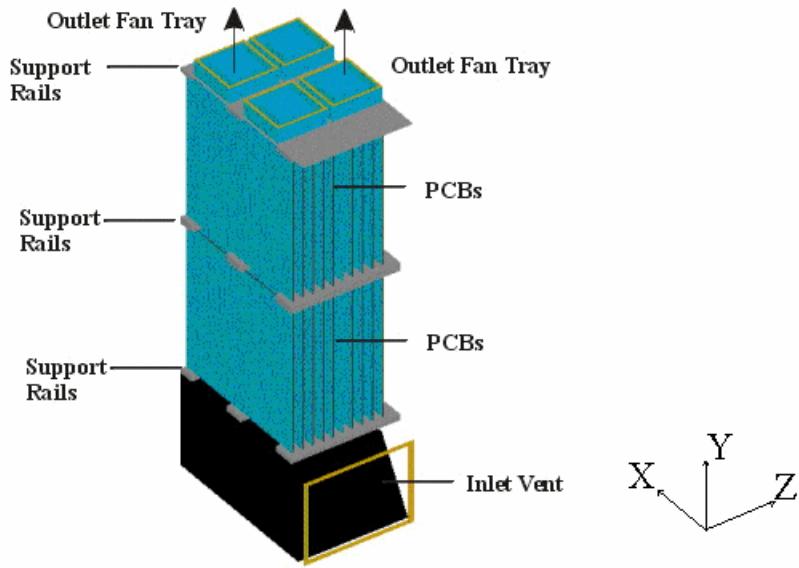
**Figure 1-1. Problem Specification**



An example is a Simcenter Flotherm model of the airflow and heat transfer inside the electronics cabinet as shown in [Figure 1-2](#).

The cabinet has two racks of PCBs supported by card guide support rails. The supply and outlet comprise an inlet vent at the bottom, and two outlet fan trays at the top.

**Figure 1-2. Modeling an Electronics Cabinet**



## Related Topics

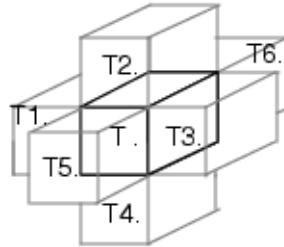
- [Defining the Requirements](#)
- [Defining the Mathematical Model](#)
- [Creating, Importing, and Exporting Geometry](#)

## Add a Solution Grid

During program solution, Simcenter Flotherm integrates the relevant differential conservation equations over each computational grid cell, assembling a set of algebraic equations which relate the value of a variable in a cell to its value in its nearest neighbor.

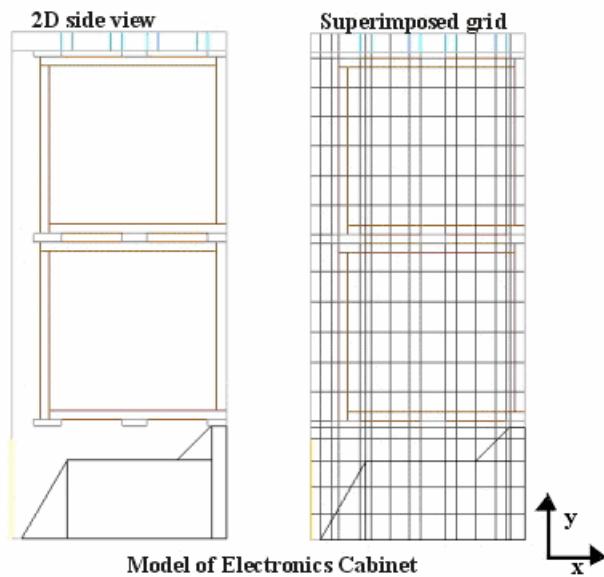
For example, temperature T shown in [Figure 1-3](#) is related to temperatures T<sub>1</sub>, T<sub>2</sub>, T<sub>3</sub>, T<sub>4</sub>, T<sub>5</sub>, and T<sub>6</sub> in the neighboring cells.

**Figure 1-3. Computational Method**



[Figure 1-4](#) shows a superimposed solution grid on an example electronics cabinet.

**Figure 1-4. Electronics Cabinet With Superimposed Grid**



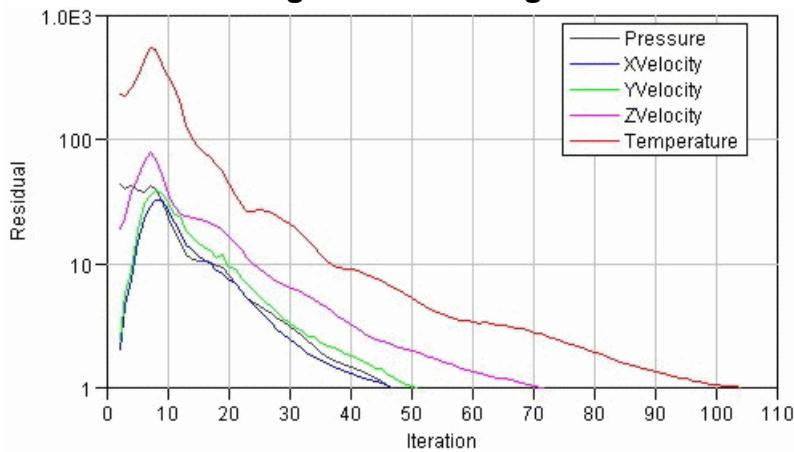
## Related Topics

[Spatial Solution Grid](#)

## Solve the Model

The program solves the algebraic equations, using iterative procedures to converge on a solution after a finite number of successive iterations.

**Figure 1-5. Solving**



## Related Topics

[Solving the Project](#)

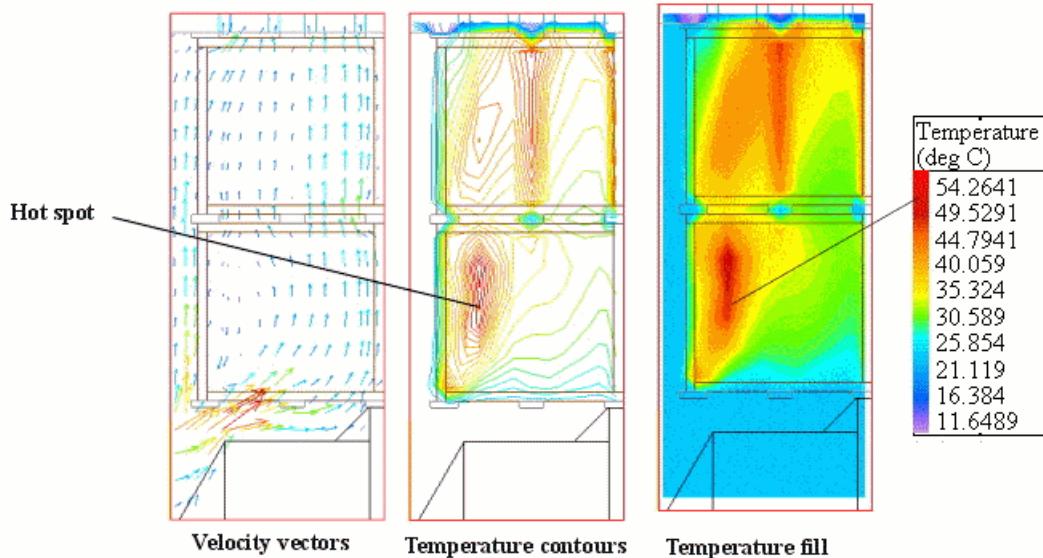
## Analyze the Results

After solution, the circulation patterns in the form of velocity vectors, isotherm, and isobar contours and fills, may be displayed to show the resultant flow behavior.

The patterns may be analyzed by the engineer, and corrective measures, such as the repositioning of fans and vents, taken.

After solution, the circulation patterns may be displayed to show the resultant flow values.

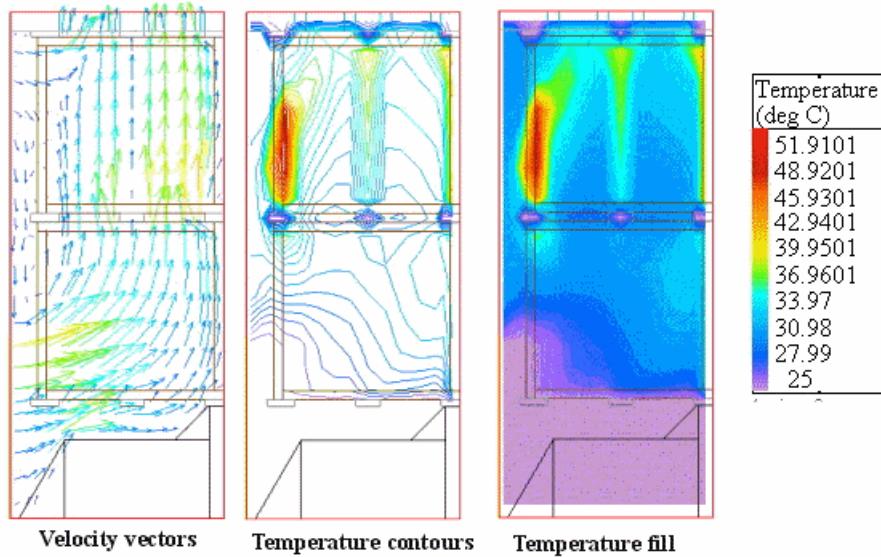
**Figure 1-6. Circulation Patterns in Electronics Cabinet**



In [Figure 1-6](#), the airflow in the top rack appears to be inhibited by the middle card guide support rail causing a hot spot in the top center of the PCBs.

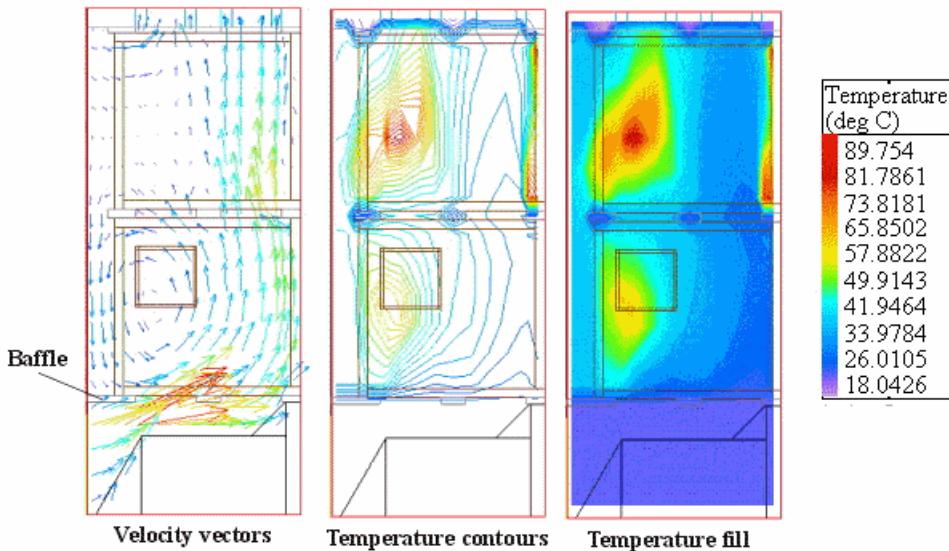
By increasing the size of the inlet vent and re-solving, a better heat distribution is obtained as shown below.

**Figure 1-7. Increased Size Inlet Vent**



Simcenter Flotherm quickly demonstrate retrograde steps. For example, adding a baffle just above the inlet vent to prevent the incoming air flowing up the cabinet in front of the racks changes the airflow to produce a stagnant region near the front of the lower rack of boards.

**Figure 1-8. Stagnant Region Near the Front of the Lower Rack of Boards**



Therefore, repeated boundary modification and re-solving can obtain the optimum design from a thermal perspective.

After building the basic model, you can use design optimization techniques in the Command Center to create different design scenarios.

## Related Topics

[Viewing Results](#)

[Reporting Project Data and Results in Tables](#)

[Design Optimization \[Simcenter Flotherm Command Center User Guide\]](#)

# Product GUI

To access: Open the product.

Simcenter Flotherm is a multi-application product, comprising the Project Manager, Command Center, MCAD Bridge, and EDA Bridge.

## Objects

- Project Manager window

This window forms the core of the program. The Project Manager shows all the information used within the model and to create geometry. The geometry is displayed as a hierarchical tree.

- Drawing Board or Graphics Display Area (GDA) pane

A detachable pane displaying the geometry as 3D objects when in Create mode.

In Analyze mode, planes of results (contours or vectors) and particle animation can be displayed.

- Profiles pane

A detachable pane showing the progress of the solution and how close to the final acceptable result the solution is.

- Messages pane

A detachable pane showing information, warning, and error messages.

- Tables pane

A detachable pane displaying results in numerical form.

- Command Center window

Use this window to optimize the model and to control the development of further investigations to study the effect of changing various parameters. The application utilizes optimization software from the Center for Qualitative Methods, CQM BV.

- MCAD Bridge window

Import information from CAD packages using via this window. This application takes the CAD data, removes unnecessary detail, breaks it up into Simcenter Flotherm objects and transfers it into the model. Some additional work is then required to specify data required to fully describe the model over and above the geometry imported. This method can be used to create whole models or just parts.

- EDA Bridge window

Construct detailed representations of PCBs using this window. PCBs can be constructed from new geometry, loaded from existing geometry or imported from EDA interface tools.



# Chapter 2

## Defining the Requirements

---

The first step when creating a project in Simcenter Flotherm is to define the requirements. You need to determine what is to be achieved and identify the key features to be modeled.

<b>Establishing the Requirements .....</b>	<b>33</b>
<b>Start Simple .....</b>	<b>33</b>
<b>Main Features to Consider When Modeling.....</b>	<b>33</b>

## Establishing the Requirements

Take the time to assess exactly what to model before starting.

A sketch that shows the major features and dimensions will help to start the case. This will allow you to decide how much needs to be included and also to orient yourself with the Simcenter Flotherm axis set.

At the outset identify the important key features of the model. It is in these areas that you may have to concentrate more grid cells to increase the resolution of the results. Also place monitor points into key areas. These allow the variables to be tracked at particular locations during the solution process. This will quickly give feedback as to whether the results are sensible and increase confidence when leaving a simulation running for an extended period.

## Start Simple

It is recommended that you start with a simple model and add complexity in stages.

Add project notes to track changes made.

By adding complexity in stages it is much easier to backtrack when there is a problem with the solution.

Only include features that are important to the model in question. Over-complication of the geometry will increase the solution time, and such items as fillets in the corners of a box do not have any effect on the thermal calculations.

## Main Features to Consider When Modeling

Important aspects of your model to consider before and during modeling.

## Extent of the Solution Domain

The solution of the variables is calculated within a cube of fluid known as the solution domain. The size of this will depend on the case in question. Another consideration in setting the solution domain will be whether only the inside of a box is required or whether some of the environment around any box needs to be included. Indeed, it is not even necessary to model the whole of the box.

## Component or Board

Modeling a component or board will not generally require the model to be extended to a solid box surface. This type of simulation may well be investigating the methods of local heat removal and, therefore, a comparison can be made by siting the objects suspended in free air or by defining a particular airflow over the objects in question.

## Computer or Electronics Module

When modeling a whole system such as a computer or electronics module the casing itself will form an important part of the model and will need to be represented. Often the solution domain will end at the casing and you will need to define the outside temperature and how the heat is transmitted from the exterior of the box – ambient setting. This is normally acceptable for cases that have forced ventilation or are sited away from significant obstructions.

## Heat Transfer on the Sides of the Box

If the outside of the box is designed to aid the heat transfer processes (for example fins) then it is difficult to define a heat transfer coefficient and the structure will have to be modeled explicitly. Hence the solution domain will have to be extended beyond the box.

## Objects Near the Box

If there are local objects near the box in question, then the solution domain will need to be extended to include these. A good example of this would be tabletop electronics with vents in the bottom. Here the gap between the bottom of the box and the table will have a significant effect on airflow within the system and will have to be included in the solution domain.

## Well Structured Data Tree

As well as considering what to model, it is also worth thinking at an early stage of how to structure the data tree. By starting with a well laid out structure the project will be much more manageable as it grows. This will again be of benefit in tracing the source of any problems. It will also allow the easy transfer of sub-assemblies between the project and libraries, making it easy to store often used components for use in future projects.

# Chapter 3

## Working on a Project

---

General information, such as how to run the product in interactive and batch modes, project operations such as loading, importing, and exporting projects and descriptions of general dialog boxes. Descriptions of elements of the Project Manager window, and data tree operations.

<b>Project Overview .....</b>	<b>36</b>
<b>Starting and Running the Product.....</b>	<b>38</b>
Starting and Running in Interactive Mode .....	38
Starting From a Command Window.....	38
Exiting .....	38
<b>FloSCRIPT .....</b>	<b>40</b>
About FloSCRIPT .....	40
Recording a Sequence of Operations to a FloSCRIPT File .....	42
Running a FloSCRIPT File .....	43
<b>Project Manager Window.....</b>	<b>44</b>
Window Layout .....	45
Property Sheets .....	46
Data Tree.....	47
New Object Palette .....	49
Project Attributes/Library Trees .....	53
Project Manager Status Bar .....	54
Workplane.....	56
Snap Grid .....	57
<b>Project Operations.....</b>	<b>59</b>
Creating a New Project .....	59
Setting the Default Project .....	60
Installing the Application Examples.....	60
Loading Projects .....	62
Changing the Solution Directory .....	63
Cataloging Projects.....	64
Importing External Projects .....	65
Saving .....	68
Exporting Projects in SAT or IGS Files .....	70
Exporting Thermal Netlists and BCI-ROM Files.....	70
Archiving .....	76
Deleting Projects.....	78
<b>Project Manager Operations .....</b>	<b>80</b>
Showing and Hiding Toolbars .....	80
Detaching/Reattaching the Drawing Board .....	81

Detaching/Reattaching the Profiles Window .....	82
Changing Global Units .....	83
Changing Local Units .....	83
Undoing and Redoing the Last Operation .....	84
Changing the Coordinate System .....	85
Using Parallel Processing .....	85
<b>Data Tree Operations .....</b>	<b>87</b>
Viewing Summary Information .....	87
Adding Assemblies .....	89
Adding Parts .....	90
Colocating Objects .....	90
Selecting Objects in the Data Tree .....	91
Promoting and Demoting Geometry in the Data Tree .....	92
Topping Assemblies or Parts .....	92
Hiding Objects From the Data Tree (Lightweight View of Assemblies) .....	93
Simple Searches for Objects or Attributes .....	93
Complex Searches for Objects or Attributes .....	94
Applying Simultaneous Changes to More Than One Object .....	96
Creating Unique Names for Objects .....	97
<b>Zoom-In Projects .....</b>	<b>98</b>
Zoom-in Project Example .....	98
Creating a Zoom-In Project .....	98
Determining the Coupling Metric .....	103
<b>Project Manager and General Dialog Boxes .....</b>	<b>104</b>
Base Project Changed Dialog Box .....	105
BCI-ROM Preferences Dialog Box .....	106
Create Category Dialog Box .....	108
Find Dialog Box .....	109
Global Units Dialog Box .....	115
License Timeout Dialog Box .....	116
Load Project Dialog Box .....	117
Message Window .....	119
New Project Dialog Box .....	120
Notes Dialog Box .....	122
Save Project Dialog Box .....	123
Thermal Netlist Preferences Dialog Box .....	125
Thermal Netlist/BCI-ROM Export Progress Dialog Box .....	126
User Preferences Dialog Box .....	127
Zoom-In Creation Dialog Box .....	136

## Project Overview

Each study is encapsulated within a unique Simcenter Flotherm project.

All data relating to the study is stored in a project directory, created by Simcenter Flotherm.

## Directory Structure

You do not need to know the directory structure to run Simcenter Flotherm, however, it is described here in case you decide to maintain different data areas for different types of projects or for individual users.

Default installations create an *MentorMA* directory in Program Files (on Microsoft® Windows®) or in */opt/* on UNIX. The location of the Simcenter Flotherm projects directory, known as the solution directory, defaults to *flosuite\_v<version>\flotherm\fouser*, but you can change this to a different directory when saving your project. On multi-user systems, separate solution directories are useful for designating different user areas.

Within the projects solution directory, each project has its own directory named after the project, but with a numeric extension. Within each project directory, the separate data types are held in subdirectories.

The following files and directories are the only ones that you might need to access manually:

- The project solution directory, if *fouser* is relocated.
- *group.bak*, a backup of the project file, held in the PDProject directory. Replace the *group* file with *group.bak* if you need to restore a project.
- Library files of geometry and attributes in the *Libraries* directory.
- *floqueue*, for setting up a remote solution environment.

Do not edit the *.LMCache* and *.LMRecent* directories, which are for system files required by the Library Manager.

## Working With Project Files

To work on a project within Simcenter Flotherm, you either create a new project or load an existing one.

The solver will not run until the project is saved.

After a project is saved, you can be reload it and, if required, export it for archive and for use in other Simcenter Flotherm, MCAD, or ECAD systems.

# Starting and Running the Product

Simcenter Flotherm can started from the GUI or from a command line.

Starting and Running in Interactive Mode .....	38
Starting From a Command Window .....	38
Exiting .....	38

## Starting and Running in Interactive Mode

Running interactively enables setup, flow calculation, and results analysis to be completed in the same program session.

### Restrictions and Limitations

- Simcenter Flotherm can only run in interactive (GUI) mode on Windows platforms.

### Procedure

1. To start the GUI from the task bar, choose **Start > All apps > MentorMA > Simcenter Flotherm <version> > Simcenter Flotherm <version>**.
2. (Optional) Create a shortcut to the Simcenter Flotherm batch file and use that:  
`<install_dir>\flosuite_v<version>\flotherm\WinXP\bin\flotherm.bat`  
where `<install_dir>` is the directory in which you have installed Simcenter Flotherm.

## Starting From a Command Window

You can start the product with a specified project from the command window.

### Procedure

1. To start a command window from the task bar, choose **Start > All apps > MentorMA > Simcenter Flotherm <version> > Simcenter Flotherm <version> Environment Shell**.
2. Run the flotherm command with the -p option.

### Related Topics

[flotherm](#)

## Exiting

Exiting is straightforward if the solver is not running and the currently open project is saved.

## Restrictions and Limitations

- If you attempt to exit Simcenter Flotherm while either the exchange factor calculation or the solver is running, you are prompted to confirm or cancel the request. If you choose **No** to cancel the exit, Simcenter Flotherm does not exit, but any open dialog boxes are closed. If you choose **Yes** to continue, the solution is interrupted and Simcenter Flotherm exits.

## Procedure

1. Choose **Project > Quit**.

Simcenter Flotherm checks for project changes before exiting the program.

2. If changes are detected, choose one of the following options:

- Click **Save** to save the project and solution data before exiting.  
If saving a new project, the [Save Project Dialog Box](#) enables you can give it a name, title, and class. An existing project is overwritten.
- Click **Discard** to not save the project before exiting, but back up and remove the solution data before exiting. The backed up results are used to interpolate from later.
- Click **Cancel** to cancel the exit request.

# FloSCRIPT

---

You can use XML files as script files to run a sequence of Simcenter Flotherm operations.

---

## Note

---

 In EDA Bridge, not all actions are supported by FloSCRIPT. See [FloSCRIPT](#) in the *Simcenter Flotherm EDA Bridge User Guide*.

---

<b>About FloSCRIPT .....</b>	<b>40</b>
<b>Recording a Sequence of Operations to a FloSCRIPT File .....</b>	<b>42</b>
<b>Running a FloSCRIPT File.....</b>	<b>43</b>

## About FloSCRIPT

Successful usage of FloSCRIPT relies on familiarity and experience with the Simcenter Flotherm implementation of scripting XML.

To help you gain experience of using FloSCRIPT, a tutorial and some demonstration examples are provided in the installation under the folder:

`<install_dir>\flosuite_v<version>\flotherm\examples\FloSCRIPT`

In addition, you are encouraged to read the XML log files to learn what can be achieved, starting with simple operations before learning more complexity. For example, the following sequence adds a cuboid to the Root Assembly:

```
<create_geometry geometry_type="cuboid">
    <source_geometry>
        <geometry name="Root Assembly"/>
    </source_geometry>
</create_geometry>
```

After you have written a script sequence you can use the schema to check its syntax.

Alternatively, you could load the FloSCRIPT schema into an XML tool which will validate on-the-fly as you write your scripts.

## Path References to Files

You can reference external files (CSV, PDML, MCAD parts/assemblies) by either relative or absolute paths.

When referencing by a relative path, the current folder can be referenced by the single-dot shortcut `(.)`, and the parent folder can be referenced by the double-dot shortcut `(..)`.

The relative path is relative to the location of the FloSCRIPT file.

Whether an absolute or relative path is recorded in a script depends on the circumstances of the recording:

- When recording a script and manually importing a file by navigating to a folder, then there is no concept of a relative path and the absolute path will be recorded.
- When recording a script during which another script is running which imports or exports a file, then the path used in the running script, whether relative or absolute, is maintained in the recording.

When a relative path is recorded in an exported file, then the same single-dot (.) and double-dot (..) shortcuts are used as described above.

#### Example of Relative Path Specification Using the Double-Dot Shortcut

The following example uses an absolute path to specify a STEP file that is to be imported:

```
<launch launch_type="mcad"/>
  <external_command process="MCAD">
    <import filename=
      "D:/floscript_files/import_files/step_files/my_cad_file.STEP"
      import_type="step"/>
  </external_command>
```

If the FloSCRIPT file is located at *D:/floscript\_files/scripts/my\_scripts*, then the import can be specified relatively by the following:

```
<import filename= "../..../import_files/step_files/my_cad_file.STEP"
  import_type="step"/>
```

#### Example of Relative Path Specification Using the Single-Dot Shortcut

The following example uses an absolute path to specify a STEP file that is to be imported:

```
<launch launch_type="mcad"/>
  <external_command process="MCAD">
    <import filename=
      "D:/floscript_files/import_files/my_cad_file.STEP" import_type="step"/>
  </external_command>
```

If the FloSCRIPT file is located at *D:/floscript\_files*, then the import can be specified relatively by the following:

```
<import filename= "./import_files/my_cad_file.STEP" import_type="step"/>
```

## FloSCRIPT Schema

The FloSCRIPT Schema, *FloSCRIPTSchema.xsd*, is provided in:

```
<install_dir>|flosuite_v<version>|flotherm\examples\FloSCRIPT\Schema
```

## FloSCRIPT Log Files

XML script files are written for each session in the folder:

`<install_dir>\flosuite_v<version>\flotherm\WinXP\bin\LogFiles`

Each file is named with a unique number:

- Simcenter Flotherm log files are named `logFile<number>.xml`.
- Command Center log files are named `CCLogFile<number>.xml`.
- MCAD Bridge log files are named `MCADLogFile<number>.xml`.
- EDA Bridge log files are named `EDALogFile<number>.xml`.

Log files can be run as FloSCRIPT files from the respective application window.

---

### Note

 Only the last five log files are retained when a new application window session is started. You can retain log files between sessions, however, you must prefix the filename so that it does not begin with the (case-insensitive) “`logFile`”, “`CCLogFile`”, “`MCADLogFile`”, or “`EDALogFile`” string.

---

The log files are identified internally by a version number. Running a “future” file version is not allowed.

## Related Topics

[Recording a Sequence of Operations to a FloSCRIPT File](#)

[Running a FloSCRIPT File](#)

# Recording a Sequence of Operations to a FloSCRIPT File

FloSCRIPT files are XML script files that can record a sequence of user operations.

## Restrictions and Limitations

- Restricted to the Project Manager, Command Center, MCAD Bridge, and EDA Bridge applications.

## Procedure

1. In the relevant application window, choose **Macro > Record FloSCRIPT**.

The Record FloSCRIPT file browser opens. The default folder is

`<install_dir>\flosuite_<version>\flotherm\WinXP\bin`

2. Enter a filename for the FloSCRIPT file. Optionally, navigate to another folder. Click **Save**.
3. Start performing the operations you want to record.  
You can pause (**Macro > Pause FloSCRIPT**) and resume (**Macro > Resume FloSCRIPT**) recording at any time.
4. When finished, choose **Macro > Stop FloSCRIPT**.

## Related Topics

[Running a FloSCRIPT File](#)

[About FloSCRIPT](#)

# Running a FloSCRIPT File

FloSCRIPT files are XML script files that can run a sequence of operations.

## Restrictions and Limitations

- FloSCRIPT files are only applicable to one application: Project Manager, Command Center, MCAD Bridge, or EDA Bridge. If you try to run a file in a different application from which it was recorded, then an error message is generated.

## Procedure

1. Choose **Macro > Play FloSCRIPT**.

The Play FloSCRIPT file browser opens. By default only \*.xml files are listed.

2. Navigate to the FloSCRIPT file and click **Open**.



### Note

FloSCRIPT files can also be run from the command line flotherm command.

---

## Related Topics

[About FloSCRIPT](#)

[flotherm](#)

[Recording a Sequence of Operations to a FloSCRIPT File](#)

# Project Manager Window

Use the Project Manager to control and build your projects. You can create new projects or load existing ones as well as display and manipulate the project geometry data using a tree hierarchy. The Project Manager opens when Simcenter Flotherm starts and it is active for the entire program session.

<b>Window Layout .....</b>	<b>45</b>
<b>Property Sheets .....</b>	<b>46</b>
<b>Data Tree .....</b>	<b>47</b>
<b>New Object Palette.....</b>	<b>49</b>
<b>Project Attributes/Library Trees .....</b>	<b>53</b>
<b>Project Manager Status Bar .....</b>	<b>54</b>
<b>Workplane .....</b>	<b>56</b>
<b>Snap Grid .....</b>	<b>57</b>

# Window Layout

The Project Manager window is displayed after the splash screen closes.

## Description

### Note

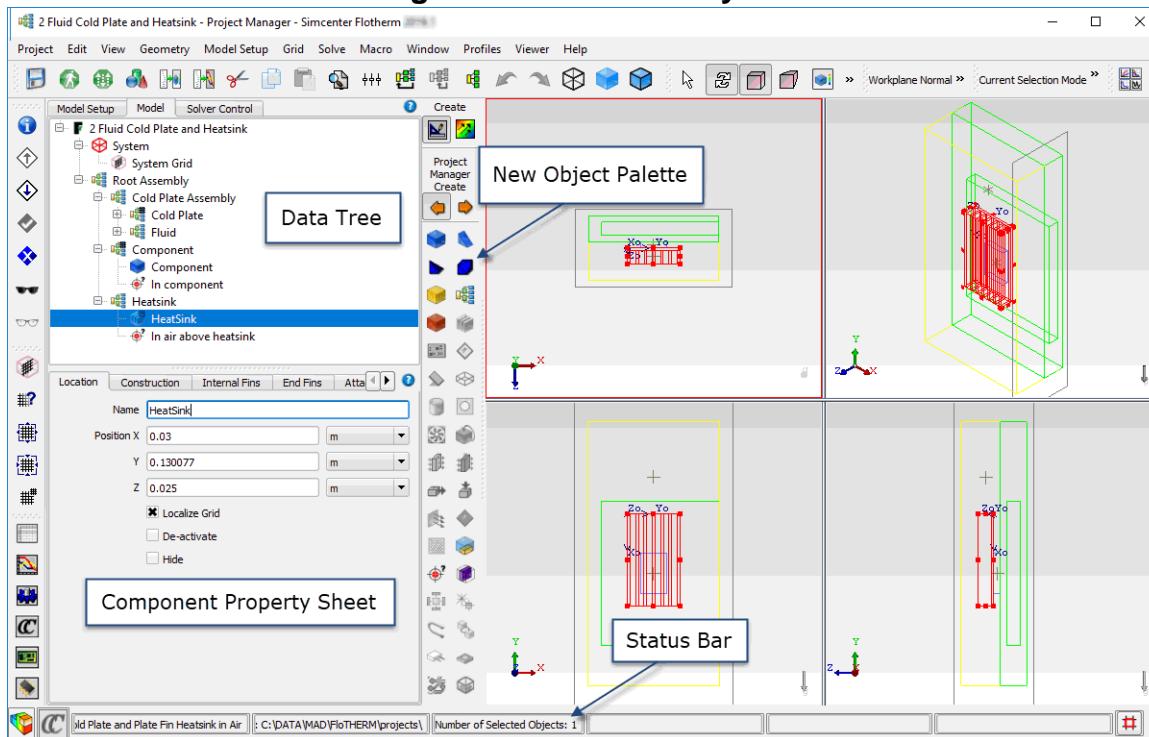
 The Graphics Display Area (GDA) serves as both a drawing board when the window is in Create mode, and as a results viewer when the window is in Analyze mode.

Figure 3-1 shows the Data Tree and GDA (Drawing Board) in Create mode. This is the layout when starting up for the first time, to maximize the Drawing Board area.

For a description of the Project Manager in Analyze mode, see “[Viewing Results](#)” on page 419.

The name of the project is shown in the top title bar and is the name of the top node in the data tree. When Simcenter Flotherm starts, it displays the default project.

**Figure 3-1. Window Layout**



## Objects

- Data Tree

Displays the geometry hierarchical structure. Refer to “[Data Tree](#)” on page 47.

- New Object Palette

For selection and addition of geometry templates to your model. Refer to “[New Object Palette](#)” on page 49.

- Drawing Board

Displays the model in 2D or 3D. Refer to “[The Drawing Board](#)” on page 254.

- Project Attributes/Library Tabs

Project Attributes are used to define non-geometric properties which can be added (attached) to objects. The Library provides a resource of objects. The visibility of these tabs is remembered between sessions. Press F7 if not visible. (Not shown in [Figure 3-1](#).) Refer to “[Project Attributes/Library Trees](#)” on page 53.

- Property Sheet

Defines the properties of selected objects and attributes. Refer to “[Property Sheets](#)” on page 46.

- Message Window

Displays information, warning, and error messages. (Not shown in [Figure 3-1](#).) Refer to “[Message Window](#)” on page 119.

- Grid Summary Dialog

Displays general grid information. (Not shown in [Figure 3-1](#).) Refer to “[Grid Summary Dialog](#)” on page 330.

- Status Bar

Displays project summary information. Refer to “[Project Manager Status Bar](#)” on page 54.

## Related Topics

[Detaching/Reattaching the Drawing Board](#)

# Property Sheets

Property sheets are the standard user interface for specifying values and parameters.

A property sheet is opened when you select an object in the data tree or a project attribute. Most of the object property sheets have more than one tab.

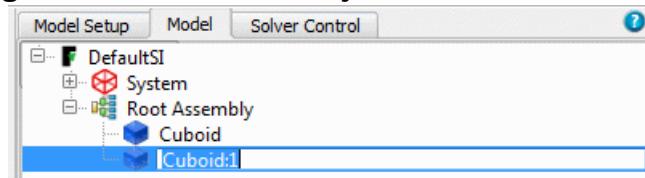
## Name Fields

Up to 32 characters may be entered in Name fields.

New geometry objects are supplied with a default name corresponding to the object type and if objects of the type already exist, ‘:<n>’ is appended to the name, where <n> is a sequence number, for example, Cuboid:1.

The name of an object can also be changed by double-clicking the object in the data tree, which puts the name into edit mode.

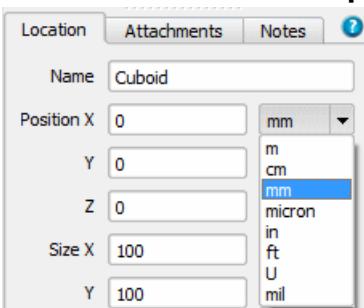
**Figure 3-2. Data Tree Object Name in Edit Mode**



## Units Dropdown List

The Units dropdown list enables you to override the default settings for the current property sheet.

**Figure 3-3. Dimension Units Dropdown List**



The values do not change, they are displayed in different units. For example, 0.1 m is displayed as 10 cm if the units are changed from meters to centimeters.

## Data Tree

Simcenter Flotherm uses assemblies and parts to organize its geometry data. These assemblies and parts are displayed in a data tree, under the **Model** tab, which can be expanded or contracted (Reset) to show or hide the hierarchical structure of the project geometry.

All projects have these characteristics:

- Project node, which is the top node and is used for project identification and importing/exporting projects,
- System node, for defining the solution domain, and
- Top-level Root Assembly node, which holds the geometry.

When Simcenter Flotherm is started, the data tree shows only the top-level geometry.

### Data Tree Icons

The following icons are used to identify property sheet options in the **Location** tab that have been applied to data tree objects. The examples shown are for assemblies.

- Localized grid applied — 
- De-activated object — 
- Hidden object — 
- Ignored geometry in assembly — 

---

#### Note

 This icon is only visible when Show Ignored Geometry is checked in the **Project Manager** tab of the User Preferences dialog box, otherwise the assembly is not shown in the data tree.

---

- Lightweight assembly — 

### Related Topics

[Data Tree Operations](#)

[Generic Property Sheet Location Tab \[Simcenter Flotherm SmartParts Reference Guide\]](#)

# New Object Palette

To access: The New Object Palette is a fixed vertical toolbar located between the data tree and drawing board.

Use the palette to add objects to the data tree and drawing board.

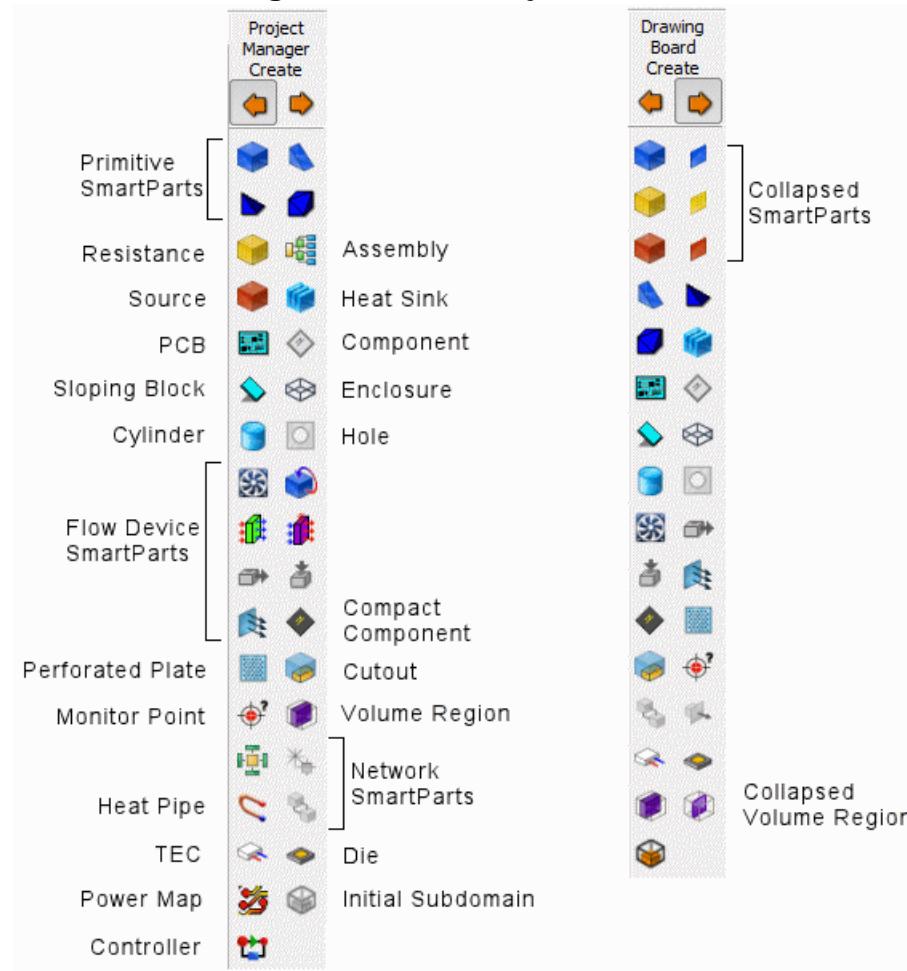
## Description

The palette provides templates for all of the available geometry objects.

- Project Manager Create
- Drawing Board Create

Hover text identifies each icon.

**Figure 3-4. New Object Palette**



Icons are dimmed when the object cannot be added. Addition is context-sensitive to the currently selected object.

## Objects

Object	Child Objects
Cuboid	A cuboid, prism, tet, inverted tet, resistance, assembly, source, recirculation device, cutout, and volume region can be added colocated with, and as a sibling to, the selected cuboid. Monitor points are added as siblings, but located at the center of the cuboid.  A hole can be added as a child to the selected cuboid.
Prism	As for cuboid, except recirculation devices and holes.
Tet	As for cuboid, except recirculation devices and holes.
Inverted Tet	As for cuboid, except recirculation devices and holes.
Resistance	As for cuboid, except recirculation devices and holes.
Assembly	Any object except a hole, supply, extract, component, network node, or network cuboid.
Source	As for cuboid, except recirculation devices and holes.
Heat Sink	As for cuboid, except recirculation devices and holes.
PCB	As for cuboid, except recirculation devices and holes, but including components.
PCB Component	As for cuboid, except recirculation devices and holes.
Sloping Block	As for cuboid, except recirculation devices and holes.
Enclosure	As for cuboid, except recirculation devices. Holes are only added when an enclosure side is selected.
Cylinder	As for cuboid, except recirculation devices and holes.
Hole	As for cuboid, except recirculation devices and holes cannot be added and the new object is added as a sibling to the parent of the hole.
Fan	As for cuboid, except recirculation devices and holes.
Recirculation Device	Supplies and extracts can be added as children.  Cuboids, prisms, tets, inverted tets, resistances, assemblies, sources, cutouts, and volume regions can be added colocated with, and as a sibling to the selected recirculation device. Monitor points can be added as siblings, but located at the center of the recirculation device.
Supply	Additional objects are as for recirculation devices, coolers, and racks, except additional supplies or extracts are not available.  Monitor points are located at the center of the elected supply.
Extract	As for Supply.
Fixed Flow	As for cuboid, except recirculation devices and holes.

Object	Child Objects
Compact Component	As for cuboid, except recirculation devices and holes.
Perforated Plate	As for cuboid, except recirculation devices and holes.
Cutout	As for cuboid, except recirculation devices and holes. All other new objects are added as siblings to the side's parent enclosure.
Monitor Point	As for cuboid, except recirculation devices and holes.
Volume Region	As for cuboid, except recirculation devices and holes.
Network Assembly	Only network nodes and monitor points.
Network Node	Only network cuboids and monitor points.
Heat Pipe	Only network cuboids and monitor points.
Network Cuboid	Monitor points are added as siblings, located at the center of the network cuboid.
TEC (Thermoelectric Cooler)	As for cuboid, except recirculation devices and holes.
Die	As for cuboid, except recirculation devices and holes.
Cooler	As for recirculation device.
Rack	As for recirculation device.
Power Map, Power Map Layer, Power Map Via	As for cuboid, except recirculation devices and holes. Objects match the bounding box dimensions of the selected Power Map object. Monitor points are located at the center of the selected object.
Controller	Only a source and a monitor point.

Object	Default Size, based on a 1.0 m × 1.0 m × 1.0 m solution domain
Assembly	Empty assembly
Compact Component	0.1 m × 0.1 m × 0.1 m. No power is attached to the component
Cuboid	0.1 m × 0.1 m × 0.1 m
Cutout	0.1 m × 0.1 m × 0.1 m
Cylinder	12-facet cylinder comprising both cuboids and prisms with radius of 0.05 m and length 0.1 m
Enclosure	Enclosure occupying a 1 m × 1 m × 1 m space, with thin walls 0.001 m thick

<b>Object</b>	<b>Default Size, based on a <math>1.0\text{ m} \times 1.0\text{ m} \times 1.0\text{ m}</math> solution domain</b>
Fan	3D, 12-facet fixed flow axial fan with hub diameter of 0.008 m, outer diameter of 0.1 m, depth of 0.1 m, and a fixed flow rate of $0.008\text{ m}^3\text{s}^{-1}$
Fixed Flow	Inflow of zero volume flow-rate covering an area of $0.1\text{ m} \times 0.1\text{ m}$
Heat Sink	$0.1\text{ m} \times 0.1\text{ m} \times 0.005\text{ m}$ with 3 internal fins 0.095 m long and uniform style
Hole	$0.1\text{ m} \times 0.1\text{ m}$ open space
Inverted Tet	0.1 m lengths in the 3 coordinate directions
Monitor Point	Cross-hair shaped monitor point
PCB	$0.1\text{ m} \times 0.1\text{ m}$ non-conducting PCB
PCB Component	One-tenth the X-Y size of the parent PCB
Perforated Plate	$0.1\text{ m} \times 0.1\text{ m}$ with square holes 0.005 m pitched $0.01\text{ m} \times 0.01\text{ m}$
Prism	0.1 m lengths in the 3 coordinate directions
Recirculation Device	0.1 m square supply in the z-plane at the reference origin, and a 0.1 m square extract in the z-plane at $z = 0.1\text{ m}$ from the reference origin with a flow rate of $0.008\text{ m}^3\text{s}^{-1}$ . The default size of both the child nodes (extract and supply) are one-tenth of the solution domain X and Y sizes.
Resistance	$0.1\text{ m} \times 0.1\text{ m} \times 0.1\text{ m}$ volume resistance
Sloping Block	$0.1\text{ m} \times 0.1414214\text{ m}$ angled at 45 degrees
Tet	0.1 m lengths in the 3 coordinate directions
Source	$0.1\text{ m} \times 0.1\text{ m} \times 0.1\text{ m}$ volume source
Volume Region	$0.1\text{ m} \times 0.1\text{ m} \times 0.1\text{ m}$
Cooler	0.1 m square supply in the z-plane at the reference origin, and a 0.1 m square extract in the z-plane at $z = 0.1\text{ m}$ from the reference origin with a flow rate of 2,900 cfm. The default size of both the child nodes (extract and supply) are one-tenth of the solution domain X and Y sizes.
Rack	0.1 m square supply in the z-plane at the reference origin, and a 0.1 m square extract in the z-plane at $z = 0.1\text{ m}$ from the reference origin with a flow rate of 480 cfm. The default size of both the child nodes (extract and supply) are one-tenth of the solution domain X and Y sizes.
Network Assembly	Empty network assembly
Network Node	Empty node

Object	Default Size, based on a 1.0 m × 1.0 m × 1.0 m solution domain
Network Cuboid	0.1 m × 0.1 m × 0.1 m block
Heat Pipe	Empty heat pipe assembly
TEC	40 mm × 40 mm × 47 mm and a ceramic thickness of 0.75 mm
Die	4.9 mm × 4.9 mm × 0.22 mm
Controller	Empty controller assembly

## Usage Notes

The listing order in the data tree reflects the setup sequence, that is, items at the top of the tree are considered created first. See “[Construction Precedence Rules](#)” on page 218 for coincident geometry.

Generally, when a new object is added to an assembly, the default size is one-tenth of the solution domain. One major exception is the enclosure, which is sized to fit the solution domain with the wall thickness one-tenth of the smallest dimension.

The new object is edited to suit the model using property sheets. You can also use the drawing board to change the geometry size and location by dragging boundaries.

In addition to creating new objects from templates, you can add geometry to the data tree using the library and importing files in various formats from the file system.

Refer to the [Simcenter Flotherm SmartParts Reference Guide](#).

## Related Topics

[Libraries](#)

[Geometry Files](#)

## Project Attributes/Library Trees

The Project Attributes/Library trees are on the right of the drawing board. The quickest way to display or hide these trees is to press F7. Each tree is displayed in a separate tab.

Attributes are used to define non-geometric properties which can be added (attached) to objects. The Project (Attributes) tree lists all attributes that are currently used on the project.

Refer to the [Simcenter Flotherm Project Attributes Reference Guide](#).

The Library tree initially lists the start-up libraries supplied with the program, but these can be added to, see “[Libraries](#)” on page 285.

## Project Manager Status Bar

To access: Located at the bottom of the Project Manager window.

Use the status bar to see summary information about the currently loaded project.

### Objects

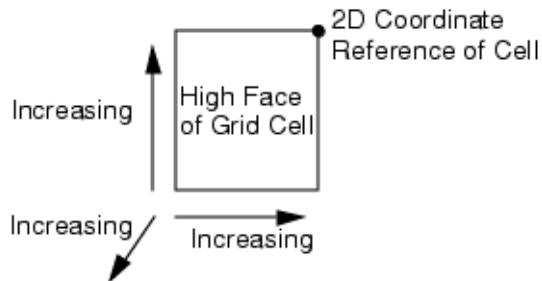
Object	Description
<b>Results icon</b>	 Indicates that no solution results are available for the project.
	 Indicates that solution results are available.
<b>Command Center icon</b>	 Indicates that Command Center input variables have not been defined.
	 Indicates that Command Center input variables have been defined.
Project Title	The project title as defined in the Save Project dialog box when the project was last saved.
Solution Directory	The solution directory as defined in the Save Project dialog box when the project was last saved.
Number of Objects or Number of Selected Objects	<ul style="list-style-type: none"><li>The Number of Objects is shown when nothing is selected. This is the total number of objects in the project, including the Root Assembly.</li><li>The Number of Selected Objects is shown when objects are selected.</li></ul>
The Status Bar is separated here when the Drawing Board (GDA) is detached from the Simcenter Floterm window.	
Total Number of Grid Cells <sup>11</sup>	The total number of grid cells for the base grid. The number of cells affects performance in terms of the time it takes to solve the model and the quantity of results data.
Maximum Aspect Ratio <sup>1</sup>	Grid cells with large aspect ratios can cause slow or sometimes divergent solver behavior. For further information and guidelines, see “ <a href="#">Maximum Aspect Ratios</a> ” on page 306.

Object	Description
Cursor At or Grid Cell High Face	<ul style="list-style-type: none"> <li>The Cursor At position is shown when the solution grid is switched off.</li> <li>The coordinates give the cursor position in the drawing board.</li> <li>The Grid Cell High Face is shown when the solution grid is on.</li> </ul> <p>The coordinates give the position of the “furthest” corner of the hovered-over grid cell when looking at the high face, see <a href="#">Figure 3-5</a>. These coordinates are the same as those used to identify grid cells in tables when in Analyze mode.</p>
<b>Maximum Aspect Ratio Highlight button</b> 	<p>Highlights, in red, the cell with the maximum aspect ratio.</p> <p>To unhighlight, Ctrl+click in the cell.</p>

1. Shown when the solution grid is switched or when the Grid Summary dialog is displayed. In addition, the grid will be re-calculated when the data tree is displaying summary information, to ensure that De-keypointed flags are up-to-date.

## Usage Notes

**Figure 3-5. Grid Cell High Face**



## Related Topics

[Save Project Dialog Box](#)

[Spatial Solution Grid](#)

## Workplane

To access: **Viewer > Show Grid Toggle** (toggle operation). Alternatively, press G.

Use the workplane to manipulate the geometry and display a solution or snap grid.

### Description

The workplane is a 2D reference plane, see [Figure 3-6](#).

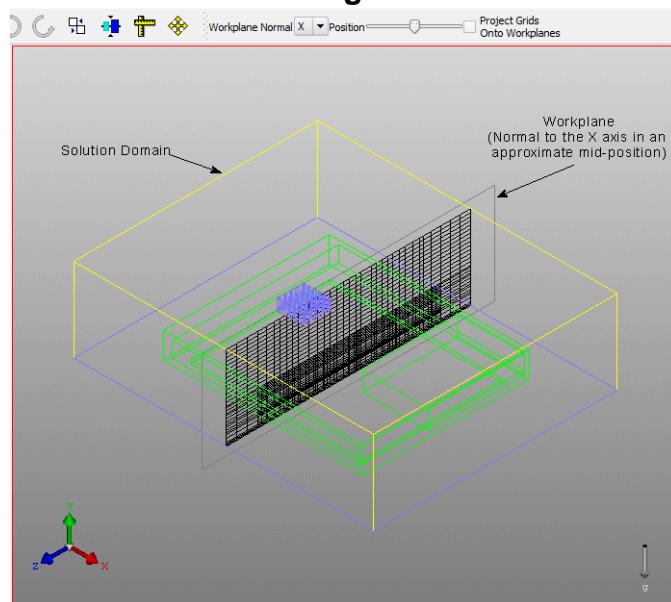
For clarity, a boundary can be shown extending beyond that of the solution domain. The visibility of this boundary is controlled by the **Display Workplanes** check box in the **Drawing Board** tab of the User Preferences dialog box.

The workplane shows the Snap Grid when the Snap Toggle (Drawing Board toolbar) is set to **Snap Grid is On**.

The workplane shows the Solution Grid when the Snap Toggle is set to either **Snap to Object is On** or **Snap is Off**.

The Snap Grid and Solution Grid are never displayed at the same time.

**Figure 3-6. Isometric View Showing Solution Domain and Workplane**



### Objects

Object	Description
Snap Grid	A geometry positional tool, see “ <a href="#">Snap Grid</a> ” on page 57. The display interval of the grid as shown on the workplane is defined by the <b>Drawing Board</b> tab of the User Preferences dialog box.

Object	Description
Solution Grid	The subdivision of the solution domain into finite elements, see “ <a href="#">Spatial Solution Grid</a> ” on page 303.

## Usage Notes

Geometry can be drawn, moved, or resized by moving the projections of the geometry onto the workplane.

Use the Work Planes toolbar to define the axis of the workplane and its position along that axis. Select Project Grids Onto Workplanes in the toolbar to project the grid onto the workplane rather than only show the grid at the workplane position. When projected, all grid lines that exist parallel to the workplane are shown projected onto the workplane.

## Related Topics

[Controls for Viewing the Spatial Solution Grid](#)

[Mirroring an Object](#)

## Snap Grid

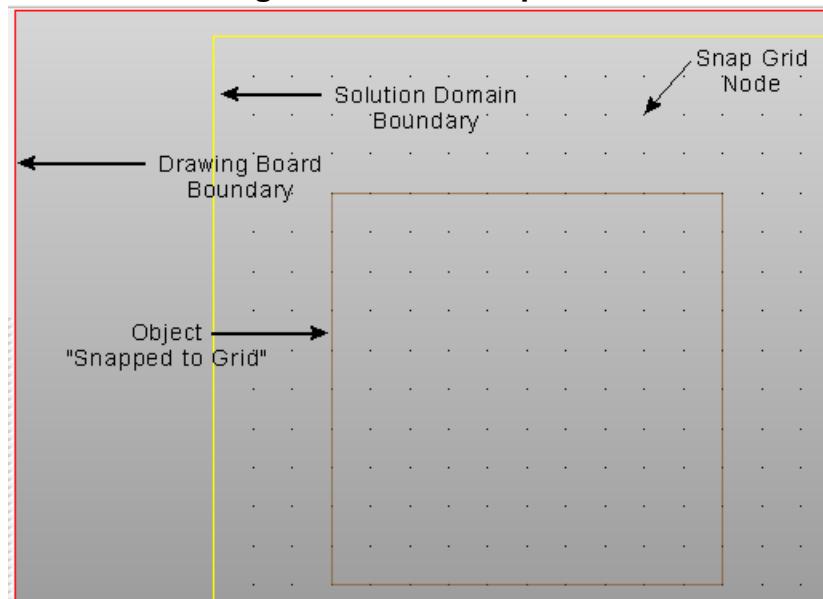
A grid of positions to which geometry can be restricted.

When the Snap Grid is active, geometry which is moved or created on the drawing board will “snap” to the nearest grid line.

The Snap Grid size is defined by the **Drawing Board** tab of the User Preferences dialog box. The Snap Grid is active when Snap to Grid is active (signified by the **Snap Toggle** icon), and is visible when Show Grid Toggle is on.

The appearance of the grid is a matrix of dots which mark the node positions of the grid, see [Figure 3-7](#). The snap grid can appear as a continuous black surface if it is very fine and you are zoomed out.

**Figure 3-7. The Snap Grid**



## Related Topics

[Drawing Board Icons](#)

[User Preferences Dialog Box - Drawing Board Tab](#)

# Project Operations

---

The following are operations carried out on the project as an entity, rather than modeling operations within an opened project.

<b>Creating a New Project .....</b>	<b>59</b>
<b>Setting the Default Project .....</b>	<b>60</b>
<b>Installing the Application Examples .....</b>	<b>60</b>
<b>Loading Projects .....</b>	<b>62</b>
<b>Changing the Solution Directory .....</b>	<b>63</b>
<b>Cataloging Projects .....</b>	<b>64</b>
<b>Importing External Projects.....</b>	<b>65</b>
<b>Saving .....</b>	<b>68</b>
<b>Exporting Projects in SAT or IGS Files .....</b>	<b>70</b>
<b>Exporting Thermal Netlists and BCI-ROM Files.....</b>	<b>70</b>
<b>Archiving .....</b>	<b>76</b>
<b>Deleting Projects .....</b>	<b>78</b>

## Creating a New Project

You can create an empty project or a model project from a template.

### Procedure

1. Press Ctrl+N to start a project from a template,

---

**Note**

 If there are any outstanding edits, the Simcenter Flotherm Save/Discard/Cancel dialog box opens. Click **Save** to keep the changes or **Discard** to delete them.

---

The New Project dialog box opens.

2. Select one of the Simcenter Flotherm templates and click **OK**.

### Results

The template is loaded into Simcenter Flotherm.

### Related Topics

[New Project Dialog Box](#)

[Saving a Project](#)

[Saving a Project as a Template](#)

## Setting the Default Project

The default project is the project that opens when you start Simcenter Flotherm.

### Procedure

1. Press Ctrl+N to open the New Project dialog box.
2. Select the template that you want to be the default project.
3. Click **Set Default**.
  - If you only want to set the default but do not want to replace the currently loaded project, click **Cancel**.
  - To replace the currently loaded project with the default, click **OK**.

### Results

The next time that you start up Simcenter Flotherm this template will be loaded by default.

### Related Topics

[New Project Dialog Box](#)

## Installing the Application Examples

Support Center contains application examples, which demonstrate Simcenter Flotherm modeling techniques. You can install these in your version of Simcenter Flotherm for reference or to use as the basis of your own models.

### Procedure

1. Exit Simcenter Flotherm.
2. Create a new directory named ‘Application Examples’ in the ‘Templates’ directory, where Simcenter Flotherm is installed.

For example:

*C:\Program Files\MentorMA\flosuite\_v<>\flotherm\flocentral\Templates*

---

#### **Tip**

 You can give the new directory that you create in step 2 a name of your choice.

The name you give becomes the tab name in the [New Project Dialog Box](#).

---

3. Download the application examples from the Related Downloads page under ‘Simcenter Flotherm Supporting Application Examples, Best Practice Guides and Tutorials’ in Support Center.

4. Extract the contents of the zip file and copy the *.pack* or *.pdml* files into the directory created in step 2.
5. Start Simcenter Flotherm.
6. Select **New** from the **Project** menu.

## Results

The [New Project Dialog Box](#) will include a tab named Application Examples (or whatever name you specified in step 2) which, when selected, will show the application example projects.

## Loading Projects

Projects can be loaded (that is, opened) as described in the following subsections.

<b>Loading an Existing Project .....</b>	<b>62</b>
<b>Owning a Project .....</b>	<b>62</b>

## Loading an Existing Project

You may need to update or just view the geometry and results generated in a previous project or program session. To access the information generated, you must load the project into program memory.

### Restrictions and Limitations

- If loading projects from other Mentor Graphics Mechanical Analysis products, first consult your local user support.
- The project must be unlocked.

Simcenter Flotherm locks projects when you are working on them to prevent them being changed by other users. If the system fails when working on a project it will still be locked when you attempt to load it again. You will be warned that it is locked and told to use **Unlock** to allow access to the project.

Before unlocking a project, check that you are not using it in another program session and, in a multi-user system, it is not being used by someone else.

### Procedure

1. If the project has been loaded recently, then it may be listed under the **Project** menu; otherwise press Ctrl+L to open the Load Project dialog box.  
If there are any outstanding edits that have been made to the current project, the Simcenter Flotherm Save/Discard/Cancel dialog box opens. Click **Save** to keep the changes or **Discard** to delete them.
2. The Load Project dialog box opens. Select a project and click **OK**.

### Related Topics

[Load Project Dialog Box](#)

## Owning a Project

Before you can load a project that is owned by someone else, you must become the owner of the project.

## Prerequisites

- Make sure Simcenter Flotherm is not running.

## Procedure

1. Copy the complete project directory into your solution directory.
2. To start the command window from the task bar, choose **Start > MentorMA > Simcenter Flotherm <version> > Simcenter Flotherm <version> Environment Shell**.
3. Go to the solution directory and run the floupdateall command as follows:

```
floupdateall -o -d .
```

The period identifies that the current directory is to be updated. The command makes you the owner of all the projects in the solution directory.

4. Close the command window and start up Simcenter Flotherm interactively.
5. Choose **Project > Load** to open the Load Project dialog box.
6. Ensure that the correct solution directory is chosen and click **Catalog** to rebuild the load list.

## Results

The updated projects will be listed for loading into Simcenter Flotherm.

# Changing the Solution Directory

The initial default setting of the Solution Directory is set up during the installation procedure.

## Restrictions and Limitations

- UNC paths are not supported.
- To use a solution directory, the user must have write permission for the directory, and the directory must not be in a system protected part of the file system.

## Procedure

1. Press Ctrl+L to open the Load Project dialog box.  
The folder path of the current Solution Directory is shown in the dialog box.
2. Click **Browse** to open the Project Solution Directory dialog box.
3. Navigate to the new solution directory and click **Select Folder**.  
The Solution Directory path is updated and any projects in that folder are displayed.
4. Click **OK** or **Cancel** to close the dialog box.

## Related Topics

[Load Project Dialog Box](#)

# Cataloging Projects

Catalog if there are projects copied or moved into the directory using the OS, but they are not listed in the Load Project dialog box.

## Procedure

1. Press Ctrl+L to open the Load Project dialog box.
2. Click **Catalog**.

## Results

All the Simcenter Flotherm projects held in the Solution Directory are listed in the Load Project dialog box.

## Related Topics

[Load Project Dialog Box](#)

[Changing the Solution Directory](#)

## Importing External Projects

Importing FloXML, PDML, and PACK files.

There is an option to import ASCII PDML although this is not required for the normal running of Simcenter Flotherm and is only issued on special license and does not normally appear in the **Import Project** menu. Contact your Simcenter Flotherm supplier if you need this format.

<b>Importing a FloXML Project .....</b>	<b>65</b>
<b>Importing a PDML Project.....</b>	<b>65</b>
<b>Importing a PACK Project.....</b>	<b>66</b>
<b>Importing a V1.4 Project.....</b>	<b>66</b>
<b>Importing a V2/V3 Project .....</b>	<b>67</b>

### Importing a FloXML Project

Project FloXML files are XML files that can be imported into Simcenter Flotherm as projects.

#### Restrictions and Limitations

- Project FloXML files must conform to the [FloXML Schema](#).
- You cannot import a Project FloXML file from a network directory using a UNC path. Either copy the file to a local directory or map the network folder containing that file.

#### Procedure

1. In the top of the data tree, right-click the project node and choose **Import Project > FloXML**.  
A file selection dialog box is displayed to find the required file.
2. Select the Project FloXML file that you want to import, then click **Open**.  
By default, the file browser lists all files.

#### Related Topics

[FloXML Files](#)

### Importing a PDML Project

PDML files provide a simple method of transferring a project from one Simcenter Flotherm installation to another.

#### Prerequisites

- A Simcenter Flotherm project exported in PDML format.

## Procedure

1. In the top of the data tree, right-click the project node and choose **Import Project > PDML**.

A file selection dialog box is displayed to find the required file.

2. Select the PDML file that you want to import, then click **Open**.

## Related Topics

[PDML Files](#)

[Archiving Project Data in PDML Format](#)

## Importing a PACK Project

PACK files include the project and any solution results.

## Procedure

1. In the top of the data tree, right-click the project node and choose **Import Project > Pack File**.

A file selection dialog box is displayed to find the required file. By default, the filter searches for \*.pack files.

2. Select the PACK file that you want to import, then click **Open**.
3. If a project of the same name as the PACK file exists in the solution directory, you will be given the option to overwrite the existing project.
  - Choose **Yes** to unpack and overwrite the existing project, which is then loaded into Simcenter Flotherm.
  - Choose **No** to ignore the import request.

## Related Topics

[Archiving Project Data in PACK Format](#)

## Importing a V1.4 Project

Version 1.4 files require a non-standard import treatment.

## Prerequisites

- A Simcenter Flotherm V1.4 \*.prb or \*.PRB file.

## Procedure

1. In the top of the data tree, right-click the project node and choose **Import Project > V1.4 File**.

A file selection dialog box is displayed to find the required file.

2. Select the V1.4 Project file that you want to import, then click **Open**.

## Importing a V2/V3 Project

Versions 2 and 3 files require a non-standard import treatment.

### Prerequisites

- A Simcenter Flotherm version 2.x or 3.x project file.

### Procedure

1. In the top of the data tree, right-click the project node and choose **Import Project > V2/V3 Project**.

A file selection dialog box is displayed to find the required file.

2. Select the V2/V3 Project file that you want to import, then click **Open**.

## Saving

Once projects created in previous versions of Simcenter Flotherm have been loaded and saved, they can no longer be run on the earlier version of Simcenter Flotherm.

<b>Saving a GDA Image to a Graphics File .....</b>	<b>68</b>
<b>Saving a Project .....</b>	<b>68</b>
<b>Saving a Project as a Template.....</b>	<b>69</b>

## Saving a GDA Image to a Graphics File

GDA images can be captured to JPEG or PNG files.

### Prerequisites

- You have set up your preferences for the appearance of the GDA image using the settings in the **Analyze Mode** tab of the User Preferences dialog box. Only the resolution settings are applicable when in Create mode.

### Procedure

1. If the GDA is divided into frames then select the frame you want to capture.
2. Click the Output Snapshot icon  or choose **Viewer > Output Snapshot**.  
The Output Snapshot dialog box is opened.
3. Navigate to a folder, enter a filename, select a file type, and click **Save**.

### Related Topics

[User Preferences Dialog Box - Analyze Mode Tab](#)

## Saving a Project

The Save Project dialog box is opened when you save a project for the first time.

### Procedure

1. Press Ctrl+S.
  - If it is the first time you have saved the project, then the Save Project dialog box is opened.
  - If it is not the first time you have saved the project, any changes are saved and there is no further action to be taken.
2. Use the Save Project dialog box to name the project, and to add a Title and any Notes that will help you identify the project when you next want to load it.

3. Optionally, change the Solution Directory.
4. Optionally, save the solution results.
5. Click **OK**.

You will be warned if you are attempting to overwrite an existing project.

## Results

When the project is saved, two project files are automatically created:

- *group* - the project data file, and
- *group.bak* - the project backup data file.

Both files are stored in the *PDProject* directory below the project solution directory.

If, for some reason, you need to restore your project, then copy the file *group.bak* over *group*.

## Related Topics

[Save Project Dialog Box](#)

# Saving a Project as a Template

Template projects can be used when creating new projects.

## Procedure

1. Choose **Project > Save As**.
2. Check the Save As Template check box.
3. Name the template and add a Title and any Notes that will help you identify the template.
4. Optionally, change the default category or click **Create** to open the Create Category dialog box to create a new category.

Categories are used to organize templates into groups.

5. Optionally, save the solution results.
6. Click **OK**.

## Results

The project is saved as a template in the specified category and will be listed in the New Project dialog box.

## Related Topics

[Save Project Dialog Box](#)

[Create Category Dialog Box](#)

[New Project Dialog Box](#)

## Exporting Projects in SAT or IGS Files

Projects can be exported to external disk files.

### Procedure

1. You have a choice:

If you want to...	Do the following:
Export an assembly as an ACIS *.sat file	<ul style="list-style-type: none"><li>Right-click the project name at the top of the Project Manager data tree and choose <b>Export Project &gt; SAT</b>.</li></ul>
Save an MCAD assembly as a *.igs format file	<ul style="list-style-type: none"><li>Right-click the project name at the top of the Project Manager data tree and choose <b>Export Project &gt; IGES</b>.</li></ul>

2. Navigate to the destination folder and click **Save**.

### Results

- Exported \*.sat files do not include the assembly hierarchy, object names, or units.
- Exported \*.igs files do not include the assembly hierarchy, object, or names.

### Related Topics

[Archiving Project Data in PDML Format](#)

[Archiving Project Data in PACK Format](#)

## Exporting Thermal Netlists and BCI-ROM Files

You can export a thermal network representation in SPICE-Eldo or VHDL-AMS format, or a set of files for a BCI-ROM analysis of the model.

You can export BCI-ROM files as:

- Matrices - Files exported in this format are compatible with mathematical programming tools such as MATLAB® and GNU Octave.  
Files of this type can also be solved with the BCI-ROM Matrices solver included with Simcenter Flotherm.

- FMU (Co-Simulation) - Files exported in this format can be solved in FMI compliant tools such as Simcenter Flomaster™, Simcenter AMESim™, and Ansys Simplorer®.
- VHDL-AMS - Files exported in this format can be solved in tools such as SystemVision® Cloud and Xpedition® AMS.

The technology used by Simcenter Flotherm to create thermal netlists and BCI-ROM files is based on the FANTASTIC method, generally described in the following publications:

1. L. Codecasa, D. D'Amore, and P. Maffezzoni, “Parameters for multi-point moment matching reduction of discretized thermal networks”, in *Proc. IEEE THERMINIC, 2002*, pp. 151-154.
2. L Codecasa, V d'Alessandro, A Magnani, N Rinaldi, PJ Zampardi, “Fast novel thermal analysis simulation tool for integrated circuits (FANTASTIC)”, *20th International Workshop on Thermal Investigations of ICs and Systems (THERMINIC) article 6972507 United Kingdom, 2014*.
3. Lorenzo Codecasa, Vincenzo d'Alessandro, Alessandro Magnani, Niccolò Rinaldi, “Matrix reduction tool for creating boundary condition independent dynamic compact thermal models”, *Thermal Investigations of ICs and Systems (THERMINIC) 2015 21st International Workshop on*, pp. 1-5, 2015.
4. JHJ Janssen, L Codecasa, “Why matrix reduction is better than objective function based optimization in compact thermal model creation”, *Thermal Investigations of ICs and Systems (THERMINIC) 2015 21st International Workshop on*, pp. 1-6, 2015.

## Restrictions and Limitations

- Materials with temperature-dependent thermal conductivity are not supported.
- Attached ambients must be uniquely identified.
- None of the following SmartParts are supported:  
Resistance, Heat Sink, Fan, Network Assembly, Heat Pipe, TEC, Die, Power Map, Recirculation Device, Cooler, Rack, Fixed Flow, Compact Component, Perforated Plate, Controller.
- Collapsed cuboids must not abut the solution domain boundary or cutout face.
- Radiation, Solar Radiation, and Joule Heating (Model Setup) are not supported.
- Activate Plate Conduction (Solver Control) is not supported.
- The calculation time required for export depends on the following:
  - Number of sources.
  - Number of unique heat transfer coefficients in the model.
  - Number of grid cells.

- Range of heat transfer coefficients.
- Acceptable relative error.
- When specifying power, either using a Source or a Thermal attribute, specify a total value (Total Source or Total Power).

**Caution**

 Although other types of power definition are allowed in the Simcenter Flotherm model, they may not be handled consistently by Eldo®. Results may then not match and therefore verification may not be possible.

---

## Prerequisites

- Access is controlled by license.
- When exporting the SPICE-Eldo format, names of thermal ports must conform to the node name conventions of Eldo, namely:

“The following characters are not allowed in node names: (), {}, ', =. Node names cannot begin with any of the following characters: \*, +, -, &, |, ". Do not use the period character (.) in a node name because it is reserved as a hierarchy separator.”

In the context of the Simcenter Flotherm model, thermal ports are any objects to which either a Source or Thermal attribute is attached.

- When exporting the VHDL-AMS format, elements or identifiers contained in the file must conform to the VHDL name conventions:
  - The elements must consist only of letters, numbers, and underscores.
  - Elements and identifiers must start with a letter.
  - Two consecutive underscores are not allowed.
  - None of the VHDL reserved keywords can be used, for example, component, case, body, Package.
  - Commas and square brackets are removed and spaces are replaced with underscores in element/identifier names during VHDL-AMS export. Multiple consecutive spaces are replaced with single underscore. The model name is not affected.

Refer to the *IEEE Standard for VHDL Language Reference Manual* for a full list of keywords:

<https://standards.ieee.org/standard/1076-2019.html/>

- In the context of the Simcenter Flotherm model, elements/identifiers are:
  - Any objects to which either a Source or Thermal attribute is attached.

- The name of the model. This limitation does not apply when exporting the SPICE-Eldo format.
- Type of Solution (Model Setup) must be Conduction Only.
- At least one powered non-collapsed Source object must be present.
- At least one non-zero Heat Transfer Coefficient must be attached to the solution domain (system boundary) or to cutouts, and at least one solid object must abut such a boundary face.

## Procedure

1. Ensure that the data tree hierarchy is appropriate and, if a verification is to be done, that Source powers match those derived from the electrical netlist.

The hierarchical order of Sources in the data tree determines the order of thermal ports in the extracted thermal list.

2. You have a choice.

If you want to...	Do the following:
Export a thermal netlist in SPICE-Eldo or VHDL-AMS format.	<ol style="list-style-type: none"> <li>1. Either choose <b>Project &gt; Export Project &gt; Thermal Netlist</b>, or right-click the project node in the data tree and choose <b>Export Project &gt; Thermal Netlist</b>. The Thermal Netlist Preferences dialog box is opened.</li> <li>2. Change, or leave as is, the default Acceptable Relative Error value and click <b>OK</b>. A file browser dialog box is opened. The first time a file is exported, the given default format is SPICE-Eldo, and the default filename is <i>&lt;project name&gt;.sp</i>.  Optionally, use the file type dropdown list to select VHDL-AMS format with a default filename of <i>&lt;project name&gt;.vhd</i> At subsequent exports, a default filename is not provided.</li> </ol>

If you want to...	Do the following:
Export BCI-ROM files.	<ol style="list-style-type: none"><li>1. Either choose <b>Project &gt; Export Project &gt; BCI-ROM</b>, or right-click the project node in the data tree and choose <b>Export Project &gt; BCI-ROM</b>. The BCI-ROM Preferences dialog box is opened.</li><li>2. Use the BCI-ROM Format dropdown list to specify the file format of the exported zip file. Choose Matrices, FMU (Co-Simulation), or VHDL-AMS.</li><li>3. Change, or leave as is, the default Acceptable Relative Error, and Minimum and Maximum HTC values and click <b>OK</b>. A file browser dialog box is opened.</li></ol> <p> <b>Note:</b> The ‘Save as type’ field is automatically set depending upon your selection in step 2.</p> <p>The default filename is <i>&lt;project name&gt;.zip</i>.</p>

3. Navigate to a folder and click **Save**.

The solver runs to create the export file(s). The Thermal Netlist/BCI-ROM Export Progress dialog box is displayed, showing a percentage progress bar.

---

**Note**

 A CFD solution is not created.

---

## Results

If successful, the following message is output:

INFO I/8016 - Thermal Netlist / BCI-ROM successfully exported

- A netlist *.sp* file contains an exported network in the form of a SPICE netlist that can be used by Eldo as part of an electrothermal simulation. If equivalent power values are used in the Simcenter Flotherm model and in the electrical netlist then temperatures calculated by Simcenter Flotherm can be compared with those calculated by Eldo to verify the thermal network.
- A netlist *.vhd* file contains the exported network in VHDL-AMS format.
- A BCI-ROM *Matrices.zip* file contains a set of files:
  - *\*.mtx* are matrices defining the ROM (Reduced Order Model).
  - *report.rom* gives a brief report of the result of the extraction process, containing:

The acceptable relative error achieved versus the value requested in the BCI-ROM Preferences dialog box, and the resultant ROM size.

The HTC range values as set in the BCI-ROM Preferences dialog box.

- *rom\_parameters.m* is a script, compatible with mathematical programming tools (such as MATLAB® and GNU Octave), that provides a means to set variables of interest when solving the BCI-ROM in those tools. Specifically, it allows the definition of the following for a ROM solution:
  - Heat transfer coefficients.
  - Ambient temperatures.
  - Initial temperature.
  - Power as a function of time. Constant, piecewise linear, sinusoidal, and exponential options are supported.
  - Transient solution duration.

Complete ROM solving harness examples are available from Support Center:

<https://support.sw.siemens.com>

- *xData.txt* provides a summary of the key parameters of the detailed model that was used to create the compact model.
- FMU .zip file contains the extracted BCI-ROM in a .fmu format. Within the .fmu file these files can be found:
  - Binaries - Contains the Co-Simulation FMU solver (in 32 and 64 bit Windows only solver)
  - Resources - \*.mtx are matrices defining the ROM (Reduced Order Model).
  - *modelDescription.xml* - Contains the definition of the FMU connections and variables required by the importing tool. Default time step of 0.01 second is included with the FMU. This file adheres to the FMI standard.
- VHDL .zip file contains:
  - \*.csv - Contains the information defining the extracted ROM. Each column in the csv file represents a matrix.
  - \*.vhd - Defines the connections of the ROM information included in the accompanying csv file and enables the ROM to be solved.

## Related Topics

[Thermal Netlist Preferences Dialog Box](#)

[BCI-ROM Preferences Dialog Box](#)

[Thermal Netlist/BCI-ROM Export Progress Dialog Box](#)

## Archiving

You can archive a project in PDML or PACK formats. PACK files can include or exclude results.

Archiving Project Data in PDML Format.....	76
Archiving Project Data in PACK Format.....	77

## Archiving Project Data in PDML Format

Export the solution directory in PDML format files if you do not need to save the results. You can automate the procedure by creating scripts.

### Procedure

1. Right-click the project name at the top of the Project Manager data tree and choose **Export Project > PDML**.  
A browser dialog box is displayed. The name of the project, with a *.pdml* extension, is used as the default filename.
2. Navigate to the destination folder and click **Save**.

### Examples

The following is an example Microsoft Windows script using flogate\_cl to export a solution directory in PDML format. You may need to change the installation directory and/or the flosuite version number for your installation.

```
@echo off

REM USER SETTINGS
set "FLOSOLNDIR=C:\Program Files\MentorMA\flosuite_v<version>\flotherm\frouser"
call C:\Program Files\MentorMA\flosuite_v<version>\flotherm\WinXP\bin\flotherm.bat -env
REM END USER SETTINGS

cd /D %FLOSOLNDIR%

if NOT EXIST PDMLDIR mkdir PDMLDIR

dir /B /AD > temp

for /f "tokens=1,2 delims=." %%i in (temp) do (
    if EXIST "%FLOSOLNDIR%\%%i.%%j\PDProject\group" (flogate_cl -i "PDML" -r "%FLOSOLNDIR%\%%i.%%j\PDProject\group" -o PDML -w PDMLDIR\%%i.pdml)
    del temp
```

- Edit the lines in the USER SETTINGS part to fit with your configuration, that is, the paths to your solution directory and to *flotherm.bat*.

- The example assumes that the solution directory is the default *flouser* directory although this may be set to any mapped drive.
- A new subdirectory called *PDMLDIR* is created in the solution directory to contain the exported files ready for archive.
- The disadvantage of archiving with PDML format files is that the solution data is not stored in the file; however, PDML files are generally less than 1 MB in size.

## Related Topics

[PDML Files](#)

[Importing a PDML Project](#)

## Archiving Project Data in PACK Format

Export in PACK file format if you need the project and the results. PACK file format is the recommended format to use when sending projects to Mentor Graphics Mechanical Analysis customer support. You can automate the procedure by creating scripts.

### Procedure

1. Right-click the project name at the top of the Project Manager data tree and choose **Export project > Pack File**. If you do not want to save results, select **Pack File (no results)**.

A browser dialog box is displayed. The name of the project, with a *.pack* extension, is used as the default filename.

As Simcenter Flotherm identifies packed projects by the *.pack* filename extension, when naming the packed project *make sure you keep* the default *.pack* extension.

2. Navigate to the destination folder and click **Save**.

### Examples

The zip utility that is installed with the Simcenter Flotherm software installation may be used to backup project data from a solution directory. For Microsoft Windows OSs, a script, similar to the following example, may be used to archive Simcenter Flotherm projects with solution data. You may need to change the installation directory and/or the flosuite version number for your installation.

## Working on a Project

### Deleting Projects

---

```
@echo off

REM USER SETTINGS
set "FLOSOLNDIR=C:\Program Files\MentorMA\flosuite_v<version>\flotherm\fouser"
call C:\Program Files\MentorMA\flosuite_v<version>\flotherm\WinXP\bin\flotherm.bat -env
REM END USER SETTINGS

cd /D %FLOSOLNDIR%

if NOT EXIST PACKDIR mkdir PACKDIR

dir /B /AD > temp

for /f "tokens=1,2 delims=." %%i in (temp) do (
    if EXIST "%FLOSOLNDIR%\%%i.%%j\PDProject\group (echo %%i.%%j | 
    "%FLOPRODUCTDIR%\bin\zip" -q -r PACKDIR\%%i.pack .\%%i.%%j -z )) 
del temp
```

- Edit the lines in the USER SETTINGS part to fit with your configuration, that is, the paths to your solution directory and to *fouser.bat*.
  - The example assumes that the solution directory is the default *fouser* directory although this may be set to any mapped drive.
- A new subdirectory called *PACKDIR* is created in the solution directory to contain the exported files ready for archive.
- PACK format files can be very large.

### Related Topics

[Importing a PACK Project](#)

## Deleting Projects

Deleting a project removes the project completely.

### Restrictions and Limitations

- You cannot delete a project while it is currently loaded in program memory.
- You cannot delete a project that was not saved under your own username.

### Procedure

1. Press Ctrl+L to open the Load Project dialog box.
2. Select the project from the project list then click **Delete**.  
A confirmation box is opened.
3. Click **Yes**.

## Results

The project is removed from the solution directory, and from the computer.

## Related Topics

[Load Project Dialog Box](#)

# Project Manager Operations

---

Configuring the user interface, and setting user preferences.

---

## Note

 The Project Manager operations associated with defining the model and the geometry are covered in “[Defining the Mathematical Model](#)” on page 137, “[Creating, Importing, and Exporting Geometry](#)” on page 211, and “[Viewing and Manipulating Geometry](#)” on page 253.

---

Showing and Hiding Toolbars .....	80
Detaching/Reattaching the Drawing Board .....	81
Detaching/Reattaching the Profiles Window .....	82
Changing Global Units .....	83
Changing Local Units .....	83
Undoing and Redoing the Last Operation.....	84
Changing the Coordinate System.....	85
Using Parallel Processing .....	85

## Showing and Hiding Toolbars

Visibility of the toolbars is controlled by a pop-up dialog box comprising a set of check boxes.

---

## Note

 The visibility of the Drawing Board, Workplanes, and Selection Modes toolbars is controlled from the Drawing Board window when detached.

If the Profiles pane is reattached, then the Profiles toolbar is added to the Project Manager window.

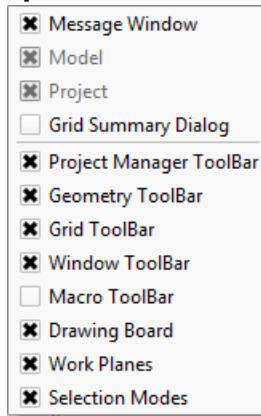
---

## Procedure

1. Right-click in the menu bar.

The choice of toolbars that can be displayed is context-sensitive.

**Figure 3-8. Example Toolbar Visibility Dialog Box**



2. Select a toolbar to be shown or hidden.

The dialog closes.

## Results

The selected toolbar is either hidden or shown, depending on its previous state. The selection action toggles between the two states.

A toolbar can be moved and either left floating or docked along an edge of the window.

# Detaching/Reattaching the Drawing Board

The drawing board can be detached from the Project Manager window to create a separate Drawing Board window. This is particularly effective when using multiple screens.

## Procedure

Choose **Window > Detach GDA**.

Alternatively, click the **Detach/Reattach GDA** icon in the Project Manager toolbar.

## Results

The Drawing Board is contained within a separate window.

The following items are transferred from the Project Manager window to the Drawing Board window:

- The Viewer menu.
- Geometry menu options that are relevant to the drawing board, such as Align. Geometry menu options that are relevant to the data tree, such as Promote, are retained in the Project Manager window.
- The Drawing Board, Workplane, and Selection Modes toolbars.

- The Results Tree and Tables are detached with the Drawing Board when the Project Manager is in Analyze mode.
- The Status Bar grid information, cursor position and the maximum aspect ratio highlight button.

The Message Window is detached at the same time.

The sizes and positions of the Drawing Board and Message Window windows are retained over sessions.

To reattach the windows, choose **Window > Reattach GDA** or click the **Detach/Reattach GDA** icon in the Project Manager toolbar.

## Related Topics

[Project Manager Status Bar](#)

[The Drawing Board](#)

# Detaching/Reattaching the Profiles Window

The Profiles window is detached from the Project Manager window by default.

## Procedure

1. To reattach the Profiles window, do one of the following:
  - Choose **Window > Reattach Profiles**, or
  - Click the **Detach/Reattach Profiles** icon in the Project Manager toolbar.If the drawing board is attached to the Project Manager, then the Profiles window is positioned underneath the drawing board, otherwise it occupies the position normally taken by the drawing board.  
If the Message Window is attached to the Project Manager, then the Profiles window is tabbed with the Message Window.
2. To detach the Profiles window, do one of the following:
  - Double-click the title bar of the docked window, or
  - Choose **Window > Detach Profiles**, or
  - Click the **Detach/Reattach Profiles** icon in the Project Manager toolbar.

## Related Topics

[Profiles Window](#)

## Changing Global Units

Global units are used throughout Simcenter Flotherm as default units in property sheets and dialog boxes. All property sheets and dialog boxes use the default global units when first shown. If you change a global unit then the property sheets and dialog boxes will use that unit the next time they are opened.

### Restrictions and Limitations

- If units are changed locally, they continue to be used locally in preference to the Global units.

### Procedure

1. Choose **Edit > Units** to open the Global Units dialog box.
2. Scroll down the list of measurement types until you find the type that you want to change.
3. Select the new units from the dropdown list and click **OK**.

### Results

- Open a property sheet or dialog box to confirm that the default units have changed.

### Related Topics

[Global Units Dialog Box](#)

[Changing Local Units](#)

## Changing Local Units

For convenience you may want to input data in non-default units. For example, the global setting for Length Unit defaults to meters, but you may want to enter values from a manufacturer's specification which is in inches.

### Restrictions and Limitations

- Local is local to a property sheet tab or dialog box. For example, if you change the units for Width, then all other "Length Units", such as for Height, Thickness, and so on, will be changed. However, Length Units in the **Location** tab of a property sheet are unaffected by a local change in the **Construction** tab of the same property sheet.

### Procedure

1. Open the relevant property sheet or dialog box.
2. *Before* entering data in the values field, select the units from the dropdown list.

**Note**  
 If you enter values before changing units, the values will be converted to the new units.

---

## Related Topics

[Changing Global Units](#)

# Undoing and Redoing the Last Operation

Multiple undo and redo is possible within the following limitations.

## Restrictions and Limitations

- Multiple undo and redo is restricted to up to 10 operations. The operations include object and model manipulations, but *exclude* view manipulations.
- A consequence of having an undo stack is that you may have to flush the stack to delete attributes attached to objects in the stack. For example, if you create an object, then attach an attribute (say, a material) then delete the object, you cannot then delete the material, because it is attached to an object in the undo stack. To delete the material, flush the undo stack using Ctrl+Shift+F.
- Operations that can be undone:
  - Project Manager (Create mode) operations:
    - Visualization state: wireframe, solid, solid with edges.
    - Editing the size or location of an object.
    - Creating an object using the New Object Palette.
    - Renaming an object.
    - Cutting an object.
    - Pasting a cut object.
    - Copying an object.
    - Paste a copied object.
    - Hiding and Revealing an object.
    - Activating and Deactivating an object.
    - Promoting and Demoting an object.
    - Localize Grid on an object.
    - Aligning a set of objects.

- Rotating an object.
- Project Manager (Analyze mode) operations:
  - Creating a plane.
  - Creating a particle source.
  - Saving a viewpoint.

### Procedure

1. If you are unsure which operation will be undone or redone then mouse hover over the relevant **Edit > Undo/Redo** menu option or **Undo/Redo** icon.  
A short description of the operation that will be undone/redone is displayed in the Status Bar at the bottom of the window. Note also that the menu option wording changes to include a very brief description of the operation, for example, “Undo Load From Library”.
2. Use the **Edit > Undo/Redo** menu option or **Undo/Redo** icon, or press Ctrl+Z to undo, or Ctrl+Y to redo the last operation.

## Changing the Coordinate System

You have a choice between using absolute or local coordinates.

### Procedure

1. Open the User Preferences dialog box by clicking the **User Preferences** icon or choosing **Edit > User Preferences**.
2. In the Display Positions In field, select either **Absolute Coordinates** or **Local Coordinates**.

## Using Parallel Processing

To speed up the solving process, provided you have multiple processors, you can specify more than one processor when running the solver.

### Restrictions and Limitations

Parallel processing is only supported on shared memory systems.

### Procedure

1. Open the User Preferences dialog box by clicking the **User Preferences** icon or choosing **Edit > User Preferences**.
2. In the Number of Processors to Use field, enter the number of processors you want to use.

## Results

You can check that multiple processors are being used by monitoring your computer using the Windows Task Manager when running the solver.

# Data Tree Operations

Objects in the data tree are manipulated by functionality provided by the project manager.	
<b>Viewing Summary Information</b>	<b>87</b>
<b>Adding Assemblies</b>	<b>89</b>
<b>Adding Parts</b>	<b>90</b>
<b>Colocating Objects</b>	<b>90</b>
<b>Selecting Objects in the Data Tree</b>	<b>91</b>
<b>Promoting and Demoting Geometry in the Data Tree</b>	<b>92</b>
<b>Topping Assemblies or Parts</b>	<b>92</b>
<b>Hiding Objects From the Data Tree (Lightweight View of Assemblies)</b>	<b>93</b>
<b>Simple Searches for Objects or Attributes</b>	<b>93</b>
<b>Complex Searches for Objects or Attributes</b>	<b>94</b>
<b>Applying Simultaneous Changes to More Than One Object</b>	<b>96</b>
<b>Creating Unique Names for Objects</b>	<b>97</b>

## Viewing Summary Information

Summary information provides a quick way of viewing power source values, object sizes, and attached attributes in table format.

### Restrictions and Limitations

- Although the units in the summary information are updated when a change is made using the Global Units dialog box, if the change is then ‘undone’, the summary information is not updated. In such circumstances you will have to hide and then re-show the summary information to see the undone change.

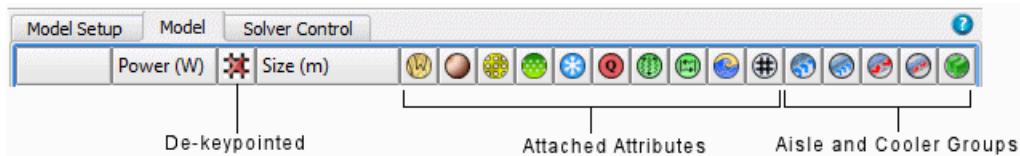
### Procedure

- If summary information is not shown, press I. Alternatively, click the **Show Summary** icon or choose **Window > Show Summary**.
- If you want to show more or less summary information, choose **Edit > User Preferences** to open the User Preferences dialog box, then select the **Summary** tab.
- Use the check boxes to select what will be displayed, then click **OK** to close the dialog box and update the summary information.
- To hide summary information, press I again.

### Results

Summary information is provided in columns, see [Figure 3-9](#) and [Table 3-1](#).

**Figure 3-9. Summary Information Column Headers**



**Table 3-1. Summary Information Column Content**

Column	Description
Power	The Power setting in the attached thermal attribute. The units are those set by the Power definition in the Global Units dialog box.  Parents of sources (for example, assemblies) show the summation of power from the children by enclosing the value in brackets (for example, [30]).
De-keypointed	A cross is used to flag that a keypoint has been deactivated. Hover Text names the de-keypointed side of the object, for example, X High.  The flagging of de-keypointed objects is filtered by the De-keypointing % Tolerance value set in the <b>Summary</b> tab of the User Preferences dialog box.
Size	The dimensions X, Y, Z of the object. The units are those set by the Length definition in the Global Units dialog box.  For Sloping Block, these dimensions correspond with those when defining the block by Sides in the Define Using dropdown of the Sloping Block property sheet <b>Construction</b> tab.
Attached attributes	A symbol shows that an attribute is attached. Hover Text identifies the name of the attribute, or the names of attributes if different attributes are attached to different faces.  <b>i Tip:</b> You can open an attribute property sheet directly by clicking on a symbol. When different attributes are attached to different faces of an object then repeated clicking cycles through the attribute property sheets.  A bordered square box signifies No Attachment.  Blank signifies that the attribute cannot be attached.
Cold Aisle Group. Cold Aisle Sub-Group. Hot Aisle Group. Hot Aisle Sub-Group	Symbols are used to show whether or not an object belongs to one of the aisle groups used when determining the Capture Index. Hover Text identifies the name of the group.
Cooler Group	A symbol is used to show whether or not a cooler belongs to a Cooler Group. Cooler Groups are used for remote rack temperature control of cooler airflow. Hover Text identifies the name of the group.

- If a child object is deactivated, then its power is not included in its parent's power total.
- Where a power value of an object has been defined as "per area" or "per volume", the power of the object is calculated using the value of the area or volume of the object, and this value is used in the summation for the assembly. In other words, the summations are always in units of power.
- Where a power value of an object has been defined as varying with temperature, either linearly or non-linearly, then no contribution is made to the power total for the parent assembly. In such cases, to use an assumed temperature to calculate the power value would introduce an unacceptable error in the summation.

## Related Topics

[User Preferences Dialog Box - Summary Tab](#)

[Global Units Dialog Box](#)

[Keypoint Deactivation \(De-Keypointing\)](#)

[Capture Index](#)

# Adding Assemblies

Simcenter Flotherm uses the concept of assemblies and parts to create the geometry. The top-level root assembly contains all the project geometry, but to create a structured design you create sub-assemblies of related functional parts, for example separate assemblies for the structure, equipment, and ventilation.

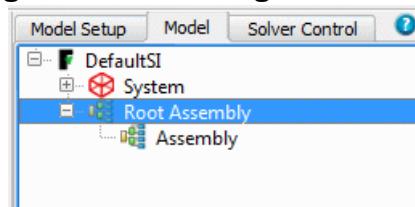
## Procedure

1. Select an assembly in the data tree (the root assembly if starting a new project).
2. Click the **Assembly** icon.

## Results

The new empty assembly is added as a child of the selected assembly, ready to hold new geometry parts.

**Figure 3-10. Adding Assemblies**



The origin of the assembly is located at the origin of the selected assembly.

## Adding Parts

New data tree objects are created as children of the item selected in the data tree.

### Restrictions and Limitations

- Power maps are added by importing of power map files, see [Power Maps](#) in the *Simcenter Flotherm SmartParts Reference Guide* and “[Joule Heating Analysis](#)” on page 183.

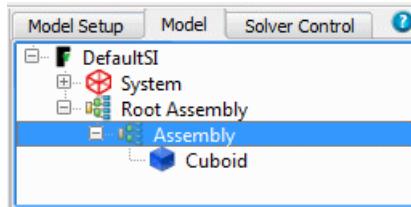
### Procedure

- Select an assembly, or a valid parent object, in the data tree.
- Click the appropriate object icon in the palette.

### Results

The new part appears as a child of the selected assembly/object.

**Figure 3-11. A Cuboid Added to an Assembly**



The origin of the part is located at the origin of the parent assembly.

---

#### Note

 The location of a new monitor point depends on whether it is created when a part or an assembly is selected. If a new monitor point is created for a selected part, then the monitor point is located at the center of the selected part. However, if a new monitor point is created for a selected assembly, then it is located at the origin of its parent assembly.

---

### Related Topics

[Creating, Importing, and Exporting Geometry](#)

## Colocating Objects

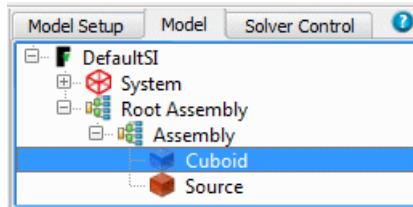
Parts can be colocated in an assembly.

### Procedure

- Select a part in the tree.
- Click an object in the palette.

For example, if you select a cuboid then add a source, then the source is added at the same location and in the same assembly as the cuboid, see [Figure 3-12](#).

**Figure 3-12. Adding Siblings**



## Selecting Objects in the Data Tree

Individual multiple selection and contiguous multiple selection are supported.

### Procedure

1. You have a choice.

If you want to...	Do the following:
Select multiple objects individually, that is, one at a time.	<ul style="list-style-type: none"> <li>• Select the first item and then press Ctrl+left-click over each additional required data item.</li> </ul> <p>If your design has a number of primitives/SmartParts/regions/cutouts/monitor points under the root assembly, you can select a group of them of the same or of mixed type.</p>
Select multiple objects in a contiguous group.	<ul style="list-style-type: none"> <li>• Select the first object, hold the Shift key, and then select the last object.</li> </ul> <p>Alternatively, you can select the last object, hold the Shift key, and then select the first object.</p>

2. To deselect an item or items, click in the blank area of the display or press Esc.

### Related Topics

[Selecting Objects in the Drawing Board](#)

[Applying Simultaneous Changes to More Than One Object](#)

## Promoting and Demoting Geometry in the Data Tree

The listing order of the geometry reflects the setup sequence.

Items at the top of the tree are generally considered created first (see “[Construction Precedence Rules](#)” on page 218 for more details).

### Restrictions and Limitations

- When using the **Promote** and **Demote** icons, promotion and demotion within the data tree are restricted to the sibling level.

### Procedure

You have a choice:

If you want to...	Do the following:
Change the listing order using the mouse.	<ul style="list-style-type: none"><li>Drag the object icon to the desired location. As the object is dragged across the tree, the mouse pointer changes to the Valid Drop Location pointer for a valid drop-site, or the Invalid Drop Location pointer for an invalid drop-site.</li></ul>
Copy an object to a new position in the data tree.	<ol style="list-style-type: none"><li>Select the object.</li><li>Hold the Ctrl key.</li><li>Drag and drop.</li></ol>
Change the listing order at the sibling level using the icons.	<ol style="list-style-type: none"><li>Select the object.</li><li>Click the <b>Promote</b> or <b>Demote</b> icon.</li></ol>

## Topping Assemblies or Parts

When viewing long data trees, you can restrict (“top”) the tree view to show only an assembly or part. By Topping assemblies you can methodically step through all the assemblies in a particular path from the highest to the lowest level in the data tree. Topping is also reflected in the drawing board and so can be used to declutter views.

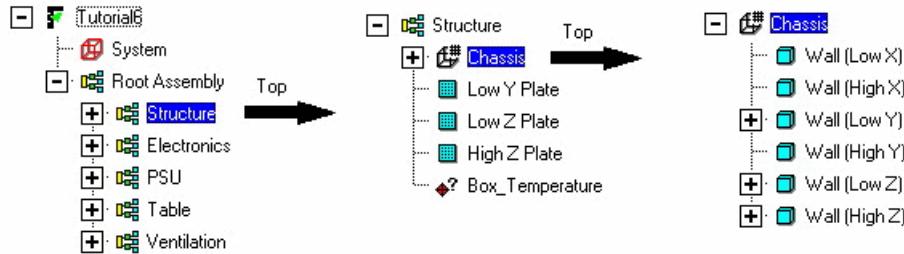
### Procedure

- Select the object to be “topped”.
- Press F3 or choose **View > Top**.

## Results

The selected object becomes the top item in the data tree as shown in [Figure 3-13](#). All objects that are above the selected object are hidden from view.

**Figure 3-13. Topping**



- To show the parent of the top item, press F5, or choose **View > Up to Parent**.
- To display the whole tree, press F4, or choose **View > Reset**.

## Hiding Objects From the Data Tree (Lightweight View of Assemblies)

Improving the readability of a data tree by hiding all child objects of an assembly.

### Procedure

1. Select an assembly.
2. In the property sheet **Location** tab, check the Lightweight check box.

### Results

The child objects of the assembly are hidden from the data tree, and the assembly is indicated as being “lightweight” by the addition of a black outline to the assembly icon

#### Note

Lightweight assemblies can be searched for using the **Extended Criteria** tab of the Find dialog box with a criteria of “Lightweight is on”.

## Simple Searches for Objects or Attributes

Simple searches, based on a name, can be used to find objects in the data tree, or attributes in the Project Attributes tree.

### Procedure

1. Press Ctrl+F to open the Find dialog box. By default, the **Quick Criteria** tab is open.

2. The data tree is searched by default. To search the Project Attributes, change **Geometry** to **Attributes** in the **Action** section of the dialog box.  
If you select **Attributes**, then the Localized and Collapsed options are unavailable as they are not applicable.
3. Enter a full or partial name in the Name field and check or uncheck the Match Case and Partial Match check boxes to widen or narrow the search.
4. By default, all types are searched, but you can select other Type options to filter for a particular object/attribute type.
5. For geometry searches, if necessary, use the Localized and Collapsed options to further filter the search.
6. Before clicking **Find**, decide what action you want to be taken:
  - In the Select options, Select All highlights all of the found objects/attributes in the data tree/Project Attributes tree. Cycle Select only highlights one found object at a time. The forward (>>) and back (<<) buttons can be used to cycle around the found objects/attributes.
  - Check the Filter check box to remove objects that do not match the find criteria from the data tree/Project Attributes tree. Children of found objects remain in the data tree.
7. To begin the search, click **Find**.

## Results

The Results Count field is updated to show the number of found objects/attributes (hits).

Depending on the Action Select selection, one or all of the matched objects/attributes will be highlighted.

## Related Topics

[Find Dialog Box - Quick Criteria Tab](#)

# Complex Searches for Objects or Attributes

Complex combinations of criteria can be specified to find objects in the data tree, or attributes in the Project Attributes tree.

## Restrictions and Limitations

- For the following SmartParts, the property sheet **Construction** tab field entries that can be searched are:
  - PCB Component — All except Pattern details.
  - Die — Sizes only, however, Power can be searched as a Common criteria.

- For Heat Sink SmartParts, the property sheet tab field entries that can be searched are:
  - Main tab — All except Bonded details (Groove Depth, and so on).
  - Internal Fins tab — All except Center Gap, Fin Insets, Taper details, Fin Style, Cells Between Fins.
  - End Fins tab — All except Offsets, Styles, Brackets.
  - Pin Geometry tab — Uniform dimensions only.
  - Pin Arrangement tab — None.
- Attribute attachments defined within property sheet Construction tabs, for example, PCB materials or PCB components that inherit PCB materials, are not found when searching Attribute Data.
- Searches for Attribute Data:
  - Control and Transient attributes have no data that can be searched.
  - Type = Source, Criteria = Activate “is off” do not return results for inactive source attachments.

## Procedure

1. Press Ctrl+F to open the Find dialog box.
2. Select the **Extended Criteria** tab.
3. The data tree is searched by default. To search the Project Attributes, change **Geometry** to **Attributes** in the **Action** section of the dialog box.  
If you select **Attributes**, the **SmartPart Data** option is unavailable as it is not applicable.
4. Choose a criteria type:
  - For data tree (Geometry) searches:
    - **Common** enables for the selection of common criteria such as **Name** and **Power**, and whether switches such as **Localize Grid** or **Hide** are on or off. The **Geometry Type** criteria enables you to specify any geometry type including extracts, supplies, and monitor points. **Position X** enables you to specify the X-origin position of an object.
    - **SmartPart Data** opens the **Type** dropdown list of SmartParts. The available Criteria are then particular to the selected SmartPart.
    - **Attribute Data** opens the **Type** dropdown list of Attributes. The available Criteria are then particular to the selected Attribute.

- For Project Attribute tree (Attributes) searches:
    - **Common** enables the selection of **Name** or **Attribute Type**.
    - **Attribute Data** opens the **Type** dropdown list of Attributes. The available Criteria are then particular to the selected Attribute.
5. Select a Criteria from the dropdown list and click the + button.

The criteria is added to the Criteria list.
  6. Continue adding criteria, as necessary, to improve the search results.
  7. Either select **Match All** for all listed criteria to be satisfied, or select **Match Any**.
  8. Before clicking **Find**, decide what action you want to be taken:
    - In the Select options, **Select All** highlights all of the found objects/attributes in the data tree/Project Attributes tree. **Cycle Select** only highlights one found object/attribute at a time. The forward (>>) and back (<<) buttons can be used to cycle around the found objects/attributes.
    - Check the **Filter** check box to remove objects that do not match the find criteria from the data tree/Project Attributes tree. Children of found objects remain in the data tree and drawing board.
  9. To begin the search, click **Find**.

## Results

The **Results Count** field is updated to show the number of found objects/attributes (hits). Depending on the Action Select selection, one or all of the matched objects/attributes will be highlighted.

## Related Topics

[Find Dialog Box - Extended Criteria Tab](#)

# Applying Simultaneous Changes to More Than One Object

The same property sheet field change can be made to selected objects.

## Restrictions and Limitations

- Not applicable to table entry fields, with the exceptions of power tables of General compact components and resistance tables of 2-Resistor compact components.

## Procedure

1. Select the objects to be edited.

Property sheet fields that are common to the selected objects are displayed and can be edited.

2. Edit the common fields as required.

## Results

You can check that the edits have been made by deselecting all of the objects and then reselecting individual objects.

## Related Topics

- [Selecting Objects in the Data Tree](#)
- [Selecting Objects in the Drawing Board](#)

# Creating Unique Names for Objects

You can append unique sequence numbers to non-unique names to aid the identification of objects.

## Procedure

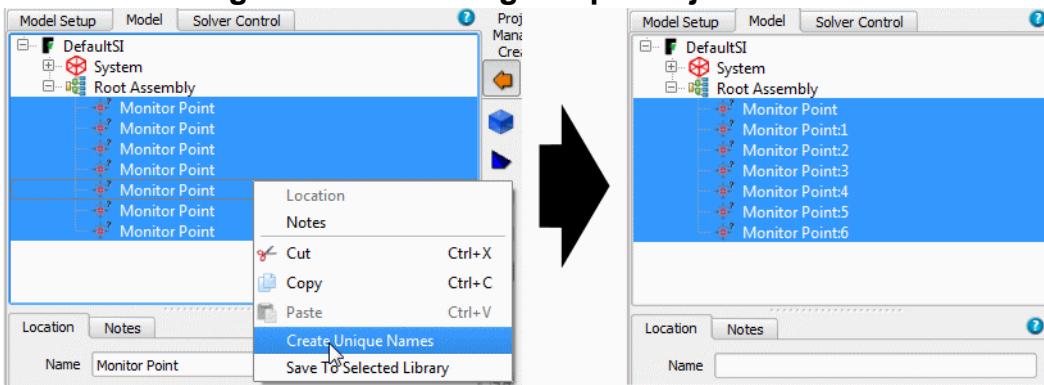
1. Multi-select objects with non-unique names.
2. Right-click and choose **Create Unique Names**.

## Results

'<:n>' is appended to the names, where <n> is a sequence number.

## Examples

**Figure 3-14. Creating Unique Object Names**



## Zoom-In Projects

---

Zoom-in projects are projects created from volume regions, and can be used to concentrate the computer analysis on a particular area of interest. Zoom-in projects can be useful if changes over the region will have little effect on the global flow fields of the original solution domain.

<b>Zoom-in Project Example .....</b>	<b>98</b>
<b>Creating a Zoom-In Project .....</b>	<b>98</b>
<b>Determining the Coupling Metric .....</b>	<b>103</b>

## Zoom-in Project Example

Zooming in on a cabinet in a data center.

Model racks or servers in the cabinet so as to ensure the correct airflow and air temperatures would be predicted. Model PCBs and other objects at a simple block level.

Once a solution has been obtained, a region around a slot could be used to create a zoom-in project. The crude PCB representation could then be replaced with a more detailed representation of the individual components and their associated powers. An accurate representation of the flow fields around the components can now be easily achieved without the overhead of solving for the whole data center. The model could be further refined by zooming-in on an individual chip and increasing its modeling complexity.

### Related Topics

[Creating a Zoom-In Project](#)

## Creating a Zoom-In Project

Create a zoom-in project when you want to zoom-in and refine the geometry and modeling complexity over a particular region without incurring the computational overhead of solving the complete project.

### Restrictions and Limitations

- To obtain a valid region from which the zoom-in project is created, the region should be sized, where possible, such that the majority of its boundary provides either a fluid-to-fluid, or a solid-to-fluid interface. This minimizes the number of components which invalidate the zoom-in approach. Errors can be removed by either removing the offending part or increasing the size of the region.
- 2D projects and Transient projects are not suitable for creating zoom-in projects.

- The project has to be in a recently state, that is, no post-solution modifications have been made to the project. The following conditions are not suitable to create an environment:
  - The grid has changed so project is not consistent with the solution data (backup directory exists).
  - A residual history does not exist, so that the project is inconsistent with the solution.
  - The project has not been translated, so that project is inconsistent with solution (tag file does not exist).
- The project must have been solved for Temperature, Pressure, Velocities, and Heat Flux. Heat Flux calculation is none-standard and is requested by a check box in the **Model Setup** tab.
- The following configurations of a region are not allowed:
  - Plane (collapsed) regions.
  - Regions on the edge of a grid space. A region at a localized grid boundary or at the solution domain boundary cannot be used to create environment.
- Only a limited number of object types are allowed to exist in the region, see [Table 3-2](#). This is to ensure that the automated grid definition and solution control capabilities are never compromised.

**Table 3-2. Allowed Geometry Criteria**

Part	Orientation with Respect to Region			
	Contained	Abutting Internally	Externally	Spanning
Cuboids/Cylinders/Prisms/Tets/Inverted Tets/Sloping Block	Yes	Yes	Yes	Yes
• Fixed Heat	Yes	No	No	No
• Joule	Yes	No	No	No
Enclosure	Yes	Yes	Yes	Yes
PCB				
• Child component with non-zero power that has Apply Over Board modeling level	Yes	Yes	Yes	No
• Child component with non-zero power that is bisected by the region	Yes	Yes	Yes	No
• Child component with non-zero power that is externally abutting the region	Yes	Yes	Yes	No
TEC	Yes	Yes	Yes	No
Die	Yes	Yes	Yes	No

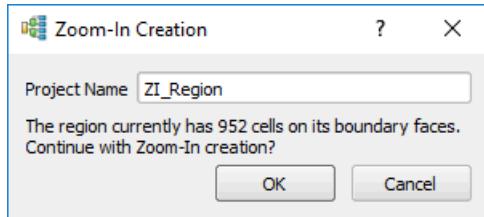
**Table 3-2. Allowed Geometry Criteria (cont.)**

Part	Orientation with Respect to Region			Spanning
	Contained	Abutting Internally	Externally	
Compact Component	Yes	Yes	Yes	No
Network Assembly	Yes	Yes	Yes	No
Heat Pipe	Yes	Yes	Yes	No
PCB Component	Yes	Yes	Yes	No
Heat Sink	Yes	Yes	No	No
Fan	Yes	No	No	No
Recirculation Device	Yes	No	No	No
Rack	Yes	No	No	No
Cooler	Yes	No	No	No
Fixed Flow	Yes	No	No	No
Resistance				
• Volume Resistance	Yes	Yes	Yes	Yes
• Collapsed or Planar Resistances	Yes	No	No	Yes
Perforated Plate	Yes	No	No	Yes
Localized Grid	Yes	No	No	No
Source	Yes	Yes	Yes	No
Cutouts	No	No	No	No

## Procedure

1. Set up and solve your project with a volume region over the place of interest.  
The volume region must satisfy all the zoom-in project requirement criteria.
2. Right-click the volume region and choose **Create Zoom-In** to open the Zoom-In Creation dialog box.  
The dialog box provides a default name for the project, which you can change, provided it is unique.

The dialog box also displays the number of cells on the region boundary, for example.



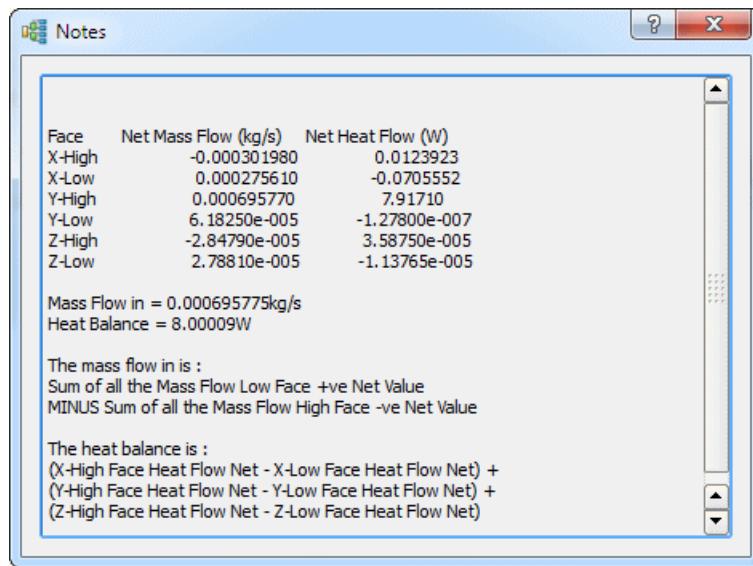
3. Optionally, give the zoom-in project an alternative name, then click **OK** to continue.

A progress bar is displayed before the zoom-in project is saved.

## Results

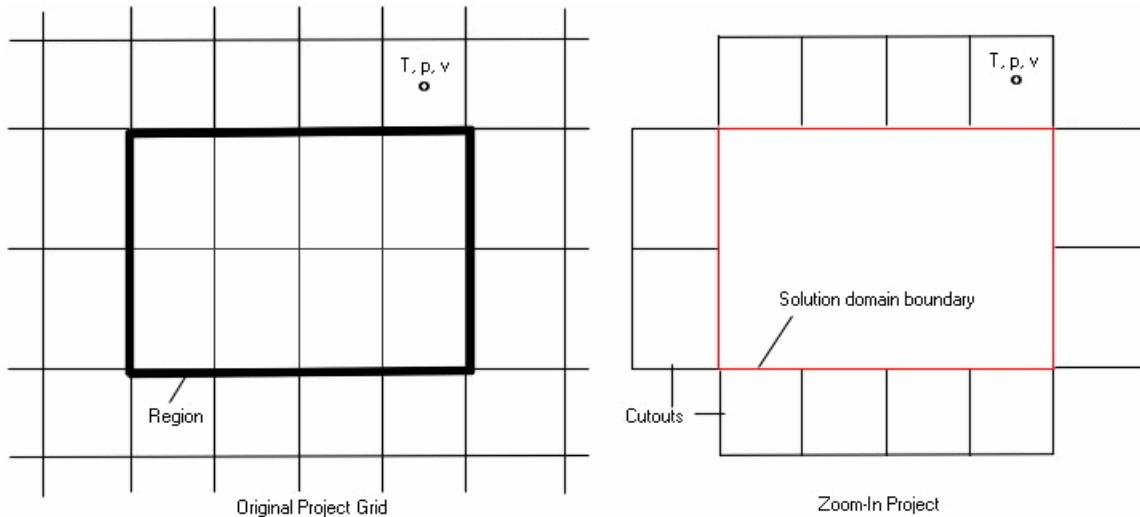
A new project is created, with the name specified in the Zoom-In Creation dialog box. The notes for the new project are appended with data relating to the original solution domain.

**Figure 3-15. Example of Parent Solution Data Appended to Zoom-In Project Notes**

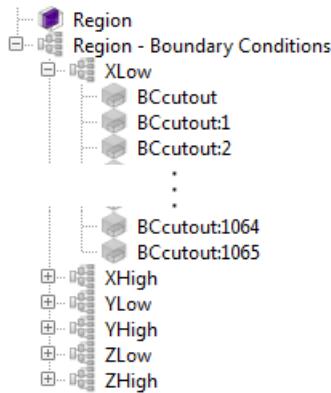


The boundary conditions at each cell face on the surface of the selected region is represented in a zoom-in project by a collection of cutout objects (each the same size of the region cell faces) externally abutting the zoom-in solution domain. These cutouts are not visible in the drawing board ([Figure 3-16](#)), but are listed in the data tree under assemblies labeled according to boundary face ([Figure 3-17](#)).

**Figure 3-16. Representation of Boundary Conditions**



**Figure 3-17. Cutouts on Data Tree**



Radically changing the geometrical shape of objects within the zoom-in region will increase the coupling (the measure of the amount a zoom-in region changes the results in the original solution domain) and careful attention should be paid to the *coupling metric*, see “[Determining the Coupling Metric](#)” on page 103.

Changing the number and locality of source/sink objects (for example, heat sources, sources of mass flow) should not invalidate the zoom-in approach provided the total heat or flow source/sink of the region remains constant.

To warn of the risk of potentially creating very large models, a message is issued if the number of cell faces on the six region sides exceeds 10,000. If this value is exceeded, the zoom-in project will be created, but processing may be slow. To suppress the warning message, you can set a higher value in the FLOMAXZOOMINCELLS environment variable.

## Related Topics

[Zoom-In Creation Dialog Box](#)

[Zoom-In Projects](#)

# Determining the Coupling Metric

The coupling metric, or the measure of the amount a zoom-in region changes the results, can be obtained using the project notes.

## Prerequisites

- A zoom-in project has been created, see “[Creating a Zoom-In Project](#)” on page 98.

## Procedure

1. Choose **Project > Load** to open the Load Project dialog box, select the project.
2. Click **Notes** to view solution data saved from the original project.  
An example is shown in [Figure 3-15](#) on page 101.
3. Solve the zoom-in project and compare the results with the data written to the project notes.

If no changes are made to the zoom-in model, the data should be identical.

## Results

The net mass and heat flows of each face of the zoom-in region can be obtained by summing the columns of the cutouts result table and comparing the values to their equivalent within the project notes.

## Related Topics

[Creating a Zoom-In Project](#)

# Project Manager and General Dialog Boxes

---

The following dialog boxes are associated with general project operations.

<b>Base Project Changed Dialog Box .....</b>	<b>105</b>
<b>BCI-ROM Preferences Dialog Box .....</b>	<b>106</b>
<b>Create Category Dialog Box .....</b>	<b>108</b>
<b>Find Dialog Box .....</b>	<b>109</b>
Find Dialog Box - Quick Criteria Tab .....	110
Find Dialog Box - Extended Criteria Tab .....	112
<b>Global Units Dialog Box .....</b>	<b>115</b>
<b>License Timeout Dialog Box .....</b>	<b>116</b>
<b>Load Project Dialog Box .....</b>	<b>117</b>
<b>Message Window .....</b>	<b>119</b>
<b>New Project Dialog Box .....</b>	<b>120</b>
<b>Notes Dialog Box .....</b>	<b>122</b>
<b>Save Project Dialog Box .....</b>	<b>123</b>
<b>Thermal Netlist Preferences Dialog Box .....</b>	<b>125</b>
<b>Thermal Netlist/BCI-ROM Export Progress Dialog Box .....</b>	<b>126</b>
<b>User Preferences Dialog Box .....</b>	<b>127</b>
User Preferences Dialog Box - Project Manager Tab .....	128
User Preferences Dialog Box - Drawing Board Tab .....	130
User Preferences Dialog Box - Summary Tab .....	132
User Preferences Dialog Box - Analyze Mode Tab .....	133
<b>Zoom-In Creation Dialog Box .....</b>	<b>136</b>

## Base Project Changed Dialog Box

To access: The dialog box is opened when Command Center scenario results have been stored, but a change has been made to the Base Project that invalidates the results.

Use this dialog box to continue using the current project or create a new project.

---

**Note**

-  If a change is made to the Base Project that invalidates Command Center results while the Command Center is closed; then, when the Command Center is reopened, the results will be marked as Stale.
- 

### Objects

Object	Description
Continue using the existing project and invalidate scenario results	Click to apply the changes to the current loaded project. The scenario solution statuses are marked as Stale.
Save As the current project with a new name to continue	Click to leave the current project unchanged, but open the Save Project dialog box to save the project with the Base Project changes in a new project. The new project becomes the current loaded project. The scenario solution statuses are marked as Stale.

## BCI-ROM Preferences Dialog Box

To access: The dialog box is opened when exporting BCI-ROM files.

Use this dialog box to specify the format of the BCI-ROM files that you are exporting, request an acceptable relative error value, and to set a global HTC range.

### Objects

Object	Description
BCI-ROM Format	<p>Use this dropdown list to choose from:</p> <ul style="list-style-type: none"><li>• Matrices - Files exported in this format are compatible with mathematical programming tools such as MATLAB® and GNU Octave. Files of this type can also be solved with the BCI-ROM Matrices solver included with Simcenter Flotherm.  <b>Note:</b> This is the default option. During the export a Matrices Files.zip file is created.</li><li>• FMU (Co-Simulation) - Files exported in this format can be solved in FMI compliant tools such as Simcenter Flomaster™, Simcenter AMESim™, and Ansys Simplorer®. During the export an FMU (Co-Simulation) Files.zip file is created.</li><li>• VHDL-AMS - Files exported in this format can be solved in tools such as SystemVision® Cloud, Xpedition®. During the export a VHDL-AMS Files.zip file is created.</li></ul>
Acceptable Relative Error	<p>The requested acceptable relative error. The default value of 0.001 is recommended for most applications.</p> <p>Larger values (up to 0.01) can be used to accelerate the extraction process.</p> <p>Smaller values (down to 0.0001) can be used when accuracy is paramount, however, use with caution, as this will impact on the calculation time required for extraction.</p>
Minimum HTC	Sets the HTC range that is exercised for all Ambient attributes attached to the model, either through the solution domain faces or through cutouts.
Maximum HTC	<p>The default range of 1-to-10,000 W/m<sup>2</sup>K is used for Delphi model generation in FloTHERM PACK, and is well suited for package level applications.</p> <p>The range can be customized for any application, for example, larger values can be used for cold plate type boundary conditions.</p>

## Related Topics

[Exporting Thermal Netlists and BCI-ROM Files](#)

## Create Category Dialog Box

To access: From the Save Project dialog box, with **Save As Template** selected, click **Create**.

Use this dialog box to create a new template category.

### Objects

Field	Description
Category Name	The name of the template category. This becomes the title of a new tab in the New Project dialog box.

### Related Topics

[Save Project Dialog Box](#)

[New Project Dialog Box](#)

[Saving a Project as a Template](#)

## Find Dialog Box

To access:

- Ctrl+F
- Click the **Find** icon.

Use this dialog box to perform a quick or detailed search for objects in the current topped assembly.

### Objects

Object	Description
<b>Quick Criteria</b> tab	See “ <a href="#">Find Dialog Box - Quick Criteria Tab</a> ” on page 110.
<b>Extended Criteria</b> tab	See “ <a href="#">Find Dialog Box - Extended Criteria Tab</a> ” on page 112.

## Find Dialog Box - Quick Criteria Tab

To access:

- Press Ctrl+F.
- Click the **Find** icon.

Use this tab to perform a quick search for objects in the current topped assembly.

### Objects

Field	Description
Type	Select from a dropdown list of SmartParts.
Name	The complete or partial name of the object(s) being searched for.
Match Case	Only find objects whose names or partial names match the case of the entry in the Name field.
Partial Match	Find objects whose names fully or partially match the entry in the Name field. Uncheck this box to find objects whose names fully match the entry in the Name field.
Localized	Access-restricted if Attributes has been selected in the Action section of this dialog box. <ul style="list-style-type: none"><li>• On — Only find objects that have Localize Grid on.</li><li>• Off — Only find objects that have Localize Grid off.</li><li>• Both — Ignore the Localize Grid setting.</li></ul>
Collapsed	Access-restricted if Attributes has been selected in the Action section of this dialog box. <ul style="list-style-type: none"><li>• On — Only find objects that are collapsed.</li><li>• Off — Only find objects that are uncollapsed.</li><li>• Both — Find collapsed and uncollapsed objects.</li></ul>
Action	
Select	The action taken when <b>Find</b> is clicked.  Left-hand list: <ul style="list-style-type: none"><li>• Select All — Select all objects that match the criteria.</li><li>• Cycle Select — Select the first object that matches the criteria. Use the &lt;&lt; and&gt;&gt; buttons to select other matches.</li></ul> Right-hand list: <ul style="list-style-type: none"><li>• Geometry — Restrict searches to the data tree.</li><li>• Attributes — Restrict searches to the Project Attributes.</li></ul>

Field	Description
Search	Use to limit the extent of the search: <ul style="list-style-type: none"><li>• Model — Search the whole model.</li><li>• Current Selection — Only search what is currently selected.</li></ul>
Filter	Removes all objects from the data tree and drawing board that do not match the search criteria except found objects, their parent assemblies and their child objects.
Results Count	Shows the number of objects that match the criteria when <b>Find</b> is clicked.

## Related Topics

[Simple Searches for Objects or Attributes](#)

[Find Dialog Box - Extended Criteria Tab](#)

[Complex Searches for Objects or Attributes](#)

## Find Dialog Box - Extended Criteria Tab

To access:

- Press Ctrl+F, then select the **Extended Criteria** tab.
- Click the **Find** icon, then select the **Extended Criteria** tab.

Use this dialog box to search the current topped assembly for objects whose properties match all or at least one of a list of user-defined search criteria.

### Objects

Field	Description
Common	Search for objects and SmartParts using criteria in the <b>Location</b> tab of a property sheet.
SmartPart Data	Access-restricted if Attributes has been selected in the Action section of this dialog box.  Search for SmartParts using criteria in the <b>Construction</b> tab of a property sheet.   <b>Tip:</b> To search for primitives, that is, Cuboids, Prisms, Tets, and Inverted Tets, select Common and use a Criteria of Geometry Type.
Attribute Data	Search for objects and SmartParts that have attribute attachments using criteria in the attribute's property sheet.
Common Results Data	Search all results tables based on one or more criteria.
Geometry Results Data	Search the results tables of SmartParts.
Type	(SmartPart Data, or Geometry Results Data) Select from a dropdown list of SmartParts.  (Attribute Data) Select from a dropdown list of Attributes.
Results Type	(Geometry Results Data) If applicable, select from a dropdown list of available results tables for the selected Type (SmartPart).

Field	Description
Criteria	Select from a dropdown list of context-sensitive criteria. For example, if you have selected Attribute Data and Material attribute type, then the context-sensitive criteria are the fields in the Material attribute property sheet, that is, Thermal Conductivity Type, Input Method, Conductivity, and so on.  For SmartParts, the criteria available are usually all of the parameters defined in the <b>Construction</b> tab of the property sheet, however, some SmartParts have restrictions, see Restrictions and Limitations under <a href="#">Complex Searches for Objects or Attributes</a> .  For results tables searches, the criteria available are Surface Temperature, Temperature, Bn, and Volume Flow Rate.
<b>Criteria</b>	
Criteria List	This list is populated by selecting Criteria and clicking the + button. When you have added a criterion, you can set context-related operations and values. The default units of values are those defined in the Global Units dialog box.
Match All	Set when all listed criteria are required to be matched.
Match Any	Set when one or more of the listed criteria are required to be matched.
Clear All	Click to empty the Criteria List.
<b>Action</b>	
Select	The action taken when <b>Find</b> is clicked.  Left-hand list: <ul style="list-style-type: none"><li>• Select All — Select all objects that match the criteria.</li><li>• Cycle Select — Select the first object that matches the criteria. Use the &lt;&lt; and &gt;&gt; buttons to select other matches.</li></ul> Right-hand list: <ul style="list-style-type: none"><li>• Geometry — Restrict searches to the data tree.</li><li>• Attributes — Restrict searches to the Project Attributes.</li></ul>
Search	Use to limit the extent of the search: <ul style="list-style-type: none"><li>• Model — Search the whole model.</li><li>• Current Selection — Only search what is currently selected.</li></ul>
Filter	Removes all objects from the data tree and drawing board that do not match the search criteria except found objects, their parent assemblies and their child objects.
Results Count	Shows the number of objects that match the criteria when <b>Find</b> is clicked.

## Related Topics

[Complex Searches for Objects or Attributes](#)

[Find Dialog Box - Quick Criteria Tab](#)

[Searching Results Tables](#)

# Global Units Dialog Box

To access: **Edit > Units**

Use this dialog box to set the default units used throughout the system, for example, as shown in SmartPart property sheets.

## Description

The units for length set here are also used by MCAD Bridge when exporting Simcenter Flotherm geometry in MCAD file formats.

## Objects

The dialog box includes a list of all the measurement types. For each measurement type there is a dropdown list of selectable units.

## Related Topics

[Changing Global Units](#)

## License Timeout Dialog Box

To access: Automatically displayed in installations that use a license server and connection with the license server has timed out.

Use this dialog box to reacquire the connection to the license server or exit Simcenter Flotherm.

### Objects

None.

# Load Project Dialog Box

To access:

- Press Ctrl+L.
- **Project > Load**

Use this dialog box to load a previously saved project, to delete a project, to unlock a project or to catalog a Solution Directory.

## Description

If there are outstanding edits to the current project, you are given the option to either **Save** or **Discard** these before the Load Project dialog box is opened.

A table lists the projects in the currently selected Solution Directory. If the table appears incomplete, click **Catalog** to update the list.

The Project Name, Date Created and Date Saved are shown for each project. The first two columns of the table denote, by icons, that:

-  Results are available.
-  Command Center Input Variables have been defined.

## Objects

Field	Description
Title	The project title of the currently selected project. The title is defined in the Save Project dialog box when a project is saved.
Solution Directory	The path of the folder containing the current list of projects. UNC paths are not supported.
Notes	Click to open the Notes dialog box for the currently selected project. Notes are added in the Save Project dialog box when a project is saved.
Browse	Click to open the Project Solution Directory dialog box to change the currently selected Solution Directory.
Delete	Click to delete the currently selected project.
Unlock	Click to unlock the currently selected project.
Catalog	Click to catalog all projects in the currently selected Solution Directory.

## Usage Notes

Access is unavailable during a solution to prevent any conflict between projects and solution results. Conversely, when this dialog box is open, solution is prevented.

## Related Topics

- [Loading an Existing Project](#)
- [Changing the Solution Directory](#)
- [Cataloging Projects](#)
- [Deleting Projects](#)

# Message Window

To access: **Window > Message Window**

This window reports general errors, warnings, and information.

## Objects

Object	Description
Information, Warning, and Error filter check boxes	The check boxes control the displayed list. Uncheck a check box to remove that type of message from display.
<b>Clear</b> button	Permanently removes all messages from the window. The messages are maintained during a program session unless they are cleared.

## Usage Notes

- To dock the window under the drawing board, double-click the title bar.
- Messages generated during initialization or solution are appended to the contents of the error list, it is therefore recommended to regularly clear the list.
- The **Help** icon for this window is only visible when the window is undocked.

## Related Topics

[Messages](#)

## New Project Dialog Box

To access:

- Ctrl+N.
- **Project > New.**

Use this dialog box to create a new project from an existing template. You can also set the default template which is loaded when you start Simcenter Flotherm.

### Description

If there are outstanding edits to the current project, you are given the option to either **Save** or **Discard** these before the New Project dialog box is opened.

The dialog box consists of a series of tabbed panels which each display a list of project templates to choose from. The current, default project type is selected when the dialog box opens.

The templates are organized in categories, each category has a separate tab.

The icons indicate whether templates have been saved with or without results by the presence or absence of a thermal plot.

### Objects

Object	Description
<b>Application Examples</b> tab (where installed)	Practical examples of Simcenter Flotherm modeling techniques.  By default, this tab and the application examples that it contains are not created during installation but you can create it and download the application examples by following the steps in “ <a href="#">Installing the Application Examples</a> ” on page 60.
<b>Defaults</b> tab	DefaultSI defines a space of 27 m <sup>3</sup> . DefaultUS defines a space of 1000 ft <sup>3</sup> . In both defaults, the space is filled with air at 20°C 50% RH and pressure of 1 atmosphere.
<b>atx</b> tab	Standard form factor template for ATX PC designs.
<b>btx</b> tab	Standard form factor template for BTX PC designs.
<b>Set Default</b>	Click to make the selected template the default template when Simcenter Flotherm opens.
<b>Delete</b>	Click to delete the selected template.

## Related Topics

[Saving a Project as a Template](#)

[Setting the Default Project](#)

[Create Category Dialog Box](#)

## Notes Dialog Box

To access: Open the Load Project dialog box (Ctrl+L) then click **Notes**.

Use this dialog box to read any reference notes attached to the current project.

### Objects

- A read-only text box.
  - Reference notes created when saving the project.  
If the project is a zoomed-in project, then the notes will also contain mass and heat flow data from the original project.

### Related Topics

[Save Project Dialog Box](#)

[Creating a Zoom-In Project](#)

# Save Project Dialog Box

To access: **Project > Save As** or, if the project is being saved for the first time, press **Ctrl+N** or click the **Save** icon.

Use this dialog box to save a project with the name you specify. The results, if any, are also saved.

## Note

-  To save a project, including its full history and solution data, to the external file system for transfer across systems, you must export it as a **\*.pack** file. See “[Exporting Projects in SAT or IGS Files](#)” on page 70.

## Objects

Field	Description
Project Name	Sets the name of the project, up to 32 characters long. Avoid using special characters, although they can be used, they are replaced by underscore characters in the filename and may not be recognized in batch commands.
Title	Sets the project title intended to later help identify the project.
Notes	Add reference information to the project. Any notes added here will be displayed in the Notes dialog box when the project is loaded.
Save As Template	Requests the project is saved as a template.
Category	(Save As Template checked) Assigns the saved template to this category. The default categories are Application Examples, Defaults, atx, and btx.
Create	(Save As Template checked) Click to open the <a href="#">Create Category Dialog Box</a> to create a new template category.
Solution Directory	(Save As Template unchecked) The directory that will hold the project data. To change the directory, click “Browse” to open the Solution Directory dialog box. The saved project becomes available from the New Project dialog box.
Save With Results	You can choose to save the project with or without results. Saving with Results creates a larger file.

## Usage Notes

If you enter the name of an existing project you will be prompted to confirm the save which will overwrite the named project.

## Related Topics

[Load Project Dialog Box](#)

[New Project Dialog Box](#)

[Saving a Project](#)

[Saving a Project as a Template](#)

## Thermal Netlist Preferences Dialog Box

To access: The dialog box is opened when exporting a thermal netlist.

Use this dialog box to request an acceptable relative error value.

### Objects

Object	Description
Acceptable Relative Error	The requested acceptable relative error. The default value of 0.001 is recommended for most applications. Larger values (up to 0.01) can be used to accelerate the extraction process. Smaller values (down to 0.0001) can be used when accuracy is paramount, however, use with caution, as this will impact on the calculation time required for extraction.

### Related Topics

[Exporting Thermal Netlists and BCI-ROM Files](#)

## Thermal Netlist/BCI-ROM Export Progress Dialog Box

To access: Displayed during the export of a thermal netlist or BCI-ROM data.

This dialog box shows solver progress data during an export, and closes when the export of data has completed.

### Objects

Field	Description
Thermal Netlist/BCI-ROM Export	Read only. A percentage estimate of how far the export has progressed.
Start Time	Read only. The time the export was started.
Elapsed Time	Read only. The elapsed time during which the solver has been running.
Elapsed CPU	Read only. The recorded actual processing time of all utilized cores/CPUs

### Related Topics

[Exporting Thermal Netlists and BCI-ROM Files](#)

# User Preferences Dialog Box

To access:

- **Edit > User Preferences.**
- Click the **User Preferences** icon.

Use this dialog box to set your preferences. The dialog box has four tabs.

## Objects

Object	Description
<b>Project Manager</b> tab	See “ <a href="#">User Preferences Dialog Box - Project Manager Tab</a> ” on page 128.
<b>Drawing Board</b> tab	See “ <a href="#">User Preferences Dialog Box - Drawing Board Tab</a> ” on page 130.
<b>Summary</b> tab	See “ <a href="#">User Preferences Dialog Box - Summary Tab</a> ” on page 132.
<b>Analyze Mode</b> tab	See “ <a href="#">User Preferences Dialog Box - Analyze Mode Tab</a> ” on page 133.

## Usage Notes

- To restore the default settings within one of the tabs, open the tab, click **Restore Defaults**, then click **OK**.
- User preference settings are saved to the following configuration file and read on start-up.

*<install dir>\flosuite\_v<version>\flotherm\config\<user\_name>-<user\_id>.cfg*

## User Preferences Dialog Box - Project Manager Tab

To access:

- **Edit > User Preferences.**
- Click the **User Preferences** icon.

The **Project Manager** tab is displayed by default.

Use this tab to set preferences for coordinates, number of processors to use, and project manager behavior.

### Objects

Object	Description
Display Positions in	X, Y, Z position coordinates in object property sheets can be displayed as: <ul style="list-style-type: none"><li>• Absolute Coordinates, which are relative to the model origin, or</li><li>• Local Coordinates, which are relative to the origin of the current assembly. A setting of local coordinates is ignored for system, initial, and grid values which are always given in absolute coordinates.</li></ul> See “ <a href="#">Coordinate Systems</a> ” on page 215.
Number Of Processors To Use	Set to the number of processors on your machine that you want to use. If 1 then serial solvers will be used. If greater than 1 then parallel solvers will be used.
On Object Creation	When an object is created you can set your preference to either select the new object or leave the current selection active.
On Object Deletion	When an object is deleted you can set your preference to either select the deleted object’s neighbor or have nothing selected.
Automatic Application Window Open	When you exit a program session with this field set to On, any windows open at that time are re-opened automatically at their last location and size when you re-start Simcenter Flotherm again. If set to Off, only the PM is opened when you start Simcenter Flotherm, and will be located as you last left it. When you open other windows, they appear at their last location and size.
Tables Velocity View Stagger	Sets the flow velocity representation for tables in Analyze mode as either the cell edge discretized value (staggered) or a cell centered average.
Show Ignored Geometry	Make ignored geometry visible in the data tree, but highlighted in red. Geometry is marked as ignored by activating <b>Ignore Geometry</b> in the Assembly property sheet.

<b>Object</b>	<b>Description</b>
Configure IDF	Enables you to set up your preferences for IDF file conversion.
Conductivity (Configure IDF)	
Board (X/Y)	Sets the default conductivity of the board.
Board (Z)	Sets the default conductivity of the board.
Package	Sets the default conductivity of the package.
Component Filter (Configure IDF)	
Filter On	Filters out small components.
Minimum (X/Y)	Filter components that have a width or length below this value.
Minimum (Z)	Filter components that have a height below this value.
Zero Thickness (Configure IDF)	
Board	The board thickness to use if the board is imported with zero thickness.
Package	The package thickness to use if the package is imported with zero thickness.
Component (Configure IDF)	
Offsets	Ensures that any component mounting offset data (available only in IDF 3.0) files will be accounted for in the physical height assigned to the cuboids representing the mechanical and electrical parts.
Descriptors	Ensures that the names of the imported data will use the following convention: <ul style="list-style-type: none"> <li>• The IDF filename is assigned to the geometry object</li> <li>• The assembly is named <i>idfboard_id</i></li> <li>• The PCB SmartPart is named <i>board_id</i></li> <li>• Mechanical components are named: <i>geometry_id_part_id_package_index</i> where <i>package_index</i> is the rank of the package in the input file.</li> </ul>

## Related Topics

[Importing IDF Files](#)

## User Preferences Dialog Box - Drawing Board Tab

To access:

- **Edit > User Preferences**, then select the **Drawing Board** tab.
- Click the **User Preferences** icon, then select the **Drawing Board** tab.

Use this tab to control the display of the snap grid, monitor points, regions, flow/source direction, gravity, solar vectors, and background color of the GDA. The settings are retained when Simcenter Flotherm closes.

### Description

Unless otherwise stated, these settings also apply to the GDA when in Analyze mode.

### Objects

Object	Description
Snap Grid Size	Sets the gap between the snap grid lines used for aligning objects when dragging or drawing in the GDA. The finer the detail required, the smaller the grid size.
Snap Grid Display Interval	The number of lines to skip before displaying the next grid line. If too fine a grid is set, a warning will be issued, and the grid is not drawn.
Display Monitor Points	Display all monitor points below the current assembly
Display Volume Regions	Display all regions below the current assembly.
Display Subdomains	Display subdomains, in orange. Default checked.  <b>Note:</b> This can also be set on the <b>Analyze Mode</b> tab (see “ <a href="#">User Preferences Dialog Box - Analyze Mode Tab</a> ” on page 133).
Display Solution Domain	Display the solution domain, in yellow. Default checked.  <b>Note:</b> This can also be set on the <b>Analyze Mode</b> tab (see “ <a href="#">User Preferences Dialog Box - Analyze Mode Tab</a> ” on page 133).
Display Workplanes	Display the workplane, in dark gray. Not applicable when in Analyze mode.
Display Flow/Source Direction	Display arrows attached to flow or source objects indicating their supply directions. The arrow does not appear for sources without an attached source attribute.
Display Local Axis	Display the local axes of selected objects.

<b>Object</b>	<b>Description</b>
Display Gravity Vector	Display an arrow (g) showing the direction of gravity.
Display Solar Vector	Display an arrow (s) showing the direction of short wave solar radiation.
Gradient Background	Sets the background of the GDA to be merged from the top background color to the bottom background color. If unchecked, the drawing board background is a uniform color.  For the GDA in Analyze mode, use the equivalent option in the <b>Analyze Mode</b> tab of this dialog box.
Background Color	The background color of the GDA.  For the GDA in Analyze mode, use the equivalent option in the <b>Analyze Mode</b> tab of this dialog box.
Top Background Color	(Gradient Background) The background color at the top of the GDA.  For the GDA in Analyze mode, use the equivalent option in the <b>Analyze Mode</b> tab of this dialog box.
Bottom Background Color	(Gradient Background) The background color at the bottom of the GDA.  For the GDA in Analyze mode, use the equivalent option in the <b>Analyze Mode</b> tab of this dialog box.
Reverse Mouse Wheel Zoom Direction	Applies to the GDA view currently under the mouse pointer. Unchecked – Rotating the mouse wheel towards you zooms into the view.  Checked – Rotating the mouse wheel towards you zooms out of the view.

## Related Topics

[Snap Grid](#)

## User Preferences Dialog Box - Summary Tab

To access:

- **Edit > User Preferences**, then select the **Summary** tab.
- Click the **User Preferences** icon, then select the **Summary** tab.

Use this tab to control the displayed summary information. The settings are retained when Simcenter Flotherm closes.

### Objects

Object	Description
De-keypointing % Tolerance	Hide de-keypointed flags that are because of extremely small differences between grid lines. The value is the percentage of the object's dimension in any axis (see Usage Notes).
Power	Select to display the power value.
De-keypointed	Select to display whether the object has been de-keypointed or not.
Size	Select to display the size of the object.
Project attributes: Material, Surface, Resistance, Ambient, Thermal, Source, Radiation, Surface Exchange, Fluid, and Grid Constraint.	Select to display the attribute attachment.
Cold Aisle Group, Cold Aisle Sub-Group, Hot Aisle Group, Hot Aisle Sub-Group, and Cooler Group.	Select to display the group membership.

### Usage Notes

If an object is 5 mm in the X direction and De-keypointing % Tolerance is set to 0.1, then the gap between de-keypointed grid lines must be greater than 0.005 mm in the X direction before de-keypointing is flagged in the summary information. If the value is set to 1, then the grid line gap must be 0.05 mm before it is flagged.

### Related Topics

[Viewing Summary Information](#)

[Keypoint Deactivation \(De-Keypointing\)](#)

## User Preferences Dialog Box - Analyze Mode Tab

To access: F11, or choose **Edit > User Preferences**, or click the **User Preferences** tool icon, then select the **Analyze Mode** tab.

Use this tab to specify the resolution of GDA snapshots, change font sizes when the GDA is in Analyze mode, and change the rendering of animations.

### Objects

Object	Description
Output Snapshot use Screen Resolution	Use the screen resolution when creating a GDA snapshot.
Snapshot Width	(Output Snapshot use Screen Resolution unchecked) User-specified resolution width, in pixels.
Snapshot Height	(Output Snapshot use Screen Resolution unchecked) User-specified resolution height, in pixels.
Ambient Light Level	Sets the intensity of the minimum light falling on the model. Most noticeable when viewing 3D Isosurface plots.
Legend Font Size	Controls the size of the legend font in the GDA views when in Analyze mode.
Title Font Size	Controls the size of the project name in the GDA views when in Analyze mode.
Show Title	Shows/Hides the project name in the GDA views when in Analyze mode.
Animation	
Number of Frames	Total number of frames in an animation. Default 18.
Frames per Second	Speed of an animation in frames per sec (fps). Default 30.  <b>Note:</b> It may not be possible to render the plot at the selected fps, it is a target value.
Swing	Sets animations to loop continuously in alternate directions. The plot location or variable values travel from one side of the animation range to the other. When it reaches the end of the range, it reverses its direction and travels back towards the start of the range.
Display Subdomains	Display subdomains, in orange. Default unchecked.  <b>Note:</b> This can also be set on the <b>Drawing Board</b> tab (see “ <a href="#">User Preferences Dialog Box - Drawing Board Tab</a> ” on page 130).

Object	Description
Display Solution Domain	<p>Display the solution domain, in yellow. Default unchecked.</p> <p> <b>Note:</b> This can also be set on the <b>Drawing Board</b> tab (see “<a href="#">User Preferences Dialog Box - Drawing Board Tab</a>” on page 130).</p>
Display Geometry Without Expanding Tree	<p>When unchecked, geometry of assemblies that are collapsed in the data tree is not shown in the GDA - instead, an outline of the assembly is shown (as a green wireframe box). When checked, geometry of all assemblies is shown in the GDA, regardless of whether they are expanded or collapsed in the data tree. Default unchecked.</p>
Gradient Background	<p>When in Analyze mode, sets the background of the GDA to be merged from the top background color to the bottom background color. If unchecked, the background is a uniform color.</p> <p>For the GDA in Create mode, use the equivalent option in the <b>Drawing Board</b> tab of this dialog box.</p>
Background Color	<p>When in Analyze mode, the background color of the GDA. For the GDA in Create mode, use the equivalent option in the <b>Drawing Board</b> tab of this dialog box.</p>
Top Background Color	<p>(Gradient Background) When in Analyze mode, the background color at the top of the GDA.</p> <p>For the GDA in Create mode, use the equivalent option in the <b>Drawing Board</b> tab of this dialog box.</p>
Bottom Background Color	<p>(Gradient Background) When in Analyze mode, the background color at the bottom of the GDA.</p> <p>For the GDA in Create mode, use the equivalent option in the <b>Drawing Board</b> tab of this dialog box.</p>
Number of Significant Figures	<p>The number of significant figures, between 1 and 7, displayed in results tables, annotations, and plot legends. Default 3.</p>
Marginal Temperature Range	<p>Applicable to EDA Components when reporting results. The margin, in degC, between the calculated junction or case temperature and a specified maximum which reports a Grade as “Marginal”, see “<a href="#">EDA Components Results Table</a>” on page 508. Default 5 degC.</p>

## Usage Notes

You must close the dialog box to effect any changes made.

## Related Topics

[Changing the Appearance of the GDA in Analyze Mode](#)

[Saving a GDA Image to a Graphics File](#)

## Zoom-In Creation Dialog Box

To access: Select a Region that can become a zoom-in project and choose **Create Zoom-In** from the context-sensitive popup menu.

Use this dialog box to create and name a zoom-in project. The dialog box also displays the number of cells on the region boundary.

### Objects

Field	Description
Project Name	A unique name for the project. The default name is ZI_<region> where <region> is the name of the volume region used to create the zoom-in project.

### Related Topics

[Creating a Zoom-In Project](#)

# Chapter 4

## Defining the Mathematical Model

---

Simcenter Flotherm requires defining a region of space (solution domain) within which mathematical model equations are to be solved. Modeling the solution domain requires the setting of the solution domain dimensions, boundary conditions, and the filling fluid properties.

<b>The Modeling Method .....</b>	<b>138</b>
<b>The Solution Domain.....</b>	<b>140</b>
Solution Domain Overall Definitions .....	140
Moving and Resizing the Solution Domain in the Drawing Board .....	140
Boundary Types .....	141
Non-Cuboid Solution Domains .....	141
Examples of Open and Symmetry Boundaries.....	141
Symmetry and Radiation .....	142
<b>Initial Values .....</b>	<b>143</b>
Initial Values and Solution Type .....	143
The Solution Variables .....	143
Starting From a Previous Solution Set .....	144
Viewing Initial Fields .....	145
Setting Initial Field Values Over Subdomains .....	146
<b>Solution Variable Types .....</b>	<b>147</b>
Base and Standard Derived Variables .....	147
Auxiliary Variables.....	147
Project Solution Variables Procedures .....	148
<b>Modeling Selections .....</b>	<b>150</b>
Types of Solution .....	150
Mass Flux .....	150
Heat Flux.....	151
Surface Temperature.....	152
Temperature Gradient .....	153
Power Density.....	153
Bn and Sc .....	155
<b>Modeling Dimensionality .....</b>	<b>156</b>
<b>Capture Index .....</b>	<b>159</b>
Concept of the Capture Index .....	159
Data Center Aisle Groups and Sub-Groups .....	160
Aisle Group CI Calculation .....	161
Assessing CI Values for Data Center Design .....	162
Capture Index Procedures .....	164

<b>Solar Radiation</b>	.....	<b>168</b>
Solar Radiation Model	.....	168
Including Solar Radiation	.....	168
<b>Airflow</b>	.....	<b>170</b>
Laminar Airflow	.....	170
Turbulent Airflow and Models	.....	170
Revised Algebraic Model	.....	171
Automatic Algebraic Model	.....	172
LVEL K-Epsilon Model	.....	172
<b>Gravity</b>	.....	<b>175</b>
Modeling Gravity	.....	175
Calculation of Gravity	.....	176
<b>Changing the Fluid Properties</b>	.....	<b>178</b>
<b>Reference Temperature and Pressure</b>	.....	<b>180</b>
Datum Pressure	.....	180
Modeling a Remote Radiation Source	.....	180
Ambient Temperature When Modeling Buoyancy Force	.....	182
Default Initial Value for Temperature	.....	182
Transient Variation of Temperature	.....	182
<b>Joule Heating Analysis</b>	.....	<b>183</b>
Joule Heating Modeled by Source SmartParts	.....	183
Joule Heating and Power Map SmartParts	.....	183
<b>Solution Domain Dialog Boxes and Property Sheets</b>	.....	<b>184</b>
System Property Sheet	.....	185
Subdomain Property Sheet	.....	187
Solar Configuration Dialog Box	.....	189
Model Setup Tab	.....	191
Solver Control Tab	.....	200
Solution Set Dialog Box	.....	208
Solution Type Changed Dialog Box	.....	209

## The Modeling Method

The solution procedures are based on CFD techniques.

These techniques require the definition of a mathematical model to describe:

- Solution region and its boundary conditions
- Properties of the fluid flow
- Variables to be solved
- Dimensionality of the model (2D or 3D)

- Solution type
- Any effects of gravity
- System temperature and pressure values
- Any radiation effects

## Project Defaults

Simcenter Flotherm provides default values to get you started. You can start from an empty space or a design type using **Project > New** to open the [New Project Dialog Box](#), then choosing a project template. For example, choosing **DefaultSI** from the New Project dialog box models a 1 m<sup>3</sup> space filled with air at 30°C and pressure of one atmosphere; if **DefaultUS** is chosen instead, the project space defaults to 8 ft<sup>3</sup>.

In both cases, the solution type is steady state, solving for temperature, pressure, and velocities, and simulating heat flow in three dimensions. Radiation effects are not included in the default model. Gravity is on and acting in the negative Y-direction.

Although the default model settings are applicable for a wide range of cases, it is common to change the settings to suit your specific requirements. For example, activating radiation, or swapping the LVEL K-Epsilon turbulence model for the Revised algebraic model.

## Main Source of Error

The mathematical models you set up are used to simulate the physical system being represented. As such, it is critical that they accurately reflect the physical reality. Errors in the input to the mathematical models are often the largest source of error in an analysis, great care should, therefore, be taken to ensure the quality of the input data.

# The Solution Domain

Simcenter Flotherm provides solution for the parts of the model contained within the solution domain. You can edit the solution domain to meet the scope of your model.

<b>Solution Domain Overall Definitions .....</b>	<b>140</b>
<b>Moving and Resizing the Solution Domain in the Drawing Board .....</b>	<b>140</b>
<b>Boundary Types .....</b>	<b>141</b>
<b>Non-Cuboid Solution Domains .....</b>	<b>141</b>
<b>Examples of Open and Symmetry Boundaries .....</b>	<b>141</b>
<b>Symmetry and Radiation .....</b>	<b>142</b>

## Solution Domain Overall Definitions

Defining the solution domain size, location, boundaries, external ambient conditions and reference temperatures and pressure.

The solution domain shape defaults to a cuboid.

The solution domain size and location, boundary face types, and external ambient conditions are defined using the [System Property Sheet](#).

The reference temperatures and pressure are defined using the [Model Setup](#) tab.

The solution domain grid constraints are defined using [System Grid Property Sheet](#).

By default, the origin of the solution domain is coincident with the geometry origin. Changing the position and the size of the solution domain enables you to focus on particular sections of the geometry. Alternatively, you can extend the solution domain beyond the geometry to examine the environment outside the model.

## Moving and Resizing the Solution Domain in the Drawing Board

Moving or changing the size of the solution domain in the drawing board can be faster and more intuitive than entering values in the System property sheet.

### Procedure

1. Select the System node in the data tree.

The solution domain is shown in red and pick points are displayed. When unselected, the solution domain is shown in yellow.

2. Make sure you are in Select mode.

If you want to...	Do the following:
Move the solution domain.	<ul style="list-style-type: none"> <li>Hover over a pick point until the mouse cursor changes to a pair of crossed double-headed arrows.</li> </ul>
Resize the solution domain.	<ul style="list-style-type: none"> <li>Hover over a pick point until the mouse cursor changes to a single double-headed arrow in the direction you want to resize. You may need to change views to get the direction you want.</li> </ul>

3. Select and drag the boundary/domain as required.

## Results

The position and/or size dimensions are updated in the **Location** tab of the System property sheet.

## Related Topics

[Changing Mouse Mode in Create Mode](#)

## Boundary Types

By default, the six boundaries of the solution domain are treated as open boundaries.

Open boundaries are boundaries of constant pressure through which air or, if blocked by a conducting object, heat can flow.

Use the [System Property Sheet Boundaries Tab](#) to set up impermeable symmetry boundaries.

## Non-Cuboid Solution Domains

For non-cuboid models, you can create cutouts to remove sections of the solution domain.

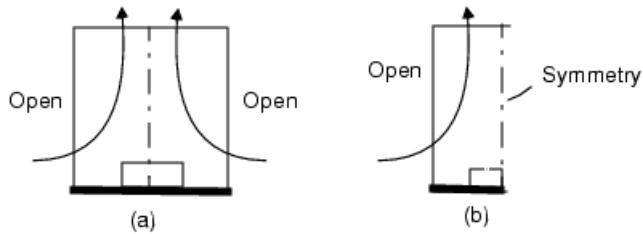
See [Cutouts](#) in the *Simcenter Flotherm SmartParts Reference Guide*.

## Examples of Open and Symmetry Boundaries

How symmetry can be used to simplify a model.

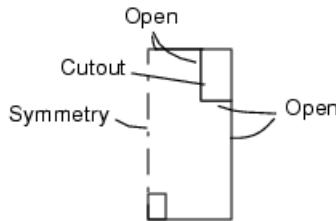
In [Figure 4-1](#), (a) models an overall boundary open on three sides. As it is symmetrical, only half the model needs solving; this is achieved using a symmetry plane, (b).

**Figure 4-1. Open and Symmetry Boundaries**



Cutouts on the solution domain boundary can also be included in a symmetry model as shown in Figure 4-2.

**Figure 4-2. Cutouts on the Solution Domain Boundary**



---

**Note**

 The two outer surfaces of the cutout are irrelevant and do not need to be considered as Open or Symmetry.

---

The drawing board distinguishes between both types of boundary using color:

- Dark Gray for Symmetry.
- Yellow for Open.

## Symmetry and Radiation

For radiation, symmetry means that every object within the domain has a mirror image on the other side of the symmetry face.

As a result, we consider the radiation between all user-defined, radiating, solid objects and all their mirror images. The exchange factors are automatically modified to add the total effect into the analysis.

### Related Topics

[The Solution Domain](#)

[Exchange Factors](#)

# Initial Values

---

The following topics discuss the initial field values of the model.

<b>Initial Values and Solution Type .....</b>	<b>143</b>
<b>The Solution Variables .....</b>	<b>143</b>
<b>Starting From a Previous Solution Set .....</b>	<b>144</b>
<b>Viewing Initial Fields.....</b>	<b>145</b>
<b>Setting Initial Field Values Over Subdomains .....</b>	<b>146</b>

## Initial Values and Solution Type

For steady state, the initial field values will not affect the final converged results, but a good initial guess will reduce the time taken to reach a solution convergence. However, for the majority of cases, the defaults are normally sufficient.

In transient calculations, the set of initial conditions are not a guess at the final solution, rather they form a part of the problem specification itself. For transient calculations, the initial fields must be set to the conditions that are known to prevail at the start of the transient.

### Initialization

The simplest method of initialization is to set the initial variable value constant over the complete domain using the [Solver Control Tab - Variable Solution Control Section](#).

You can set the initial variables manually on a variable-by-variable basis or load in the results from a previous project using the [Solution Set Dialog Box](#).

## The Solution Variables

Solution variables are calculated during the solution.

The following is a list of the solution variables with meanings and default (automatic) values for their initial fields:

- Pressure — Field of pressures, stored and solved by default. The default initialization value is  $10^{-10}$ . Not applicable when Type of Solution is Conduction Only.
- X Velocity — Field of x-directed velocity resolutes, stored and solved by default. The default initialization value is  $10^{-10}$ . Not applicable when Type of Solution is Conduction Only.
- Y Velocity — Field of y-directed velocity resolutes, stored and solved by default. The default initialization value is  $10^{-10}$ . Not applicable when Type of Solution is Conduction Only.

- Z Velocity — Field of z-directed velocity resolutes, stored and solved by default provided dimensionality is 3D. The default initialization value is  $10^{-10}$ . Not applicable when Type of Solution is Conduction Only.
- KE Turb — Field of Kinetic Energy of turbulence, activated by selecting the LVEL K-Epsilon turbulence model. The default initialization value is program calculated.
- Diss Turb — Field of dissipation of turbulence, activated by selecting the LVEL K-Epsilon turbulence model. The default initialization value is program calculated.
- Temperature — Field of temperatures, stored and solved provided it is selected as the thermal variable.

The default initialization value is the external ambient temperature set in the Global Units dialog box, the default of which is 35°C. Not applicable when Type of Solution is Flow Only.

- Density — Density field, stored when the Ideal Gas Law option is selected as a Density Type in the Fluid attribute property sheet. The default initialization value is the ambient density.
- Turb Vis — Turbulent Viscosity field, activated when any of the turbulence models is activated.
- Potential — Electrical potential field, activated when Joule Heating is enabled.

In addition to the solved variables (like velocities, temperature, and pressure), access to other field variables is also given in this dialog box. For example, if the ideal gas law option has been selected for the fluid property, Density will appear towards the end of the variable list. Density refers to the field of densities that is stored, and which, therefore, can be initialized.

## Related Topics

[Solver Control Tab - Variable Solution Control Section](#)

[Model Setup Tab - Turbulence Section](#)

[Subdomain Property Sheet Initial Values Tab](#)

[Joule Heating Analysis](#)

## Starting From a Previous Solution Set

You can start a solution using initial variable values obtained from a previously solved project (solution set).

### Restrictions and Limitations

- If the selected solution set has a different grid from the current project, the automatic grid interpolation will be invoked when the project is re-initialized.

- If a variable cannot be initialized from the solution set, it is set to the default automatic initialization value.

## Procedure

1. In the **Solver Control** tab, Initial Variables section, select Selected Solution Set as the Initial Value.
2. Click **Select** to open the Solution Set dialog box, select a project, and click **OK** to close the dialog box.

The Solution Set field is updated with the project name.

When you do this, all variables will take their initial values from the solution set. If you do not want a variable to use the solution set, proceed as follows.

- a. Select the variable in the **Solver Control** tab (Variable field).
  - b. Select either Automatic or User Specified as the Initial Value.
  - c. If User Specified, then enter a value.
  - d. Repeat for each variable that you do not want to load from the predefined solution set.
3. Re-Initialize the project before solving.

## Related Topics

[Solver Control Tab - Variable Solution Control Section](#)

[Solution Set Dialog Box](#)

# Viewing Initial Fields

How to check the initial values of variables before a solution is started.

## Procedure

1. Choose **Solve > Re-Initialize**.  
The fields are initialized and loaded into program memory according to your settings.
2. The initial values are shown in property sheets of variables in the Scalar Fields node of the Results Tree (Analyze mode).

## Related Topics

[Solver Control Tab - Variable Solution Control Section](#)

[Subdomain Property Sheet Initial Values Tab](#)

## Setting Initial Field Values Over Subdomains

You can initialize variables over sub-regions of the solution domain, within the solution domain and specify initial values of variables within those subdomains.

### Procedure

1. To create a subdomain, select System in the data tree and click the **Initial Subdomain** icon in the New Object Palette.

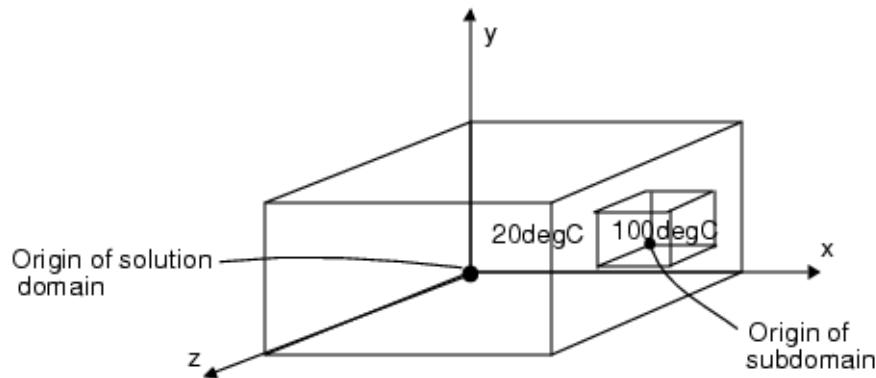
If the subdomain is created using the Project Manager, it is one-tenth the size of the solution domain and located at its origin.

Subdomains are rendered as a peach-colored cuboids in the drawing board. Visibility of Subdomains in the drawing board is controlled by the Display Subdomains check box in the **Drawing Board** tab of the User Preferences dialog box.

2. Define the location and size of the subdomain, and then use the **Initial Values** tab of the property sheet to define the initial value for each selected variable.

[Figure 4-3](#) shows a solution domain in which a subdomain has been added. The value of temperature is set to 20°C in the overall domain, and to 100°C in the subdomain.

**Figure 4-3. Initial Values in Subdomain**



### Related Topics

[Subdomain Property Sheet](#)

[Solver Control Tab - Variable Solution Control Section](#)

# Solution Variable Types

The following topics describe types of solution variables, and how to obtain them.

<b>Base and Standard Derived Variables .....</b>	<b>147</b>
<b>Auxiliary Variables .....</b>	<b>147</b>
<b>Project Solution Variables Procedures .....</b>	<b>148</b>

## Base and Standard Derived Variables

Simcenter Flotherm calculates the base variables (temperature, pressure, and velocities) during solution depending on the type of model chosen.

Additional derived variables (speed, heat, mass fluxes, surface temperature, capture index) can also be derived from the base variables during solution and stored.

For example, heat fluxes can be stored to provide the data for the calculation of the total flow rate and heat flux respectively, across defined planar and volume regions.

## Auxiliary Variables

After solution, auxiliary variables can be calculated from the base and derived variables.

### Related Topics

[Additional Auxiliary Variables](#)

## Project Solution Variables Procedures

The following procedures relate to the solution variables.

Determining Base Solution Variables .....	148
Determining Additional Derived Variables.....	148

### Determining Base Solution Variables

The base solution variables are the variables that are automatically selected for calculation, and are dependent on the type of solution you choose.

#### Procedure

1. In the Project Manager, open the [Model Setup Tab](#).
2. Select the Type of Solution required:
  - Flow and heat transfer.
  - Flow only, for isothermal cases.
  - Conduction only, for cases made up of primarily solid objects and solving for temperature.
3. Select the Dimensionality:
  - 2D, for cases where the geometry and physical conditions are regular in the third coordinate direction and save computation time.
  - 3D, for more modeling in all three dimensions, but computationally more expensive.
4. Choose whether to include Radiation effects for situations with radiative heat transfer.
5. Select a steady or transient (time-dependent) Solution Type if you are trying to capture a transient effect.
6. Choose to include Solar Radiation if the effects of shortwave radiation on the thermal environment are to be modeled.

### Determining Additional Derived Variables

In addition to the automatically-selected base variables, you can select additional variables to be calculated.

#### Procedure

1. Open the [Model Setup Tab](#).
2. Go to the Capture Index section of the tab and check Calculate to model cold and/or hot aisle groups. Check Sub-Groups to model sub-groups.

3. Go to the Stored Variables section of the tab.

- Check Mass Fluxes to determine the total flow rate going through each board gap in a PCB rack.
- Check Heat Fluxes to carry out a heat flux budget analysis on complex 3D objects, such as detailed models of electronic packages.

## Modeling Selections

The following selections are made using the **Model Setup** tab.

<b>Types of Solution</b> .....	<b>150</b>
<b>Mass Flux</b> .....	<b>150</b>
<b>Heat Flux</b> .....	<b>151</b>
<b>Surface Temperature</b> .....	<b>152</b>
<b>Temperature Gradient</b> .....	<b>153</b>
<b>Power Density</b> .....	<b>153</b>
<b>Bn and Sc</b> .....	<b>155</b>

## Types of Solution

Overviews of Flow and Heat Transfer, Flow Only and Conduction Only.

There are three types of solution:

- Flow and Heat Transfer solves temperature, pressure, and velocities appropriate for the dimensionality selected (the default).
- Flow Only calculates the flow of isothermal (constant temperature) fluid. Any thermal effects are ignored, as are gravity effects.
- Conduction Only simulates the conduction of heat for the entire solution domain filled with stationary (non-flowing) material.

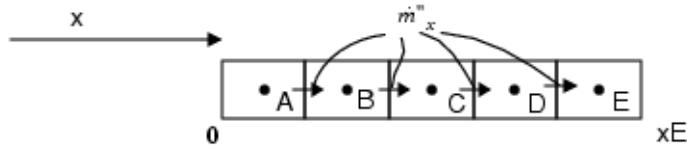
See [Background of Computational Fluid Dynamics \(CFD\)](#) in the *Simcenter Flotherm Background Theory Reference Guide* for details of the variables and their precise storage locations.

## Mass Flux

Mass Flux values are not normally stored, but can be if the Store Mass Flux check box in the **Model Setup** tab is checked.

In the same way as the velocity components and heat transfer coefficients, the mass fluxes are associated with high cell faces of the individual grid cells. This is illustrated in [Figure 4-4](#) for five grid cells spanning the x-direction of the domain.

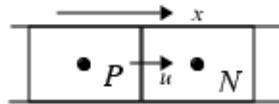
**Figure 4-4. Storage Mass Fluxes**



## Heat Flux

Heat Flux values are not normally stored, but can be if the Store Heat Flux check box in the **Model Setup** tab is checked.

The heat flux is the total heat flux including both convective and conductive transport. For example, consider a grid cell labeled  $P$  and its neighbor cell in the high  $x$ -direction labeled  $N$ . Let  $u$  denote the  $x$ -directed velocity component at the intervening cell face. The total heat flux across this face is calculated as follows:



For  $u \geq 0$ :

$$\dot{J}'' = \dot{m}''_x C_p (T_p - T_{min}) + \dot{q}''_x$$

For  $u < 0$ :

$$\dot{J}'' = \dot{m}''_x C_p (T_N - T_{min}) + \dot{q}''_x$$

where:

$T_p$  and  $T_N$  denote the temperatures at  $P$  and  $N$  respectively.

$T_{min}$  denotes the minimum domain temperature.

$C_p$  denotes the specific heat of the fluid.

$\dot{m}''_x$  and  $\dot{q}''_x$  respectively denote the mass flow per unit area and the diffusive (conductive) heat flow per unit area (as described earlier in this section under heat transfer coefficients).

The switching of temperature according to the sign of the velocity resolute is the embodiment of the upwind principle used in one form or another in most CFD methodologies.

Within and on the surface of cuboidal blocks, the fluid mass flow rate is of course zero so that  $(\dot{J}'_x)$  reduces to the conductive flux,  $\dot{q}'_x$

## Grid Association

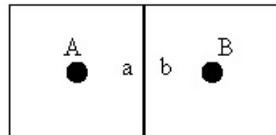
As for velocity resolutes, mass fluxes, and heat transfer coefficients, the total heat fluxes pertain to cell faces, not to cell centers; for a given cell, the value displayed will be that of the high face.

# Surface Temperature

Calculation of surface temperature values.

Surface Temperature values are normally stored, that is, the Store Surface Temperature check box in the **Model Setup** tab is checked by default.

Consider two cells A and B:



Suppose the solver computes the temperature at A and B to be  $T_A$  and  $T_B$  respectively.

The solver computes the surface temperature at a and b as follows.

Suppose the computed heat transfer coefficient between A and B is  $h_{AB}$  and that between A and a is  $h_{Aa}$  and between B and b is  $h_{Bb}$ .

A 1D heat balance gives:

$$h_{AB} (T_A - T_B) = h_{Aa} (T_A - T_a)$$

The only unknown is  $T_a$

Rearranging the equation gives:

$$T_a = [(h_{Aa} - h_{AB}) T_A + h_{AB} T_B] / h_{Aa}$$

Similarly:

$$T_b = [(h_{Bb} - h_{AB}) T_B + h_{AB} T_A] / h_{Bb}$$

## Temperature Gradient

Temperature gradient values are not normally stored, but can be if the GradT check box in the Stored Variables section of **Model Setup** tab is checked.

When these values are stored, they are viewable in Analyze mode.

- Grad T is a vector indicating the size and direction of the steepest temperature gradient calculated from cell centers. It is stored as three scalar field components: X Grad T, Y Grad T, and Z Grad T.
- Mag Grad T, the magnitude of the temperature gradient, is a scalar field stored at cell centers.

## Power Density

Power Density values are not normally stored, but they can be if the Power Density check box in the Stored Variables section of the **Model Setup** tab is checked.

Objects with thermal attributes that are supported by power density calculations are shown in [Table 4-1](#).

Once stored, the calculated values are available in Analyze mode for post-processing such as creating plane plots, and so on.

**Table 4-1. Power Density Support for Objects**

Object	Thermal Attribute	Power Density Supported
Cuboid	Fixed Temperature	No
	Fixed Heat Flow - Power/Area	Yes
	Fixed Heat Flow - Total Power	Yes
	Conduction	Yes
	Joule Heating - Current or Voltage	Yes
Prism	Fixed Temperature	No
	Fixed Heat Flow - Power/Area	Yes
	Fixed Heat Flow - Total Power	Yes
	Conduction	Yes
Sloping Plate	Fixed Temperature	No
	Conduction	Yes
Cylinder	Fixed Temperature	No
	Conduction	Yes

**Table 4-1. Power Density Support for Objects (cont.)**

Object	Thermal Attribute	Power Density Supported
Tet	Fixed Temperature	No
	Fixed Heat Flow - Power/Area	No
	Fixed Heat Flow - Total Power	No
	Conduction	Yes
Inverted Tet	Fixed Temperature	No
	Fixed Heat Flow - Power/Area	No
	Fixed Heat Flow - Total Power	No
	Conduction	Yes
Source	Source/Volume (or Source/Area for a collapsed source)	Yes
	Total Source	Yes
	Fixed Value	No
	Linear Source	Yes
	Non-Linear Source	Yes
PCB Component	Apply over Board	Yes
	Discrete	Yes
	Discrete - Solid Component	Yes
Fan		Yes
Compact Component		Yes
Network Assembly <sup>1</sup>	Conduction	Yes
	Fixed Temperature	No
TEC <sup>2</sup>		Yes
Die	Uniform Dissipation	Yes
	Non-Uniform Dissipation - Discrete Sources	Yes
	Non-Uniform Dissipation - Total Coverage Sources	Yes
Power Map <sup>3</sup>		Yes

1. Power Density for a Network Assembly is calculated for an uncollapsed Network Cuboid, which is a child of a Network Node that has a thermal attribute attached.
2. Only the Joule heating part, that is, the hot side of the TEC, will show Power Density values.
3. Power Maps are modeled as a combination of copper cuboids, each cuboid having an associated power source.

**Note**

 See [Power Maps](#) in the *Simcenter Flotherm SmartParts Reference Guide* and “[Joule Heating Analysis](#)” on page 183.

---

Thermal attributes are attached to objects using the following property sheets:

- Cuboid, Prism, Sloping Plate, Cylinder, Tet, Inverted Tet, Network Node:  
Thermal Attribute property sheet; see [Thermal Attributes](#) in the *Simcenter Flotherm Project Attributes Reference Guide*.
- Volume and Plane Sources:  
Source Attribute property sheet; see [Source Attributes](#) in the *Simcenter Flotherm Project Attributes Reference Guide*.
- PCB Component:  
PCB Component SmartPart property sheet; see [PCB Component Property Sheet](#) in the *Simcenter Flotherm SmartParts Reference Guide*.
- Die:  
Die SmartPart property sheet; see [Die Property Sheet](#) in the *Simcenter Flotherm SmartParts Reference Guide*.

## Bn and Sc

Normalized parameters that give an indication of thermal bottlenecks and paths.

Bn and Sn are not normally stored, but can be if the Bn and Sc check box in Stored Variables section of the **Model Setup** tab is checked. This check box only becomes active when Heat Fluxes and temperature gradient (GradT) values are stored.

- **Bn** — is a normalized parameter that indicates thermal bottlenecks
- **Sc** — is a normalized parameter that indicates possible path of heat flow

These values are available in Analyze mode when stored.

## Modeling Dimensionality

The choice of dimensionality is one of the most important decisions you have to make when performing a CFD calculation.

As to what choice is appropriate depends on what you want from the calculation, that is, whether you require the resolution of 3D effects present in the flow or whether a 2D simulation will provide the necessary information to the accuracy required for the application under consideration.

Selection of the model dimensionality automatically activates solution for the pertinent airflow variables. Selection of 2D automatically activates solution for the x-directed ( $u$ ) and y-directed ( $v$ ) velocity components and the pressure ( $P$ ). Selection of 3D automatically activates solution for the three velocity components ( $u$ ,  $v$ , and  $w$ ) and the pressure ( $P$ ).

Temperature control is achieved using the Type of Solution menu.

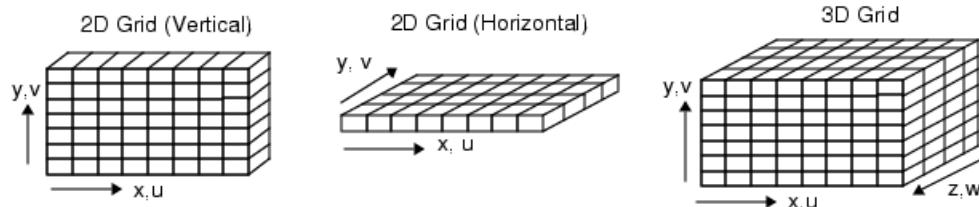
### Efficiency

For a given level of grid refinement 3D simulations are more computer intensive than 2D simulations. So, the choice made directly affects the time required for the program to provide the solution of the equations governing the flow.

### Grid Layout

The layout of the grid and velocities for the three Cartesian grid options is shown in [Figure 4-5](#).

**Figure 4-5. Grid Layout**



$x$ ,  $y$ , or  $z$  can be vertical according to the selection of the gravity direction in the [Model Setup Tab - Gravity Section](#).

The 2D option uses the  $x$ - $y$  coordinates in the 2D plane with the  $z$ -coordinates normal to the plane. The 2D option can be used:

- To represent a vertical slice through an enclosure when little variation of the flow conditions is expected horizontally in the perpendicular direction,
- Alternatively, to set the thickness of the 2D vertical slice to cover the entire extent of the enclosure normal to the 2D plane,

- To analyze the flow in a horizontal plane when the vertical variations are considered to be unimportant.

In all cases it is important to recognize that the 2D x-y grid has a depth in the third direction (z), which is set in the [System Property Sheet](#). The values of temperature ( $T$ ), pressure ( $P$ ), and velocities ( $u$ ,  $v$ , and  $w$ ) in each grid cell represent the average value that pertains throughout the depth of the grid cell.

## Computed Values

If  $f$  generically stands for  $T$ ,  $u$ ,  $v$ ,  $w$ , or  $P$ , the value computed for each grid cell in the 2D x-y plane is the depth-averaged value  $\bar{f}$ , that is:

$$\bar{f} = \left( \int_0^{Z_E} f dz \right) / Z_E$$

where  $Z_E$  is the end coordinate specifying the depth of the 2D slice, determined from the settings in the [System Property Sheet](#).

## Special Note on Ambients

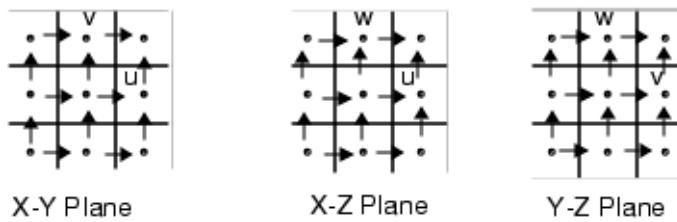
Care should be taken when switching from 3D to 2D when ambients are attached to the Z-Low, Z-High, or Default All boundaries of the Overall Solution Domain.

The software calculates the heat transfer through the Z, as well as the X and Y boundaries, which may not be desired.

## Fluid Velocity

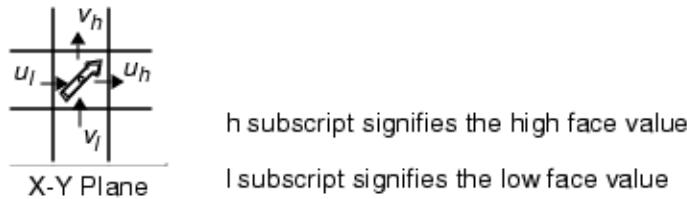
The program solves for the resolutes  $u$ ,  $v$ , and  $w$  of the air velocity and, as explained in [Background of Computational Fluid Dynamics \(CFD\)](#) in the *Simcenter Flotherm Background Theory Reference Guide*, these quantities are stored at the grid cell faces as illustrated in [Figure 4-6](#):

**Figure 4-6. Fluid Speed**



The velocity vector displayed in contour plots when in Analyze mode is the value at the grid cell centers as clarified in the following diagram (Z omitted).

**Figure 4-7. Velocity Vector**



The magnitude of the velocity vector (that is, the air speed) is:

$$\sqrt{\left(\frac{u_l + u_h}{2}\right)^2 + \left(\frac{v_l + v_h}{2}\right)^2 + \left(\frac{w_l + w_h}{2}\right)^2}$$

The fluid speed can be displayed directly as the mouse passes over a plane plot grid (Analyze mode) when Speed is the selected scalar field for the plot.

# Capture Index

The Capture Index (CI) can be used to assess the cooling performance in data centers involving discrete models of racks and cooling devices.

<b>Concept of the Capture Index.....</b>	<b>159</b>
<b>Data Center Aisle Groups and Sub-Groups .....</b>	<b>160</b>
<b>Aisle Group CI Calculation .....</b>	<b>161</b>
<b>Assessing CI Values for Data Center Design.....</b>	<b>162</b>
<b>Capture Index Procedures .....</b>	<b>164</b>

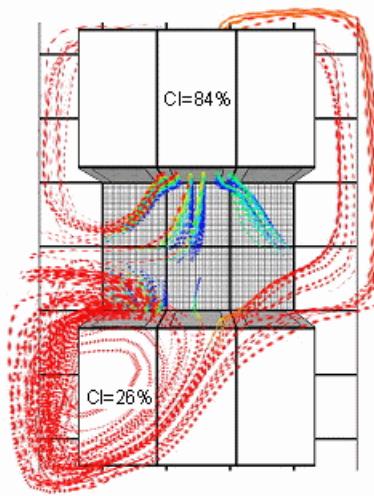
## Concept of the Capture Index

There are two types of CI (Cold Aisle and Hot Aisle) that can be defined for assessing cooling performance in data centers.

### Cold Aisle Capture Index

This is the fraction of air (expressed as a percentage figure) ingested by the rack which originates from local cooling resources (for example, perforated floor tiles or local coolers). This figure is generally used to assess data centers with raised floor ventilation systems. See [Figure 4-8](#) for an example showing greater CI (84%) for a rack placed centrally in the aisle compared with one at the end of a row (26%). The lower figure being because of more mixing of cold air with ambient air before entering the rack.

**Figure 4-8. Cold Aisle CI Examples**

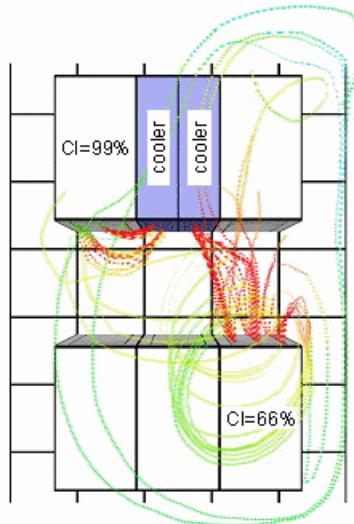


### Hot Aisle Capture Index

This is the fraction of air (expressed as a percentage figure) exhausted by a rack which is captured by local extracts (for example, local coolers or return vents). See [Figure 4-9](#) for an

example showing greater CI (99%) for a rack placed adjacent to a cooler compared with one placed opposite (66%). The lower figure being because of more mixing of hot air with ambient air before entering the cooler.

**Figure 4-9. Hot Aisle CI Examples**



The CI calculation is available for Fixed Flows, Recirculation Devices, Perforated Plates, Coolers, and Racks.

## Data Center Aisle Groups and Sub-Groups

Aisles are identified by attaching group or sub-group names to flow device objects.

### Aisle Groups

Simcenter Flotherm has been configured generally to allow any possible combination of racks and coolers for the calculation of CI values. All that you are required to do is to select appropriate groups of parts by labeling them with a cold aisle or hot aisle group name. The software determines the CI values for each rack as a post-processing calculation and outputs the data in tables and associated CSV files.

### Aisle Sub-Groups

Capture Indexes are, by default, calculated for each rack in an aisle group. In this default behavior, each rack is considered as belonging to a separate (unnamed) sub-group.

To know the average CI value for a number of racks, you must allocate a named aisle sub-group to those racks. Racks belonging to such a named aisle sub-group will have a calculated CI that is the average for all the racks in the sub-group, and you should be aware of this when viewing the solution figures in Analyze mode tables.

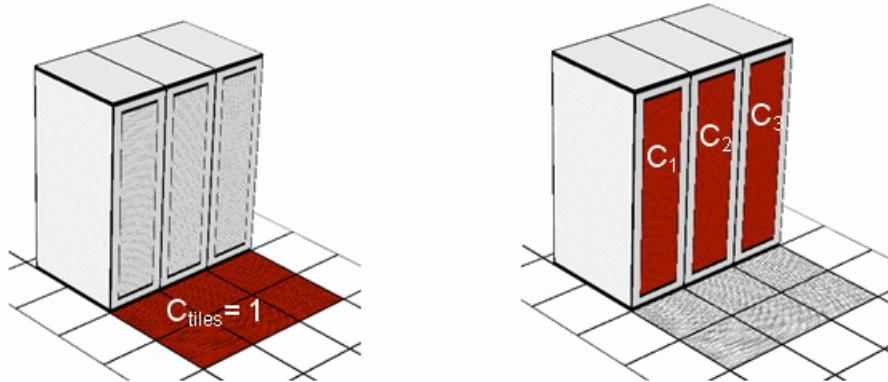
# Aisle Group CI Calculation

Calculations of capture index for cold and hot aisle groups.

## Cold Aisle Group Calculation

In this example, tiles and Rack flow devices are grouped together with a Cold Aisle Group name.

**Figure 4-10. Example Cold Aisle Group**



The CI calculation is performed as follows, refer to [Figure 4-10](#):

1. As a post-processing solution, the software assigns a single concentration value (for example,  $C_{tiles} = 1$ ) to the local cooling airflow devices and solves for concentration only.
2. The concentration is determined at each rack inlet in the group.
3. CI values are calculated from:

$$CI_i = \frac{C_i}{C_{tiles}} = C_i$$

where:

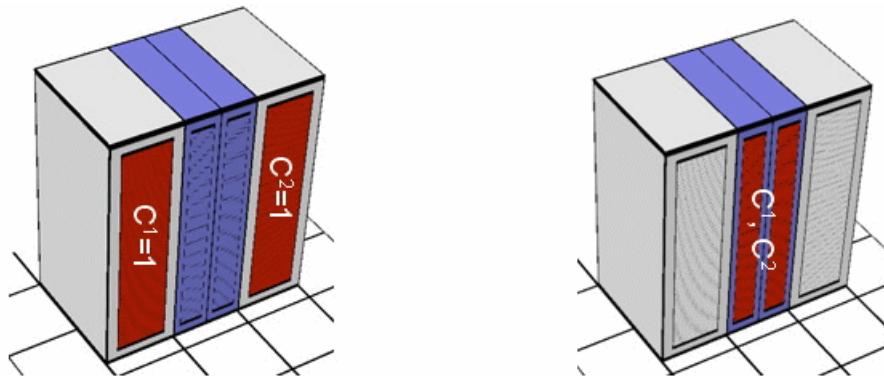
$CI_i$  is the Capture Index of Rack  $i$ .

$C_i$  is the concentration at the Rack  $i$  inlet.

## Hot Aisle Group Calculation

In this example, Coolers and Rack flow devices are grouped together with a Hot Aisle Group name.

**Figure 4-11. Example Hot Aisle Group**



The CI calculation is performed as follows, refer to [Figure 4-11](#):

1. As a post-processing solution, the software assigns a unique concentration to each rack exhaust airflow (for example,  $C_i = 1$ ) and solves for concentration for each rack in sequence.
2. The concentration of each species from each rack is determined at the defined cooler inlets in the group.
3. CI values are calculated from:

$$CI_i = \sum_{j=1}^N \frac{C_j^i Q_j}{Q_i}$$

where:

$C_j^i$  is the concentration of species  $i$  computed at the inlet of Cooler (or return vent)  $j$ .

$Q_i$  is the volumetric airflow rate through Rack  $i$ .

$Q_j$  is the volumetric airflow rate through Cooler (or return vent)  $j$ .

$N$  is the number of coolers (or return vents) in a defined hot aisle group.

## Assessing CI Values for Data Center Design

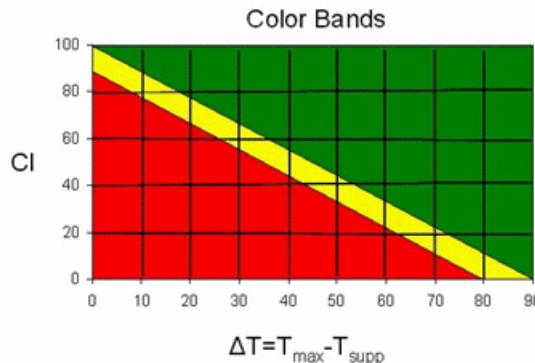
In general, a CI value of 100% for all racks will guarantee good cooling performance, however, this is very difficult, and often unnecessary, to achieve.

Generally, with local coolers or return vents, the hot aisle CI is the preferred approach. Otherwise, the cold aisle CI should be used, particularly with raised-floor data centers without local return vents.

For values below 100%, it is recommended that an assessment is made which takes into account the maximum target temperature (that is, how hot you are willing to let your rack inlet get) and the (average) supply temperature. This can be done by charting CI values versus  $\Delta T$  to give CI ratings, see [Figure 4-12](#).

**Figure 4-12. Accessing CI Values**

**Good, Marginal and Bad**



The interface lines in [Figure 4-12](#) are given by the formulae:

$$CI \text{ (green/yellow)} = -10/9 \Delta T + 100$$

$$CI \text{ (yellow/red)} = CI \text{ (green/yellow)} - 10$$

## Capture Index Procedures

The calculation of CI is switched on using a **Model Setup** tab check box, and the objects to be involved in the analysis must be identified by attachment to hot or cold aisle groups or sub-groups.

Attaching an Aisle Group or Sub-Group Name to an Object .....	164
Checking Aisle Group Attachments in Data Tree Summary Information .....	165
Viewing Capture Index Results .....	166
Adding Capture Index to Command Center Scenarios .....	167

## Attaching an Aisle Group or Sub-Group Name to an Object

Names for aisle groups and sub-groups are user-defined.

### Restrictions and Limitations

- Objects include Fixed Flows, Recirculation Devices, Perforated Plates, Coolers, and Racks.
- Perforated Plates are always used as floor tiles and therefore can only belong to Cold Aisle groups.

### Procedure

1. Open the **Model Setup** tab.
2. Make sure that Calculate is checked in the Capture Index section. If naming sub-groups then also make sure that the Sub-Groups check box is checked.
3. Select the object to be associated with a hot or cold aisle group/sub-group and open the **Group** tab of the property sheet. One or more objects can be selected, provided they are of the same type.
4. For the appropriate group/sub-group, select an existing, or create a new, group/sub-group name to attach to the object.

### Related Topics

- [Concept of the Capture Index](#)  
[Data Center Aisle Groups and Sub-Groups](#)

## Checking Aisle Group Attachments in Data Tree Summary Information

Aisle Group attachments can be seen in the summary information table provided they have been switched on for display.

### Prerequisites

- The relevant Hot/Cold Aisle Group/Sub-Group check boxes in the **Summary** tab of the User Preferences dialog box are checked.

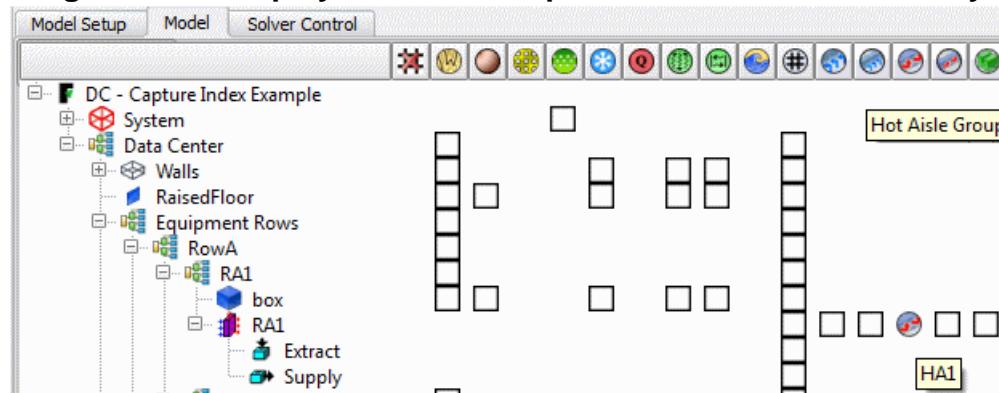
### Procedure

- Select part of the data tree and either click the **Show Summary** icon or press the I key.
- Optionally, to improve readability by filtering the data tree, do the following:
  - Press Ctrl+F to open the Find dialog box. Stay in the **Quick Criteria** tab.
  - For Type, select Fixed Flow, Recirculation Device, Perforated Plate, Cooler, or Rack.
  - Check the Filter check box.
  - Click **Find**.

### Results

Hover Text displays group names, for example, see [Figure 4-13](#).

**Figure 4-13. Display of Aisle Groups in the Data Tree Summary**



### Related Topics

[Attaching an Aisle Group or Sub-Group Name to an Object](#)

## Viewing Capture Index Results

On completion of the Simcenter Flotherm solution, a post-processing calculation is automatically started which outputs results that can be displayed in tabular format when in Analyze mode.

### Procedure

1. In Analyze mode, select the geometry (Rack or Recirculation Device geometry types).
2. Click the respective geometry type tab in the Tables pane/window.

### Results

The Capture Index (Hot) and Capture Index (Cold) percentage values are shown in the last two columns of the table.

The following text files are created:

- *<project\_directory>\<project\_name>\DataSets\BaseSolution\PDTemp\DCM.log* contains information on the CI calculation process and is the repository of any error messages in the event of a failure to calculate the CI values.
- *<project\_directory>\<project\_name>\DataSets\BaseSolution\PDTemp\<project\_name>\_cold|hot.csv* is a CSV file that contains a detailed breakdown of the data associated with the cold or hot aisle groups.

For a cold aisle analysis, the following data is available for each defined group:

- Ci — assigned concentration value, local cooling device (kg/kg)
- Cj — calculated concentration, rack (kg/kg)
- Qi — volumetric flow rate, local cooling device ( $m^3/s$ )
- Qj — volumetric flow rate, rack ( $m^3/s$ )
- Ti — average temperature, local cooling device ( $^\circ C$ )
- Tj — average temperature, rack ( $^\circ C$ )
- Fij% — capture index value for rack associated with a single local cooling device = Cj expressed as a percentage.
- CI% — total capture index value = summation of Fij% values over all cooling devices in the group.

For a hot aisle analysis, the following data is available for each defined group:

- Ci — assigned concentration value, rack (kg/kg)
- Cj — calculated concentration, cooler inlet (kg/kg)
- Qi — volumetric flow rate, rack ( $m^3/s$ )

- $Q_j$  — volumetric flow rate, cooler inlet ( $\text{m}^3/\text{s}$ )
- $T_i$  — average temperature, rack ( $^\circ\text{C}$ )
- $T_j$  — average temperature, cooler ( $^\circ\text{C}$ )
- $F_{ij}\%$  — capture index value for cooler associated with a single rack =  $C_j$  expressed as a percentage.
- $CI\%$  — total capture index value = summation of  $F_{ij}\%$  values over all cooling devices in the group.

## Related Topics

[Concept of the Capture Index](#)

[Aisle Group CI Calculation](#)

## Adding Capture Index to Command Center Scenarios

CI output is fully supported in the Command Center, with access to the CI values in the Output Variables panel and subsequent output in the Scenario Table.

### Procedure

1. To create scenario CI outputs, open the Command Center application window, and click the **Output Variables** tab.
2. Select the respective geometry (Rack or Recirculation Device).  
Hot Capture Index and Cold Capture Index check boxes are listed in the **Data** tab.
3. You can select these as output variables and view their values in the **Scenario Table** tab after the scenarios have been solved.

## Solar Radiation

---

By default, Simcenter Flotherm does not model the effects of the Sun.

<b>Solar Radiation Model</b>	.....	168
<b>Including Solar Radiation</b>	.....	168

## Solar Radiation Model

Solar radiation is affected when passing through transparent materials and resistances and perforated plates.

When calculating the effects of the sun, two types of solar data are used:

- The program-calculated sun position is used to determine either any over-shadowing of surrounding buildings or the shading effect of canopies, and so on.
- Solar intensity is used to calculate solar gains and the effect on temperature in external electronic equipment such as roadside cabinets.

For further background to solar data refer to [CIBSE Guide A Section 2.7](#) and [CIBSE Guide J Section 5](#).

---

### Note

 The effect of solar radiation is only calculated on solid (3D) objects. If studying the solar effect on a 2D surface, do not use a collapsed cuboid, use a 3D cuboid with a small depth instead.

---

## Related Topics

- [Including Solar Radiation](#)
- [Solar Configuration Dialog Box](#)

## Including Solar Radiation

The effects of the Sun on the model can be included in the solution.

### Restrictions and Limitations

- Solar field results are not removed on Re-Initialize. To force the re-calculation of the solar field after initializing, you must change the model, for example, by attaching or detaching a material, before solving.
- Although the solar calculator can run on multiple cores, as specified by the **Project Manager** tab of the User Preferences dialog box, a maximum of 8 cores are used.

## Procedure

1. Choose **Model Setup > Solar Configuration**.

The Solar Configuration dialog box opens.

2. Make sure that Solar Radiation Enabled is checked, enter data, then click **OK**.
3. Solve the model.

## Results

- After solution, the dimensionless SolarViz scalar field is available for plotting in Analyze mode.
- The following log file is created:

*<project\_directory>\<project\_name>\DataSets\BaseSolution\PDTemp\solarlogit*

## Related Topics

[Solar Configuration Dialog Box](#)

[Solar Radiation](#)

## Airflow

---

The flow type can be Laminar or Turbulent.

<b>Laminar Airflow</b> .....	<b>170</b>
<b>Turbulent Airflow and Models</b> .....	<b>170</b>
<b>Revised Algebraic Model</b> .....	<b>171</b>
<b>Automatic Algebraic Model</b> .....	<b>172</b>
<b>LVEL K-Epsilon Model</b> .....	<b>172</b>

## Laminar Airflow

In laminar flow, the mixing between fluid elements takes place at the microscopic level by molecular diffusion, and is, therefore, controlled by the physical properties of the fluid, namely the laminar (that is, molecular) viscosity for the diffusion of momentum and the conductivity of heat.

It is rare for flow to be entirely laminar in electronic enclosures. Calculation of a representative Reynold's number for forced convection or a Grashof number for natural convection can be used to determine the flow regime (laminar or turbulent) for the built environment.

### Related Topics

[Turbulent Airflow and Models](#)

## Turbulent Airflow and Models

Turbulent flow is an irregular, fluctuating, and chaotic motion, which in Simcenter Flotherm is treated as being superimposed on the underlying mean motion.

The temperatures, velocities, and pressures computed when turbulence is simulated are representative of this underlying mean motion. The mean values of these quantities are averages over time, long compared with a typical turbulent fluctuation, but short compared with any transient variation of the mean flow.

Turbulence is responsible for macroscopic mixing of the fluid elements, and this is a much more powerful mixing mechanism than molecular diffusion. This phenomenon is modeled by means of a turbulent viscosity (or eddy viscosity) which augments the laminar viscosity. The turbulent viscosity is not a property of the fluid but a function of the flow conditions. Typically, it ranges from being 100 to 1000 times larger than the molecular viscosity.

Unless it is clear that the flow is laminar (not just unidirectional) then always select turbulent to account for these larger values of effective viscosity and the resulting increased mixing.

The turbulence model is defined in the **Model Setup** tab.

See [Turbulence Viscosity Model](#) in the *Simcenter Flotherm Background Theory Reference Guide* for a mathematical description of the modeling methods.

## Related Topics

- [Revised Algebraic Model](#)
- [Automatic Algebraic Model](#)
- [LVEL K-Epsilon Model](#)
- [Laminar Airflow](#)
- [Model Setup Tab - Turbulence Section](#)

## Revised Algebraic Model

A significant turbulence contributor comes from airflow over solid surfaces. The Revised Algebraic model calculates the turbulent viscosity to account for this.

It computes a length scale, varying from point to point, based on objects in the system. This scale, together with the locally computed velocity, is used to compute a turbulent viscosity. The Revised Algebraic approach is based on the LVEL turbulence model described in [Ref. 1].

When Revised Algebraic is selected in the **Model Setup** tab, the program calculates the turbulent viscosity using typical flow values. You must set the characteristic velocity and the characteristic length to values that typify the flow under consideration:

Simcenter Flotherm calculates the maximum turbulent viscosity from:

$$0.01 \times (\text{density}) \times (\text{characteristic velocity}) \times (\text{characteristic length})$$

that is, applies a “cap” to the field of turbulent viscosity.

The macroscopic mixing of heat energy is presumed to be controlled by the general turbulent mixing process, so that Simcenter Flotherm calculates the turbulent conductivity from the following expression:

$$(\text{turbulent viscosity}) \times (\text{specific heat}) / 0.9$$

where 0.9 is the base default value for the turbulent Prandtl number.

For many enclosure designs the calculated results are found to be insensitive to the exact specifications of these quantities.

In this case all that is required is that they are set to values representative of the problem in question. The reason for this insensitivity is that enclosures are often packed full of PCBs, power supplies, cables, and so on, so that the primary turbulent transports taking place are between the surfaces of the solids and the fluid. Simcenter Flotherm calculates the wall friction

and heat transfer by means of the logarithmic friction law, which does not directly involve the value of the turbulent viscosity.

For detailed calculations or where the turbulence is dominated by velocity shears, such as those occurring at the boundaries of free jets, the Revised KE Model should be used.

## Related Topics

[Automatic Algebraic Model](#)

[LVEL K-Epsilon Model](#)

## Automatic Algebraic Model

The Automatic Algebraic model is identical to the Revised Algebraic, except the calculated turbulent viscosity field is not capped.

The turbulent viscosity is based on the distance from the wall and the local speed. It is particularly appropriate for cases that are cluttered with objects and has the advantage that there are no user settings. The disadvantage is that it may produce artificially high turbulent viscosities away from surfaces, particularly in large open spaces.

Although the Automatic Algebraic option is suitable for most problems, the Revised Algebraic option may be preferable when modeling bulk fluid movement or accurate treatment at air-solid surfaces.

## Related Topics

[Revised Algebraic Model](#)

[LVEL K-Epsilon Model](#)

## LVEL K-Epsilon Model

The KE model is blended when calculating the effect of turbulence near to a wall.

Near the wall, the effect of turbulence is calculated by the LVEL model described for Revised Algebraic, but this is blended with the KE approach.

The LVEL K-Epsilon model calculates the turbulent viscosity for the fluid cells not immediately adjacent to solid surfaces as a function of two field variables: the kinetic energy of the turbulence ( $k$ ) and its rate of dissipation ( $\varepsilon$ ). These two field variables are determined by the solution of two additional differential equations that these variables satisfy.

This enables, more properly, for the turbulent variations in effective viscosity caused by velocity shears from free jets, and so on, in open spaces.

At flow boundaries, turbulent kinetic energy and turbulent dissipation are convected in by the flow. The level of these values are set by the user when defining the ambient using the property sheet, or in the appropriate flow property sheet. If left at the default of zero, an estimate is made based on the flow rate calculated from the average inflow speed.

**Note**

 KE models are rarely needed and are used for cases where turbulence is to be modeled over large empty volumes within an enclosure.

---

## LVEL K-Epsilon and Stratification

Where there are large empty volumes within an enclosure the representation of the turbulence may be more critical. In such cases, you are advised to perform calculations in which the specified characteristic velocity and length are varied over ranges corresponding to the problem in question. The sensitivity of the calculations to the uncertainties in turbulence representation can then be deduced by comparing these results. If these are significant you should consider using the KE model of turbulence.

## LVEL K-Epsilon Additional Solution Variables

The LVEL K-Epsilon model defines the turbulent viscosity at each point from two additional variables that characterize the local state of turbulence:

- KE Turb, the kinetic energy of turbulence, and
- KE Diss, its rate of dissipation

In the KE model, these two additional fields are solved at each cell thereby providing a turbulent viscosity that differs from cell to cell.

KE Turb and KE Diss are automatically set by Simcenter Flotherm for:

- Boundaries with the KE Turb and KE Diss left to their default value of zero, that is, no ambients have been attached to change the KE Turb and Diss values
- Initial values
- False time steps

KE Turb and KE Diss are calculated based on average inlet velocity values as follows:

$$\text{kinetic energy of turbulence} = 10^{-3} \times (\text{velocity})^2$$

$$\text{dissipation rate of turbulence} = \frac{0.1643 \times [10^{-3} \times (\text{velocity})^2]^{3/2}}{l_i}$$

where:

$$l_i = 0.1 \times \sqrt{(\text{nominal inlet area})}$$

These automatic boundary settings are usually suitable because the generation of turbulence within the calculation domain is normally sufficiently high to make precise settings of the inlet values inconsequential.

The initial and false time step values are set the same as the values for the velocities.

## Turbulence Model References

1. D. Agonafer, L. Gan-Li and D.B. Spalding. "The LVEL Turbulence Model for Conjugate Heat Transfer at Low Reynolds Numbers". *EEP-Vol. 18, Application of CAE/CAD Electronic Systems*. ASME 1996.

## Related Topics

[Revised Algebraic Model](#)

[Automatic Algebraic Model](#)

# Gravity

---

Gravity gives rise to natural convection within enclosures whenever temperature differences are present.

**Modeling Gravity**..... [175](#)

**Calculation of Gravity**..... [176](#)

## Modeling Gravity

Specifying direction.

When natural convection is the only mechanism whereby fluid flows, it is essential to solve for temperature and to model gravity, however, even in a forced convection system, gravity may have a significant effect in regions not reached by airflow from the fans.

There are some applications that you may want to treat as isothermal (that is, constant temperature). In this case, select the Flow Only option as the Type of Solution in the **Model Setup** tab, and gravity will be ignored.

By default, gravity is on in the negative Y-direction. However, the gravity solution causes the momentum equations to become strongly coupled with the temperature equations via the buoyancy force term. This results in larger numbers of iterations to reach a converged solution than if gravity is absent.

Gravity is defined using the [Model Setup Tab - Gravity Section](#).

### Gravity Direction

You can align the gravity force to act in any of the coordinate directions, either directed along or opposite to the direction of the coordinate axis. In the 2D case it makes no sense to set gravity in the Z direction.

You can also set resolutes in the X-, Y-, and Z-direction of the unit vector that is aligned with the direction of gravity. If the vector specified is not a unit vector, then the program will normalize to a unit vector before starting the solution.

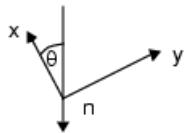
A unit vector is a vector of unit length whose resolutes in the three coordinate directions are known as the direction cosines. Let the unit vector be denoted by  $n$  and its resolutes in the x-, y-, and z-direction by  $n_x$   $n_y$  and  $n_z$  - the values of which are to be entered in the corresponding data fields in the menu. For the example shown in [Figure 4-14](#), the following are the settings to be made:

$$n_x = -\cos\theta$$

$$n_y = -\sin\theta$$

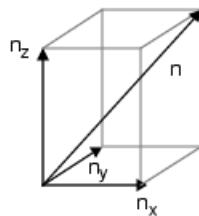
$$n_z = 0.0$$

**Figure 4-14. One Vector Component is Zero**



The general situation is where all three components of  $n$  are non-zero.

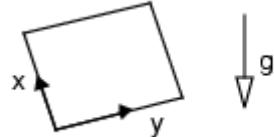
**Figure 4-15. Non-Zero Vector Components**



### Example of Angled Gravity

A typical use of angled gravity is when an enclosure is a VDU monitor inclined to the horizontal, where gravity has resolutes in both the X and Y directions.

**Figure 4-16. Angled Gravity**



## Calculation of Gravity

How gravity on fluids is treated.

### Constant-Density Option

When the constant-density option is selected for the fluid property (Density Type in the Fluid attribute property sheet), the gravitational force per unit volume of air is calculated from the following expression, known as Boussinesq's approximation:

$$9.81 \times (\text{expansivity}) \times (\text{density}) \times (T - \text{reference temperature})$$

where:

- *expansivity* is the volumetric expansivity of the fluid. It is a property of the fluid and, therefore, is set in Fluid Properties. For an ideal gas, the expansivity is the reciprocal of the absolute temperature. The default value for is set to 1/300 K.
- *density* is the constant-density value as set in the Fluid attribute property sheet.
- *reference temperature* has the default value of 35°C. It is set as the Global Ambient Temperature in the [Model Setup Tab - Global System Settings Section](#).
- $T$  denotes the in-cell value of the temperature and, by default, is in degrees Centigrade. The value is automatically calculated.

## Variable-Density Option

When a non-constant air density option is selected for the fluid property, the gravitational force per unit volume is calculated from the following expression:

$$9.81 \times (\rho - \text{reference density})$$

where:

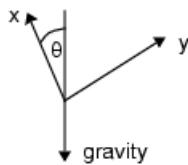
- *reference density* is calculated by Simcenter Flotherm, based on the value for the Global Ambient Temperature as defined in the [Model Setup Tab - Global System Settings Section](#).
- $\rho$  denotes the in-cell value of the density which is calculated by Simcenter Flotherm from the ideal gas law formula selected in Fluid Properties.

## Application of the Gravity Force

When gravity points along one of the coordinate axes (that is, the [NORMAL] option is selected), the formula for the gravitational force is applied to the associated velocity equation. Therefore, if the negative x-direction is selected, Simcenter Flotherm applies the negative of the force to the  $u$ -velocity equation.

When gravity is set at an angle to the coordinates, Simcenter Flotherm applies the resolved components of the force in the respective velocity equations. For example, if the gravity force is in the x-y plane inclined at an angle  $\theta$  to  $x$  ([Figure 4-17](#)), Simcenter Flotherm applies the resolved components to the  $u$  and  $v$  velocity equations (but not to the equation for the  $w$  velocity). For the  $u$  equation, the factor 9.81 in the force term is replaced by  $9.81 \cos\theta$ ; in the  $v$  equation it is replaced by  $9.81 \sin\theta$ .

**Figure 4-17. Application of Gravity Force**



## The Acceleration Due to Gravity

By default, the acceleration due to gravity used by Simcenter Flotherm is  $9.81 \text{ ms}^{-2}$ , which is the value at sea level.

If necessary, the constant can be reset in the [Model Setup Tab - Gravity Section](#) in order to simulate conditions of high acceleration as pertains, for example, in rocket launches.

## Gravity Modeling Considerations

The appearance of the reference temperature (for constant density option) and the reference density (for variable density option) in the force formula is designed to have the effect of reducing the gravitational force exerted on the fluid by either partial or total removal of the hydrostatic component. The object of this is to represent the motion-inducing part of the gravity force as accurately as possible.

This procedure is desirable because the hydrostatic force is usually very much larger than the motion-inducing part of the force.

The distinction between the motion-inducing and hydrostatic part of the gravity force is illustrated by the following example. A tank of water at room temperature ( $35^\circ\text{C}$ ) if left alone will attain the state of hydrostatic equilibrium, the gravity force being exactly balanced by the increase of pressure with depth below the surface. It is small inequalities of temperature in the water that causes water motion - the light water rising and the heavier falling.

Reducing the gravity force in this way results in the removal of hydrostatic head from the pressure field. Since the default value for the ambient temperature is  $35^\circ\text{C}$ , in the example of the tank of water at  $35^\circ\text{C}$ , the use of the force formula results in a reduced pressure field that is zero throughout the tank in the equilibrium case.

## Changing the Fluid Properties

To accurately model the cooling/heating effect by the surrounding air, you must define its density, conductivity, specific heat, viscosity, and expansivity.

The default fluid property values for the Default\_SI projects are for air at  $30^\circ\text{C}$  and 1 atm.

The fluid property currently in use as the enclosure coolant is that selected in the Fluid field of the **Model Setup** tab. You can select another Fluid attribute if listed, create a new Fluid attribute, or load a Fluid attribute from a library.

## Reference Temperature and Pressure

---

The model reference temperature and pressure are set in the **Model Setup** tab.

<b>Datum Pressure</b> .....	<b>180</b>
<b>Modeling a Remote Radiation Source</b> .....	<b>180</b>
<b>Ambient Temperature When Modeling Buoyancy Force</b> .....	<b>182</b>
<b>Default Initial Value for Temperature</b> .....	<b>182</b>
<b>Transient Variation of Temperature</b> .....	<b>182</b>

## Datum Pressure

The pressure relative to the datum pressure is referred to as the gauge pressure.

More often than not, when working with electronic enclosures, the pressures at the enclosure vents are zero gauge pressure, that is,  $0 \text{ N m}^{-2}$ , which is the default for pressure for all flow boundaries. These gauge pressures are also “reduced” in the sense that the hydrostatic component is subtracted out of them, so that even at a vertical vent, the gauge pressure is exactly zero all the way up vertically. See “[Gravity Modeling Considerations](#)” on page 178.

Simcenter Flotherm uses the datum values to calculate the absolute value of pressure at each grid cell:

$$\text{absolute pressure} = \text{datum pressure} + \text{gauge pressure}$$

The absolute pressure is required when the ideal gas law option is selected for the attached Fluid property.

See [Fluids](#) in the *Simcenter Flotherm Project Attributes Reference Guide*.

---

### Note

 Simcenter Flotherm *should* add back in the hydrostatic component in the absolute pressure for use in the gas law, however, this effect is entirely negligible for most cases of practical interest.

---

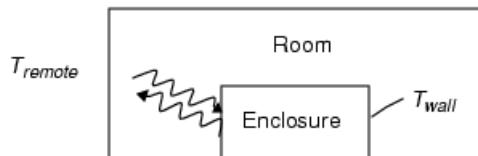
## Modeling a Remote Radiation Source

The heat loss/gain through radiative transfer is achieved by setting the remote source temperature in the **Model Setup** tab and then applying an emissivity value to the exterior surface using a Surface attribute.

## Within a Room

An example of its use is an enclosure within a room with walls at a constant temperature as shown in Figure 4-18.

**Figure 4-18. Modeling a Remote Radiation Source**



In this case there is radiant exchange between the enclosure walls and the remote surfaces.

Setting the external radiant temperature enables the net radiant flux to the surface to be calculated from the following expression:

$$\sigma F \epsilon ((T_{remote})^4 - (T_{wall})^4)$$

where:

- $\sigma$  is the Stefan-Boltzmann constant.
- $F$  is the view factor.
- $\epsilon$  is the emissivity of the surface.
- $T_{remote}$  is the temperature of the remote surface, which is distinct from the bulk temperature of the fluid surrounding the enclosure.
- $T_{wall}$  is the calculated temperature of the external surface of the enclosure (not available to the user).

## External Ambient at Night

For an external cabinet at night, the external radiant temperature is that of the sky. This will be dependent on the local conditions, cloud cover, and so on.

## Further Information

For more background on radiative cooling/heating, consult books under [Further Reading](#) in the *Simcenter Flotherm Background Theory Reference Guide*, especially those by G. Ellison and *Transport Phenomena* by Bird, Stewart, and Lightfoot.

## Ambient Temperature When Modeling Buoyancy Force

The ambient temperature is used in the definition of the buoyancy force resulting from temperature differences within the enclosure.

If gravity has been switched on (using the [Model Setup Tab - Gravity Section](#)) and the constant density option is active (the default), Simcenter Flotherm calculates the buoyancy force at each point of the enclosure as:

$$9.81 \times (\text{density}) \times (\text{expansivity}) \times (T - T_{\text{ambient}})$$

where  $T$  is the temperature of the fluid at the point in question.

Clearly, if the fluid within the enclosure is everywhere at the ambient temperature, the buoyancy force is everywhere zero.

## Default Initial Value for Temperature

The ambient temperature is used as the default initial value for the temperature field.

At the start of the calculation the temperature in every grid cell is set equal to the ambient temperature, irrespective of whether the cell contains fluid or solid. This default can be overridden by use of the options in the [Solver Control Tab - Variable Solution Control Section](#).

## Transient Variation of Temperature

Independently varying ambient air temperature and radiant temperature.

The modeling of electronics equipment, such as pole-mounted communications base stations, over periods of time extending to several hours requires both the ambient air temperature (External Ambient Temperature) and the radiant temperature (External Radiant Temperature) to vary dynamically.

To model the variation dynamically, attach Ambient and Radiant Transient functions

The ambient air temperature and the radiant temperature are defined to vary independently, because the radiant temperature is analogous to the so-called “Sky Temperature” and has a different diurnal variation from the ambient air value.

For more information, see [Surface Attributes](#) in the *Simcenter Flotherm Project Attributes Reference Guide*.

## Joule Heating Analysis

Joule heating can be modeled by creating Source SmartParts within electrically conducting geometry, or by importing power map CSV files to create Power Map SmartParts.

**Joule Heating Modeled by Source SmartParts.....** **183**

**Joule Heating and Power Map SmartParts.....** **183**

## Joule Heating Modeled by Source SmartParts

Electrical potential and current can be defined at conductive solids to model the thermal effects of Joule heating.

When Joule Heating is switched on, you can add a Source attribute that defines an electrical potential or current and attach that attribute to a Source SmartPart.

3D fields of electrical potential, current density, Joule heating, and electrical resistivity will be calculated during the solution, based on the model definition.

### Related Topics

[Source Attributes \[Simcenter Flotherm Project Attributes Reference Guide\]](#)

## Joule Heating and Power Map SmartParts

Power maps of PCB nets, exported as CSV files from HyperLynx PI, can be imported into Simcenter Flotherm to predict the temperatures of (high) current carrying copper nets in PCBs (that is, to predict the effect of Joule heating).

HyperLynx PI is a simulation tool that predicts electrical current flow (DC voltage drop) on nets within a PCB.

“Net” is used to mean a contiguous 3D object in a PCB, nearly always made of copper, in which electrical current flows.

### Related Topics

[Power Maps \[Simcenter Flotherm SmartParts Reference Guide\]](#)

# Solution Domain Dialog Boxes and Property Sheets

---

GUI interfaces associated with the solution domain and the solution.

<b>System Property Sheet</b> .....	<b>185</b>
<b>Subdomain Property Sheet</b> .....	<b>187</b>
<b>Solar Configuration Dialog Box</b> .....	<b>189</b>
<b>Model Setup Tab</b> .....	<b>191</b>
<b>Solver Control Tab</b> .....	<b>200</b>
<b>Solution Set Dialog Box</b> .....	<b>208</b>
<b>Solution Type Changed Dialog Box</b> .....	<b>209</b>

# System Property Sheet

To access: Select the System node in the data tree.

Use this property sheet to set the location and size of the solution domain. The solution domain defines the extent of the geometry model included in the Simcenter Flotherm calculations, for example, either the complete model or just a subset of it.

## Objects

Field	Description
<b>Location</b> tab	See <a href="#">Generic Property Sheet Location Tab</a> in the <i>Simcenter Flotherm SmartParts Reference Guide</i> .
<b>Boundaries</b> tab	See “ <a href="#">System Property Sheet Boundaries Tab</a> ” on page 186.

## Related Topics

[The Solution Domain](#)

## System Property Sheet Boundaries Tab

To access: Select the System node in the data tree, then the **Boundaries** tab.

Use this property sheet to define all or individual (six) faces of the solution domain and their ambient attribute(s).

### Objects

Field	Description
Faces	<ul style="list-style-type: none"><li>• Open — an open face of constant pressure through which air can flow.</li><li>• Symmetry — a frictionless, impermeable, and adiabatic planar surface through which neither air nor heat can flow.</li></ul>
Ambient	If required, attach an ambient attribute to all or individual faces.

### Usage Notes

By default the Faces and Ambient settings apply to all faces. To define individual faces and ambients, expand the fields to include definitions for each of the six faces: Xo High, Xo Low, Yo High, Yo Low, Zo High, and Zo Low.

### Related Topics

[Examples of Open and Symmetry Boundaries](#)

[Boundary Types](#)

## Subdomain Property Sheet

To access: Select a subdomain.

Use this property sheet to define a subregion in the solution domain over which constant initial values can be applied.

### Objects

Field	Description
<b>Location tab</b>	See <a href="#">Generic Property Sheet Location Tab</a> in the <i>Simcenter Flotherm SmartParts Reference Guide</i> .
<b>Initial Values tab</b>	See “ <a href="#">Subdomain Property Sheet Initial Values Tab</a> ” on page 188.

## Subdomain Property Sheet Initial Values Tab

To access: Select a subdomain, then the **Initial Values** tab.

Use this property sheet tab to set a constant subdomain value for each of the variables listed. The variables listed depend on the Type of Solution, Turbulence Model and if the fluid density is set to follow the Ideal Gas Law.

### Description

The values are those to be used before starting the solution, and are particularly important when running a transient solution - when running a steady-state solve they only affect the speed of solve. The values takes precedence over those implied by the uniform whole-domain values set in the [Solver Control Tab - Variable Solution Control Section](#).

### Objects

Field	Description
Pressure	(Flow and Heat Transfer or Flow Only) The initial value of pressure.
X, Y and Z Velocity	(Flow and Heat Transfer or Flow Only) The initial values of x-, y-, and z-directed resolutes.
Temperature	(Flow and Heat Transfer or Conduction Only) The initial value of temperature
Density	(Ideal Gas Law fluids or multiple fluids) The initial value of density.
KE Turb	(LVEL K-Epsilon Turbulent flow) The initial value of kinetic energy of turbulence.
Diss Turb	(LVEL K-Epsilon Turbulent flow) The initial value of dissipation of turbulence.
Turb Vis	(Any Turbulent flow model) The initial value of turbulent viscosity.

### Related Topics

[The Solution Variables](#)

[Setting Initial Field Values Over Subdomains](#)

# Solar Configuration Dialog Box

To access: Either choose **Model Setup > Solar Configuration**, or check Solar Radiation in the **Model Setup** tab and click **Click To Edit**.

Use this dialog box to model the effects of short-wave radiation from the sun.

## Objects

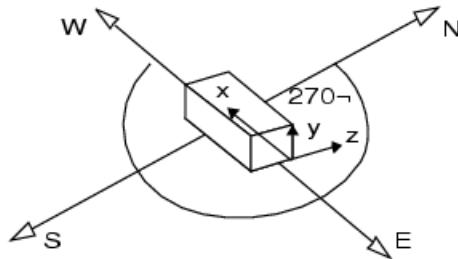
Field	Description
Solar Radiation	
Solar Radiation Enabled	Activates the calculation of the effects of solar radiation for a specified sun position and intensity during program solution. It enables the option fields describing the orientation and latitude of the enclosure, the position of the sun in the sky, and the solar intensity from which the solar radiation is calculated.
Model Orientation from North	Sets the orientation of the building by defining the angle that a wall parallel to the chosen axes makes to the North.
Angle Measured From	The axis of the wall to be aligned with North.
Angle	The alignment angle from North. For example, the following are the angles of the other compass points: <ul style="list-style-type: none"> <li>• East — 90°</li> <li>• South — 180°</li> <li>• West — 270°</li> </ul>
Solar Position	Simcenter Flotherm calculates the position of the sun from the location of the enclosure and the time of year you enter.
Latitude	Sets the latitude of the building.
Date	Sets the day of year. Pick from the calendar. Click arrows to change the month. The year is ignored.
Solar Time	Sets the time of day using a 24-hour clock. The sun is at its zenith at 12:00.
Solar Calculation Type	<ul style="list-style-type: none"> <li>• Solar Intensity — Enables user-specified solar intensity.</li> <li>• Cloudiness — Activates the automatic calculation of the solar radiation intensity normal to the sun's rays, according to the solar position settings.</li> </ul>
Solar Intensity	(Solar Intensity) The solar radiation falling on a notional plane perpendicular to the direction of the radiation.

Field	Description
Cloudiness	(Cloudiness) To observe the effect of any cloud cover, the calculated solar intensity can be reduced by a specified proportion by setting a cloudiness factor between 0 and 1. As the cloudiness is increased towards 1 (maximum cloudiness) the solar intensity is reduced. For example, setting a cloudiness of 0.3 would reduce the intensity by 30%, that is to 70% of the clear day value.
<b>Calculated Values</b>	
Calculated Solar Intensity	By default, gives the calculated solar intensity for a clear day based on the latitude, day of the year, and time of day. This value can be reduced by setting a cloudiness factor.
Azimuth Angle	The solar azimuth from the North, measured clockwise, starting from zero at North.
Solar Altitude	The angle of the sun above the horizon, with zero representing on the horizon.

## Usage Notes

Figure 4-19 shows a building with walls parallel to the X-axis oriented towards the West.

**Figure 4-19. Example Alignment**



## Related Topics

- [Model Setup Tab](#)
- [Solar Radiation](#)
- [Including Solar Radiation](#)

# Model Setup Tab

To access: Click the **Model Setup** tab.

Use this tab to define the principal features of the mathematical model.

## Objects

Field	Description
Type of Solution	Set the type of solution to determine the variables solved. There are three options: Flow and Heat Transfer, Flow Only, and Conduction Only. Refer to “ <a href="#">Types of Solution</a> ” on page 150.
Dimensionality	Sets the overall solution domain to either two or three dimensions. See “ <a href="#">Modeling Dimensionality</a> ” on page 156, which discusses the importance of this setting and how it affects the solution.
Radiation	Choose the radiation modeling type for boundaries to which you have attached radiation attributes from the Radiation popup menu. There are three options: <ul style="list-style-type: none"> <li>• Radiation Off — The appropriate modeling choice for most forced convection systems and initial simulation of all types.</li> <li>• Radiation On — Calculates the view factors between all pairs of surfaces for which radiation has been activated. If the path between the surfaces is blocked, then the view factor is calculated based upon the proportion of radiation that can pass.</li> <li>• Radiation On - High Accuracy — Calculations are more accurate. The calculation of the radiation exchange factor could take up to four times longer.</li> </ul> See <a href="#">Radiation Attribute Property Sheet</a> in the <i>Simcenter Flotherm Project Attributes Reference Guide</i> .
Joule Heating	Include Joule Heating calculation in the solution. When Joule Heating is enabled: <ul style="list-style-type: none"> <li>• Potential can be selected as a Source Type in a Source attribute.</li> <li>• Potential can be selected as a Variable in the <b>Solver Control</b> tab.</li> </ul> See “ <a href="#">Joule Heating Modeled by Source SmartParts</a> ” on page 183.
Solar Radiation	Switches on solar radiation calculation to include in the solution. Click <b>Click to Edit</b> to open the <a href="#">Solar Configuration Dialog Box</a> .
Transient Solution	Sets the solution to be a Transient solution, rather than the default Steady-State solution. See “ <a href="#">Transient Analysis</a> ” on page 335. Click <b>Click to Edit</b> to open the <a href="#">Transient Solution Dialog Box</a> .
Fluid	The Fluid attribute defining the circulating fluid (normally air) in the solution domain.
Gravity section — refer to “ <a href="#">Model Setup Tab - Gravity Section</a> ” on page 193.	

Field	Description
Turbulence section — refer to “ <a href="#">Model Setup Tab - Turbulence Section</a> ” on page 194.	
Global System Settings section — refer to “ <a href="#">Model Setup Tab - Global System Settings Section</a> ” on page 195.	
Capture Index	
Calculate	Activate this option to request the calculation of the Capture Index (CI).
Sub-Groups	(Calculate) Enables attachment of Aisle Sub-Group names.
<b>Stored Variables</b> section — refer to “ <a href="#">Model Setup Tab - Stored Variables Section</a> ” on page 197.	
<b>Auxiliary Variables</b> section - refer to “ <a href="#">Model Setup Tab - Auxiliary Variables Section</a> ” on page 199.	

## Usage Notes

### Radiation

---

#### Note

---

-  For geometrically complex situations, activation of full radiation will require more computational effort to calculate the exchange factors.

For more information on the strategy of radiation modeling, see [Radiation Model](#) in the *Simcenter Flotherm Background Theory Reference Guide*.

---

## Related Topics

[Solution Variable Types](#)

[Modeling Selections](#)

[Capture Index](#)

[Exchange Factors](#)

## Model Setup Tab - Gravity Section

To access: Click the **Model Setup** tab, then scroll down to the Gravity field.

Use this section of the **Model Setup** tab to activate or deactivate the effects of gravity. When activated, gravity can be simulated as a force acting on the fluid either parallel or angled to any of the coordinate directions.

### Objects

Field	Description
Gravity	<ul style="list-style-type: none"><li>Off — Removes the effects of gravity from the solution.</li><li>Normal (default) — Activates gravity to align with one of the three coordinate directions selected from the popup.</li><li>Angled — Activates gravity in the direction set by vector resolutes.</li></ul>
Direction	(Normal) The choice is in the positive or negative X, Y, or Z directions.
Direction XN, YN, ZN	(Angled) Sets the resolute of the vector in the x-direction, y-direction, and z-direction respectively. The vector is normalized into a unit vector at the start of the next solve step. For an example, see <a href="#">Angled Flow</a> in the <i>Simcenter Flotherm SmartParts Reference Guide</i> .
Value	(Normal or Angled) The value of the gravity constant used for the program solution. <ul style="list-style-type: none"><li>Automatic — The system default value of <math>9.81 \text{ ms}^{-2}</math>.</li><li>User Specified — A user value.</li></ul>
Gravity (g)	(User Specified) Sets the Gravity value.

### Related Topics

[Gravity](#)

## Model Setup Tab - Turbulence Section

To access: Click the **Model Setup** tab, then scroll down to the Turbulence field.

Use this section of the **Model Setup** tab to define the airflow model.

### Objects

Field	Description
Turbulence	<ul style="list-style-type: none"><li>Turbulent (the default) — where the motion of the fluid layers are affected by irregular, fluctuating, and chaotic eddy currents. For most models of electronic enclosures, whether ventilated using forced or natural convection, the air movement exhibits turbulent motion.</li><li>Laminar — where the fluid layers move steadily without interference.</li></ul>
Turbulence Model	(Turbulent) <ul style="list-style-type: none"><li>Automatic Algebraic (the default) — for Simcenter Flotherm-calculated viscosity where the solution domain is highly cluttered with objects.</li><li>Revised Algebraic — for accurate treatment at air-solid interfaces.</li><li>LVEL K-Epsilon — as Revised Algebraic, but using the KE model.</li></ul>
Velocity	(Revised Algebraic) The <i>characteristic velocity</i> that typifies the flow under consideration. For example, for an enclosure containing racks of PCBs, the <i>characteristic velocity</i> is ideally the mean velocity in the inter-PCB gap.
Length	(Revised Algebraic) The <i>characteristic length</i> that typifies the flow under consideration. For example, for an enclosure containing racks of PCBs, the <i>characteristic length</i> is the average gap between PCBs.
KE Model Stratification	(LVEL K-Epsilon) Include buoyancy generation terms. Use this option only where the temperature differences in the fluid are large (approximately 50° C); this option should not be required for the majority of Simcenter Flotherm applications.

### Related Topics

[Airflow](#)

## Model Setup Tab - Global System Settings Section

To access: Click the **Model Setup** tab, then scroll down to the Pressure field.

Use this section of the **Model Setup** tab to set the basic reference values that are used throughout the solution domain.

### Objects

Field	Description
Pressure	The external fluid pressure, which is used to calculate the absolute pressure values for use in variable-density models.  All settings of pressure must then be made relative to this datum pressure. For most calculations, the default value of 1 atmosphere will be appropriate.
Default Radiant Temperature	The external radiant temperature, that is, the temperature of a remote radiating source, used in radiation models to calculate the net radiant flux incident on the external sides of an enclosure. This temperature is used, with the emissivity value for the surface of the enclosure set in the Surface Attribute property sheet, to calculate the radiative exchange between the exterior surface of an enclosure and the remote surface. In the case of external cabinets at night, the temperature is that of the sky.
Radiant Transient	The Transient attribute applied to vary the Default Radiant Temperature as a function of time during transient calculations.
Default Ambient Temperature	The ambient temperature to be used by default for: <ul style="list-style-type: none"> <li>• Fluid temperature at inflow boundaries</li> <li>• Reference temperature in the buoyancy force</li> <li>• Initial fields of temperature</li> </ul> If no ambient attribute is attached to the sides of the solution domain then the temperature value specified here is the temperature of air external to the domain.
Ambient Transient	The Transient attribute applied to vary the Default Ambient Temperature as a function of time during transient calculations.

### Usage Notes

The Default Radiant and Ambient Temperatures can be changed locally on a selected face of the overall boundary or on a cutout on the boundary by attaching Ambient attributes. Any such radiant or ambient temperature attributes attached to the sides of the domain take precedence over the temperature set here.

### Related Topics

[Reference Temperature and Pressure](#)

## Transient Variation of Temperature

## Model Setup Tab - Stored Variables Section

To access: Click the **Model Setup** tab, then scroll down to the Stored Variables check boxes.

Use this section of the **Model Setup** tab to calculate and store additional data during a solve. By default, only Surface Temperatures is selected (Yes) to be calculated.

### Objects

Field	Description
Mass Fluxes	Activate only when a flow is present and, therefore, deactivate for conduction-only cases. Storage is allocated for the display of three additional field variables for: <ul style="list-style-type: none"> <li>• Mass fluxes (flow rates per unit area) across cell faces normal to the x-direction,</li> <li>• Mass fluxes across cell faces normal to the y-direction, and</li> <li>• In 3D simulations, mass fluxes across cell faces normal to the z-direction.</li> </ul>
Surface Temperatures	Allow access to the true surface temperatures of all conducting objects for examination in Tables and post-processing in Analyze mode.
Power Density	Calculate power density data for each cell.
Heat Fluxes	Activate when flow and heat transfer is present. Therefore deactivate for flow-only cases. Activates the storage allocation for the display of three additional field stores for: <ul style="list-style-type: none"> <li>• Heat fluxes (heat flow per unit area) across cell faces normal to the x-direction,</li> <li>• Heat fluxes across cell faces normal to the y-direction, and</li> <li>• In 3D simulations, the heat fluxes across cell faces normal to the z-direction.</li> </ul>
GradT	Enable the calculation and storing of the Grad T and Mag Grad T temperature gradient post-processing fields
Bn and Sc	(Heat Fluxes and GradT must be selected) Enable the calculation and storing of Bn and Sc parameters.

### Related Topics

[Mass Flux](#)

[Surface Temperature](#)

[Power Density](#)

[Heat Flux](#)

[Temperature Gradient](#)

[Bn and Sc](#)

## Model Setup Tab - Auxiliary Variables Section

To access: Click the **Model Setup** tab, then scroll down to the Auxiliary Variables check boxes.  
Use this section of the **Model Setup** tab to calculate auxiliary variables after a solve.

### Objects

Field	Description
Flow Angle	Calculate to evaluate the performance of a laminar (uni-directional) flow scheme
Total Pressure	Calculate to measure condensation risk.

### Related Topics

[Additional Auxiliary Variables](#)

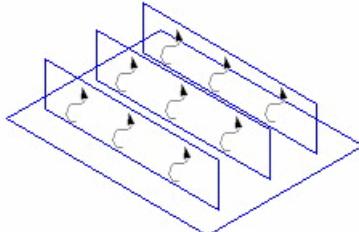
## Solver Control Tab

To access: Open the **Solver Control** tab.

Use this tab to control the solution.

### Objects

Field	Description
Solver Option	<p>Sets the type of solution. For most calculations, the default Multi Grid solver is the best option.</p> <ul style="list-style-type: none"><li>• Multi Grid — an iteration procedure that uses multi-grid acceleration to solve the linear equations for temperature. For problems with conjugate heat transfer, it can improve convergence and significantly reduce overall computation time.</li><li>• Segregated Conjugate Residual — an iteration procedure that uses conjugate residual acceleration to solve the linear equations for pressure and temperature.</li></ul>
Outer Iteration	<p>The maximum number of outer iterations for a steady calculation and the maximum number at each time step for a transient calculation. Each outer iteration results in the solution of all differential equations expressed in quasi-linear form.</p> <p>The default is set to 500. For the majority of calculations, several hundred iterations will be needed to converge the solution to a satisfactory level, however, the exact setting of the number of outer iterations is not critical because:</p> <ul style="list-style-type: none"><li>• The calculation stops when the normalized errors (see <a href="#">Solver Control Tab - Variable Solution Control Section</a>) fall below unity.</li><li>• If the algorithm performs the specified number of iterations without satisfying the convergence criterion, you can continue the calculation using Solve Start.</li></ul>
Fan Relaxation	An advanced control that is sometimes required when using non-linear fans; see “ <a href="#">Fan Relaxation</a> ” on page 414 for details. Setting this parameter to less than 1, for example 0.7, may help achieve convergence.

Field	Description
Activate Plate Conduction	<p>Activates the linear embedded conduction solver, enabling full conjugate heat transfer in 2D solid objects, as shown below:</p>  <p>This option provides a more physically correct representation for heat associated with solid, thin objects, but there is an additional grid mesh overhead that will result in increased memory requirements. Details of the mesh size is provided during the solve by information message I/9033.</p> <p>The Joule Heating and Surface Exchange settings on collapsed cuboids have no effect when plate conduction is activated.</p> <p>For information on the precedence rules, see “<a href="#">Context Rules for Plate Conduction</a>” on page 227.</p> <p>By default, Activate Plate Conduction is switched off and a simplified linear solver is used, which does not enable full conjugate heat transfer of 2D solid objects, but requires less memory.</p> <p> <b>Caution:</b> When plate conduction is activated, a Heat Pipe or Network Assembly is included in the project, you must use Re-Initialize and Solve when editing a solved project, clicking the <b>Solve Start</b> icon may lead to incorrect results.</p>
Use Double Precision Solver	Refer to “ <a href="#">Double-Precision Solver</a> ” on page 418. This setting is saved with the project.
Network Assembly Block Correction	This setting may be useful when the project contains one or more network assemblies and the solution is very slow to finish, typically, the temperature residual showing little sign of reducing. Note that in some cases its activation may increase the solution time.
Multi Grid Damping	<p>An advanced control that adds increased damping control when Solver Option is set to Multi Grid (the default setting). Only use this option if the Multi Grid solution exhibits divergent behavior. Alternatively, use the Segregated Conjugate Residual solver.</p> <p> <b>Note:</b> Although this setting prevents divergence, it may result in some slow down in convergence in normal situations. For this reason, this setting should only be activated when required. See “<a href="#">Multi Grid Damping</a>” on page 412.</p>

Field	Description
Freeze Flow	<p>An advanced control that deactivates the solution of flow but retains the stored variables, hence effectively freezing the flow solution. Freeze flow can be useful in the following situations:</p> <ul style="list-style-type: none"><li>Improving convergence of the Temperature equation for very complex conjugate heat transfer problems. In some physically very complex cases, it can be difficult to achieve complete convergence in the temperature equation as a result of fluctuating errors in the fluid velocity solution. To achieve convergence, you can freeze the flow field and solve for the temperature equation only.</li><li>For cases in which buoyancy is largely insignificant, extremely rapid parametric studies can be performed by freezing the flow field, making a change to a heat source for example, and solving for T only.</li></ul> <p> <b>Caution:</b> You should only use the Freeze Flow option in cases strongly dominated by forced convection. Buoyancy forces involve direct coupling between the temperature and vertical velocity field, and failure to solve for the velocities would invalidate any results from the parametric study.</p>
Error Field	<p>An advanced control that stores the field of residuals for a selected variable. After solution, the Field Error scalar field contains the local residual errors for the selected variable. To identify areas where the program may have difficulty in converging, create a plane plot showing the Error Field scalar values and set the Fill Type to Contour Lines. You can then focus attention on the grid, project setup, and assumptions in the areas of difficulty. See “<a href="#">Error Field</a>” on page 416.</p>
Error Variable	(Error Field) Select the variable type.

Field	Description
Estimated Free Convection Velocity	<p>Estimated Free Convection Velocity (EFCV) is used to calculate the False Time Step Relaxation values and Termination Residuals.</p> <p>The EFCV needs to be set for situations in which the flow is driven solely by natural convection, when the buoyancy forces are of importance. That is, when there is no forced flow, for example, as generated by fans or fixed flow SmartParts.</p> <p>Typically, in electronic enclosures, the free convection velocity will vary between 0.1 m/s to 1.0 m/s depending on the heat dissipated, the enclosure height, and the resistances to airflow within the enclosure. The software assumes a default of 0.2 m/s.</p> <p>You need only estimate EFCV; you do not need to know it exactly, as Simcenter Flotherm calculates the airflow pattern. If you set EFCV to the maximum anticipated velocity, it will have the effect of heavy under-relaxation so that you should get convergence, albeit slowly. However, choosing a large EFCV may lead to excessively high termination criteria. Care should be taken to ensure the solution does not converge too easily.</p> <p>After solution, you are advised to check the average fluid velocities produced by the software, and adjust the EFCV and re-solve if significantly different from the original estimate.</p> <p>See “<a href="#">False Time Step Relaxation</a>” on page 410.</p>
Monitor Point section	— see “ <a href="#">Solver Control Tab - Monitor Point Solution Control Section</a> ” on page 204.
Variable Solution Control section	— see “ <a href="#">Solver Control Tab - Variable Solution Control Section</a> ” on page 205.

## Related Topics

[Techniques for Controlling the Solution](#)

## Solver Control Tab - Monitor Point Solution Control Section

To access: Click the **Solver Control** tab, then scroll down to the Monitor Point heading.

Use this section of the **Solver Control** tab to use Monitor Points to control the steady-state or transient solution.

### Description

This section is hidden if Type of Solution is set to Flow Only (**Model Setup** tab) because temperatures are not monitored during Flow Only solutions.

### Objects

Field	Description
Steady State Solution Control	
Monitor Point Convergence for Temperature	Expands the property sheet to allow the setting of criteria for stopping the solver when a desired level of temperature convergence is reached at all monitor points.
Required Accuracy	(Monitor Point Convergence for Temperature) Sets the accuracy to which the monitor points are judged to be converged.
Number of Iterations	(Monitor Point Convergence for Temperature) Sets the sample size over which the convergence criteria are measured.
Residual Threshold	(Monitor Point Convergence for Temperature) Sets the residual value below which the program starts testing for monitor point convergence, default 10.
Transient Solution Control	
Monitor Point Transient Termination Criteria	Only active when a Transient Solution is selected. Opens a two-column table of Monitor Point and Temperature.
Monitor Point Temperature Table	Existing Monitor Points in the model are selectable from the dropdown list. The goal temperature for each monitor point can be set.  The default temperature for the first monitor point is the global ambient temperature if it has not been previously set. The temperature values are those defined in the Global Units dialog box.

### Related Topics

[Monitor Point Convergence for Temperature](#)

[Transient Analysis Procedures](#)

## Solver Control Tab - Variable Solution Control Section

To access: Click the **Solver Control** tab, then scroll down to the Variable field.

Use this section of the **Solver Control** tab to control the solution of a field variable.

### Description

The solution duration can be controlled by the false-time step, the termination residual, and the number of inner iterations.

### Objects

Field	Description
Variable	Select the variable to which the following settings will apply. The variables available depend on: <ul style="list-style-type: none"> <li>• The Type of Solution.</li> <li>• The Turbulence Model.</li> <li>• If the fluid density of any Fluids in the model is set to follow the Ideal Gas Law.</li> <li>• If Joule Heating is switched on.</li> </ul> See “ <a href="#">The Solution Variables</a> ” on page 143.
Inner Iterations	Sets the maximum number of inner iterations on the linearized equation solver for the selected variable. See “ <a href="#">Inner Iterations</a> ” on page 413.
False Time Step	(Not Pressure variable) Sets a transient-like parameter, that is used to damp or under-relax the solution of any variable except Pressure. Set to correspond to the timescale of the problem, for example, (typical length)/(typical velocity).
Type	<ul style="list-style-type: none"> <li>• Automatic — based on fans in forced flow cases and the EFCV in natural convection cases.</li> <li>• User Specified — enables you to override the automatic settings.</li> </ul>
Damping Slider Bar	(Automatic) Use to change the automatic setting by either dividing or multiplying it by a factor of up to 20. Larger values give light damping, small values give heavy damping.
Value	(User Specified, read only if Automatic) User-specified value.
Termination Residuals	Sets the convergence tolerance, that is, the acceptable total error in solution for a variable.

Field	Description
Type	<ul style="list-style-type: none"> <li>Automatic — calculated as:           <ul style="list-style-type: none"> <li>For Pressure: 0.5% estimated typical mass flow.</li> <li>For Velocities: 0.5% estimated typical momentum.</li> <li>For Temperature: 0.5% of fixed heat source.</li> <li>For Potential (Joule Heating on): 0.5% of the total current sources.</li> </ul> </li> <li>Note: If the model contains only Fixed Value Potential electrical sources, then the estimated termination residual may be too conservative. You can change the default value and use monitor points to check convergence.</li> <li>User Specified — enables you to override the automatic settings.</li> </ul>
Factor Slider Bar	(Automatic) Use to change the automatic setting by either dividing it by up to a factor of 10, or multiplying it by a factor of up to 20.
Value	(User Specified, read only if Automatic) User-specified value.
Linear Relaxation	This is a legacy field. The field is only displayed if a pre-V10 project with a non-default setting for Linear Relaxation has been loaded.
Initial Variables	
	Sets initial values for individual field variables to be solved and stored. These initial values either help convergence in steady-state solutions or provide the starting conditions for transient calculations.
All Initial Values	<p>Sets the method of initialization of all variables:</p> <ul style="list-style-type: none"> <li>Automatic — generate values by program.</li> <li>User Specified</li> <li>Selected Solution Set — only active when <b>Select</b> has been used to load the results from another project. Sets the variable values to that of the loaded set.</li> </ul> <p>When displaying <b>Set Individually</b>, at least one variable's method of initialization differs from the others.</p>
Initial Value	Sets the method of initialization of the currently-selected variable: Automatic, User Specified or Selected Solution Set.
Value	(User Specified) User-specified value.
Solution Set	(Selected Solution Set) The name of the loaded (selected) solution set.
	<p><b>Note:</b> Solve is denied if this field is empty when Select Solution Set is active, and Error message E/1027 is output during re-initialize. Select a different option in the <b>Initial Value</b> dropdown, or select a solution set. This applies to each variable.</p>

Field	Description
Select button	(Selected Solution Set) Click <b>Select</b> to open the <a href="#">Solution Set Dialog Box</a> to select the results data from another project as the initial values for the next solution run.

## Related Topics

[False Time Step Relaxation](#)

[The Solution Process](#)

[Initial Values](#)

## Solution Set Dialog Box

To access: From the Variable Solution Control section of the **Solver Control** tab, select a Selected Solution Set for the Initial Variables Initial Value and click **Select**.

Use this dialog box to load solution data from a selected project.

### Description

The solution data of the selected project is loaded into the current project and used as the initial conditions for the next solver run.

---

#### Note

 Initializing from a stored solution works most efficiently when there are small physical differences between the current project and the one being used to initialize from. If there are large differences, especially geometric (for example, repositioned fans), it may be more efficient to re-initialize using the initial variables Automatic initial value option in the [Solver Control Tab - Variable Solution Control Section](#).

---

### Objects

Consists of a scroll list from which you can choose the project whose solution set is to be loaded.

### Related Topics

[Initial Values](#)

# Solution Type Changed Dialog Box

To access: Opens automatically when the Transient Solution check box is checked in the **Model Setup** tab and the project contains a steady-state solution, or when Transient Solution is unchecked and the project contains a transient solution.

Use this dialog box when changing the solution type from steady-state to transient, or vice versa, to continue using the same project or save as a new project.

## Description

The dialog box has three buttons.

## Objects

Object	Description
Continue using the existing project	Click to continue using the loaded project. The solution will be overwritten.
'Save As' the current project with a new name to continue	Click to save the project, but with a different name, and keep the project loaded. The Save Project dialog box is opened, allowing you to change the project name and description. The previous solution is retained with the original project.
Defer decision until later	Click to continue using the loaded project.

## Related Topics

[Model Setup Tab](#)

[Save Project Dialog Box](#)



# Chapter 5

## Creating, Importing, and Exporting Geometry

---

You can create and manipulate the model geometry using the project data tree and the drawing board. Simcenter Flotherm uses a rectangular coordinate method for modeling, and all geometry is constructed from fundamental cuboid and prism shapes.

<b>Overview of Geometry</b> .....	<b>212</b>
<b>Coordinate Systems</b> .....	<b>215</b>
<b>Construction Precedence Rules</b> .....	<b>218</b>
Construction Precedence Rules Overview .....	218
Context Rules for Primitives .....	219
Context Rules for SmartParts .....	225
<b>Geometry Files</b> .....	<b>229</b>
PDML Files .....	229
FloXML Files .....	229
FloFEA Files .....	232
ECAD File Formats .....	232
ECXML Files .....	233
JEDEC Part Model XML Files (ThermalSection) .....	234
Files Imported via MCAD Bridge .....	235
Files Imported via EDA Bridge .....	235
Power Map Files .....	235
<b>Geometry Create and Import Operations</b> .....	<b>236</b>
Creating Complex Shapes .....	236
Promoting and Demoting Objects .....	237
Copying Objects .....	237
Creating a Pattern of Objects .....	238
Mirroring an Object .....	239
Importing FloXML Geometry Files .....	240
Importing IDF Files .....	241
Importing PDML Geometry Files .....	242
Importing ECXML Files .....	242
Importing Network Assemblies From JEDEC Part Model XML Files .....	243
Importing T3Ster Files .....	243
Importing V1.4 Library Files .....	244
Importing V2/V3 Project Assembly Files .....	244
<b>Export Operations</b> .....	<b>245</b>
Exporting Geometry .....	245

Exporting an Assembly to FloTHERM PACK . . . . .	246
Exporting Network Assemblies to JEDEC Part Model XML Files . . . . .	247
Converting Files to PDML Format . . . . .	248
Exporting ECXML Files . . . . .	248
Creating FloXML From Spreadsheets . . . . .	249
<b>Pattern Creation Dialog Box . . . . .</b>	<b>251</b>

## Overview of Geometry

The geometry is built by locating and defining the geometry using primitives, assemblies, and SmartParts. In addition, you can import models of current carrying conductors as Power Maps to use to predict the effect of Joule heating.

### Creating Geometry

Geometry can be created in the model or imported into the model.

The geometry of the model can be built by:

- Adding object templates from the [New Object Palette](#). You can use the New Object Palette to add objects to the data tree (Project Manager Create) or to the drawing board (Drawing Board Create).
  - **Project Manager Create.** When you create in the data tree, the object has a default size and position.
  - **Drawing Board Create.** You create on the drawing board by drawing the outline of the object (the exceptions are Monitor Points, which are one-dimensional and are created by clicking on the drawing board). You can snap to another object or use the Snap Grid as you create the object. To exit from drawing board create mode, press Esc.
- Importing objects from [Libraries](#) into the data tree.
- Importing external ECAD, MCAD, or PDML files. See “[Geometry Files](#)” on page 229.

### Primitives

Primitives are the basic building blocks from which all Simcenter Flotherm geometry is built, either specifically or inherently using assemblies or SmartParts.

**Table 5-1. Primitives**

Primitive	Usage
Solid Cuboid	Block, internal plate, and external wall.
Collapsed Solid Cuboid	Thin internal plates and thin external walls.

**Table 5-1. Primitives (cont.)**

<b>Primitive</b>	<b>Usage</b>
Solid Prism	Flow deflection devices, non-rectilinear solution domain, corners of the cowl in highly detailed fans.
Tet	Tetrahedral objects, that is, with three right-angled triangular faces that are aligned with the three coordinate axes.
Inverted Tet	Heptahedron objects, that is, a cuboid with one of its corners removed.
Resistance	Volume Resistances such as cable bundles, heating and cooling coils, and filters.
Collapsed Resistance	Planar Resistances such as vents, perforated plates, and thin filters.
Source	Volume Sources for general heat gains.
Collapsed Source	Planar Sources such as hot surfaces.

**Note**

 Where an object acts as a barrier to flow and is not required to store or distribute heat, using a collapsed representation is often a more efficient computational model, especially if the object is very thin compared with the size of the solution domain.

## Assemblies

Assemblies are combinations of primitives, SmartParts, or other assemblies, combined to create a more complex object or set of objects.

The Simcenter Flotherm geometry construction method is based on the concept of a logical hierarchy of assemblies of parts and sub-assemblies (in other words assemblies within assemblies) with the root assembly as the top-level assembly encompassing the complete geometry structure.

All objects (primitives, SmartParts, and other assemblies) are created and associated with an assembly.

When local coordinates are used, the location of an object is relative to the origin of the assembly to which it belongs.

## SmartParts

SmartParts are complicated assemblages of primitives which can be generated parametrically, with different levels of modeling to choose the level of detail required.

The supplied SmartParts fall into two categories:

- Geometry Macros, for defining abstract shapes
- Electronic Parts, which are practical models of electronic components

By default, SmartParts are represented collectively as an overall structure, but some can be decomposed to allow individual modification of their composite parts.

**Table 5-2. SmartParts**

SmartPart	Usage
Geometry Macros	
Cylinder <sup>11</sup>	Circular PCBs.
Thick/Thin Sloping Blocks <sup>22</sup>	Thick/thin angled plates.
Blocks with Holes <sup>1</sup>	Rectangular holes.
Enclosures <sup>1</sup>	Thick or thin walls, with or without openings, other materials or resistances inserted.
Electronic Parts	
Axial/Swirl Fan	Axial fans including non-linear fan characteristics and swirl settings.
Rectangular Fan	Rectangular or squirrel cage fans.
Extruded or Bonded Heat Sink <sup>1</sup>	Detailed model constructed from cuboid primitives.
PCBs Conducting and Non-conducting <sup>1</sup>	Non-conducting for simple representation in forced convection systems or full conducting with simple 3D modeling for the board and its components.
PCB Component	For discrete definition of components on PCBs.
Recirculation Device	Heat exchanger.
Fixed Flow	For known fixed flows.
Compact Component	Compact resistor network representation of packages.

1. Available for decomposition.

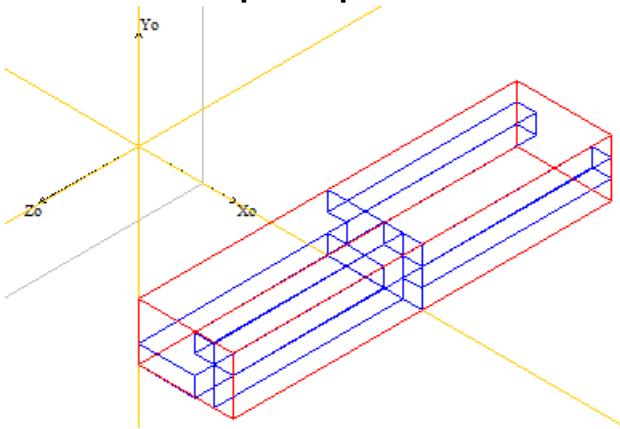
2. Only Thick Sloping Blocks are available for decomposition.

## Power Maps

Power maps are geometry and power definitions imported from Mentor Graphics HyperLynx® PI (Power Integrity) files.

In Simcenter Flotherm, a power map is modeled as a combination of copper cuboids (Figure 5-1), each having an associated power source, and is used to predict the effect of Joule heating on PCBs.

**Figure 5-1. Power Map Composite Cuboid Geometry**



## Related Topics

[FloTHERM SmartParts Reference Manual](#)

# Coordinate Systems

Geometry is defined by two coordinate systems: Local Object Coordinates and System Coordinates.

## Local Object Coordinates

Each geometry object has a local coordinate system, identified by  $X_o$ ,  $Y_o$ ,  $Z_o$ , which defines the object size and orientation of the object within the model.

The position of an object in the model is defined by the relative position of its local origin to the system origin.

When an object is moved, for example, by translation or rotation, the local coordinate system ( $X_o$ ,  $Y_o$ ,  $Z_o$ ) moves with the object.

## System Coordinates

Geometry is located in the overall model using two right-hand coordinate system types:

- **Absolute Coordinates** — are relative to the model origin.  
Absolute coordinates are the default coordinates.
- **Local Coordinates** — are dependent on the display method being used.

Local coordinates provide useful reference points for complex objects. Each sub-assembly has its own local coordinate system which enables easy location and replication of objects within that assembly.

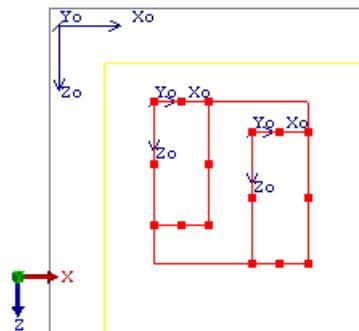
Local coordinates are the same as the absolute coordinates if the model is viewed from the root assembly (providing the root assembly is still at the default origin). Note, however, that the local coordinates are always relative to the parent assembly.

To switch between the coordinate reference types, use the setting in the **Project Manager** tab of the [User Preferences Dialog Box](#).

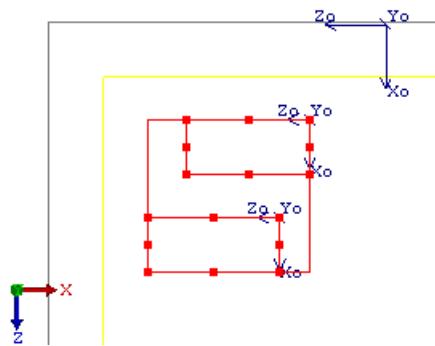
## Examples of Coordinate Systems

The following example demonstrates how the local and object coordinates are defined in the context of an assembly that is rotated.

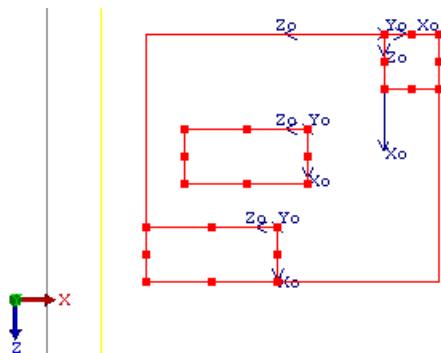
1. Consider an assembly containing two cuboids. By default, the object coordinate axes are the same as the system axes.



2. Collapse the assembly and rotate it 90° clockwise. The object coordinate axes rotate 90°, but in local coordinates the location of the cuboids has not changed.



3. In the data tree, add another cuboid to the assembly. The new cuboid is created at the origin of the assembly, but its own object coordinate axes are the same as the system axes.



---

**Note**

 System objects that model the solution domain (that is, overall solution domain, cutouts, subdomains, regions, and grid) are *always* located by absolute coordinates.

---

# Construction Precedence Rules

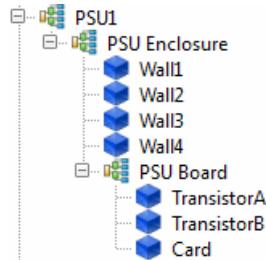
Construction precedence is significant when objects of the same type coincide.

<b>Construction Precedence Rules Overview.....</b>	<b>218</b>
<b>Context Rules for Primitives.....</b>	<b>219</b>
<b>Context Rules for SmartParts .....</b>	<b>225</b>

## Construction Precedence Rules Overview

Objects at the bottom of data tree are considered to have been created last and, in general, take precedence.

**Figure 5-2. Construction Precedence Rules**



If Wall4 and Card have coincident boundaries, then Card overwrites the settings made by Wall4 in the coincident region. Note, however, that this is not always the case for objects of different types as described by the following precedence rules:

1. Solid objects (solid cuboids and prisms) overwrite other solid objects according to the tree order.
2. Fans always overwrite collapsed resistances irrespective of tree order.
3. Flow objects (fans, resistances, free boundaries associated with cutouts and fixed flow devices) overwrite each other in tree order with the exception of point 2.
4. Uncollapsed cuboids and prisms overwrite fans and resistances irrespective of order.
5. Fans and resistances located on collapsed cuboids will overwrite the collapsed cuboid providing that they appear lower in the data structure.
6. Fixed flows take precedence over free boundaries created on cutouts.

## Related Topics

- [Context Rules for Primitives](#)  
[Context Rules for SmartParts](#)

## Context Rules for Primitives

SmartParts are provided for the construction of common complex shapes, however, unusual shapes can be built up by the insertion of multiple overlapping cuboids, triangles, and collapsed cuboids. Heat transfer in complex shapes constructed in this way are subject to rules.

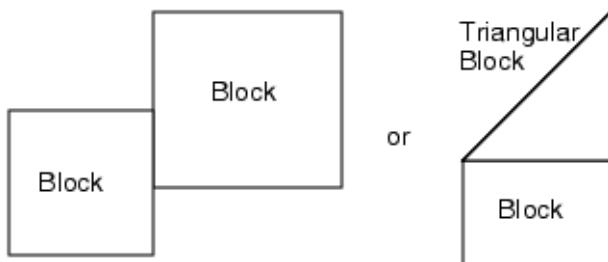
<b>Blocks or Prisms in Surface Contact .....</b>	<b>219</b>
<b>Overlapping Intersecting Blocks or Prisms.....</b>	<b>220</b>
<b>Block Placed on the Surface of a Collapsed Cuboid.....</b>	<b>222</b>
<b>Cuboid Block Intersected by a Collapsed Cuboid .....</b>	<b>222</b>
<b>Cuboids in Contact With Sloping Blocks .....</b>	<b>223</b>
<b>Cuboids in Contact With Domain Edge .....</b>	<b>224</b>

### Blocks or Prisms in Surface Contact

To explain how heat transfer is treated by the program when two bodies are in contact, we will look at two example cases.

Consider two cuboid blocks in contact, and a cuboid block and prism in contact.

**Figure 5-3. Cuboid Blocks in Surface Contact**



When two cuboid blocks are in contact, see [Figure 5-3](#), the surface between the two objects is treated as follows:

- If the thermal attributes of both objects are set to conducting, the program calculates the heat transfer between the objects, assuming that no thermal resistance exists at the interface.

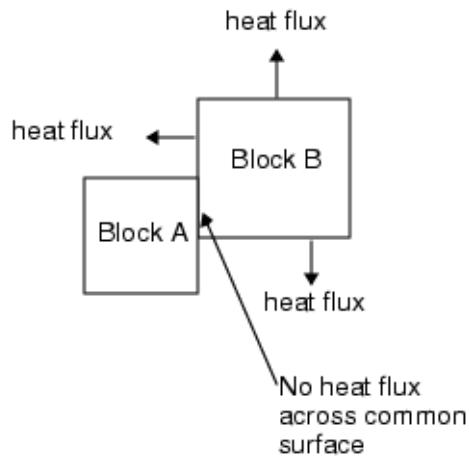
To introduce a resistance arising from the presence of an adhesive, specify the resistance as a surface property on one or more surface objects.

- If the objects are not conducting, or only conducting in one of the two objects, the program does not compute the heat conduction between them. The exposed surfaces of the objects (that is, the surfaces not in contact with other blocks or prisms) exchange heat with the surrounding fluid according to the thermal options selected.

- Heat transfer across the interface between the objects when only one of them is conducting can be contrived by specifying a source per unit area for temperature on the surface of the object.
- The Fixed Temperature and the Fixed Heat Flow (cuboids only) thermal attributes apply only to exposed surfaces of the objects, that is, the surfaces which are contacted by the fluid, and not in contact with another block or prism. In [Figure 5-4](#), a heat flux is set for Block B, all of which goes into the fluid and none of which goes into Block A.

This statement is true irrespective of the thermal option selected for Block A, that is, there is no flux of heat from Block B to Block A whether Block A is conductive, fixed flux, or fixed temperature.

**Figure 5-4. No Heat Flux Across Common Surfaces**



If the Total Power is set for Block B (rather than flux per unit area), all this power goes into the fluid, that is, the program sets the heat fluxes for the exposed faces of Block B such that the power output to the fluid equals the total power set by the user.

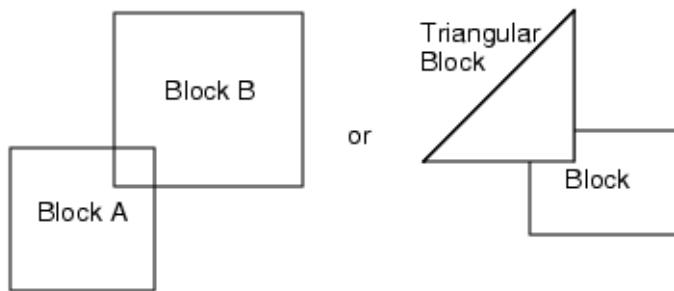
## Overlapping Intersecting Blocks or Prisms

To explain how heat transfer is treated by the program when two bodies overlap, we will look at two example cases.

Consider when two blocks overlap, and an overlapping block and prism.

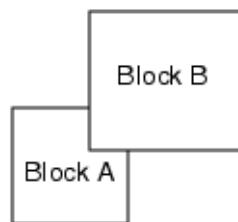
The treatments for blocks or prisms in surface contact also apply to this configuration, however, the order in which the objects are specified is important.

**Figure 5-5. Overlapping Blocks**



In the overlapping region, the thermal properties (that is, the conductivity, the density and the specific heat) of the second object in the hierarchy takes precedence over that of the first object. Therefore, the interface between the objects is defined by the perimeter of the second defined object. If Block A is entered before Block B (Figure 5-6) then the thermal properties of Block B apply in the region of intersection.

**Figure 5-6. Overlapping Blocks — Block B Specified Second**



- Block A conducting, Block B not conducting

The temperatures for Block A are not calculated in the region of intersection, but the power specified for Block A is applied in total to the non-intersecting portion of Block A (that is, no power is lost).

- Block B conducting, Block A not conducting

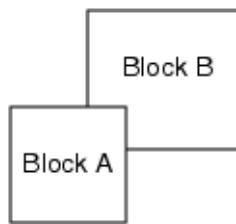
The temperature is calculated throughout Block B, including the region of intersection, with the total power uniformly applied over the whole of Block B.

- Blocks A and B both conducting

The temperatures are solved throughout and the power in the region of intersection is additive for the two cuboids, that is, the power for Block A and the power for Block B are added to one another in the region of intersection.

If Block B is entered before Block A, the thermal properties of Block A apply in the region of intersection.

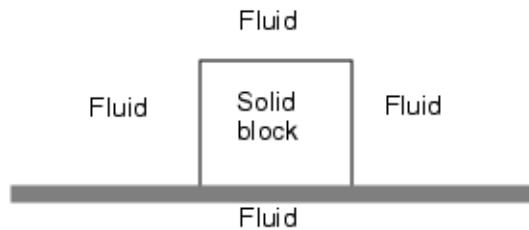
**Figure 5-7. Overlapping Blocks — Block A Specified Second**



## Block Placed on the Surface of a Collapsed Cuboid

The heat transfer depends on the thermal model defined for the thermal attribute attached to the block.

**Figure 5-8. Block Resting on a Collapsed Cuboid**



- Conduction thermal model.  
The collapsed cuboid reduces the heat transfer from the block to the fluid on the other side of the collapsed cuboid. If the thermal conductivity of the collapsed cuboid is set to zero, heat transfer is completely eliminated across the surface of intersection.
- Fixed Heat Flow or Fixed Temperature thermal model.  
The heat flux from the block is unaffected by the collapsed cuboid.

## Cuboid Block Intersected by a Collapsed Cuboid

Heat transfer for the case of a block with a penetrating collapsed cuboid.

**Figure 5-9. Block With a Penetrating Collapsed Cuboid**



In this situation, shown in [Figure 5-9](#), the thermal resistance of the collapsed cuboid alters the thermal specification of the block as follows:

- For a conducting block, a collapsed cuboid cutting through a block locally alters the thermal resistance according to the settings made for the collapsed cuboid providing the collapsed cuboid is defined lower in the tree hierarchy than the block. Hence, if the collapsed cuboid has zero conductivity, heat conduction across the collapsed cuboid within the block is prevented.
- If the fixed flux or fixed surface temperature option has been set for the block, the presence of collapsed cuboid within the block is ignored.

**Note**

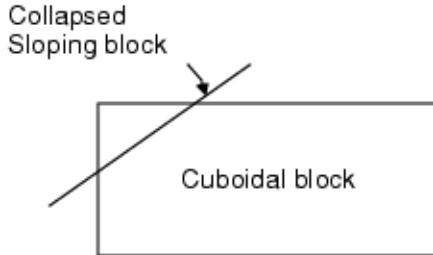
If plate conduction is activated in the **Solver Control** tab, then the collapsed cuboid will overwrite the block regardless of hierarchy precedence.

---

## Cuboids in Contact With Sloping Blocks

Heat transfer in the case of an intersection of a thin (collapsed) sloping block with a cuboid block.

**Figure 5-10. Cuboid in Contact With Sloping Block**



Depending on the specification of the sloping block, the program will act in different ways:

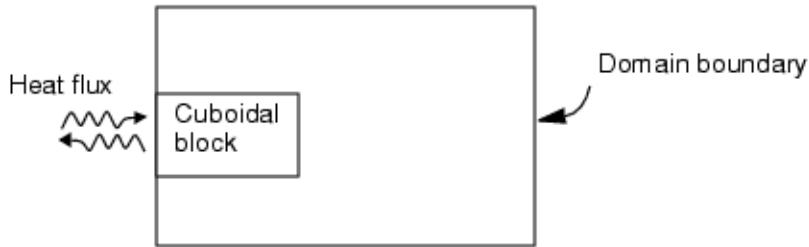
- Conducting (Thick or Thin)  
The thermal resistance of the block will be accounted for providing it is defined lower in the hierarchy than the cuboid.
- Non-conducting (Thin)  
The thin sloping block (or plate) will be ignored.
- Non-conducting (Thick)  
The sloping block will act as an insulator providing it is defined lower in the hierarchy.

## Cuboids in Contact With Domain Edge

Heat transfer where conductive cuboids are in contact with the domain edge or cutout is specified by attaching an ambient attribute.

The attached ambient sets the temperature and heat transfer coefficient. If no ambient is attached, or the heat transfer coefficient is 0.0, then no heat will be transferred from the cuboid block.

**Figure 5-11. Cuboids in Contact With Domain Edge**



## Context Rules for SmartParts

The positions of SmartParts in the model are subject to context rules.

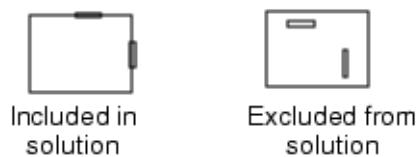
Context Rules for Flow Devices .....	225
Context Rules for Overlapping Fixed Flows.....	225
Context Rules for Overlapping Supplies and Extracts of Recirculation Devices .....	226
Context Rules for Overlapping Sources .....	226
Context Rules for Overlapping Components .....	226
Context Rules for Network Assemblies and Heat Pipes.....	226
Context Rules for Plate Conduction.....	227

## Context Rules for Flow Devices

Additional context rules are required to ensure that some flow objects are included in the solution.

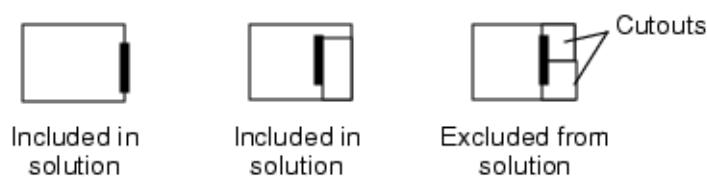
- Recirculation devices and fixed flows must sit on the edge of cuboids, cutouts, or the domain boundary.

**Figure 5-12. Context Rules for Recirculation Devices and Fixed Flows**



- Fans and resistances must be on the edge of *one and only one* cutout or domain boundary, or entirely within the solution domain.

**Figure 5-13. Context Rules for Fans and Resistances**



## Context Rules for Overlapping Fixed Flows

If two or more fixed flows overlap, their volume flow rates are additive, and, by default, the cumulative flow is averaged across the cross-sectional area of the overlapping apertures.

The FLOFIXEDFLOWSOVERWRITE environment variable enforces hierarchical precedence such that a fixed flow with the highest precedence maintains its flow through its defined area, an

overlapping fixed flow with a lower precedence maintains its flow through the remaining area, and so on. The total flow rate, as set, is still maintained, but its distribution is determined by the precedence of the fixed flows.

The use of FLOFIXEDFLOWSOVERWRITE only applies to fixed flows where the volume flow rate is set, it does not apply when the flow is set by velocity.

## Related Topics

[Environment Variables](#)

[Construction Precedence Rules](#)

## Context Rules for Overlapping Supplies and Extracts of Recirculation Devices

The context rules are dependent on whether the overlaps are from the same or from different recirculation devices.

- If two or more supplies (or extracts) from the same recirculation device overlap, they override, but in such a way that the total mass flow required by that recirculation device is maintained.
- If two or more supplies (or extracts) from different recirculation devices overlap, their flow rates are additive.

## Context Rules for Overlapping Sources

When two or more sources overlap, the effect of each source is added together to provide the total source.

For example, the power is additive in the region of overlapping heat sources.

## Context Rules for Overlapping Components

Overlapping discrete solid components are treated in the same way as blocks and prisms.

See “[Overlapping Intersecting Blocks or Prisms](#)” on page 220.

Overlapping components which are discrete, but not solid, are treated in the same way as overlapping sources; the power is additive in the region of overlap.

## Context Rules for Network Assemblies and Heat Pipes

The network cuboids exhibit the normal order of precedence rules within the network assembly or heat pipe, that is, cuboids lower down the tree will overwrite those above.

The network assembly and heat pipe use the Embedded Conduction Solver. As a consequence of this, their cuboids cannot be overwritten by other solid geometry lower down the data tree, unless that has also been solved by the Embedded Conduction Solver.

## Context Rules for Plate Conduction

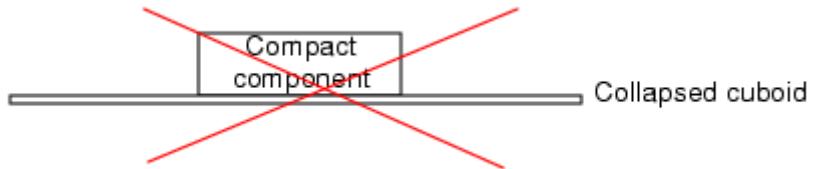
When Activate Plate Conduction is checked on in the **Solver Control** tab, all collapsed cuboids are treated as uncollapsed objects.

The following context rules apply:

- A collapsed cuboid will always overwrite the uncollapsed cuboid regardless of hierarchy precedence.
- A network assembly abutting a collapsed cuboid will conduct into the collapsed cuboid.



- A compact component abutting a collapsed cuboid is an invalid use case and will not be solved.



- Collapsed cuboids in edge-to-edge contact must abut by at least one grid cell, in their uncollapsed state, for there to be conduction between them.

**Figure 5-14. Example Layouts Where Conduction Between Collapsed Cuboids Will Occur**



**Figure 5-15. Example Layout Where Conduction Between Collapsed Cuboids Will Not Occur**

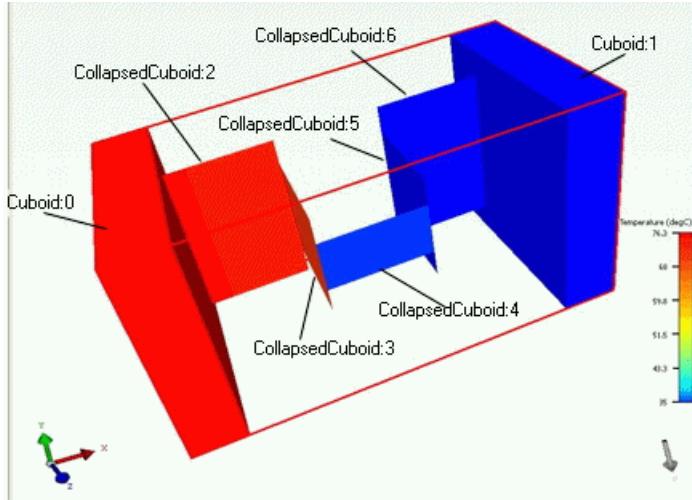


**Note**

-  To ensure that the collapsed cuboids abut by at least one grid cell when in their uncollapsed state, view them in their uncollapsed state in the drawing board with the solver grid switched on.
- 

In this example, a group of collapsed cuboids have been placed incorrectly such that heat will not flow from beyond CollapsedCuboid:3.

**Figure 5-16. Plate Conduction Example Showing Break in Conduction**



In cases such as this, uncollapse all the objects and check their arrangements against the example layouts given in [Figure 5-14](#) and [Figure 5-15](#).

## Geometry Files

---

Simcenter Flotherm can import and export geometry in different file formats.

<b>PDML Files</b> . . . . .	<b>229</b>
<b>FloXML Files</b> . . . . .	<b>229</b>
<b>FloFEA Files</b> . . . . .	<b>232</b>
<b>ECAD File Formats</b> . . . . .	<b>232</b>
<b>ECXML Files</b> . . . . .	<b>233</b>
<b>JEDEC Part Model XML Files (ThermalSection)</b> . . . . .	<b>234</b>
<b>Files Imported via MCAD Bridge</b> . . . . .	<b>235</b>
<b>Files Imported via EDA Bridge</b> . . . . .	<b>235</b>
<b>Power Map Files</b> . . . . .	<b>235</b>

## PDML Files

PDML (Physical Design Modeling Language) files are used to transfer information to and from the external file system for either the complete project or any selected assembly in the data tree.

By default, PDML files are binary, however, ASCII access can be made available with a special license key. Contact your local Simcenter Flotherm supplier for more information on the ASCII access license key if required.

Mentor Graphics uses PDML files to provide library objects for your modeling; some of these are available through starter libraries.

Other Mentor Graphics products, such as FloTHERM PACK and EDA Bridge, provide models for Simcenter Flotherm using PDML.

### Related Topics

[Archiving Project Data in PDML Format](#)

[Importing a PDML Project](#)

[Importing PDML Geometry Files](#)

## FloXML Files

FloXML files are XML files that can be imported into Simcenter Flotherm.

FloXML files must conform to the FloXML Schema. There are two FloXML file types:

- **Project FloXML files** — which contain geometry and solution domain information. Project files must contain a minimum FloXML project definition.

- **Geometry FoXML files** — which only contain geometry information.

The name defined within the XML file, for example:

```
- <xml_case>
  <name>Heatsink</name>
  .
  .
  .
</xml_case>
```

is used to name the Simcenter Flotherm project/assembly that is created from the file.

An alternative to importing an XML file directly using the GUI, is to convert the XML file to PDML format using the flogate\_cl utility, for example:

```
flogate_cl -i XML -r my_example.xml -o PDML -w my_example.pdml
```

See “[Converting Files to PDML Format](#)” on page 248. The resultant PDML file can then be imported into Simcenter Flotherm.

## Minimum Project FoXML Definition

A valid project FoXML file must contain a minimum set of default definitions.

An example set of default definitions is supplied within file *Default.xml* in the following location:

```
<install_dir>\flosuite_v<version>\flotherm\examples\FoXML\FoXML Files\Project
FoXML Examples
```

## FoXML Schema

The schema \*.xsd files are in the following location:

```
<install_dir>\flosuite_v<version>\common\WinXP\lib\flotherm\Plugin\
XML_PLUGIN_SCHEMA
```

---

### Note

 The FoXML Schema supports the definition of all objects and settings that can be made through the GUI (objects, attributes, grid settings, model settings, solver settings, and so on) *except* auxiliary variables and Power Maps.

---

## FoXML File Examples

Examples of FoXML files are provided in the installation:

```
<install_dir>\flosuite_v<version>\flotherm\examples\FoXML\FoXML Files
```

The examples, which are grouped as Assembly or Project FloXML files, are described in [Table 5-3](#) and [Table 5-4](#).

**Table 5-3. Assembly FloXML Examples**

<b>Filename</b>	<b>Description</b>
<i>2R-Model.xml</i>	A two-resistor model, based on a network assembly and three network nodes (Junction, Case, and PCB). The junction node has a power output of 3.2 W. The junction-to-board resistance is 22.2 K/W, and the junction-to case-resistance is 1.2 K/W. A monitor point is attached to the junction.
<i>Advanced-Resistance.xml</i>	A resistance object and advanced resistance attribute with loss coefficients based on device velocity.
<i>Block.xml</i>	A 20 mm × 10 mm × 1 mm aluminum block.
<i>Nested-Assemblies.xml</i>	Three levels of nested assemblies, the innermost assembly containing a cuboid.

**Table 5-4. Project FloXML Examples**

<b>Filename</b>	<b>Description</b>
<i>All-Objects-Attributes-Settings-FullModel.xml</i>	A root assembly containing a set of different objects with attributes to illustrate a wide variety of XML definitions.
<i>Default.xml</i>	Not a project, but the minimum definitions required by valid Project FloXML files.
<i>Heatsink-Windtunnel-FullModel.xml</i>	A 10-folded-fin heat sink on a 40 mm × 40 mm base with a 1 W heat source, and a 0.001 m <sup>3</sup> /s fixed volume flow in the fin direction.
<i>PDML-Referencing-FullModel.xml</i>	An example of an project XML file that includes a reference to a PDML file. It is assumed that the PDML file is named <i>PCB.pdml</i> and is located in the <i>C:\</i> folder:  <pre>&lt;pdml&gt; &lt;name&gt;PDML&lt;/name&gt; &lt;file&gt;c:\PCB.pdml&lt;/file&gt; &lt;position&gt;&lt;x&gt;0&lt;/x&gt;&lt;y&gt;0&lt;/y&gt;&lt;z&gt;0&lt;/z&gt;&lt;/position&gt;</pre>

## Related Topics

[FloXML FAQs](#)

[Importing FloXML Geometry Files](#)

[Importing a FloXML Project](#)

[flogate\\_cl](#)

## FloFEA Files

Simcenter Flotherm can export FloFEA (Finite Element Assembly) files.

FloFEA files can be imported into third-party mapping software.

### Related Topics

[Exporting Geometry](#)

## ECAD File Formats

You can import ECAD geometry from IDF board description files or using one of the supplied direct interfaces.

The Simcenter Flotherm Project Manager can import IDF data, whilst the Simcenter Flotherm EDA Bridge offers direct interfacing in addition to a more advanced IDF import capability.

For information on the supplied direct interfaces via EDA Bridge, see [EDA Board Designs](#) in the *Simcenter Flotherm EDA Bridge User Guide*.

For an overview of running EDA Bridge, see [Overview Procedure When Using EDA Bridge](#) in the *Simcenter Flotherm EDA Bridge User Guide*.

### The IDF 2.0 and 3.0 Board Description Files

The IDF description of a board consists of two files:

1. A board file (*.brd* or *.emn*) to describe:
  - o The shape of the board.
  - o Positions and orientations for mechanical and electrical components.
  - o Positions of drilled holes (vias).
  - o Any restrictions on component placement, and so on.

2. A library file (.lib or .emp) which contains geometrical and (optionally) thermal details of all the components. The following is an example of component definition with the optional thermal details highlighted in the IDF library file:

```
.ELECTRICAL
PLCC84 EPF8282A_LCC_5000_7.4_22.4_ THOU 150.0000
0 -650.0000 -625.0000 0.0000
0 650.0000 -625.0000 0.0000
0 650.0000 625.0000 0.0000
0 -650.0000 625.0000 0.0000
0 -650.0000 -625.0000 0.0000
PROP POWER_OPR 5000
PROP THETA_JC 22.4
PROP THETA_JB 7.4
.END_ELECTRICAL
```

Both thermal resistances THETA\_JC and THETA\_JB must be present.

Simcenter Flotherm must read in both files to combine their specifications and produce an assembly comprising both geometry and thermal data (if available in the original IDF files).

## ECXML Files

You can import and export assemblies and projects to and from Simcenter Flotherm in Electronics Cooling XML (ECXML) open neutral file format.

ECXML files must conform to the Simcenter Flotherm ECXML Schema. There are two ECXML file types:

- **Project ECXML files** — which contain solution domain, material, and geometry information.
- **Assembly ECXML files** — which contain material and geometry information.

The <name> defined within the ECXML is used to name the project/assembly that is created from the file.

### ECXML Schema

The Simcenter Flotherm ECXML Schema, *ECXML.xsd*, is provided in:

```
<install_dir>\flosuite_v<version>\common\WinXP\lib\flotherm\Plugin\
ECXML_PLUGIN_SCHEMA
```

### Related Topics

[Importing ECXML Files](#)

[Exporting ECXML Files](#)

## JEDEC Part Model XML Files (ThermalSection)

Simcenter Flotherm Network Assembly SmartParts can be imported from, and exported to, JEDEC Part Model xml files. Only the *ThermalSection* of the xml file is read and written by the Simcenter Flotherm import and export processes.

### References

The following publications are available from the JEDEC website:

- *Part Model Thermal Guidelines for Electronic-Device Packages - XML Requirements, JEP30-T100, February 2018* contains the ThermalSection XML Schema definition.
- *Part Model Guidelines for Electronic-Device Packages - XML Requirements, JEP30, February 2018* contains the complete XML Schema definition.

### Model Types

Only DELPHI and 2-Resistor model types are handled.

### Naming Conventions

The following naming is used when exporting Network Assemblies to JEDEC XML files:

- When exporting only DELPHI model types, or a mixture of DELPHI and 2-Resistor model types:
  - The first Network Assembly is named “Core Network”.
  - The next Network Assembly in the hierarchy is named “Connection Node”.
  - Subsequent Network Assemblies are named “Connection Node <n>” where <n> is 1, 2, 3, and so on.
- When exporting only 2-Resistor model types:
  - If there is only one Network Assembly, then it inherits the name of the parent geometry assembly.
  - If there are two Network Assemblies, the first is named “Core Network” and the second is named “Connection Node”.
  - If there are more than two Network Assemblies, then the subsequent Network Assemblies are named “Connection Node <n>” where <n> is 1, 2, 3, and so on.
- Child Network Cuboids inherit names from their parent Network Node as follows:

<parent\_network\_node\_name> <n>

where <n> is 1, 2, 3 and so on.

## Related Topics

[Importing Network Assemblies From JEDEC Part Model XML Files](#)

[Exporting Network Assemblies to JEDEC Part Model XML Files](#)

## Files Imported via MCAD Bridge

You can import a number of MCAD geometry file formats via MCAD Bridge.

See [Import and Export](#) in the *Simcenter Flotherm MCAD Bridge User Guide*.

## Files Imported via EDA Bridge

Direct Interfaces are available to Cadence Allegro PCB Designer, Cadence Allegro Package Designer, Mentor Graphics Board Station, Mentor Graphics Expedition Enterprise Flow and Zuken CR5000 Board Designer.

See [Importing Designs](#) and [Exporting Designs](#) in the *Simcenter Flotherm EDA Bridge User Guide*.

## Power Map Files

Simcenter Flotherm can import Joule Heating power values as predicted by HyperLynx® PI,

For further details, see [Power Maps](#) in the *Simcenter Flotherm SmartParts Reference Guide*.

# Geometry Create and Import Operations

---

The following operations are associated with creating the geometry of a model.

<b>Creating Complex Shapes</b> .....	<b>236</b>
<b>Promoting and Demoting Objects</b> .....	<b>237</b>
<b>Copying Objects</b> .....	<b>237</b>
<b>Creating a Pattern of Objects</b> .....	<b>238</b>
<b>Mirroring an Object</b> .....	<b>239</b>
<b>Importing FloXML Geometry Files</b> .....	<b>240</b>
<b>Importing IDF Files</b> .....	<b>241</b>
<b>Importing PDML Geometry Files</b> .....	<b>242</b>
<b>Importing ECXML Files</b> .....	<b>242</b>
<b>Importing Network Assemblies From JEDEC Part Model XML Files</b> .....	<b>243</b>
<b>Importing T3Ster Files</b> .....	<b>243</b>
<b>Importing V1.4 Library Files</b> .....	<b>244</b>
<b>Importing V2/V3 Project Assembly Files</b> .....	<b>244</b>

## Creating Complex Shapes

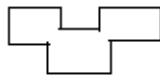
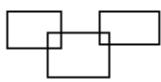
Irregular shaped objects can be created by uniting or deleting overlapping geometry using the MCAD Bridge application window. There are three main stages to creating a complex shape: loading geometry, combining MCAD bodies, and returning geometry to the data tree.

### Prerequisites

- A project containing the overlapping geometry that will make up the complex shape.

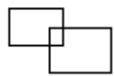
### Procedure

1. Click the **Launch MCAD Bridge** icon to start MCAD Bridge.
2. Choose **External > Import Project** to load the project geometry into MCAD Bridge.  
The individual Simcenter Flotherm objects now become MCAD Bridge bodies.
3. Remaining in MCAD Bridge, select intersecting bodies to make the following changes:
  - To unite intersecting geometry into one body, choose **Tools > Unite Bodies**.



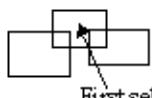
Three intersecting bodies united into a single body

- To delete all but the intersecting bodies, choose **Tools > Intersect Bodies**.



Two bodies replaced by a single body occupying the region of intersection

- To delete all the intersections bodies, select your main body first, then select intersecting bodies, then choose **Tools > Subtract Bodies**.



Three bodies replaced by a single body occupying the region that remains after subtracting the regions of intersection

4. Choose **Tools > Decompose** to revert the geometry back to Simcenter Flotherm geometry.
5. Choose **Tools > Transfer Assembly** to return the geometry to the Project Manager in its new form.
6. In the Project Manager data tree, load the complex shape into the library, from where it can be copied into other projects.

## Promoting and Demoting Objects

The creation order sequence in the data tree can be changed by promotion and demotion.

The position of an object in the data tree hierarchy may be critical to the representation of your model, for example, a lower item may overwrite one higher up, as described in “[Construction Precedence Rules](#)” on page 218.

### Procedure

You have a choice.

- Drag and drop items in the data tree.
- Select an item then click the **Promote** or **Demote** icon to move the selected item up or down the hierarchy, one position at a time.

### Results

Moving an object above or below an assembly makes the object a child of the assembly, rather than an adjacent sibling.

## Copying Objects

You can make a single copy of one or more objects.

## Procedure

To copy one or more objects to a different assembly or to a different location:

If you want to...	Do the following:
Copy to a different assembly in the data tree.	<ol style="list-style-type: none"><li>1. Select the object(s) to be copied and press Ctrl+C.</li><li>2. Select the new parent assembly and press Ctrl+V.</li></ol> <p>The pasted object(s) are added to the bottom of the assembly in the same order they were prior to being copied.</p>
Copy to a different location.	<ol style="list-style-type: none"><li>1. Select the object(s) in the drawing board.</li><li>2. Ctrl+drag the selected object(s) to the required location.</li></ol>

## Creating a Pattern of Objects

You can create a 1-dimensional or 2-dimensional pattern of equally spaced objects from a single object.

---

### Note

 Patterning can create a large grid cell count. It is recommended that you only use the necessary level of detail (for example, for fans, use a simpler representation) and, where appropriate, use a localized grid.

---

## Restrictions and Limitations

- You cannot create a pattern of the multiple-selected objects.
- You cannot create a pattern of the root assembly.
- You cannot create a pattern of the following SmartParts:
  - PCB Component
  - Network Node
  - Power Map

## Procedure

1. Select an object.

This will be the first element of the pattern.

2. Choose **Edit > Pattern** to open the Pattern Creation dialog box.

3. Enter the number of copies and spacing (pitch) along the selected direction(s), then click **Apply**.

## Results

The pattern is generated and displayed in the drawing board. The new objects are added at the same level in the data tree hierarchy as the original object.

## Related Topics

[Pattern Creation Dialog Box](#)

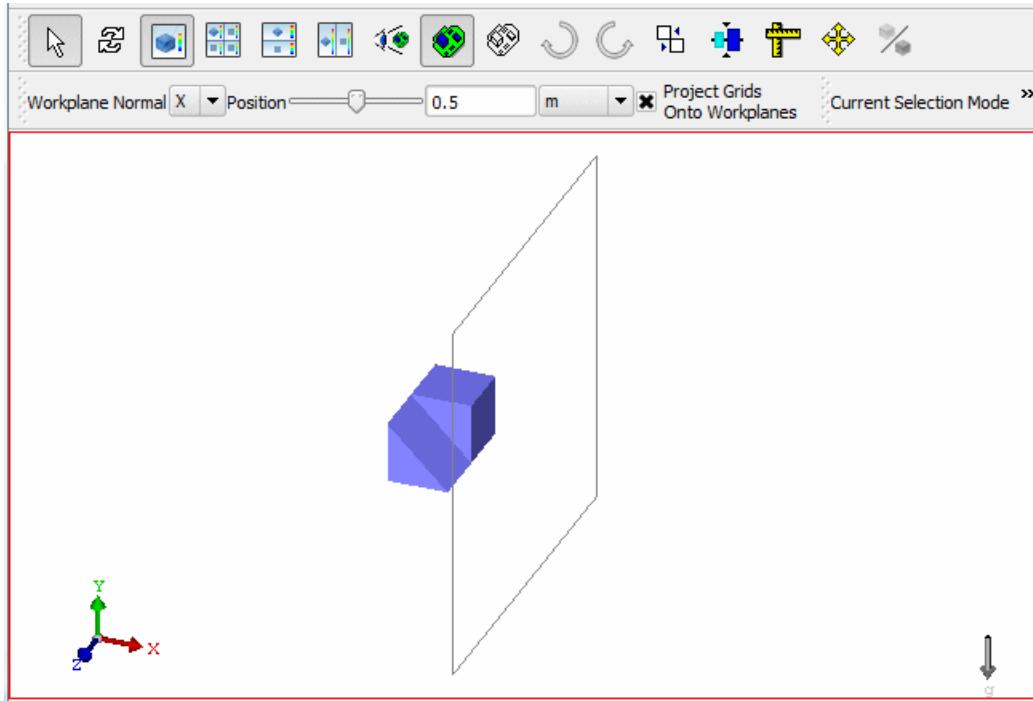
# Mirroring an Object

A mirrored copy can be made of an object or group of objects, using the workplane as the mirror plane.

## Procedure

1. Use the Workplane toolbar controls to create a mirror plane midway between the object(s) to be copied and the location of the required object.

**Figure 5-17. Objects Before Mirroring**

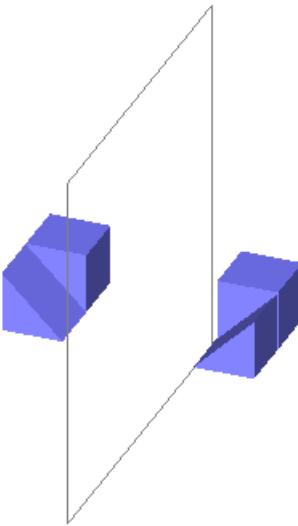


2. Select the object(s) and choose **Geometry > Mirror**, or click the **Mirror** icon in the Drawing Board toolbar.

## Results

A mirrored copy of the selected object(s) is made.

**Figure 5-18. Objects After Mirroring**



## Related Topics

[Workplane](#)

# Importing FloXML Geometry Files

Geometry FloXML files are XML files that can be imported into Simcenter Flotherm as assemblies.

## Restrictions and Limitations

- FloXML files must conform to the FloXML Schema.
- You cannot import a Geometry FloXML file from a network directory using a UNC path. Either copy the file to a local directory or map the network folder containing that file.

## Procedure

You have a choice:

If you want to...	Do the following:
Use the context-sensitive menu.	<ol style="list-style-type: none"><li>1. In the data tree, right-click the assembly to hold the imported geometry and choose <b>Import Assembly &gt; FloXML</b>.</li><li>2. Select and load a Geometry XML file. By default, the file browser lists all files.</li></ol>
Use the mouse.	<ul style="list-style-type: none"><li>• Ctrl+drag the XML file from a Windows Explorer window.</li></ul>

## Related Topics

[Importing External Projects](#)

[FloXML Files](#)

# Importing IDF Files

Geometry import from IDF version 2.0 and 3.0 file formats is supported.

## Prerequisites

- Optionally, configure IDF import preferences as follows:
  - a. Check the Configure IDF check box in the **Project Manager** tab of the User Preferences dialog box. Data entry fields and check boxes for conductivity, filtering by size, board, and package default thicknesses, use of offset data in V3.0 files, and the naming convention are made available.
  - b. Use the dialog box fields as appropriate.

## Procedure

1. In the data tree, right-click the assembly to hold the imported geometry and choose **Import Assembly > IDF**.  
IDF opens file browsers to select and import IDF Version 2.0 (\*.lib and \*.brd) or 3.0 (\*.lib and \*.brd or \*.emp and \*.emn) files. See “[ECAD File Formats](#)” on page 232.
2. Using the browser, select the board description you want to import:
  - .brd file for IDF V2.0.
  - .lib or .emn file for IDF V3.0.
3. Click **Open**.  
The browser remains open.
4. Select the appropriate geometrical file:
  - .lib file, if a .brd file was the first choice.
  - .emp file, if a .emn file was the first choice.
5. Click **Open**.

## Results

The geometry appears in the data tree below the selected assembly.

You can inspect and edit any data imported into Simcenter Flotherm using the software as normal. For IDF files, use the PCB property sheet **Construction** tab to inspect and edit the board dimensions, composition, and component details.

## Related Topics

[User Preferences Dialog Box - Project Manager Tab](#)

# Importing PDML Geometry Files

PDML geometry files can be imported into assemblies.

## Restrictions and Limitations

- ASCII PDML is only issued on special license and is not required for the normal running of Simcenter Flotherm. Contact your Simcenter Flotherm supplier if you need this format.

## Procedure

1. In the data tree, right-click the assembly that you want to hold the imported geometry and choose **Import Assembly > PDML**.  
A file browser is opened with a \*.pdml filter.
2. Select the file and click **Open**.

## Related Topics

[PDML Files](#)

# Importing ECXML Files

Importing assemblies and projects in Electronics Cooling XML (ECXML) file format.

## Restrictions and Limitations

- ECXML files must conform to the Simcenter Flotherm ECXML Schema.
- Encrypted ECXML files are not supported.

## Procedure

1. You have a choice:

If you want to...	Do the following:
Import an assembly.	<ol style="list-style-type: none"><li>1. Select a parent assembly.</li><li>2. Right click and choose <b>Import Assembly &gt; ECXML</b>.</li></ol>
Import a project.	<ol style="list-style-type: none"><li>1. Choose <b>Project &gt; Import Project &gt; ECXML</b>. If the current project has not been saved, you are prompted to save the project.</li></ol>

A file browser dialog box is opened.

2. Navigate to a folder, select an ECXML file, and click **Open**.

## Results

When importing an assembly, the imported assembly becomes a child of the selected assembly.

## Related Topics

[ECXML Files](#)

[Exporting ECXML Files](#)

# Importing Network Assemblies From JEDEC Part Model XML Files

Network Assembly SmartParts can be created by importing JEDEC xml files.

## Prerequisites

- An *xml* file that conforms to the *ThermalSection* of the JEDEC XML Schema, as described in “[JEDEC Part Model XML Files \(ThermalSection\)](#)” on page 234.

## Procedure

1. In the data tree, right-click the assembly which you want to hold the imported geometry and choose **Import Assembly > JEDEC Part Model Thermal**.

A file browser is opened with a \*.xml filter.

2. Select the file and click **Open**.

## Results

The *ThermalSection* lines of the *xml* file are used by Simcenter Flotherm to create Network Assemblies, Network Nodes and Network Cuboids. All other lines of the file are ignored.

If there is no *ThermalSection* in the imported file, then a Warning message is output and nothing is created.

## Related Topics

[Exporting Network Assemblies to JEDEC Part Model XML Files](#)

# Importing T3Ster Files

Files generated by Mentor Graphics T3Ster-Master software can be imported into assemblies.

## Procedure

1. In the data tree, right-click the assembly which you want to hold the imported geometry and choose **Import Assembly > T3Ster**. A file browser is opened with a \*.xCTM filter.  
A \*.xCTM file is a file representing a compact thermal resistor/capacitor model of an IC package, generated by Mentor Graphics T3Ster-Master software.
2. Select the file and click **Open**.  
A Network Assembly object is added to the assembly.

## Related Topics

[Importing Library Files](#)

## Importing V1.4 Library Files

Simcenter Flotherm version 1.4 library files can be imported into assemblies.

## Procedure

1. In the data tree, right-click the assembly to hold the imported geometry and choose **Import Assembly > V1.4 File**.
2. In the file browser, select the required Simcenter Flotherm V1.4 \*.lib or \*.LIB file (the default file filter is for \*.??b files) and click **Open**.

## Results

The contents of an imported .lib file are added to the selected assembly. The imported items are named as the library file without the .lib extension and appended with a sequence number.

For further information on file conversion details, contact Mentor Graphics customer support.

## Importing V2/V3 Project Assembly Files

Simcenter Flotherm version 3 \*.assembly files are assemblies of primitives and SmartParts exported from Simcenter Flotherm 3.x using the Select Assembly dialog box.

## Procedure

1. In the data tree, right-click the assembly to hold the imported geometry and choose **Import Assembly > V2/V3 Project**.
2. In the file browser to select and import a Simcenter Flotherm version 2.x or 3.x project.

## Results

After importing the assembly files into Simcenter Flotherm data tree, the resultant geometry can be dragged or copied into the Library Manager for use in the current version library.

# Export Operations

The following operations are associated with exporting the geometry of a model.

<b>Exporting Geometry .....</b>	<b>245</b>
<b>Exporting an Assembly to FloTHERM PACK.....</b>	<b>246</b>
<b>Exporting Network Assemblies to JEDEC Part Model XML Files .....</b>	<b>247</b>
<b>Converting Files to PDML Format .....</b>	<b>248</b>
<b>Exporting ECXML Files .....</b>	<b>248</b>
<b>Creating FloXML From Spreadsheets .....</b>	<b>249</b>

## Exporting Geometry

Geometry assemblies can be exported from Simcenter Flotherm either in file format or using one of the supplied direct interfaces.

This section describes the file interface via the Project Manager. For information on the supplied interfaces via EDA Bridge, see [Exporting Designs](#) in the *Simcenter Flotherm EDA Bridge User Guide*.

### Procedure

1. Right click the assembly containing the geometry to be exported and choose **Export Assembly**.
2. Choose the appropriate file type from the submenu.

If you want to...	Do the following:
Export the assembly as a FLOFEA (*.flofea) file.	Choose <b>FloFEA</b> from the submenu.
Save an MCAD assembly as a *.igs format file	Choose <b>IGES</b> from the submenu.
Export the assembly geometry in a PDML format file of the current version of Simcenter Flotherm.	Choose <b>PDML</b> from the submenu.
Export the assembly as an ACIS *.sat file.	Choose <b>SAT</b> from the submenu.

A file browser opens with a default filename.

3. Select the destination for the output file, optionally change the default filename, and click **Save**.

### Results

- \*.sat files do not include the assembly hierarchy, object names or units.

- \*.igs files do not include the assembly hierarchy, object, or names.

## Related Topics

[PDML Files](#)

[Converting Files to PDML Format](#)

# Exporting an Assembly to FloTHERM PACK

Generating a PDML file for uploading to FloTHERM PACK

## Restrictions and Limitations

- The exported assembly cannot contain assemblies or child objects (even if they are deactivated or ignored<sup>1</sup>) that have attached Material attributes with the following properties. If this is the case, then the export is stopped and error message (E/17016) is output, highlighting the attribute by name.
  - Isotropic non-linear thermal conductivity where the non-linear conductivity is defined by *more than two points* using the Table input method.

If only two points are defined, then the following linear equation definition of thermal conductivity is derived from the Table points:

$$k = Value + Coeff \times (T - T_{ref})$$

where *Value* is the conductivity at  $T_{ref}$ , and *T* is the temperature of the solid material.

$T_{ref}$  and *Value* are the temperature and thermal conductivity at the first point in the table. *Coeff* is the gradient of the chart, that is, the ratio of the difference in thermal conductivity divided by the difference in temperature.

- Orthotropic or biaxial thermal conductivity where the conductivity is non-linear in one or more directions.
- The error message E/17015 is output when you attempt to export an EDA SmartPart for import into FloTHERM PACK. However, you can export EDA Component SmartParts that are Detailed Components, but you must select the underlying assembly of the SmartPart, made visible by extending the SmartPart (Lightweight off).

## Procedure

1. Right click the assembly containing the geometry to be exported and choose **Export Assembly**.

2. Right click an assembly and choose **Export Assembly > FloTHERM PACK**

A file browser opens with a default filename.

- 
1. Set by check boxes in the **Location** tab of the respective property sheet.

3. Select the destination for the output file, optionally change the default filename, and click **Save**.

## Results

A PDML assembly file is created that can be uploaded to FloTHERM PACK

# Exporting Network Assemblies to JEDEC Part Model XML Files

Network Assemblies within a single parent geometry assembly can be exported to a JEDEC Part Model xml file.

## Restrictions and Limitations

- Network Assembly monitor points are not exported.  
Anything other than Network Assemblies and their child Network Nodes and Network Cuboids or collapsed Network Cuboids that are contained within the parent geometry assembly are ignored.
- The only attached attributes that are exported are Thermal attributes with a Thermal Model of Conduction and a Total Power definition.
- Overlapping faces are supported but must conform to the following:
  - Only one face on top of another is supported. The case of multiple faces on top of a face is not supported.
  - The faces must be in different Network Nodes.
  - The Network Nodes must be in the same parent Network Assembly.
- Deactivated Network Assemblies will be exported, and will be activated if re-imported.
- The exported <x>, <y> and <z> values of <PackageBodyCenterOffset-to-Origin> are 0.0000000000000000.
- Network Assemblies and Network Cuboids will be renamed in the exported file as described in “[JEDEC Part Model XML Files \(ThermalSection\)](#)” on page 234.

## Procedure

1. Right click the parent geometry assembly containing the Network Assemblies to be exported and choose **Export Assembly > JEDEC Part Model Thermal**.  
A file browser opens with a default filename.
2. Select the destination for the output file, optionally change the default filename, and click **Save**.

## Results

Only the *ThermalSection* is included in the exported file.

## Related Topics

[Importing Network Assemblies From JEDEC Part Model XML Files](#)

# Converting Files to PDML Format

The **flogate\_cl** utility converts external file format data to PDML for subsequent import into Simcenter Flotherm. This is particularly useful when creating a number of library files for distribution to other Simcenter Flotherm users.

## Procedure

1. Open a command window and set the Simcenter Flotherm environment by typing:

```
<install dir>\flosuite_v<version>\flotherm\WinXP\bin\flotherm -env
```

2. Then type the required **flogate\_cl** command line. Refer to “[flogate\\_cl](#)” on page 613 for a complete description of the command.

# Exporting ECXML Files

Exporting assemblies and projects in Electronics Cooling XML file format.

## Restrictions and Limitations

- Encrypted ECXML files are not supported.
- EDA Electrical Vias Assembly and EDA Layer Assembly EDA SmartParts are not exported.

## Procedure

1. You have a choice:

If you want to...	Do the following:
Export an assembly.	<ol style="list-style-type: none"><li>1. Select the assembly.</li><li>2. Right click and choose <b>Export Assembly &gt; ECXML</b>.</li></ol>
Export the whole project.	<ol style="list-style-type: none"><li>1. Choose <b>Project &gt; Export Project &gt; ECXML</b>.</li></ol>

A file browser dialog box is opened.

The default filename is *<assembly name>.ecxml* or *<project name>.ecxml*.

2. Navigate to a folder and click **Save**.

## Related Topics

[ECXML Files](#)

[Importing ECXML Files](#)

# Creating FloXML From Spreadsheets

Example files of how spreadsheets can be used to script the creation of XML models are supplied in the installation.

## Procedure

1. Open example spreadsheets from the following folder:

`<install_dir>\flosuite_v<version>\flotherm\examples\FloXML\Spreadsheets`

2. Read their descriptions in [Table 5-5](#).

FloXML models can be solved on the command line to enhance the level of automation possible when working with FloXML models. Refer to the command line descriptions under “[Solving in Batch Mode](#)” on page 367 for details.

## Examples

The supplied *Solve-Folded\_Fin\_Heat\_Sink.xls* file in the *Spreadsheets* folder is an example of how this level of automation can be achieved.

**Table 5-5. Supplied XML Spreadsheet Examples**

File	Description
<i>Advanced-Resistance.xls</i>	Creates a Geometry FloXML file containing a resistance object and advanced resistance attribute based on speed and pressure data.
<i>Data_Center_SI_Units.xls</i>	Creates a ready-to-solve model of a data center. Room dimensions, Rack, Cooler, and Perforated Tiles locations, and performance data are parametrically described in several tabs using SI units. This example is designed to automatically create localized grid spaces around the defined equipment rows to ensure an efficient solution.
<i>Data_Center_US_Units.xls</i>	Creates a ready-to-solve model of a data center. Room dimensions, Rack, Cooler, and Perforated Tiles locations, and performance data are parametrically described in several tabs using US units. This example is designed to automatically create localized grid spaces around the defined equipment rows to ensure an efficient solution.
<i>Heatpipe-LShaped.xls</i>	Creates individual Geometry FloXML files for a set of L-shaped heat pipes, specified by their dimensions and thermal properties.

**Table 5-5. Supplied XML Spreadsheet Examples (cont.)**

File	Description
<i>IGBT-Creator.xlsxm</i>	Creates a typical IGBT model. Inputs for the IGBT stack geometry, materials, power, and so on are used to create a transient model for use with the Detailed Model Calibration feature in the Command Center.
<i>Materials.xlsxm</i>	Creates a Geometry FloXML file for a set of materials, specified by their conductivity (which can be orthotropic), density, and specific heat.
<i>Solve-Folded_Fin_Heat_Sink.xlsxm</i>	Creates, and batch solves, a series of parametrically defined (one simulation per column) models containing a folded fin heat sink in a numerical wind tunnel. The temperature results are extracted from outputted monitor point CSV files and placed in the spreadsheet results so as to populate the Thermal Resistance vs Flow Rate chart.
<i>Subsv93-Update.xlam</i>	A spreadsheet addin containing subroutines that correctly form FloXML file sections for all of the supported objects, attributes, and model settings. This can be installed in another spreadsheet to make the subroutines available by default in the spreadsheet VBA programming environment.
<i>Windtunnel-AdvancedResistance-FT.xlsxm</i>	References a supplied <i>pcb1.pdml</i> file that is automatically characterized to determine advanced resistance flow properties with the ability to output a compact model of the pdml assembly.
<i>XML_Subs_FLOCOREv93_Update.bas</i>	Contains content identical to that in <i>Subsv93-Update.xlam</i> . This file is a module that can be imported into a single spreadsheet VBA project, as opposed to having the subroutines available by default.

# Pattern Creation Dialog Box

To access: **Edit > Pattern**

Use this dialog box to create a two dimensional matrix array pattern of a single geometry object. The pattern is directed along any two coordinate axis in either the positive or negative directions.

---

## Note

---



This dialog box is deactivated if more than one object is selected in the display area.

---

## Objects

Field	Description
First Direction	The pattern for the first direction of a bidirectional pattern and the only pattern for a unidirectional pattern.
Second Direction	The pattern for the second direction of a bidirectional pattern.
Number	The total number of objects created in the direction selected from a popup list containing: +X, -X, +Y, -Y, +Z, -Z. By default, no copies are made.
Pitch	The spacing between the objects. It is the distance between the start edges of each object in the given direction.

## Related Topics

[Creating a Pattern of Objects](#)



# Chapter 6

## Viewing and Manipulating Geometry

---

Geometry is displayed and manipulated in the drawing board.

<b>The Drawing Board .....</b>	<b>254</b>
Interaction of the Drawing Board With the Data Tree .....	254
Drawing Board Views .....	255
Drawing Board Color Conventions .....	255
The Active Pane .....	256
Snap Modes .....	256
<b>Viewing Geometry .....</b>	<b>258</b>
Changing Geometry Rendering .....	258
Changing Mouse Mode in Create Mode .....	259
Tabbing Between Views .....	259
Controls for Viewing the Spatial Solution Grid .....	260
Changing the Orientation of a Drawing Board Pane .....	262
Aligning the View .....	262
Rotating the View Incrementally .....	263
Changing the View Using the Mouse .....	263
Resizing the View .....	264
Hiding Geometry .....	264
Changing Geometry Transparency .....	265
Changing the GDA Background .....	266
<b>Manipulating Geometry .....</b>	<b>268</b>
Selecting Objects in the Drawing Board .....	268
Moving Objects Using the Mouse .....	269
Aligning an Object Using the Keyboard .....	271
Aligning One or More Objects to Another Object .....	271
Moving Objects by Specified Coordinate Distances .....	271
Resizing Objects Using the Mouse .....	272
Copying Objects in the Drawing Board .....	274
Measuring the Distance Between Geometry Vertices or Edges .....	275
Rotating Objects .....	276
<b>Drawing Board Dialog Boxes .....</b>	<b>278</b>
Align Dialog Box .....	279
Measure Dialog Box .....	280
Move Dialog Box .....	281
Rotate View Dialog Box .....	282
Drawing Board Icons .....	283

# The Drawing Board

The Drawing Board (the GDA in Create mode) provides 2D and 3D views of the model which allow you to check the location of the geometry and the density of the solution grid.

The model image can be rendered as wireframe (the default), solid, or solid with edges.

Interaction of the Drawing Board With the Data Tree .....	254
Drawing Board Views .....	255
Drawing Board Color Conventions .....	255
The Active Pane .....	256
Snap Modes .....	256

## Interaction of the Drawing Board With the Data Tree

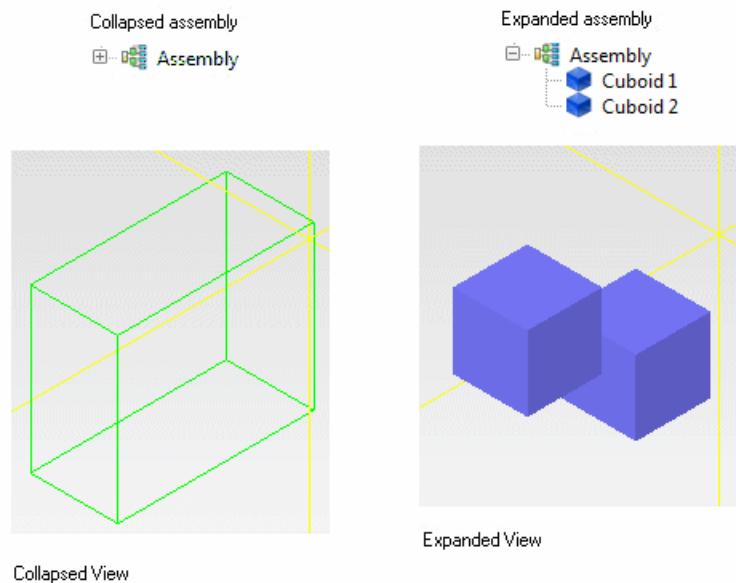
The drawing board is fully interactive with the data tree; selecting an object in the data tree also selects it in the drawing board, and vice versa.

Selecting objects in the data tree is usually easier, depending on the complexity of the model.

When you select of an object in the drawing board, the data tree will scroll to show the selected object in the data tree pane. When more than one object is selected in the drawing board, the first will be placed at the top of the view in the data tree, in such cases you may need to scroll down the data tree to see the other selected objects.

When an assembly is collapsed in the data tree (indicated by '+'), then only the outline of the assembly is shown (as a green wireframe box) in the drawing board. For example, [Figure 6-1](#) shows that it is only when the assembly is expanded that the contents become visible.

**Figure 6-1. Collapsed and Expanded Assemblies in the Data Tree and Drawing Board**



## Drawing Board Views

The drawing board has four view configurations. The view configuration is independent between Create and Analyze modes.

- Examiner View — for examining details, the single pane fills the drawing board.
- Grid View — four panes. By default, these show X, Y and Z plane third-angle projections and an isometric view.
- Split Horizontally — two panes arranged horizontally.
- Split Vertically — two panes arranged vertically.

All views show the grid if the grid visibility switch (G) is on.

### Related Topics

[Tabbing Between Views](#)

[Changing the Orientation of a Drawing Board Pane](#)

[Drawing Board Icons](#)

## Drawing Board Color Conventions

Object types are color-coded.

The color conventions used are listed in [Table 6-1](#).

**Table 6-1. Drawing Board Color Conventions**

Color	Object
Green	Assemblies
Yellow	Resistances, perforated plates, open cutouts and open overall boundary
Brown	Sources
Cyan	SmartParts (except perforated plates)
Dark Blue	Solid cuboids and prisms
Dark Gray	Symmetric cutouts and symmetric overall boundary
Peach	Initial subdomains
Purple	Regions

## The Active Pane

Only one of the drawing board panes is active at any one time.

To make a pane the active pane, click anywhere in the pane. The active pane has a red border.

Alternatively, the Tab key activates the different pictures in sequence, stepping from one picture to the next. The latter method will not affect which items are currently selected.

All model construction operations are applied to the active pane, and any visible changes made are then copied to all other panes.

## Snap Modes

Snap objects to the snap grid, or snap objects to other objects.

---

**Tip**

---

 You can use Alt+arrow keys to move objects incrementally.

---

### Snap to Grid

As you move objects, they are drawn towards the snap grid.

As you resize objects, the point you are dragging is drawn towards the snap grid.

Snapping towards the grid location works whether the grid is displayed or not.

## Snap to Object

As you move objects, they will snap to the edge of other objects when they are moved within a certain tolerance. If an object is not within the tolerance, the moved object will be placed as though the snap is off.

## Related Topics

[Snap Grid](#)

## Viewing Geometry

Changing the views of the GDA.

<b>Changing Geometry Rendering .....</b>	<b>258</b>
<b>Changing Mouse Mode in Create Mode .....</b>	<b>259</b>
<b>Tabbing Between Views .....</b>	<b>259</b>
<b>Controls for Viewing the Spatial Solution Grid .....</b>	<b>260</b>
<b>Changing the Orientation of a Drawing Board Pane.....</b>	<b>262</b>
<b>Aligning the View.....</b>	<b>262</b>
<b>Rotating the View Incrementally .....</b>	<b>263</b>
<b>Changing the View Using the Mouse .....</b>	<b>263</b>
<b>Resizing the View.....</b>	<b>264</b>
<b>Hiding Geometry .....</b>	<b>264</b>
<b>Changing Geometry Transparency .....</b>	<b>265</b>
<b>Changing the GDA Background .....</b>	<b>266</b>

## Changing Geometry Rendering

Changing the visual representation of geometry in the GDA.

### Procedure

1. Select the geometry.

If the geometry is a parent object with child objects, make all child objects visible by expanding the parent object.

If no geometry is selected, then all of the geometry is rendered in the same way.

To deselect selected geometry, click in the background of the viewer.

2. You have a choice:

If you want to...	Do the following:
Change geometry appearance to wireframe.	<ol style="list-style-type: none"><li>1. Press W. Alternatively, click the <b>Wireframe</b> icon, or choose <b>View &gt; Wireframe</b>.</li></ol>
Change geometry appearance to solid.	<ol style="list-style-type: none"><li>1. Click the <b>Solid</b> icon. Alternatively, choose <b>View &gt; Solid</b>.</li></ol>

If you want to...	Do the following:
Change geometry appearance to solid with edges.	1. Press S. Alternatively, click the <b>Solid With Edges</b> icon, or choose <b>View &gt; Solid With Edges</b> .

## Results

Geometry is superimposed by surface temperature plots.

## Related Topics

[Hiding Geometry](#)

[Changing Geometry Transparency](#)

# Changing Mouse Mode in Create Mode

There are two possible mouse modes when creating a project: Select and Manipulate.

## Procedure

1. To select objects in the GDA, choose **Viewer > Select Mode** or click the **Select Mode** icon 
2. To change the GDA view, choose **Viewer > Manipulate Mode** or click the **Manipulate Mode** icon 
3. To toggle between Manipulate and Select modes, press **F9**.

In both modes, the mouse wheel can be used to zoom in and out of views. Zooming is relative to the mouse pointer position. To return to the default size, press **R**.

## Related Topics

[Drawing Board Icons](#)

[Selecting Objects in the Drawing Board](#)

[Moving Objects Using the Mouse](#)

[Changing Mouse Mode in Analyze Mode](#)

# Tabbing Between Views

The TAB key provides a quick way of changing the active view.

## Procedure

1. Click a pane in the GDA.

2. Press the TAB key repeatedly.

## Results

The behavior depends on the view:

- In Examiner view (single pane), the view cycles around Positive X, Positive Y, Positive Z, and Isometric with each key press.
- In Grid view, each of the four panes becomes the active pane in turn.
- In Split Horizontally view or Split Vertically view, each of the two panes becomes the active pane in turn.

## Related Topics

[Drawing Board Views](#)

# Controls for Viewing the Spatial Solution Grid

The spatial solution grid is viewed on the Workplane.

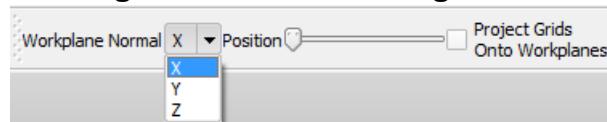
The grid is shown in black when the background is light, and is shown in white when the background is dark.

The following are the controls available when viewing the grid:

- The Direction normal to the Workplane.
- The Position of the Workplane along this direction.  
The position can also be moved, one cell at a time, by pressing the arrow keys.
- Whether or not the grid is Projected onto the Workplane.

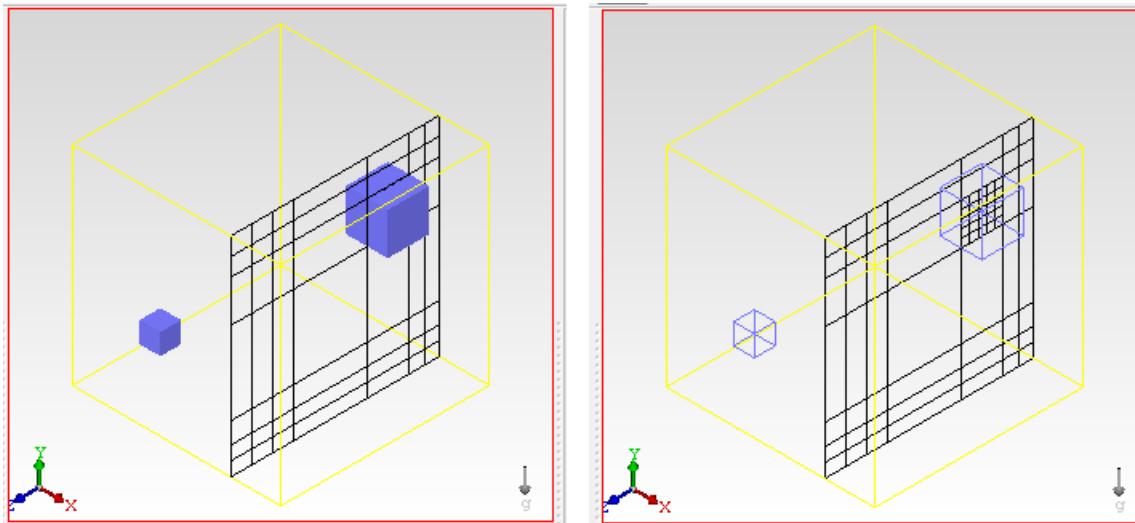
These controls are selectable from the toolbar.

**Figure 6-2. Grid Viewing Icons**



When viewing the grid in Solid view you will not be able to see local grids. To see local grids you can either change to Wireframe view, see [Figure 6-3](#), or use a Projected view (for example, see [Figure 6-4](#)).

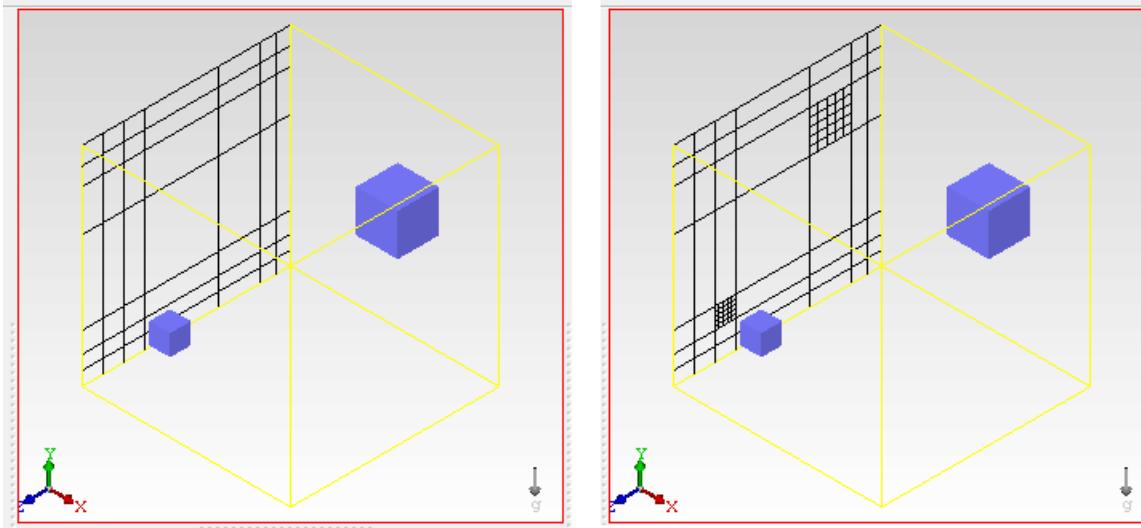
**Figure 6-3. Solid and Wireframe Views and Grid Visibility**



## Grid Projection

The left view in [Figure 6-4](#) shows the unprojected grid at the origin of the X plane, this is the grid at that position. The right view in [Figure 6-4](#) shows the projected view. Note how the detailed (and localized) grid at the cuboids is projected onto the Workplane. In this example it does not matter that the view is Solid because the grid is not cutting any objects.

**Figure 6-4. Unprojected and Projected Grid Views**



## Related Topics

[Workplane](#)

## Changing the Orientation of a Drawing Board Pane

Each drawing board pane can be changed to show a different view of the model.

### Procedure

Select the pane to be the active pane.

The active pane border is red.

- To change the projection of a Drawing Board View:
  - To view first-angle projection, press F.
  - To view third-angle projection, press T.
- To change the view direction:
  - To view in the positive X, Y or Z directions, press X, Y or Z respectively.
  - To view in the negative X, Y or Z directions, press Shift+X, Y or Z respectively.
  - To display an isometric view, press Shift+I.

### Results

The red/yellow/blue colored axes indicator at the bottom left corner of the pane shows the orientation of the view.

### Related Topics

[Drawing Board Views](#)

[Changing the View Using the Mouse](#)

## Aligning the View

The view can be aligned along one of the major axis or with the gravity vector.

### Procedure

You have a choice:

If you want to...	Do the following:
Align the view along a major axis.	Choose one of the <b>Viewer &gt; View</b> menu options or press one of the view shortcut keys.  The view is set to the chosen direction and is refitted. For example, choosing <b>Viewer &gt; Positive Z View</b> or pressing the Z key displays the scene viewed from the positive z-axis direction.

If you want to...	Do the following:
Align the view so that gravity is vertically downwards.	Choose <b>Viewer &gt; Align With Gravity</b> .

## Rotating the View Incrementally

You can rotate a GDA view with respect to the model or the viewer.

### Procedure

1. Select a view in the GDA.
2. Choose **Viewer > Rotate View** to open the Rotate View dialog box.
3. Select the Rotate Mode (Model or View), depending around which set of axis you want the rotation to be made.
4. Select the Angle increment (in degrees or radians).
5. Use a rotate buttons to incrementally rotate the view.
6. If necessary, continue using the dialog box controls to obtain the view required, then, when finished, click **Close** to close the dialog box.

### Related Topics

[Rotate View Dialog Box](#)

## Changing the View Using the Mouse

You can use the mouse to pan, zoom, and rotate the GDA view.

### Prerequisites

- The mouse mode is Manipulate mode. If you are using a two-button mouse, simulate the middle mouse button by pressing the left and right buttons simultaneously.

### Procedure

Proceed as follows:

If you want to...	Do the following:
Pan the view.	Shift+Left-click and drag the mouse.

If you want to...	Do the following:
Zoom in and out of the view.	You have a choice: <ul style="list-style-type: none"><li>• Rotate the mouse wheel. The image zooms in or out relative to the position of the mouse pointer.</li><li>• Ctrl+Shift+Left-click and then:<ul style="list-style-type: none"><li>• Drag upwards to zoom out.</li><li>• Drag downwards to zoom in.</li></ul></li><li>• Right-click and drag over an area (creating a rubber band), then release. The rubber-banded area expands to fill the display area.</li></ul>
Rotate the view.	Left-click and drag the mouse.

## Related Topics

[Changing Mouse Mode in Create Mode](#)

[Changing the Orientation of a Drawing Board Pane](#)

## Resizing the View

You can resize the view to the domain or to a selected object.

### Procedure

1. To resize the display to the domain, press R, choose **Viewer > Refit** or click the Refit icon 

---

#### Note

---

 A domain refit is done automatically if you swap between Orthographic  and Perspective  views.

---

2. To resize the display to a selected object, select the object, then press V or choose **Viewer > View Selected**.

## Hiding Geometry

Decluttering the drawing board by hiding selected objects.

## Restrictions and Limitations

- Hidden geometry remains active in the project solution.
- You cannot hide the Root Assembly.

## Procedure

1. Select the object(s) to be hidden.

More than one object can be hidden at a time.

2. Press F12, or click the **Hide** icon.

Alternatively, for individual objects, you can check the Hide check box in the **Location** tab of the object property sheet.

## Results

The selected object is hidden from views in the drawing board when in either Create or Analyze mode. In the data tree, the object is dimmed, but remains selectable.

- To reveal hidden objects, select the objects and press SHIFT+F12, or click the **Reveal** icon. Alternatively, you can uncheck the Hide check box in the **Location** tab of the object property sheet.

## Related Topics

[Topping Assemblies or Parts](#)

[Changing Geometry Transparency](#)

# Changing Geometry Transparency

All geometry is opaque by default. You can change the transparency of objects, groups of objects or assemblies. This is an alternative to hiding geometry completely.

## Restrictions and Limitations

- Transparency setting are not retained over sessions.

## Procedure

1. Select the geometry to be made transparent.

To change the transparency of all geometry, select the Root Assembly.

2. In the **Location** tab of the property sheet, move the Geometry Transparency slider bar to the right to increase the transparency of the selected object(s).

The transparency does not change until you release the mouse.

Child objects inherit the same level of transparency as their parent. Therefore, to reset all geometry transparency settings to opaque, select the Root Assembly, move the slider bar

to the far right and mouse-release, then move the slider bar to the far left and mouse-release.

## Related Topics

[Hiding Geometry](#)

# Changing the GDA Background

The GDA can have a uniform or a top-to-bottom gradient background.

## Procedure

1. Choose **Edit > User Preferences** to open the User Preferences dialog box.
2. Open the **Drawing Board** tab.
3. You have a choice:

If you want to...	Do the following:
Have a uniform colored background.	<ol style="list-style-type: none"><li>1. Leave the Gradient Background check box unchecked.</li><li>2. Click the Background Color palette button to open the Modify Color dialog box.</li><li>3. Use the Modify Color dialog box to pick an existing basic or custom color, or to define a new color</li><li>4. Click <b>OK</b> to close the Modify Color dialog box.</li></ol> <p>The color of the Background Color palette button changes to the newly selected color.</p>
Have a gradient background.	<ol style="list-style-type: none"><li>1. Check the Gradient Background check box.</li><li>2. Click the Top Background Color palette button to open the Modify Color dialog box.</li><li>3. Use the Modify Color dialog box to pick an existing basic or custom color, or to define a new color</li><li>4. Click <b>OK</b> to close the Modify Color dialog box.</li></ol> <p>The color of the Top Background Color palette button changes to the newly selected color.</p> <ol style="list-style-type: none"><li>5. Repeat Steps 2 to 4 to change the Bottom Background Color palette button.</li></ol>

4. Click **OK** to save your changes and close the User preferences dialog box.

# Manipulating Geometry

---

Moving geometry in the GDA.

<b>Selecting Objects in the Drawing Board .....</b>	<b>268</b>
<b>Moving Objects Using the Mouse.....</b>	<b>269</b>
<b>Aligning an Object Using the Keyboard .....</b>	<b>271</b>
<b>Aligning One or More Objects to Another Object .....</b>	<b>271</b>
<b>Moving Objects by Specified Coordinate Distances.....</b>	<b>271</b>
<b>Resizing Objects Using the Mouse .....</b>	<b>272</b>
<b>Copying Objects in the Drawing Board.....</b>	<b>274</b>
<b>Measuring the Distance Between Geometry Vertices or Edges.....</b>	<b>275</b>
<b>Rotating Objects .....</b>	<b>276</b>

## Selecting Objects in the Drawing Board

Selecting objects in the drawing board can be more convenient than selecting objects in the data tree, particularly if they are physically close to each other.

### Restrictions and Limitations

- ‘Side-On’ unselected 2D objects are not shown in Solid view, however, they are shown in Wireframe view.
- Rubber Band resize on multiple objects is restricted, if you select more than 100 objects by rubber-banding then grab handles will not be present.

### Prerequisites

- The mouse must be in Select Mode.

### Procedure

- You have a choice.

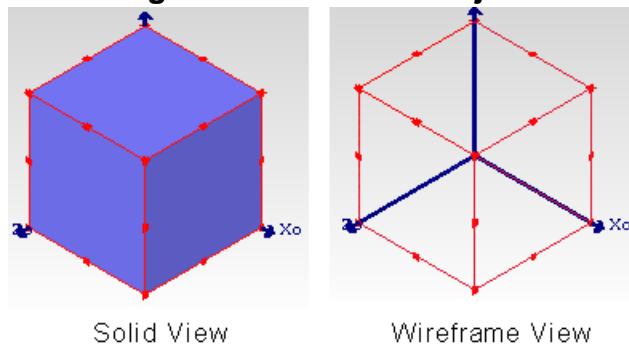
If you want to...	Do the following:
Select a single object.	<ul style="list-style-type: none"><li>Left-click the object.</li></ul>
Select more than one object.	<ul style="list-style-type: none"><li>Either:<ul style="list-style-type: none"><li>Ctrl+left-click each object, or</li><li>Rubber-band the objects.</li></ul></li></ul>

- If you are unable to select the required object, either change the view or use the data tree.

## Results

Selected objects have red boundary lines and grab handles.

**Figure 6-5. Selected Object**



To unselect all selected objects, press ESC or click in an unoccupied area in the drawing board.

## Related Topics

[Changing Mouse Mode in Create Mode](#)

[Selecting Objects in the Data Tree](#)

# Moving Objects Using the Mouse

Moving one or more selected objects by mouse drag in the drawing board.

## Procedure

1. In Select mode, select the object or set of objects you want to move.

To indicate that you can move the selected object(s), the mouse icon will change to crossed arrows as shown in [Figure 6-6](#). When this icon change occurs depends on the rendering:

- For Solids, you can have the cursor anywhere within the object boundaries except near a grab handle.
- For Wireframes, you must be on a line, but not near a grab handle.

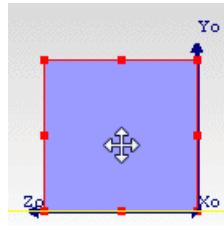
---

**Tip**

 For small objects, select with the middle mouse to force a move.

---

**Figure 6-6. Mouse Icon When Moving an Object**

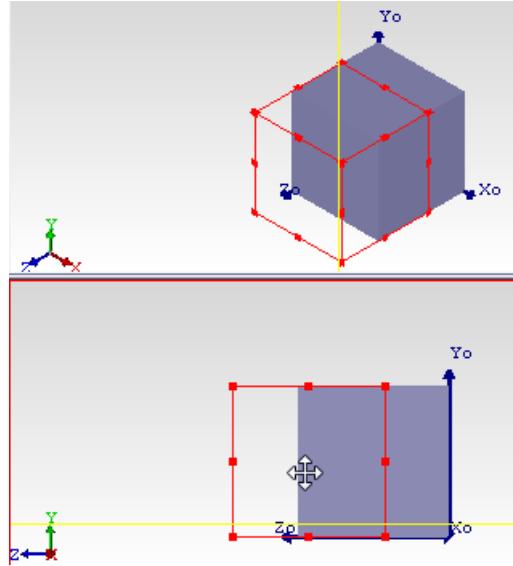


2. By default, the **Snap Toggle** setting is **Snap to Object is On**. To snap to the grid, or to switch off the snap functionality, click the **Snap Toggle** icon.
3. You have a choice.

If you want to...	Do the following:
Move the object(s) unconstrained.	<ul style="list-style-type: none"><li>• Drag the mouse.</li></ul>
Move the object(s) constrained to one of the major axes.	<ul style="list-style-type: none"><li>• Press and hold the Shift key, then drag the mouse.</li></ul>

Figure 6-6 shows an example of when an object is moving, in this case it has been constrained to the Z direction.

**Figure 6-7. Moving a Cuboid**



If the Snap Toggle setting is **Snap to Object is On** or **Snap Grid is On**, then the object may ‘jump’ to the object or grid line.

4. Release the mouse to complete the move.

The object(s) can also be moved to line up with other objects using the [Align Dialog Box](#). This allows you to align an edge or center of selected geometry with an edge or center of another piece of geometry.

## Related Topics

- [Moving Objects by Specified Coordinate Distances](#)
- [Changing Mouse Mode in Create Mode](#)
- [Snap Modes](#)

# Aligning an Object Using the Keyboard

Moving an object using the keyboard is particularly useful when you want to align an object with another object or with the grid.

## Procedure

1. In Select mode, click on the object in the drawing board.
2. Select Snap to Object or Snap to Grid.
3. Use the Alt+ arrow keys to move the object.

The object will move in steps in the direction of the arrow, unless it meets another object or a grid line, depending on the selected Snap to function.

# Aligning One or More Objects to Another Object

Moving an object using the keyboard is particularly useful when you want to align an object with another object or with the grid.

## Procedure

1. Select the plane in which the objects are to be aligned by selecting the appropriate drawing board view.
2. Select the object to which you want other object(s) to align.
3. Select the other object(s).
4. Click the **Align** icon to open the Align dialog box and click the appropriate button.  
The objects will become aligned.
5. Click **Close**.

## Related Topics

- [Align Dialog Box](#)

# Moving Objects by Specified Coordinate Distances

Moving one or more selected objects by distances entered as data.

## Restrictions and Limitations

- The distances that can be defined are in the absolute (not local) coordinate directions.

## Procedure

- Select one or more object to be moved.

- Click the **Move** icon.

The Move dialog box is opened.

- Enter distance(s) in absolute coordinate directions and click **Apply**.

The selected object moves. You can continue using the dialog box to move the selected object again, if required.

- Click **Close** to close the dialog box.

## Related Topics

[Move Dialog Box](#)

[Measuring the Distance Between Geometry Vertices or Edges](#)

# Resizing Objects Using the Mouse

Resizing one or more selected objects by mouse drag in the drawing board.

## Restrictions and Limitations

- Re-circulation devices and compact components cannot be resized by dragging.  
Re-circulation devices cannot be resized in this way because the supplies and extracts are fixed locations. Similarly, compact components have thermal resistances which are specific to a particular physical package geometry, and so would have no relevance if the SmartPart was resized in this way.
- The allowed resize directions are shown in [Table 6-2](#).

**Table 6-2. Object Axes Directions**

Geometry	Resize Direction
Cuboids, Prisms, Resistances, Sources, Heat Sinks, PCB Components, Sloping Blocks, Enclosures, Cylinders, Blocks with Holes, Holes, Cutouts, Volume Regions.	X, Y and Z
PCBs, Network Cuboids, Rectangular Fans, Fixed Flows, Perforated Plates.	X and Y
Axial Fans.	Z only
Plane Regions	None

- Collapsed objects can be resized in the directions other than the collapsed direction.

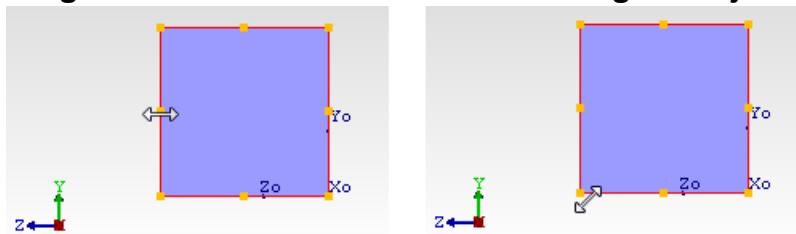
## Procedure

- In Select mode, select the object or set of objects you want to move.

If selected objects are of the same type, then X, Y, Z Size dimensions that are common are shown in the **Location** tab of the property sheet. These dimensions are updated after resizing.

To indicate where you can resize the selected object(s), the mouse icon will change to a double-header arrow. The mouse icon change occurs when you are near to a grab handle.

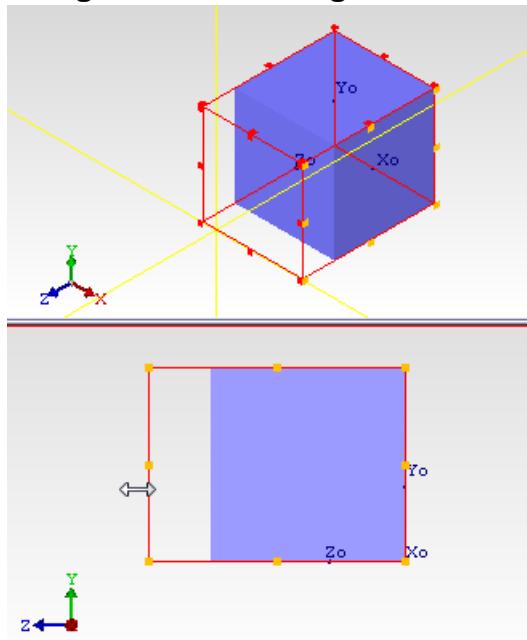
**Figure 6-8. Mouse Icons When Resizing an Object**



- By default, the **Snap Toggle** setting is **Snap to Object is On**. If you want to snap to the grid or for the snap functionality to be switched off then click the **Snap Toggle** icon.
- Drag the mouse to resize the object(s). In some cases, for example, corners, you can constrain the resize direction by pressing and holding the Shift key before dragging the mouse.

[Figure 6-6](#) shows an example of when an object is resizing.

**Figure 6-9. Resizing a Cuboid**



If the **Snap Toggle** setting is **Snap to Object is On** or **Snap Grid is On** is switched on, then the moving face of the object will ‘jump’ to the object or grid line when it gets close.

4. To complete the resize, release the mouse.
5. Also, when dragging a SmartPart boundary, if the requested resize will result in the SmartPart becoming inconsistent, then the resize will be rejected. For example, making an enclosure too small so that its walls overlap. Setting thinner walls will allow smaller enclosures.
6. Sometimes, when the resizing is rejected, the SmartPart moves. This occurs when dragging from the object's low co-ordinate direction because you are changing the location of its origin.

## Related Topics

[Selecting Objects in the Drawing Board](#)

[Changing Mouse Mode in Create Mode](#)

[Snap Modes](#)

## Copying Objects in the Drawing Board

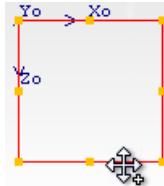
You can copy geometry in the drawing board using copy and drag.

### Procedure

1. Make sure that the mouse is in Select mode.

2. Select the object to be copied.
3. Press and hold the Ctrl key then hover over the object until the copy cursor is shown, see [Figure 6-10](#).

**Figure 6-10. Copy Cursor**



4. Select the object to be copied by left clicking, and drag the object to its new location.
5. Release the mouse button *before* releasing the Ctrl key.

If the Ctrl key is released first, then the object will move without being copied. If this happens, then the move can be undone (Ctrl+Z).

## Results

- Single items are copied into the parent assembly.
- Two or more selected objects in the same assembly are also copied into the parent assembly.
- Two or more selected objects from different assemblies are copied into the current assembly on display, that is the current ‘topped’ assembly or the root assembly.

## Measuring the Distance Between Geometry Vertices or Edges

The distance between two vertices or edges can be measured and then displayed as an annotation in the drawing board.

### Restrictions and Limitations

- You can only display the measurement between:
  - Two vertices.
  - One vertex and one edge. The measurement is between the vertex and the mid-point of the edge.
  - Two edges. The measurement is between the mid-points of the edges.

### Procedure

1. If measuring the distance between vertices/edges on a single object then that object must be selected. If measuring the distance between two objects then both objects must be selected.

2. Click the **Measure** icon.

The Measure dialog box is opened. If there is an existing measurement, click **Clear** to empty the fields.

When you mouse-hover over objects in the drawing board, a light-blue cross is displayed when near to a vertice, and edges turn light-blue.

3. Select the first vertice/edge by clicking the mouse when the light-blue cross/edge is visible.

The cross/edge changes color to black and stays fixed on the object. You can deselect by clicking the vertice/edge again.

4. Select the second vertice/edge.

The absolute distance and the coordinate distances between the two selections are calculated and displayed in the Measure dialog box.

A line is drawn connecting the two selections, and the absolute distance is shown as an annotation.

5. Click **Close** to close the Measure dialog box.

## Results

After closing the Measure dialog box, the line and annotation remain visible in the drawing board.

If you open the Measure dialog box, the absolute and coordinate distances between the two points are shown in the dialog box fields.

The distances remain in the dialog box fields and the drawing board line and annotations remain visible until you ‘Clear’ the dialog box.

## Related Topics

[Measure Dialog Box](#)

[Moving Objects by Specified Coordinate Distances](#)

## Rotating Objects

An object or group of objects can be rotated about their axes.

### Restrictions and Limitations

- Rotation is restricted to increments of 90 degrees about the center of the selected object(s).
- The axis about which the object is rotated is in the direction of the active view. If you are viewing in negative axis directions, a clockwise rotation action will result in objects appearing to rotate counterclockwise.

## Procedure

1. Select the object(s) to be rotated.
2. Select an appropriate view, see Restrictions and Limitations above.
3. Click the **Rotate Clockwise** or **Rotate Counterclockwise** icon. Repeat as necessary.

## Drawing Board Dialog Boxes

---

The following dialog boxes are used when manipulating geometry.

<b>Align Dialog Box .....</b>	<b>279</b>
<b>Measure Dialog Box.....</b>	<b>280</b>
<b>Move Dialog Box .....</b>	<b>281</b>
<b>Rotate View Dialog Box.....</b>	<b>282</b>
<b>Drawing Board Icons.....</b>	<b>283</b>

## Align Dialog Box

To access: **Geometry > Align** or click the **Align** icon.

Use this dialog box to line up objects in the drawing board with the first selected object.

### Objects

Button	Icon	Description
Horizontal Alignment		
Center		Aligns about a horizontal line that bisects the first selected object.
Top		Aligns to the horizontal line at the top edge of the first selected object.
Bottom		Aligns to the horizontal line at the bottom edge of the first selected object.
Vertical Alignment		
Center		Aligns about a vertical line that bisects the first selected object.
Right		Aligns to the vertical line at the right edge of the first selected object.
Left		Aligns to the vertical line at the left edge of the first selected object.
Horizontal and Vertical Alignment		
Centers		Aligns about horizontal and vertical lines that bisect the first selected object.

### Usage Notes

The alignment is in the plane of the currently selected drawing board view.

### Related Topics

[Aligning One or More Objects to Another Object](#)

## Measure Dialog Box

To access: **Geometry > Measure** or click the **Measure** icon.

Use this dialog box to display the distance between two selected vertices or edges in the drawing board. When an edge is selected, the measured distance is to the mid-point of the edge.

### Objects

Field	Description
Absolute Distance	Read-only. The measured distance between the two selected points.
Distance X, Y, Z	Read-only. The measured distance between the two points along the X, Y, and Z axes respectively.
Clear	Clears the measurement values from the dialog box and the absolute distance line and annotation from the drawing board.
Close	Closes the dialog box.

### Usage Notes

If no objects are selected when the dialog box is opened, a message requests you to select one or more objects.

### Related Topics

[Measuring the Distance Between Geometry Vertices or Edges](#)

## Move Dialog Box

To access: **Geometry > Move** or click the **Move** icon.

Use this dialog box to move selected geometry by user-defined distances.

### Objects

Field	Description
Move by X, Y, Z	Distances in <i>absolute</i> coordinate directions.

### Usage Notes

- When you **Apply** the move, the dialog box remains open.

### Related Topics

[Moving Objects by Specified Coordinate Distances](#)

## Rotate View Dialog Box

To access: **Viewer > Rotate View**

Use this dialog box to rotate the GDA view. You have two possible frames of reference for rotation: the model itself or the viewer.

### Objects

Field	Description
Angle	The increment angle rotated for each button press. Default 90 degrees.
Rotate Mode	The frame of reference of rotation: the model or the viewer. <ul style="list-style-type: none"><li>• Model — Rotate about the major axes of the geometry: X, Y and Z.</li><li>• View — Rotate about the major axes of the viewer. The axes move as they do with an airplane or ship, and are measured in terms of pitch, yaw, and roll.</li></ul>
Model Rotate Mode Buttons:	
 + X Rotate	Rotate about the x-axis clockwise as viewed down the axis towards the origin.
 - X Rotate	Rotate about the x-axis anti-clockwise as viewed down the axis towards the origin.
 + Y Rotate	Rotate about the y-axis clockwise as viewed down the axis towards the origin.
 - Y Rotate	Rotate about the y-axis anti-clockwise as viewed down the axis towards the origin.
 + Z Rotate	Rotate about the z-axis clockwise as viewed down the axis towards the origin.
 - Z Rotate	Rotate about the z-axis anti-clockwise as viewed down the axis towards the origin.
View Rotate Mode Buttons:	
 + Pitch Angle	Pitch downwards.

<b>Field</b>	<b>Description</b>
 - Pitch Angle	Pitch upwards.
 + Yaw Angle	Yaw left-to-right.
 - Yaw Angle	Yaw right-to-left.
 + Roll Angle	Roll clockwise.
 - Roll Angle	Roll anti-clockwise.

## Related Topics

[Rotating the View Incrementally](#)

## Drawing Board Icons

The drawing board icons are normally located between the Project Manager and Workplanes toolbars.

## Related Topics

[Drawing Board Views](#)



# Chapter 7

## Libraries

---

Simcenter Flotherm libraries store commonly used geometry which you can copy to the model.

<b>Library Overview</b> .....	<b>285</b>
<b>Library Folder Operations</b> .....	<b>287</b>
Adding a New Library .....	287
Adding Items to a Library From the Current Project .....	287
Importing Library Files .....	288
Copying Libraries .....	288
Refreshing a Library Folder .....	289
Deleting a Library Folder .....	289
Exporting a Library Folder .....	289
<b>Library Item Operations</b> .....	<b>290</b>
Copying Geometry From Libraries to the Data Tree .....	290
Copying an Attribute From a Library Folder to the Data Tree .....	291
Moving and Copying Library Items Between Library Folders .....	291
Saving Project Geometry to a Library Folder .....	292
<b>Starter Libraries</b> .....	<b>292</b>
<b>Generic Data Center Library</b> .....	<b>295</b>
Generic CRAC Units .....	295
Generic Cabinets .....	296
Equipment for High Detail Cabinet .....	297
Generic Floor Grille .....	297
Generic Floor Support .....	297
<b>Gilberts Data Center Library</b> .....	<b>297</b>
<b>Library Property Sheets and Dialog Boxes</b> .....	<b>299</b>
Library Data Property Sheet .....	300
Create Library Dialog Box .....	301

## Library Overview

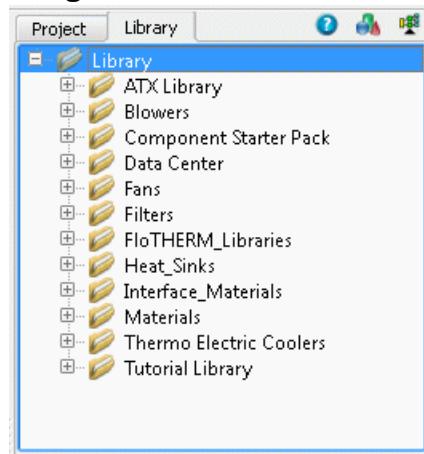
Mentor Graphics supplies standard start-up libraries. You can also build libraries to suit your own requirements.

Libraries enable easy transfer of geometry from one project to another.

The Simcenter Flotherm libraries share the right-hand pane with the project attributes, each having a separate tab, see [Figure 7-1](#). If this pane is not displayed, either click the **Show Project**

**Attributes/Library** icon or press F7. The libraries and their contents are organized in a tree structure, and, provided they are valid operations, library items can be dragged from there onto the data tree. Although you can copy and move libraries and their contents to any positions in the tree, the tree is rearranged into alphabetical order when the library is “Refreshed”, or when a new session is started.

**Figure 7-1. The Libraries**



## Attribute/Property Duplication

Each attribute or property is given its own unique identifier, so that when you import the same attribute or property more than once, it is duplicated in the library. For example, if you import two materials both with the same surface attributes attached, the surface attribute will be duplicated in the library.

# Library Folder Operations

The following operations act on library folders.

<b>Adding a New Library</b> .....	<b>287</b>
<b>Adding Items to a Library From the Current Project</b> .....	<b>287</b>
<b>Importing Library Files</b> .....	<b>288</b>
<b>Copying Libraries</b> .....	<b>288</b>
<b>Refreshing a Library Folder</b> .....	<b>289</b>
<b>Deleting a Library Folder</b> .....	<b>289</b>
<b>Exporting a Library Folder</b> .....	<b>289</b>

## Adding a New Library

New libraries are added as subfolders to existing folders.

### Restrictions and Limitations

- Microsoft® Windows has a file pathname limitation of 259 characters. To conform with this, you may be prevented or limited from creating libraries close to this limit.

### Prerequisites

- The parent folder must not be Read Only.

### Procedure

1. Right-click a library folder and select **New Library**.

The [Create Library Dialog Box](#), is opened.

2. Enter the library name for the new library and click **OK**.

A subfolder with the entered name is created under the selected folder.

## Adding Items to a Library From the Current Project

Use this procedure to make project items available to other projects.

### Procedure

Drag and drop the item from the data tree onto the library folder icon.

### Results

When items are dragged across the data tree, the mouse pointer changes to indicate a valid or invalid drop-site.

## Related Topics

[Adding an Attribute From a Library to the Project \[Simcenter Flotherm Project Attributes Reference Guide\]](#)

# Importing Library Files

Import a library file previously exported from another Simcenter Flotherm session or from Mentor Graphics T3Ster-Master software.

## Prerequisites

- The destination library folder must not be Read Only.
- Imported file types include FloXML (\*.xml), Library (\*.library), PDML (\*.pdml) or T3Ster (\*.xCTM).

## Procedure

1. Right-click the destination library folder, select **Import from File** then choose a file type.
2. Use the file selection dialog box to select the relevant file, then click **Open**.

A new subfolder is created in the Library.

## Related Topics

[Exporting a Library Folder](#)

# Copying Libraries

Library folders can be copied to create new libraries.

## Prerequisites

- The destination library folder must not be Read Only.

## Procedure

1. Right-click a library folder and select **Copy**.
2. Right-click the destination library folder and select **Paste**.

The copied folder is added to the Library tree.

---

### Tip

 The standard Ctrl+C, Ctrl+X and Ctrl+V shortcut keys can also be used to copy libraries.

---

## Refreshing a Library Folder

Refresh a library folder if you know library items are present in the library directories but are not on display in the Library tree, or if you want to reorganize the libraries in alphabetical order (the default organization). All libraries are refreshed when you start Simcenter Flotherm.

### Procedure

Right-click a library folder and select **Refresh**.

### Results

- Any unlisted library items are added to the Library tree.
- The library folders and contents are arranged in alphabetical order.

Names beginning with lower case characters are listed after all names beginning with upper case characters, that is, “a\_Library” is listed after “Z\_Library”.

## Deleting a Library Folder

Deleting a library folder also deletes it from your file system.

### Restrictions and Limitations

- The library folder to be deleted must not be Read Only.

### Procedure

1. Right-click a library folder and select **Delete**.

A confirmation dialog box is opened.

2. Click **Yes** to complete the deletion.

The library folder and all items are removed from the Library and the file system.

## Exporting a Library Folder

Export a library for archive to the external file system.

### Procedure

1. Right-click the library folder and select **Export to File**.

The Export Library selection dialog box is opened.

2. Navigate to a location and enter a filename.

3. Click **Save**.

The output file will be in the library file format, with file extension *\*.library*.

## Library Item Operations

---

The following operations act on library items.

You cannot delete libraries that are read-only.

<b>Copying Geometry From Libraries to the Data Tree.....</b>	<b>290</b>
<b>Copying an Attribute From a Library Folder to the Data Tree .....</b>	<b>291</b>
<b>Moving and Copying Library Items Between Library Folders .....</b>	<b>291</b>
<b>Saving Project Geometry to a Library Folder .....</b>	<b>292</b>

## Copying Geometry From Libraries to the Data Tree

Geometry objects (for example, fans) are provided in the installed libraries.

### Restrictions and Limitations

- When copying using the **Transfer To Project** menu option or by double-clicking:
  - If nothing in the data tree is selected, then the geometry will, by default, be copied to the root assembly.
  - If an invalid item is selected in the data tree, then nothing will be copied.

### Procedure

You have a choice.

If you want to...	Do the following:
Copy the geometry by double-clicking (simplest).	<ol style="list-style-type: none"><li>1. Select an assembly in the data tree into which you want the geometry to be copied.</li><li>2. Double-click the geometry in the Library.</li></ol>
Copy the geometry by drag and drop.	<ul style="list-style-type: none"><li>• Drag and drop the geometry from the Library to the data tree. The program prevents invalid dragging and dropping.</li></ul>
Copy the geometry using the context-sensitive menu.	<ol style="list-style-type: none"><li>1. Select an assembly in the data tree into which you want the geometry to be copied.</li><li>2. Right-click the geometry in the Library and select <b>Transfer To Project</b></li></ol>

## Copying an Attribute From a Library Folder to the Data Tree

Attributes that are copied from a library to geometry will automatically be added to the Project attributes tree.

### Restrictions and Limitations

- Dropping attributes onto multiple selections is not allowed.
- Attributes can only be attached to geometry that enables that type of attribute.

### Procedure

You have a choice.

If you want to...	Do the following:
Copy the attribute by double-clicking (simplest).	<ol style="list-style-type: none"><li>1. Select the object to which the attribute is to be attached.</li><li>2. Double-click the library attribute. If the selected object does not allow the attachment of the attribute, then the attribute is added to the Project attributes tree.</li></ol>
Copy the attribute by drag and drop.	<ul style="list-style-type: none"><li>• Drag and drop the attribute from the Library to the appropriate geometry in the data tree.</li></ul>
Copy the attribute using the context-sensitive menu.	<ul style="list-style-type: none"><li>• Right-click the attribute in the Library and select <b>Transfer To Project</b>. The attribute will be added to the Project attributes and can then be selected from the geometry property sheet <b>Attachments</b> tab.</li></ul>

## Moving and Copying Library Items Between Library Folders

Library house-keeping tasks.

## Procedure

You have a choice.

If you want to...	Do the following:
Move an item.	<ol style="list-style-type: none"> <li>1. Right-click the library item and select <b>Cut</b>.</li> <li>2. Right-click the destination folder and select <b>Paste</b>.</li> </ol> <p>Alternatively, drag the item between library folders.</p>
Copy an item.	<ol style="list-style-type: none"> <li>1. Right-click the library item and select <b>Copy</b>.</li> <li>2. Right-click the destination folder and select <b>Paste</b>.</li> </ol> <p>Alternatively, hold down the Ctrl key and drag the item between library folders.</p>

## Saving Project Geometry to a Library Folder

Saving geometry to a library makes it available for other projects when they are loaded.

### Procedure

1. Select the destination library folder in the Library tree.
2. In the data tree, right-click the geometry you want to save to the library node and select **Save to Selected Library**.

### Results

The geometry is copied to the selected library folder.

Alternatively you can drag and drop a geometry into a library folder.

## Starter Libraries

The libraries are initially stocked with standard entries to get you started, but you can build the libraries to suit your own requirements.

Organizations can also set up their own standard entries to be shared among their engineering teams.

**Table 7-1. Starter Libraries**

Library	Contents
ATX Library	Different styles of chassis and equipment suitable for ATX_Part, SFF_Part_Library and microATX_Part_Library based form factors.
Blowers	Libraries of blowers by the following suppliers: EBM, Sanyo Denki and YS Tech USA.

**Table 7-1. Starter Libraries (cont.)**

<b>Library</b>	<b>Contents</b>
Component Starter Pack	Different types of 2 Resistor Compact Components.
Data Center	APC, Generic, and Gilberts data center components. <ul style="list-style-type: none"> <li>• For a description of the contents of the Generic library, see “<a href="#">Generic Data Center Library</a>” on page 295.</li> <li>• For a description of the Gilberts GF Series Floor Grille, see “<a href="#">Gilberts Data Center Library</a>” on page 297.</li> </ul>
Fans	Libraries of axial fans by the following suppliers: Comair Rotron, Delta, ETRI, Micronel, NMB, Papst, and Sanyo Denki.
Filters	Libraries of resistance attributes representing the flow resistance of UAF foam filters (Planar), and libraries of assemblies containing resistances representing UAF foam and polyester media filters.
Simcenter_Flotherm_Libraries	Libraries for each attribute type, containing commonly used attributes pertaining to electronics.
Heat_Sinks	Subdirectories for models of Alpha Novatech, HS Marston, and IERC heat sinks.  The IERC ***README-NOTE assembly is an assembly that only contains a note about the IERC heat sinks. To read the note, copy the assembly into your project, then open the <b>Notes</b> tab of the property sheet.
Interface_Materials	Libraries of surface finishes representing Bergquist, Chomerics, Dow Corning, and Thermagon interface materials. The libraries are further subdivided into interface types.
Materials	Links providing quick access to the Materials library files listed under Simcenter Flotherm_Libraries.
Thermo Electric Coolers	Models of Marlow and Mercor TECs.
Tutorial Library	Contains an assembly, cbga1, as an example of a detailed component.

- *Library Content Disclaimer*

The data included in the delivered libraries has undergone review to ensure accuracy and currency. However, since there are many data sources available, some of the information may be out of date or not match your current knowledge. You are, therefore, advised to consult other sources to confirm information where there is any doubt as to the validity of the data supplied. For data associated with particular part manufacturers, you are advised to check with the supplier web-sites for the latest information.

Mentor Graphics does not assume responsibility for omissions or inaccuracies in the data included in these libraries nor any consequences resulting from their use within the software.

# Generic Data Center Library

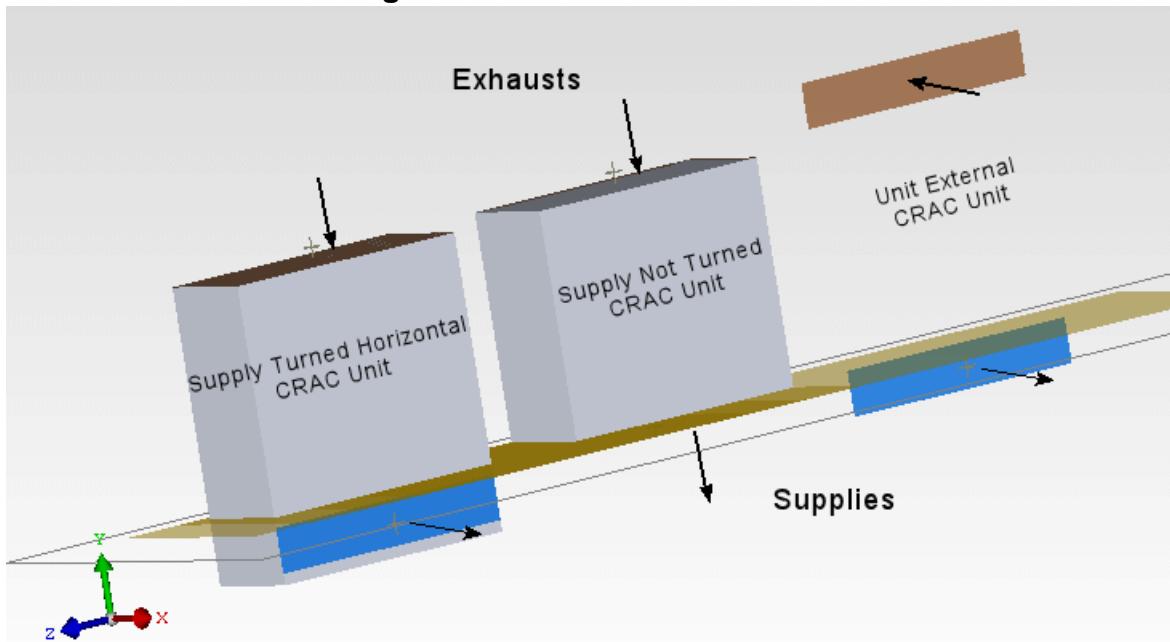
A generic data center starter pack is provided, comprising CRAC units, cabinets, floor grilles and floor supports.

<b>Generic CRAC Units .....</b>	<b>295</b>
<b>Generic Cabinets .....</b>	<b>296</b>
<b>Equipment for High Detail Cabinet.....</b>	<b>297</b>
<b>Generic Floor Grille.....</b>	<b>297</b>
<b>Generic Floor Support.....</b>	<b>297</b>

## Generic CRAC Units

Three generic assemblies are provided: Supply Not Turned, Supply Turned Horizontal, and Unit External.

**Figure 7-2. Generic CRAC Units**



The Supply Not Turned and Supply Turned Horizontal assemblies use a cuboid to represent the body of the unit and a Cooler SmartPart to represent the fans and coils. The Supply Not Turned assembly has vertical exhaust and supply. The Supply Turned Horizontal assembly has a vertical exhaust and a horizontal supply (that is, it assumes that turning vanes are used to turn the flow).

The Unit External assembly does not have a physical body but can be used to represent a CRAC unit that is outside the solution domain, that is, its supply and exhaust should be placed on the edge of the domain.

All the generic units have a default flow rate of 8.5 m<sup>3</sup>/s and maximum cooling load of 80 kW is set. The set point for the supply air is 15°C. Monitor points are provided on the supply and return.

The return air is at the top of the unit and supply is assumed to be beneath the floor (y = 0).

## Generic Cabinets

There are three detailed modeling levels: High, Medium, and Low.

All generic data center cabinets are created with the origin at a bottom corner. The front of a cabinet is coincident with the local Low-Z wall.

### Front to Back High Detail Cabinet

A high detail assembly consisting of an outer chassis (600 mm wide × 800 mm deep × 2100 mm high) with perforated doors front and back. Inside the individual servers are mounted between rack supports. To aid positioning the assembly contains a set of sources (no source attached) spaced 1 U apart which when activated act as a placement guide for internal equipment.

The assembly has four 4 U servers, but these can be exchanged with others from the Equipment for High Detail Cabinet assembly.

A hole, with associated resistance, is provided in the bottom to allow for leakage resulting from cable entry. This will overwrite a floor providing the cabinet is below the floor in the hierarchy and the floor is collapsed (as the template).

Because of the higher number of objects used this cabinet, a localized grid is provided, defined at the assembly level.

### Front to Back Medium Detail Cabinet

The medium detail assemblies consists of an outer chassis with three sources inside. This enables a lower, middle, and upper heat load. Three fans (allowing different flow rates through the three levels to coincide with the different sources) across the back pull the air through the system. The default source attached inside the cabinet defines equal heat on each third and also straightens the flow preventing vertical flow. The front door is split into three to allow analysis of the air temperature into the lower, middle, and upper door sections.

A number of sizes are available with heat loads of 1 kW or 3 kW. The airflow is based on a 10°C temperature rise between the front and back of the unit.

### Front to Back Low Detail Cabinet

This represents the lowest level of detail used to model a cabinet. A single heat source (with zero vertical flow) is inside the outer cabinet. A fan pulls the air through from front to back. The door is represented as a single resistance.

A number of sizes are available with 1 kW or 3 kW of heat. The airflow is based on a 10°C temperature rise between the front and back of the unit.

### Related Topics

[Equipment for High Detail Cabinet](#)

## Equipment for High Detail Cabinet

This generic data center library contains 1 U, 2 U, 3 U and 4 U server boxes, intended to be used in conjunction with the High Detail cabinet.

Each server box consists of an outer chassis with a heat source of 150 W per U of height. Air is pulled from front to back with a fan designed to give a 10°C temperature rise through the server.

### Related Topics

[Generic Cabinets](#)

## Generic Floor Grille

Two floor grilles are provided using Perforated Plate SmartParts and one floor grille is provided using the Fixed Flow SmartPart.

The two perforated plate floor grilles, 25% and 56% (open area), are designed for use with a plenum, that is, air is modeled on both sides of the grille. The fixed flow floor grille, perf tile - no plenum, is designed for use when there is no plenum, that is, no air is modeled behind the grille. All three of the grille templates are 2 ft square.

## Generic Floor Support

The generic data center floor support is designed to fit into a 600 mm floor void.

The total height is 575 mm to allow fitting below the tile edges found in the data center template and prevent grid extending beyond the floor void.

The support has a cross section of 50 mm x 50 mm and the origin of the assembly is centrally placed at y = - 600 mm (to place below floor level).

## Gilberts Data Center Library

A data center floor grille is supplied as an assembly of parts. The origin is positioned on the top surface of the grille in a corner.

The top of the grille is represented by a Resistance primitive, which will overwrite a floor if the floor is modeled using a collapsed resistance and is defined before the grille in the data tree hierarchy.

The assembly contains a set of deactivated resistance primitives to represent dampers at different settings. The appropriate resistance primitive needs to be activated. Different grilles in the model can have different damper positions if needed.

The construction of the floor grille is such that flow will be straightened by the grille in one direction only. If the flow is introduced from a side such that it is perpendicular to the bars then the flow will be straightened. When the flow is orientated along the bars, the straightening effect will only be as a result of other factors. In the model, the bars run along the local z-axis of the assembly, therefore, straightening has been defined for the local x-direction.

*Note: This library part has been created following a series of physical tests. These tests were to ensure that an adequate engineering representation was produced rather than exact recreation of the flow patterns under every conceivable condition. For example, air was supplied to the tested floor grille via a plenum box which was 500 mm deep and the same size as the grille. Different depth floor voids and the effect of adjacent floor grilles were not tested. A report on the testing carried out is available on request.*

# Library Property Sheets and Dialog Boxes

---

GUI interfaces associated with libraries.

<b>Library Data Property Sheet .....</b>	<b>300</b>
<b>Create Library Dialog Box .....</b>	<b>301</b>

## Library Data Property Sheet

To access: Select a library folder in the Library tree.

Use this property sheet to define the name and location of the library folder.

### Objects

Field	Description
Library Name	The name of the library.
Directory	Sets the directory to hold the new library items. Enter the full path of the directory. By default, Simcenter Flotherm uses: <i>&lt;install_dir&gt;/flosuite_v&lt;version&gt;/flotherm/flocentral/Libraries</i>
Read Only	Check to set the new library as read only. After confirmation of the settings, the library will be represented in the Library Manager as a white folder to indicate that the library is locked and cannot be changed. Uncheck to allow changes to the library.

### Usage Notes

You must have write access to the directory.

The program expects the directory to contain only valid library format files (\*.library, \*.pdml geometry). If there are non-library format files present in this directory, an error message will be displayed.

### Related Topics

[Create Library Dialog Box](#)

# Create Library Dialog Box

To access: Right-click an existing (non-read-only) library folder and select **New Library**.

Use this dialog box to create and name a new library folder.

## Objects

Field	Description
Library Name	The name of the library.
Directory	Sets the directory to hold the new library items. This must be the full path of the directory. By default, Simcenter Flotherm uses: <i>&lt;install_dir&gt;/flosuite_v&lt;version&gt;/flotherm/flocentral/Libraries</i> You must have write access to the directory.
Browse	An alternative to entering the full path in the Directory field, click to open the Directory dialog box and select the directory.

## Related Topics

[Library Data Property Sheet](#)



# Chapter 8

## Spatial Solution Grid

---

Grid operations on the drawing board, where the grid can be optimized prior to solving.

<b>Spatial Solution Grid Overview</b> .....	<b>304</b>
<b>Overview of Grid Modification</b> .....	<b>304</b>
<b>Grid Calculation Order</b> .....	<b>305</b>
<b>Grid Suitability</b> .....	<b>305</b>
<b>Grid Constraints</b> .....	<b>307</b>
<b>Grid Localization</b> .....	<b>309</b>
General Effect of Localizing a Grid .....	309
Nested Localized Grid Spaces .....	310
Abutment of Localized Grid Spaces .....	310
Keypoints and Localized Grids .....	311
Localization of Inflated Grids .....	311
Grid Space Overlap When Localizing .....	312
Localization Near the Edge of the Solution Domain .....	313
Objects Located Inside a Localized Grid .....	313
Object Location Restrictions When Using Localized Grids .....	314
Grid Smoothing and Localized Cells .....	315
Minimum Cell Size and Localized Cells .....	316
<b>Grid Inflation</b> .....	<b>317</b>
<b>Grid Smoothing</b> .....	<b>319</b>
<b>Keypoint Deactivation (De-Keypointing)</b> .....	<b>320</b>
<b>Grid Operations</b> .....	<b>322</b>
Debugging the Grid of Large Aspect Ratio Cells .....	322
Setting System Grid Constraints .....	323
Setting an Object Grid Constraint .....	323
Localizing the Grid Around an Object .....	324
Inflating the Grid Around an Object .....	325
Deflating the Grid Around an Object .....	325
Smoothing the Grid .....	325
<b>Solution Grid Property Sheets and Dialog Boxes</b> .....	<b>327</b>
System Grid Property Sheet .....	328
Grid Summary Dialog .....	330
Grid Changed Dialog Box .....	332
Previous Software Version Dialog Box .....	333

# Spatial Solution Grid Overview

The spatial solution grid subdivides the solution domain into a set of non-overlapping contiguous finite volumes called grid cells.

The CFD techniques used by Simcenter Flotherm calculate the conservation equations over each grid cell. See [Background of Computational Fluid Dynamics \(CFD\)](#) in the *Simcenter Flotherm Background Theory Reference Guide*.

Grid distribution is important as it affects the numerical accuracy, determining how close the discretized numerical solution is to the true, continuous variation implied by the differential equations.

By default, Simcenter Flotherm sets a default solution grid to align with the edges of the geometry you create. You can then refine this default grid in regions of complex flow, high gradients, and areas of particular interest. However, the more detailed the grid, the more computer time is required, so the choice of grid is a trade-off between accuracy and speed.

The spatial solution grid can be viewed:

- In the drawing board by pressing G.
- In Analyze mode, by selecting Show Grid in the [Plane Plot Property Sheet](#). In addition, the grid details of location and type can be displayed in the Grid results tables, see “[Reporting Project Data and Results in Tables](#)” on page 477.

## Dimensionality

The Dimensionality setting in the [Model Setup Tab](#) affects the grid. 2D calculates a 3D grid cell and then reduces it to a depth of one grid cell as shown in [Figure 4-5](#) on page 156.

## Grid Interpolation

Simcenter Flotherm automatically detects any changes to the geometry that affect the grid, and displays the [Grid Changed Dialog Box](#) for interpolating solution results to the new grid.

When a pre-V12.0 legacy project with results is being loaded for the first time, and a grid change is detected, then the [Previous Software Version Dialog Box](#) is displayed.

# Overview of Grid Modification

User control over the dimensions of the spatial solution grid.

The following functionality enables you to modify the base grid to suit your needs:

- Grid Constraint, either at system level or applied to individual objects, to constrain the sizes of grid cells or the number of cells.

- Grid Localization, to restrict grid lines from extending outwards from objects to the edges of the solution domain.
- Grid Inflation, to extend the grid beyond objects, either by a set distance or by a percentage of their size.
- Grid Smoothing, to remove large variations in neighboring cell sizes and large aspect ratios between directions.
- Deactivating keypoints, to remove grid lines associated with keypoints.

## Related Topics

[Grid Operations](#)

# Grid Calculation Order

It is useful to know the order in which the grid is calculated to appreciate how modifications affect the final grid.

The final grid is calculated in the following order:

1. The initial grid is based on the geometry keypoints. That is, grid lines are constructed coincident with geometry boundaries.
2. Then any constraints on objects and assemblies are applied.
3. Then system grid constraints are applied.
4. If Grid Smoothing has been defined, then that is applied next.
5. Finally, any deactivated keypoints are ignored.

# Grid Suitability

Use the Grid Summary dialog to check that the grid is suitable for the current application.

You can check:

- Total grid size, see [Total Grid Size](#).
- Size of the smallest grid cells for reasons of accuracy, see [Smallest Grid Cell](#).
- Maximum aspect ratios are not high for critical cells, see [Maximum Aspect Ratios](#).

Also, see “[Debugging the Grid of Large Aspect Ratio Cells](#)” on page 322.

## Total Grid Size

In attempting to create as realistic a model as possible, you may be tempted to generate a detailed grid, adding more grid cells than necessary. Approximates for adequate grid modeling requirements are provided by the application examples supplied with your installation or on Support Center:

<https://support.sw.siemens.com>

## Smallest Grid Cell

The smallest grid information, shown in the [Grid Summary Dialog](#), should be used to check that there are no cells smaller than  $10^{-6} \times$  the overall scale for the system being modeled.

For example, for a system which is of the order of 1 m, the smallest grid cell should be larger than 1 micron.

If this modeling rule is ignored, then the grid cell volumes can begin to approach the minimum resolution which can be handled by the computer and serious problems in the solution may occur.

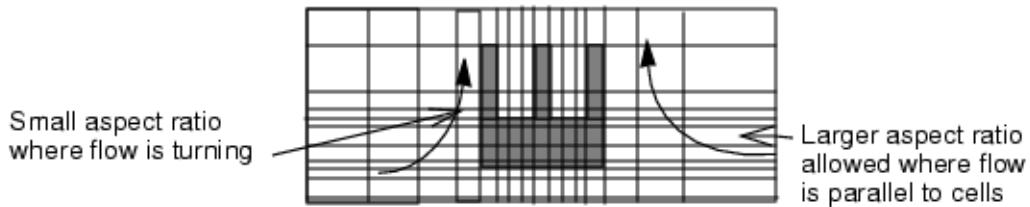
## Maximum Aspect Ratios

A bad grid, typified by having cells with high aspect ratios, can cause slow or sometimes divergent solver behavior.

For most system-level analysis, aspect ratios exceeding 10:1 are rarely required. For package-level analysis, larger aspect ratios are unavoidable, but should be confined to regions well away from the object(s) of interest.

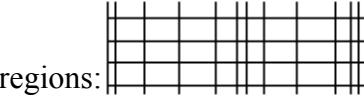
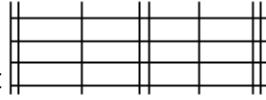
Also, high aspect ratios on the edges of the solution domain often cannot be avoided and have no detrimental effects on the results, see [Figure 8-1](#).

**Figure 8-1. Maximum Aspect Ratios**



[Table 8-1](#) provides guidelines for aspect ratio levels.

**Table 8-1. Guidelines on Aspect Ratio Levels**

	<b>Recommended</b>	<b>Avoid</b>
Grid Distribution	Smooth transitions between regions: 	Abrupt transitions between regions: 
Maximum Grid Cell Aspect Ratio	Ideally < 20	> 200
Minimum Grid Cell Size	$> 10^{-6} \times \text{Domain Size}$	$< 10^{-6} \times \text{Domain Size}$

If large aspect ratios are necessary because of geometry interaction, try to make sure that they occur in regions where the flow is simple and unidirectional, and that no large temperature gradients exist, otherwise you may have solution convergence problems.

For example, consider a single package or device modeled in detail in free air. The ideal grid should focus in towards the device, becoming gradually larger with distance away from the device. Large aspect ratio grid cells can be encountered at some distance away from the device, but this area is usually of little concern and should not have any extreme flow or temperature variations. Localizing the grid can improve the grid further, with a localized grid around the device, aspect ratios at large distances from the device are improved.

## Related Topics

[Debugging the Grid of Large Aspect Ratio Cells](#)

# Grid Constraints

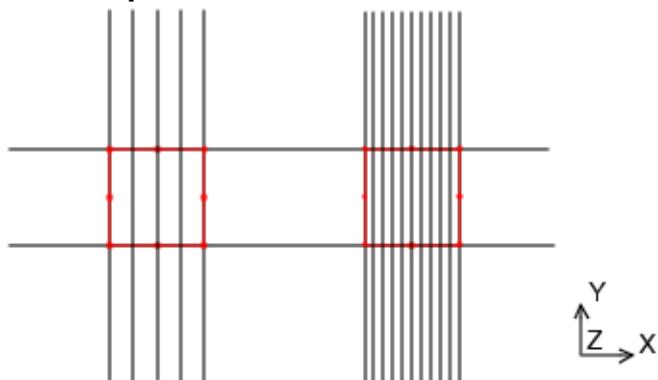
It is important to ensure adequate grid coverage wherever objects are placed in the model. Grid constraints define the minimum grid requirement across a selected geometry.

There are Object grid constraints and System grid constraints.

Use Object grid constraints to attach minimum grid requirements to individual objects.

[Figure 8-2](#) shows two cuboids with attached grid constraints in the X-direction. The left cuboid has a minimum of four cells and the right cuboid has a minimum of ten.

**Figure 8-2. Example Grid Constraints Attached to Two Cuboids**



Grid constraints determining the solution accuracy level can be applied over the complete solution domain using the System Grid property sheet, which provides control of:

- Smallest cell size allowed,
- Maximum cell size or minimum cell number,
- Smoothing of adjacent cell length ratio and inter-direction length aspect ratios.

## Related Topics

[Setting System Grid Constraints](#)

[Setting an Object Grid Constraint](#)

[System Grid Property Sheet](#)

# Grid Localization

By default, grid lines generated by an object, whether they are due to keypoints or constraint lines, extend to the edges of the solution domain, however, this may not be necessary and can be prevented by “localizing” the grid.

<b>General Effect of Localizing a Grid .....</b>	<b>309</b>
<b>Nested Localized Grid Spaces.....</b>	<b>310</b>
<b>Abutment of Localized Grid Spaces.....</b>	<b>310</b>
<b>Keypoints and Localized Grids .....</b>	<b>311</b>
<b>Localization of Inflated Grids.....</b>	<b>311</b>
<b>Grid Space Overlap When Localizing.....</b>	<b>312</b>
<b>Localization Near the Edge of the Solution Domain .....</b>	<b>313</b>
<b>Objects Located Inside a Localized Grid .....</b>	<b>313</b>
<b>Object Location Restrictions When Using Localized Grids .....</b>	<b>314</b>
<b>Grid Smoothing and Localized Cells .....</b>	<b>315</b>
<b>Minimum Cell Size and Localized Cells .....</b>	<b>316</b>

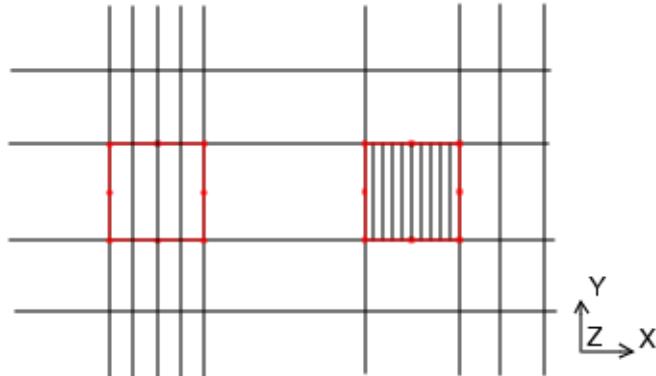
## General Effect of Localizing a Grid

Localizing a grid can be an effective way of reducing the number of cells.

Figure 8-3 shows the effect of localizing the grid for an isolated cuboid. The ten grid constraint lines inside the cuboid on the right are not needed in the air outside it, therefore, a grid space has been defined by the cuboid such that only the keypoints defining the edges of the grid space continue to the edges of the solution domain.

This is the simplest case, localizing a grid may be affected by other grid modifications and nearby geometry.

**Figure 8-3. Localized Grid**



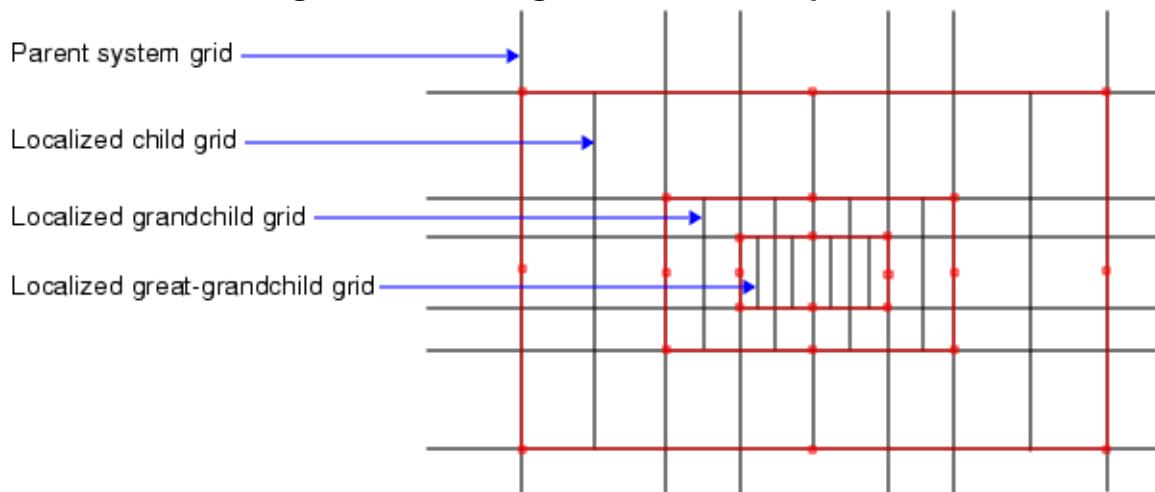
## Nested Localized Grid Spaces

To allow the nesting of localized grid spaces, a parent-child grid relationship is followed.

The basic system grid is known as the Base Grid, and any localized grid spaces created over this basic grid become children of the system grid. If localized grid spaces are created over existing localized grid spaces, then in turn, the localized grid spaces themselves have children and the basic grid becomes a grandparent.

[Figure 8-4](#) shows three localized grids created in order of size. All grid lines from a parent grid space will pass through all child grid spaces. Localized grids are, therefore, always finer than their parent spaces.

**Figure 8-4. Nesting Localized Grid Spaces**



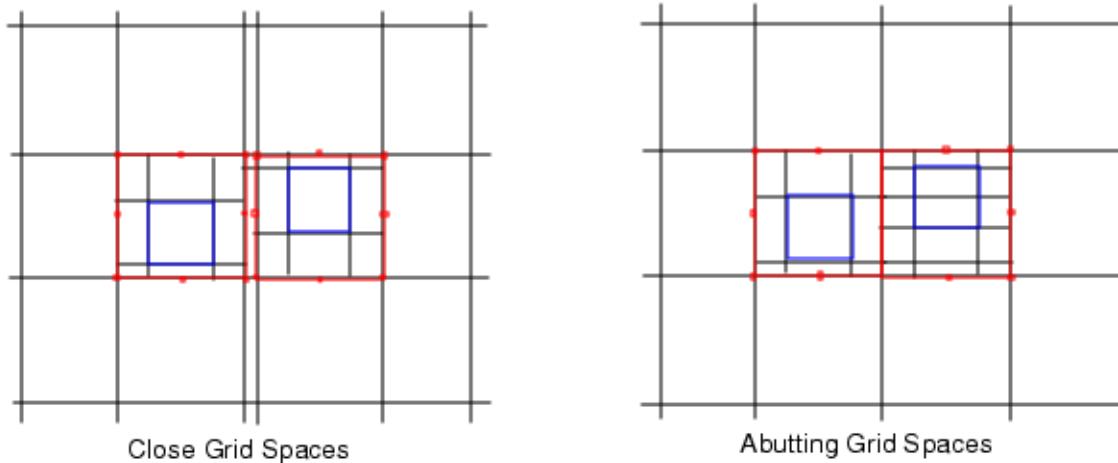
## Abutment of Localized Grid Spaces

Where localized grid spaces abut, all of the grid lines from one grid space pass through (or bleed into) the other.

The solver treats the interface between abutting grid spaces in the same way as it treats the interface between parent and child grid spaces.

The choice of which grid space's lines bleed into the other is based on minimizing the number of additional cells created by the bleeding. Under certain circumstances, the finer grid may bleed into the coarser abutting grid, but only when the resulting total number of grid cells in the model is lower than the coarse cells bleeding into the finer grid space. [Figure 8-5](#) illustrates the differences in grids when two localized grid spaces are close to each other, and are abutting.

**Figure 8-5. Close and Abutting Localized Grid Spaces**



## Keypoints and Localized Grids

The extent of a localized grid space is determined by the keypoints on the corners of the object or inflation box.

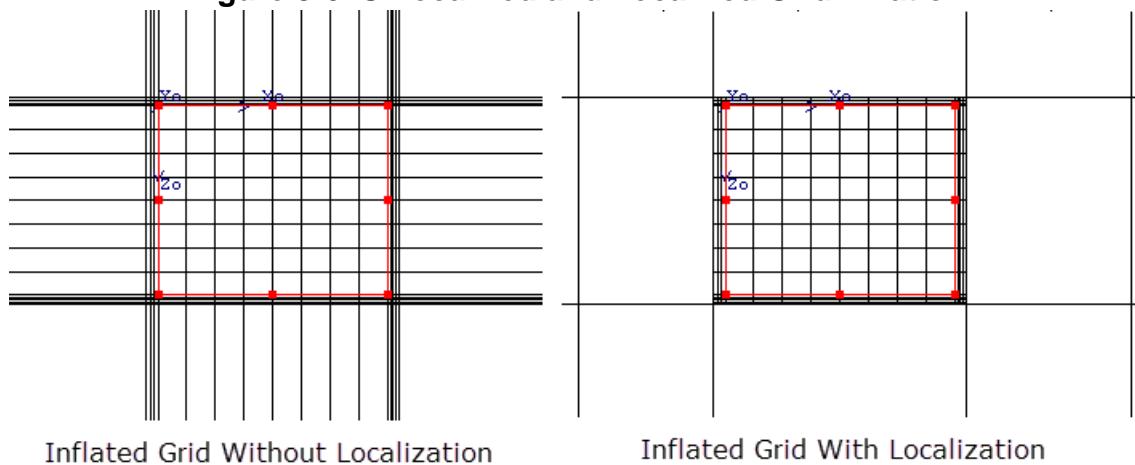
Keypoints cannot be removed from the face boundaries of localized grid spaces. As this has not always been the case, legacy projects with results that are imported may cause the grid to be interpolated, and the [Previous Software Version Dialog Box](#) to be displayed.

## Localization of Inflated Grids

Localization can be applied to inflated grids to restrict the grid lines to the object space and inflated area.

When grid constraints have been set to extend a distance beyond the object using the Inflation Grid panel in the Grid Constraint property sheet then localizing the grid for the object restricts the grid lines as shown in [Figure 8-6](#).

**Figure 8-6. Unlocalized and Localized Grid Inflation**



## Related Topics

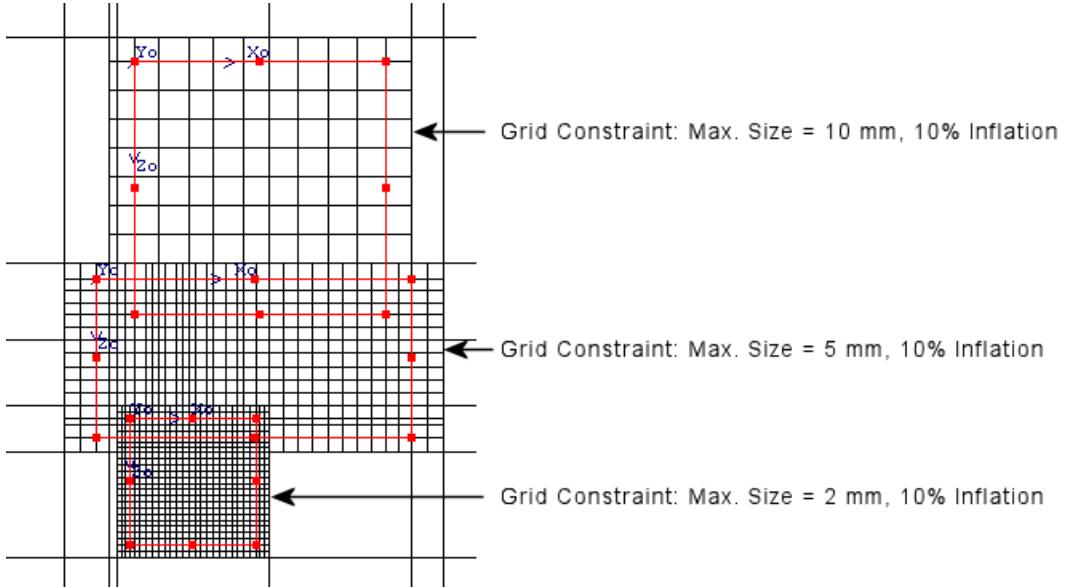
[Grid Constraint Property Sheet \[Simcenter Floterm Project Attributes Reference Guide\]](#)

## Grid Space Overlap When Localizing

When localized grid spaces overlap, the smallest grid cell specification is used to define the overlapped space.

This is shown in [Figure 8-7](#). Note that the application of grid constraints for the overlapped grid spaces is independent of the hierarchical positions of the objects in the data tree, it is always based on the smallest grid size.

**Figure 8-7. Localized Overlapping Localized Grids**



## Handling of Legacy Overlapped Localized Grids

Prior to V11.0, overlapping of two grid spaces was not permitted. The localization settings for an object lower in the data tree hierarchy was ignored if the spaces overlapped.

How these projects are handled after importing into V11.0 and higher depends on whether or not results are included:

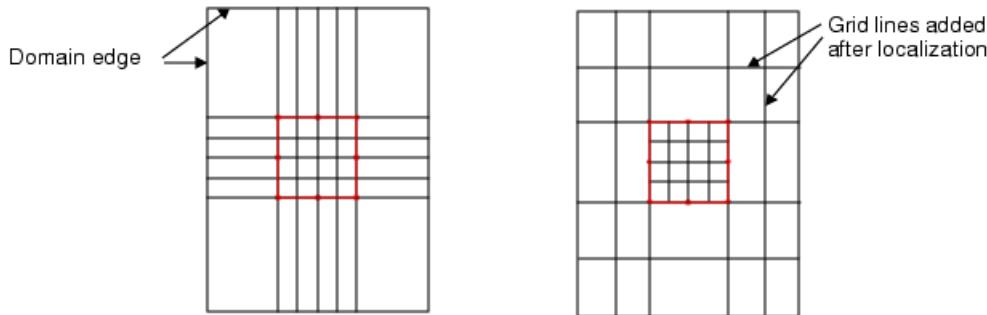
- If results are NOT included (for example, by importing PDML) then the grid is updated, allowing overlapping.
- If results are included (for example, by importing a PACK file with Results), then the legacy grid is retained to be compatible with the results.

## Localization Near the Edge of the Solution Domain

If a localized grid space is one grid cell away from the edge of the solution domain, then an additional grid line is inserted to ensure there are two grid cells.

This effect of localization is shown in [Figure 8-8](#). The addition of the extra grid cells ensures an accurate CFD solution between the localized grid space and the edge of the domain.

**Figure 8-8. Localization Near the Solution Domain Edge**



---

### Note

 An extra grid line will always be inserted under such circumstances, regardless of any minimum cell size tolerance defined. Care should, therefore, be taken to ensure a localized grid space is exactly coincident with the edge of the domain, if that is what is required.

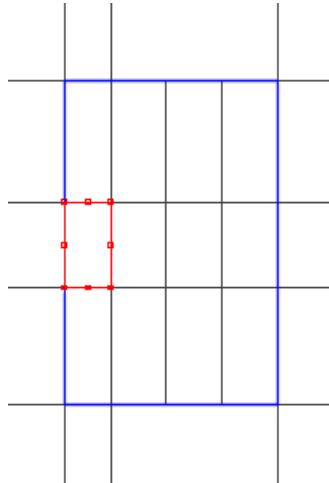
---

## Objects Located Inside a Localized Grid

When objects are placed inside a localized grid, the grid is modified.

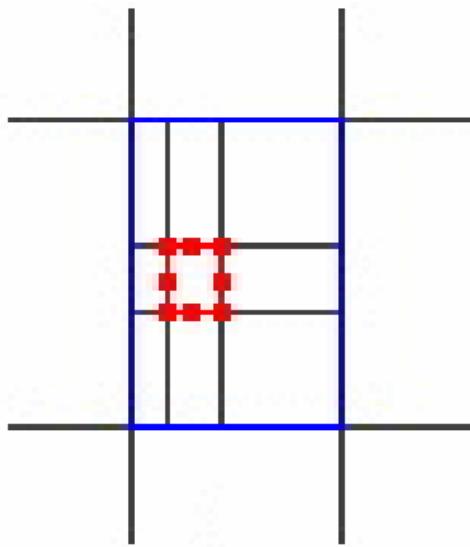
If an object internally abuts a localized grid, then all of its keypoints will pass into the parent grid, as shown in [Figure 8-9](#), where the blue cuboid is localized.

**Figure 8-9. Object Internally Abutting a Localized Grid**



If the object lies more than one grid cell inside a localized grid space, then all of its keypoints will be contained by the localized grid as shown in [Figure 8-10](#).

**Figure 8-10. Object Lying More Than One Grid Cell Inside a Localized Grid**



The easiest way to ensure that objects are wholly within a localized grid is to inflate the object that defines the localized grid, see “[Grid Inflation](#)” on page 317.

## Related Topics

[Object Location Restrictions When Using Localized Grids](#)

# Object Location Restrictions When Using Localized Grids

Care must be taken when placing certain objects near localized grids.

The following objects must be located wholly inside or on the edge of a single localized grid space:

- Fan primitive part of a fan SmartPart
- Recirculation device supply and extract
- Fixed Flows
- Compact components
- Compact heat sinks
- Thin sloping blocks

If any of these objects break this restriction, then the solver will not function and a relevant error message displayed in the Message Window. The restriction is more severe for General Model - Peripheral Package Type compact components, these are not permitted to touch the edge of a localized grid space. In such cases, it is good practice to inflate the localized grid beyond the compact component and into the parent grid, a good boundary layer will then be captured.

Localizing the grid only applies to 3D objects. Error messages are output if you attempt to localize the grid on non-3D objects such as collapsed objects, and any localized grid constraints will be turned off.

## Related Topics

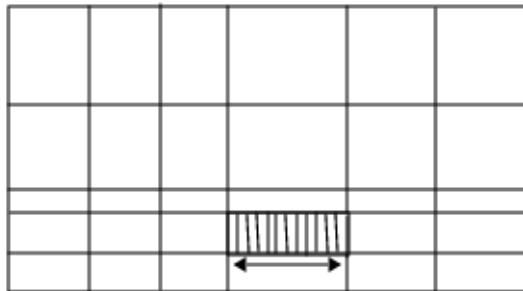
[Objects Located Inside a Localized Grid](#)

## Grid Smoothing and Localized Cells

When smoothing the grid spacing across the solution domain, the existence of small localized grid cells may cause the addition of grid cells in the parent grid.

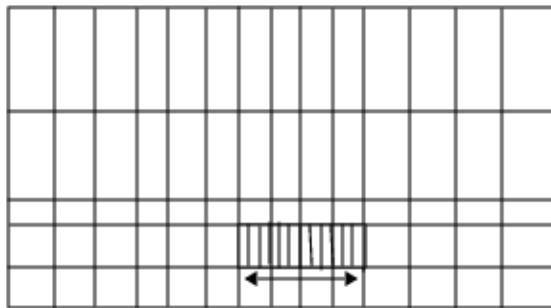
As an example, consider [Figure 8-11](#), which shows a localized (child) grid within a base (parent) grid. Before smoothing, the localized grid is for an object with grid constraints in the X-direction only.

**Figure 8-11. Localized Grid Before Smoothing**



If Medium smoothing is applied, additional cells are added to the parent grid as shown in Figure 8-12.

**Figure 8-12. Localized Grid After Smoothing**



## Minimum Cell Size and Localized Cells

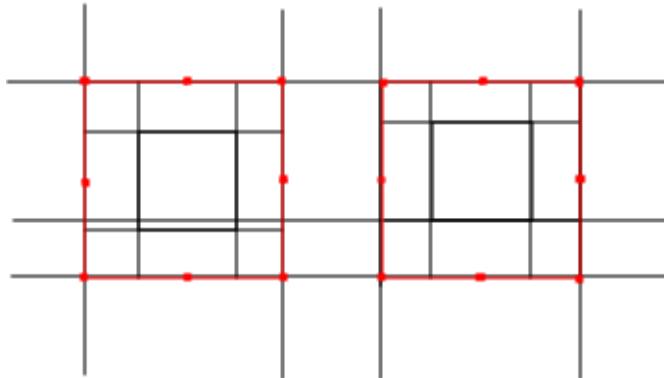
The minimum cell size in the base grid is determined by the system grid setting. Localized grid spaces can reduce (but not increase) their minimum allowable cell size via the grid constraint attached to the localized object which defines the space.

Nested grid spaces can further reduce the minimum cell size, relative to their parent grid space. This provides a way of restricting small cells to localized grid spaces; the minimum cell size setting should generally be confined to the grid space on which it is defined.

However, there are a small number of situations where the minimum cell size constraint may be violated, for example, this can happen where localized grid spaces abut. Because all of the grid lines from one grid bleed into the other (see “[Abutment of Localized Grid Spaces](#)” on page 310), it is possible that small cells can bleed into a grid space in which they are smaller than the minimum cell size constraint.

Another situation is where there are closely-spaced objects in different grid spaces, see Figure 8-13.

**Figure 8-13. Closely-Spaced Objects in Different Grid Spaces**



Each of the localized grid spaces contains a cuboid which generates its own gridlines. The two cuboids are slightly offset from each other in the Y-direction. The base grid needs a grid line in the Y-direction to satisfy the system grid maximum cell size, so it selects a grid line which coincides with the cuboid in the right-hand localized grid space. However, since all base grid lines must pass through child grids, this results in a small cell (which may be smaller than the minimum cell size) in the left-hand grid space. This type of behavior is very sensitive to the exact placement of objects, the system grid and other grid constraint settings, and the relative order of the objects in the data tree.

## Related Topics

[Localizing the Grid Around an Object](#)

# Grid Inflation

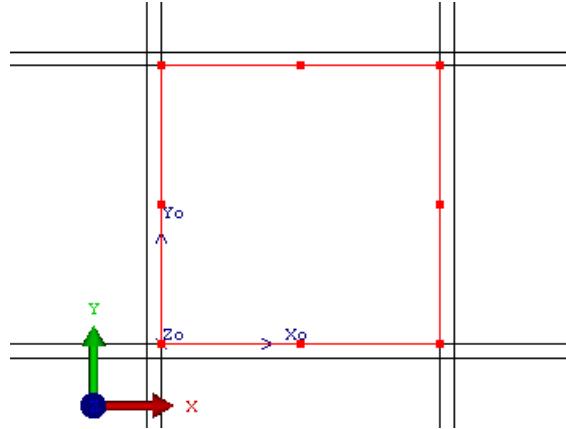
When localizing grid space, the localized grid space is the same size as the object. However, the grid space can be extended beyond an object by specifying an inflation layer.

Grid inflation can be added to all sides by choosing **Grid > Inflated Grid**. To add inflation separately to high and low sides, specify Low Side and High Side Inflation within a grid constraint.

## Default Grid Inflation

If inflation has not been applied to a side of the selected object, then, when adding uniform grid inflation (**Grid > Inflated Grid**), a default inflation of 5% on the high and low coordinate side length, with a minimum number of one cell, is applied, see [Figure 8-14](#).

**Figure 8-14. Default Grid Inflation**



New grid constraints are created by the program and attached to the x-, y- and z-coordinates in cases where:

- No grid constraint is attached to the object being inflated, or

- A grid constraint is attached but it is also attached to another object.

If a grid constraint is attached and not attached to another object then that grid constraint will be used. In this case undefined inflation settings will be replaced with the default settings.

In such cases the new grid constraint is named:

*<object name>\_<letter>*

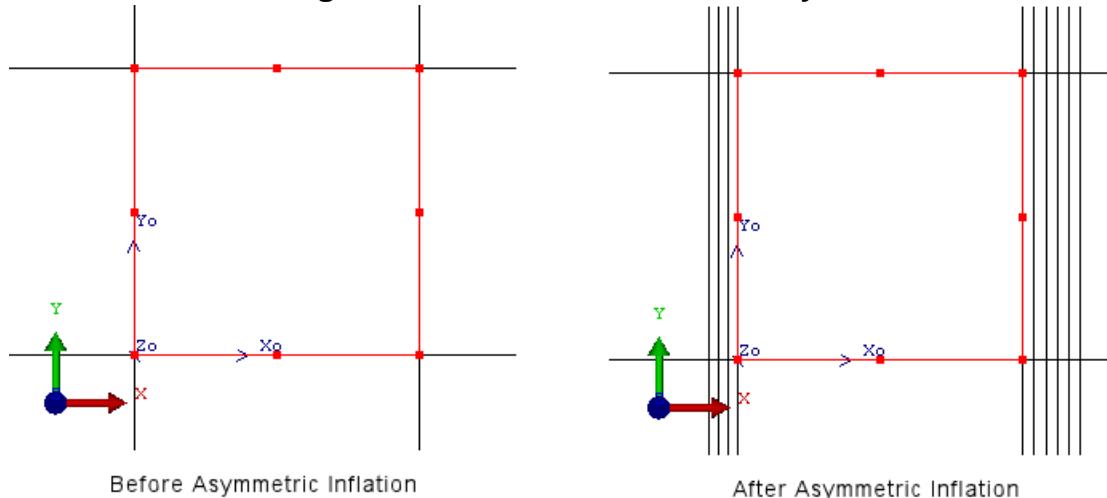
where *<letter>* is a, b, or c. For example, Cuboid:1\_a.

If the object you want to inflate has three different grid constraints attached in three different directions *and* those three grid constraints are attached to other objects in the project, then the inflated attachments will contain the name pattern of *<object name>\_a*, *<object name>\_b*, and *<object name>\_c* for the X, Y and Z directions respectively.

## Asymmetric Grid Inflation

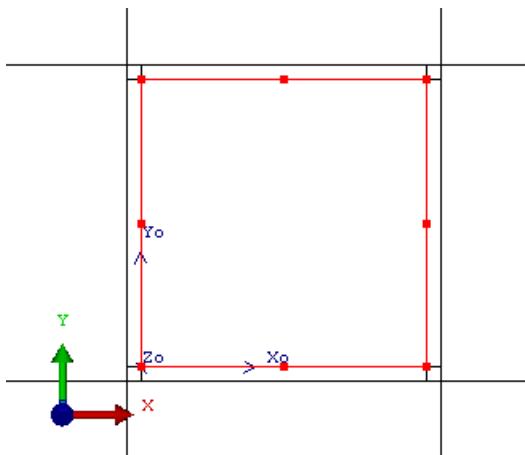
You can specify different inflation on each side of an object. For example, in [Figure 8-15](#), a Grid Constraint attribute has been attached only to the Xo direction of a cuboid. On the Low Side, three cells have been added over an additional 10% of the cuboid. On the High Side, five cells have been added over an additional 20% of the object.

**Figure 8-15. Grid With Inflation Layers**



## Grid Inflation and Localization

If grid constraints defining inflation layers are attached to an object, but the object has Localize Grid switched on, then the inflation box still generates extra grid lines beyond the extent of the object, see [Figure 8-16](#). This is useful for some kinds of grid generation.

**Figure 8-16. Grid Inflation and Localization**

## Related Topics

[Grid Constraint Property Sheet \[Simcenter Floterm Project Attributes Reference Guide\]](#)

[Inflating the Grid Around an Object](#)

[Deflating the Grid Around an Object](#)

[Grid Localization](#)

# Grid Smoothing

After all other grid conditions have been applied, you can smooth the grid by removing large variations in neighboring cell size in each direction, and removing large aspect ratios between directions.

The System Grid property sheet provides buttons and slider bars to control:

- The smallest cell size allowed
- The maximum cell size or minimum cell number
- Smoothing of adjacent cell length ratio and inter-direction length aspect ratios.

## V3 Grid Smoothing Algorithm

The V3 smoothing algorithm should be used in preference to the earlier V2 smoothing algorithm, and is the default.

The V3 grid smoothing is almost identical to the V2 algorithm, however, it does not create as many extremely small cells during the smoothing calculation. This sometimes happens when very small cells exist prior to V2 smoothing and can cause a large total number of grid cells.

## V2 Grid Smoothing Algorithm

The V3 smoothing algorithm should be used in preference to the V2 smoothing algorithm, and is the default. The V2 algorithm remains available so that projects created in V2 that use smoothing can be represented in V3 without imposing a grid change.

The grid smoothing algorithm is run after the keypoints and grid constraints have been calculated. The grid smoothing algorithm only adds additional grid cells. It will not change grid lines previously calculated.

The single slider bar performs the grid smoothing by simultaneously varying a number of parameters using the following two calculation methods:

- The first calculation is based on the length ratios of neighboring cells in a single grid direction. It will compare the lengths of the two cells in this direction, and if they are significantly different (based on a tolerance varied by the slider bar), it will split the largest of the cells in half (provided it is larger than the small cell size also varied by the slider bar). This will be repeated for all cells in this direction until the tolerances specified by the slider bar are met.
- The second calculation is based on the aspect ratios of cells in a given grid direction to cells in the other two grid directions. If these aspect ratios are large (based on a tolerance varied by the slider bar), it will split the large cell in this direction in half (provided it is larger than the small cell size also varied by the slider bar). This will be repeated for all cells in this direction until the tolerances specified by the slider bar are met.

### Related Topics

- [Smoothing the Grid](#)
- [Grid Suitability](#)
- [System Grid Property Sheet](#)

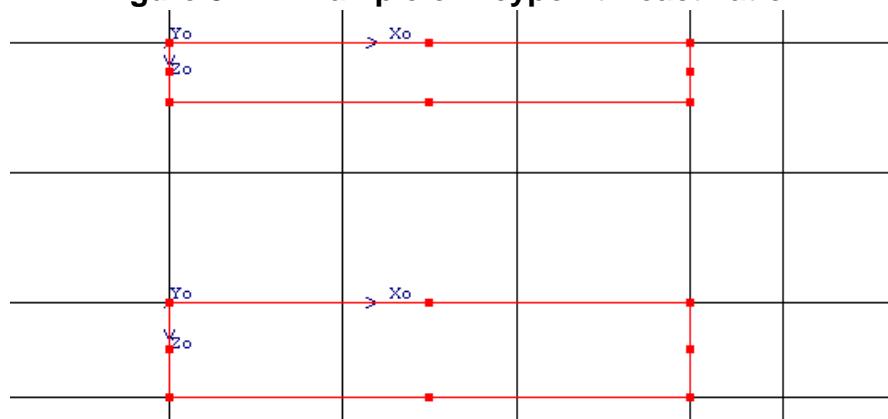
## Keypoint Deactivation (De-Keypointing)

Keypoint deactivation occurs when objects are close together to reduce an excessive number of grid cells.

Keypoint deactivation occurs when a dimension of an object is smaller than the Minimum Grid Size set in the System Grid property sheet. In such cases no grid line coincides with the keypoint.

[Figure 8-17](#) shows an example where deactivation is invoked on one cuboid, but not on another. The Z dimension of the lower cuboid is greater than the minimum grid size.

**Figure 8-17. Example of Keypoint Deactivation**



## Related Topics

[System Grid Property Sheet](#)

[User Preferences Dialog Box - Summary Tab](#)

## Grid Operations

The following procedures can be used to control the solution grid.

<b>Debugging the Grid of Large Aspect Ratio Cells .....</b>	<b>322</b>
<b>Setting System Grid Constraints .....</b>	<b>323</b>
<b>Setting an Object Grid Constraint.....</b>	<b>323</b>
<b>Localizing the Grid Around an Object .....</b>	<b>324</b>
<b>Inflating the Grid Around an Object.....</b>	<b>325</b>
<b>Deflating the Grid Around an Object .....</b>	<b>325</b>
<b>Smoothing the Grid .....</b>	<b>325</b>

## Debugging the Grid of Large Aspect Ratio Cells

Use the following procedure to “debug” large aspect ratio grid cells.

### Procedure

1. If the maximum aspect ratio, shown in the status bar, is thought to be too high, then view the Grid Summary Dialog, which lists the grid cells with the largest aspect ratios. You can also click the **Highlight the Maximum Aspect Cell** icon on the right of the status bar.
2. Select a cell in the Grid Summary Dialog to highlight it in the drawing board, and list the objects responsible.
3. Select each object responsible to highlight it in the drawing board.
4. Click the **Zoom To** button to center and zoom-in on the cell (and object if also selected).
5. You have a choice:

If you want to...	Do the following:
Move the objects responsible.	<ul style="list-style-type: none"><li>• Move the objects to reduce the bad aspect ratios, and increase the cell sizes if necessary</li></ul>
Not move the objects responsible.	<ul style="list-style-type: none"><li>• Localize objects, assemblies, or regions that contain small grid cells. This can be a very effective method of reducing aspect ratios in the other areas of the solution domain.</li></ul>

### Related Topics

[Grid Suitability](#)

[Grid Summary Dialog](#)

[Grid Localization](#)

## Setting System Grid Constraints

How to change the grid constraints that apply to the whole system.

### Procedure

1. Open the System Grid property sheet.
2. Refine the solution grid using one of the following:
  - Set a predefined standard grid type (coarse, medium, or fine) or,
  - Manually change the grid by successive degrees of grid density.

### Related Topics

[Grid Constraints](#)

[System Grid Property Sheet](#)

[Setting an Object Grid Constraint](#)

## Setting an Object Grid Constraint

Grid constraints are applied to individual objects, assemblies, and cutouts on the solution domain by attaching Grid Constraint attributes.

### Procedure

1. Select an object in the data tree or drawing board.
2. Select the **Attachments** tab of the object property sheet.

If grid constraints can be applied to the object then a Grid Constraint section of the tab will be shown.

3. Use the dropdown list to do one of the following:
  - Select an existing Grid Constraint attribute.
  - Create a new Grid Constraint attribute.

Follow the instructions given in [Creating a New Attribute for an Object](#) described in the *Simcenter Flotherm Project Attributes Reference Guide*.

- Add an attribute from a library.

Follow the instructions given in [Adding an Attribute From a Library to the Project](#) described in the *Simcenter Flotherm Project Attributes Reference Guide*.

4. Click **Edit** to view the Grid Constraint property sheet and edit if required.
5. To apply grid constraints in different directions, then expand the Grid Constraint section by pressing + and attach grid constraints as required.

## Related Topics

[Grid Constraints](#)

[Grid Constraint Property Sheet \[Simcenter Flotherm Project Attributes Reference Guide\]](#)

[Attribute Operations \[Simcenter Flotherm Project Attributes Reference Guide\]](#)

[Setting System Grid Constraints](#)

# Localizing the Grid Around an Object

Localize the grid around an object to stop grid lines from extending unnecessarily far from the object.

## Restrictions and Limitations

Localizing a grid may be affected by the geometry, as described under “[Grid Localization](#)” on page 309.

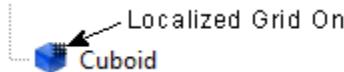
## Procedure

1. Select the object.
2. Do one of the following:
  - Press L, or
  - Open the **Location** tab of the property sheet and check the Localize Grid option.

## Results

- In the data tree, a localized grid is denoted by the addition of a grid in the top-left corner of the object’s icon, for example, see [Figure 8-18](#).

**Figure 8-18. Cuboid With Localized Grid Switched On**



- In the drawing board, with grid switched on (G) you will be able to see that grid lines are restricted to within the object, for example, the right-hand cuboid shown in [Figure 8-3](#).

## Related Topics

[Grid Constraints](#)

[Grid Inflation](#)

## Inflating the Grid Around an Object

Inflate the grid around an object to extend the grid in the vicinity of the object.

### Restrictions and Limitations

- By default, the grid is inflated by 5% of the object's size with a single cell. To use a different inflation, attach a Grid Constraint attribute to the object and define the amount of inflation using the Low Side Inflation and/or High Side Inflation sections of the property sheet. Make sure that the grid constraint is applied to the relevant coordinate(s).

### Procedure

1. Select the object around which you want to add inflation layers.
2. Click the **Inflate Grid** icon or choose **Grid > Inflate Grid**.
3. Use the drawing board to view the area surrounding the selected object and check the inflation is as expected.

### Related Topics

[Grid Inflation](#)

[Grid Constraint Property Sheet \[Simcenter Flotherm Project Attributes Reference Guide\]](#)

[Deflating the Grid Around an Object](#)

## Deflating the Grid Around an Object

Deflate the grid around an object to remove the effect of grid inflation.

### Procedure

1. Select the object around which you want to remove the inflation layers.
2. Click the **Deflate Grid** icon or choose **Grid > Deflate Grid**.

### Results

Inflation layers around the selected object will be removed. If there are no inflation layers then nothing happens.

### Related Topics

[Grid Inflation](#)

[Inflating the Grid Around an Object](#)

## Smoothing the Grid

Smoothing the grid should be the last stage of defining your grid.

## Restrictions and Limitations

- You will be using the System Grid property sheet, whose settings potentially affect the whole grid.
- There is a trade-off between accuracy with speed of solution. It is better to start with a coarse grid setting so that you can get a solution in a reasonable time. You can then refine the grid and resolve successively to the point at which the main results of interest do not vary with successive refinements.

## Prerequisites

- You should have previously applied any local settings such as object constraints, inflation, and localization as well as system constraints.

## Procedure

1. Open the System Grid property sheet.

You will be able to see the effect of smoothing as you change settings on the dialog box.

2. Select each of the **Coarse**, **Medium**, and **Fine** buttons in turn to see how these buttons affect the grid density.

The slider bars underneath move position as you click each button. The buttons provide a quick way of moving all the slider bars in a single operation.

3. If the buttons do not provide a suitable grid density, then use the slider bars to achieve the desired grid. Movement to the right refines the grid.

- Move the Minimum Size slider bar to set the minimum size of grid cell.
- The second slider bar gives you a choice of setting cell size.
  - Move the Maximum Size slider bar to set the cell size by specifying the maximum cell size.
  - Move the Minimum Number slider bar to set the cell size by specifying the minimum number of cells.
- Move the Smoothing slider bar to set the number of additional grid cells by adding cells to optimize the length ratio of neighboring cells, and reduce aspect ratios in the selected direction. You can choose between using the V3 (default) or V2 grid smoothing algorithm, however, the V3 smoothing algorithm should be used in preference.

## Related Topics

[Grid Suitability](#)

[System Grid Property Sheet](#)

[Grid Smoothing](#)

# Solution Grid Property Sheets and Dialog Boxes

---

GUI interfaces used to define the spatial solution grid.

The [Grid Constraint Property Sheet](#) is described in the *Simcenter Flotherm Project Attributes Reference Guide*. Grid information is also shown in the [Project Manager Status Bar](#).

<b>System Grid Property Sheet .....</b>	<b>328</b>
<b>Grid Summary Dialog .....</b>	<b>330</b>
<b>Grid Changed Dialog Box .....</b>	<b>332</b>
<b>Previous Software Version Dialog Box .....</b>	<b>333</b>

## System Grid Property Sheet

To access: Click the **System Grid** tool icon or select **System > System Grid** in the data tree.

Use this property sheet to refine the solution grid by setting a predefined standard grid type (coarse, medium, or fine) or manually changing the grid by successive degrees of grid density.

### Description

The property sheet is intended to make the grid-defining process as automatic as possible. However, it is still likely that grid constraints attached to objects may have to be used if object or local grid refinement is required. Users are advised to attach object constraints (see [Setting an Object Grid Constraint](#)) and then use the System Grid property sheet.

### Objects

Object	Description
<b>None, Coarse, Medium, and Fine</b> buttons	Sets the grid in all directions using standard preset grid constraints. <ul style="list-style-type: none"><li>Click <b>None</b> to reset all the system grid settings, so only the basic system grid remains.</li><li>Click <b>Coarse</b>, <b>Medium</b>, or <b>Fine</b> to reset the slider bars to predefined locations. These locations reference lookup table % values that will result in the requested grid densities for the majority of model types.</li></ul>
Direction	<ul style="list-style-type: none"><li>X, Y, Z or Override All – Sets the direction in which the change of grid constraints are to apply.</li></ul> <p>The Minimum Size and Min. No./Max. Size settings reflect a percentage of the solution domain in the direction selected.</p> <p>When Override All is selected, the displayed sizes reflect the settings for the last visit to a specified direction. Therefore, if the solution domain is different in size for the three directions, the size shown will only be reflected exactly for one direction, tending to produce cells of the same aspect ratio as the solution domain itself.</p>
Minimum Size	Sets the minimum size of a grid cell either by value or slider bar position (see Usage Notes).
Min. No./Max. Size	Sets the cell size by specifying either the maximum cell size or the minimum number of cells. Select the option from the dropdown list, then either enter the value or use the slider bar (see Usage Notes).
Smoothing	Check to apply grid smoothing.
Version	(Smoothing) Choose between the default V3, or V2 grid smoothing algorithms. The V2 algorithm remains available so that projects created in V2 that use smoothing can be represented in V3 without imposing a grid change.

Object	Description
Smoothing Slider Bar	(Smoothing) Sets the number of additional grid cells by adding cells to: <ul style="list-style-type: none"> <li>• Optimize the length ratio of neighboring cells, and</li> <li>• Reduce aspect ratios in the direction selected in the Direction dropdown list.</li> </ul> The grid is refined (more smoothing) as the slider bar is moved to the right. The amount of smoothing is used for all grid spaces.
Maximum Aspect Ratio	Displays the size of the largest cell aspect ratios. For example, the location of the largest X/Z aspect ratio is at the same location as the smallest Z-cell.
Total No. Cells	Displays the total number of cells in the project.

## Usage Notes

### Minimum Size

When the slider bar is moved, a lookup table containing a list of % values is referenced. Minimum cell size is then determined by multiplying the solution domain size by the % value as the slider is moved. The lookup table is ordered so that when the slider bar is moved towards the left-hand side, a coarser grid is created (that is, a larger minimum cell size); when moved towards the right, a finer grid is created (that is, a smaller minimum cell size).

For any localized grid spaces, the smaller of its own value and its parent's value is used. (See Minimum Size under “Grid Constraint Property Sheet” described in the *Simcenter Flotherm Project Attributes Reference Guide*.) The system grid only sets the base grid value; it may set other grid space values if they do not override it.

### Minimum Number/Maximum Size

The lookup table referenced by the slider bar is ordered so that the grid is refined when the slider is moved towards the right-hand side. The maximum size, like the minimum size, is calculated by multiplying the lookup table % value by the length of the solution domain as the slider is moved. If the minimum number is selected and the slider moved, the value of the minimum number is read directly from the lookup table.

This value may be overridden by grid constraints (if they are finer).

## Related Topics

[Grid Smoothing](#)

## Grid Summary Dialog

To access: **Grid > Summary** or click the **Grid Summary** icon.

Use this dialog to check the grid statistics for the base solution domain grid or a localized grid.

### Objects

Field	Description
For Grid	Select the grid of interest: <ul style="list-style-type: none"><li>• Base Grid — Shows information for the base grid covering the outside of all objects.</li><li>• Objects with Localized Grid — Shows the statistics for a particular localized grid.</li></ul> See “ <a href="#">Grid Localization</a> ” on page 309 for an explanation of nesting of localized grid cells.
Total Grid Size	
Total	The total number of grid cells (that is, $NX \times NY \times NZ$ ) for the selected grid space.
Cumulative Total	The total number of cells for the selected space and any ‘child’ spaces ‘nested’ inside the currently selected grid space. Cells in a parent space occupied by a child space are not counted.
NX	The number of grid cells in the x-direction.
NY	The number of grid cells in the y-direction.
NZ	The number of grid cells in the z-direction.
Largest Aspect Ratios	
Aspect Ratio and Direction	The largest aspect ratios, in order of decreasing size. The Direction is long_side_axis/short_side_axis. For example, a cell face with a Direction of Y/Z is in the x-plane with its longest side parallel with the y-axis and its shortest side parallel with the z-axis.
Smallest Grid Cells	
Cell Size and Direction	The smallest grid cells, in order of increasing size. The dimensions are in units of Length set by the Global Units dialog box, but changes made using the Global Units dialog box are not updated here if the dialog box is open. The Direction is that of the smallest side of the cell.
Objects Responsible	

Field	Description
1 and 2	Object(s) whose keyplane is responsible for the currently selected large aspect ratio or small cell. <ul style="list-style-type: none"><li>• One object — Implies that the other responsible object is the grid space.</li><li>• No objects — Implies that the grid space is responsible.</li></ul>
1 and 2 check boxes	Click a check box to highlight the object in the data tree (if below a topped assembly) and the drawing board.
Zoom To	Click to zoom to the selected cell (either a large aspect ratio cell or a small cell) in the drawing board. Any views showing the cell are zoomed in. If an object responsible is also checked then the zoom includes that object.

## Usage Notes

- Select a cell in the Largest Aspect Ratios or Smallest Grid Cells list to highlight the cell in the drawing board. If the grid is not currently being shown then it will be set to be shown.
- To unselect a selected cell, press Ctrl+Click the selected list item.
- To dock the window under the drawing board, double-click the title bar.
- The **Help** icon for this dialog is only visible when the dialog is undocked.

---

### Tip

 When using Zoom To, set Grid View for the drawing board to make sure that the selected cell is visible.

---

## Related Topics

[Grid Suitability](#)

[Debugging the Grid of Large Aspect Ratio Cells](#)

## Grid Changed Dialog Box

To access: Opens when results are stored but a change has been made that can cause a change to the grid.

Results may be interpolated because of a change in the grid. Use this dialog box to continue using the current project or create a new project.

### Description

The dialog box has three buttons.

### Objects

Object	Description
Continue using the existing project	Click to apply all changes, including the interpolated results, to the current loaded project.
'Save As' the current project with a new name to continue	Click to leave the current project unchanged, but display the Save Project dialog box to save the project, along with the interpolated results, in a new project.  The new project becomes the current loaded project.
Defer decision until later	Click to continue using the loaded project without changing the grid.   <b>Note:</b> Even if you click this button, the grid will change if you solve the project.

### Related Topics

[Spatial Solution Grid Overview](#)

## Previous Software Version Dialog Box

To access: Opens when a model, which includes results, has been loaded from a pre-V12.0 version of Simcenter Flotherm, and a change has been made that will cause a change to the grid.

Results may be interpolated because of a change in the grid. Use this dialog box to continue using the current project, which will no longer be backward compatible, or create a new project.

### Description

The dialog box has three buttons.

### Objects

Object	Description
Update the project. The project cannot subsequently be opened in the earlier version.	Click to continue using the current project.  <b>Note:</b> The project will not be compatible with earlier versions of Simcenter Flotherm.
Save the project with a new name then update the new project. The original project and results will not be updated and can be opened in the earlier version.	Click to continue with a newly created copy of the project. The original project is retained and the new project becomes the current loaded project.
Cancel Load	Click to unload the current project, and load the default project.

### Related Topics

[Spatial Solution Grid Overview](#)



# Chapter 9

## Transient Analysis

---

Use a transient analysis for cases where the alteration of flow and heat transfer with time is of interest (for example, for such events as a power up).

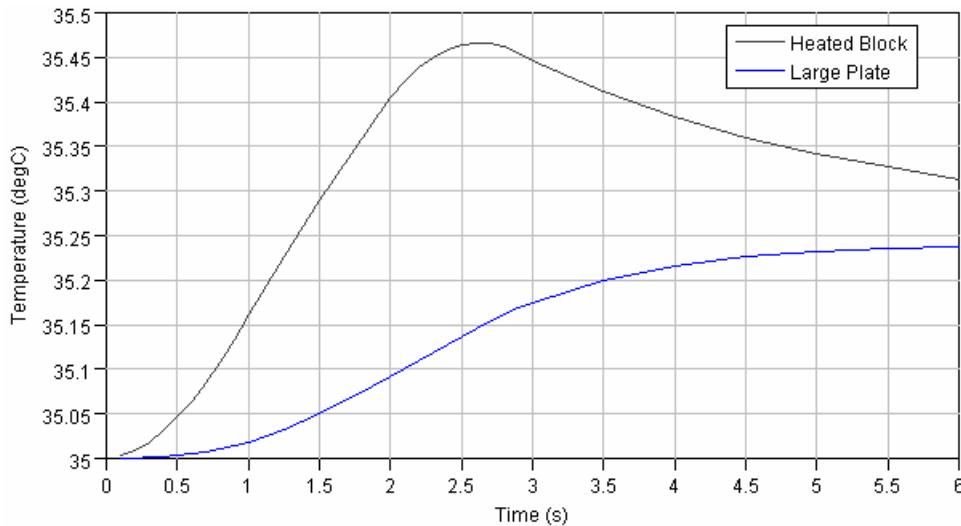
<b>Transient Analysis Introduction</b> .....	<b>335</b>
<b>Total Transient Time and the Transient Solution Period</b> .....	<b>336</b>
<b>Time Steps and the Time Distribution Plot</b> .....	<b>337</b>
<b>Transient Analysis Residuals</b> .....	<b>339</b>
<b>Transient Analysis Resolution</b> .....	<b>340</b>
<b>Time Patches</b> .....	<b>342</b>
<b>Time Step Distribution Types</b> .....	<b>343</b>
<b>Transient Keypoints and Keypoint Tolerance</b> .....	<b>344</b>
<b>Transient Functions (Time-Varying Attributes)</b> .....	<b>344</b>
<b>Save Times</b> .....	<b>345</b>
<b>Transient Analysis Procedures</b> .....	<b>346</b>
Obtaining Initial Data From Steady-State Analysis .....	346
Setting Up a Transient Analysis Time Grid .....	346
Creating Time Patches .....	347
Setting Save Times .....	348
Solving a Transient Analysis .....	349
Viewing Plots of Transient Save Time Data .....	350
Viewing Transient Save Time Data in Tables .....	350
<b>Transient Solution Dialog Box</b> .....	<b>352</b>

## Transient Analysis Introduction

In transient simulations, the settings of the initial conditions are part of the problem specification, since earlier-time values influence later-time values. This is not the case for steady state simulations, where the initial conditions only affect the speed at which the solution converges. The use of Monitor Points is fundamental to transient analyses, because Monitor Points enable you to observe how variables change over time.

Monitor Point values are calculated at the end of each Time Step and are used to create curves of Monitor Plot versus Time. The time steps determine the resolution of the plot.

**Figure 9-1. Transient Analysis of Temperature Versus Time for Two Monitor Points**



## Total Transient Time and the Transient Solution Period

The Total Transient Time is the total time over which you want to perform a transient analysis. The Solution Period is the period over which the solver runs. To examine 3D results at intervals over the total transient time, specify a solution period that is shorter than the total transient time.

After the end of the first run (solution period) you can examine the results in Analyze mode, then start the solver and it will run for the next solution period using the end conditions of the first run for the initial conditions of the next run.

**Figure 9-2. Sequence of Short Solution Periods**

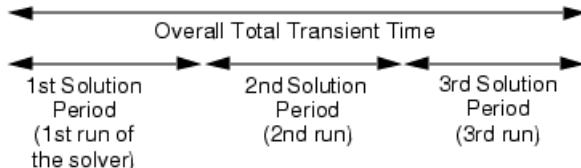


Figure 9-3 shows how this is set up on the Transient Solution dialog box. The example is of a 12-second total transient time divided into three four-second solution periods.

### Note

The Period Start Time is the start time of the *previous* run and is automatically updated at the end of each solver run.

The Period Duration is the duration of the *next* run and is user-defined. For example, the 12 second total time could have been divided into runs of duration 3, 4 and 5 seconds.

**Figure 9-3. Period Start Time Updating in Transient Solution Dialog Box**

Total Start Time	0	s
Total End Time	12	s
Period Start Time	0	s
Period Duration	4	s

Display After 1st Solver Run

Total Start Time	0	s
Total End Time	12	s
Period Start Time	4	s
Period Duration	4	s

Display After 2nd Solver Run

Total Start Time	0	s
Total End Time	12	s
Period Start Time	8	s
Period Duration	4	s

Display after 3rd (Final) Solver Run

The plots in the Profiles window are updated at the end of each solver run.

## Time Steps and the Time Distribution Plot

Monitor Point variable values are calculated at the end of each Time Step.

Time Steps are represented by blocks on the Time Step Distribution Plot ([Figure 9-4](#)), which is part of the Transient Solution dialog box. This plot is a 2D visualization of a 1D plot ([Figure 9-5](#)), the y-axis of the 2D plot showing the duration of the time step.

---

**Note**

 Plots that contain more than 10,000 lines are not shown. In these cases, the Time Step Distribution Plot is empty.

---

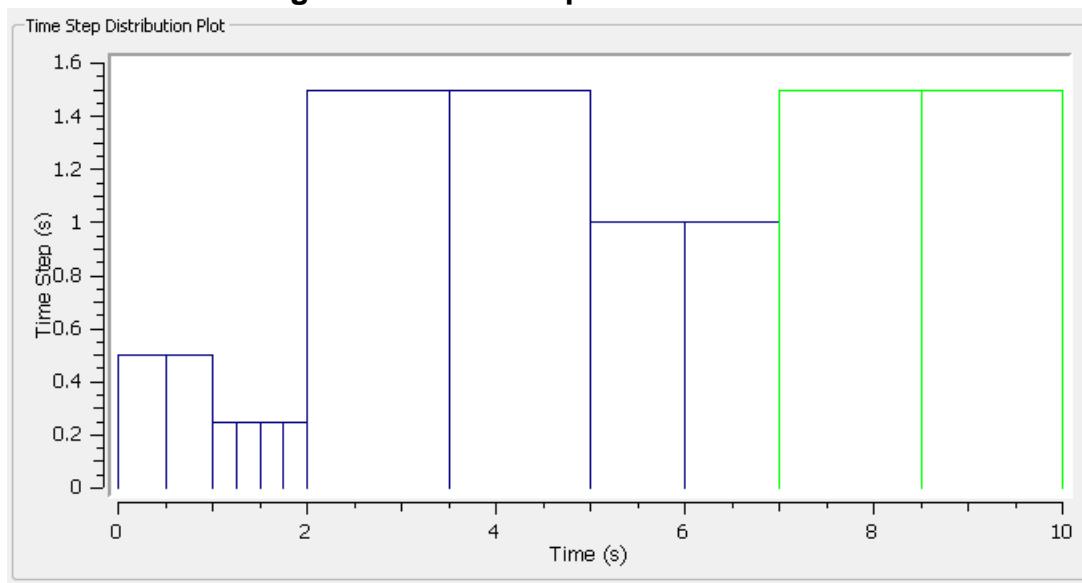
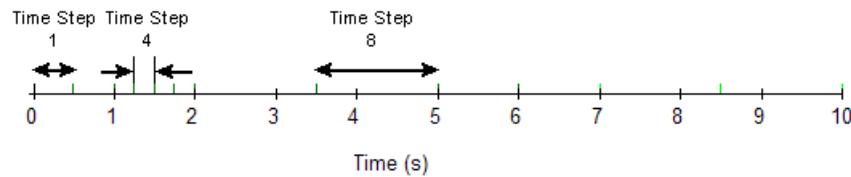
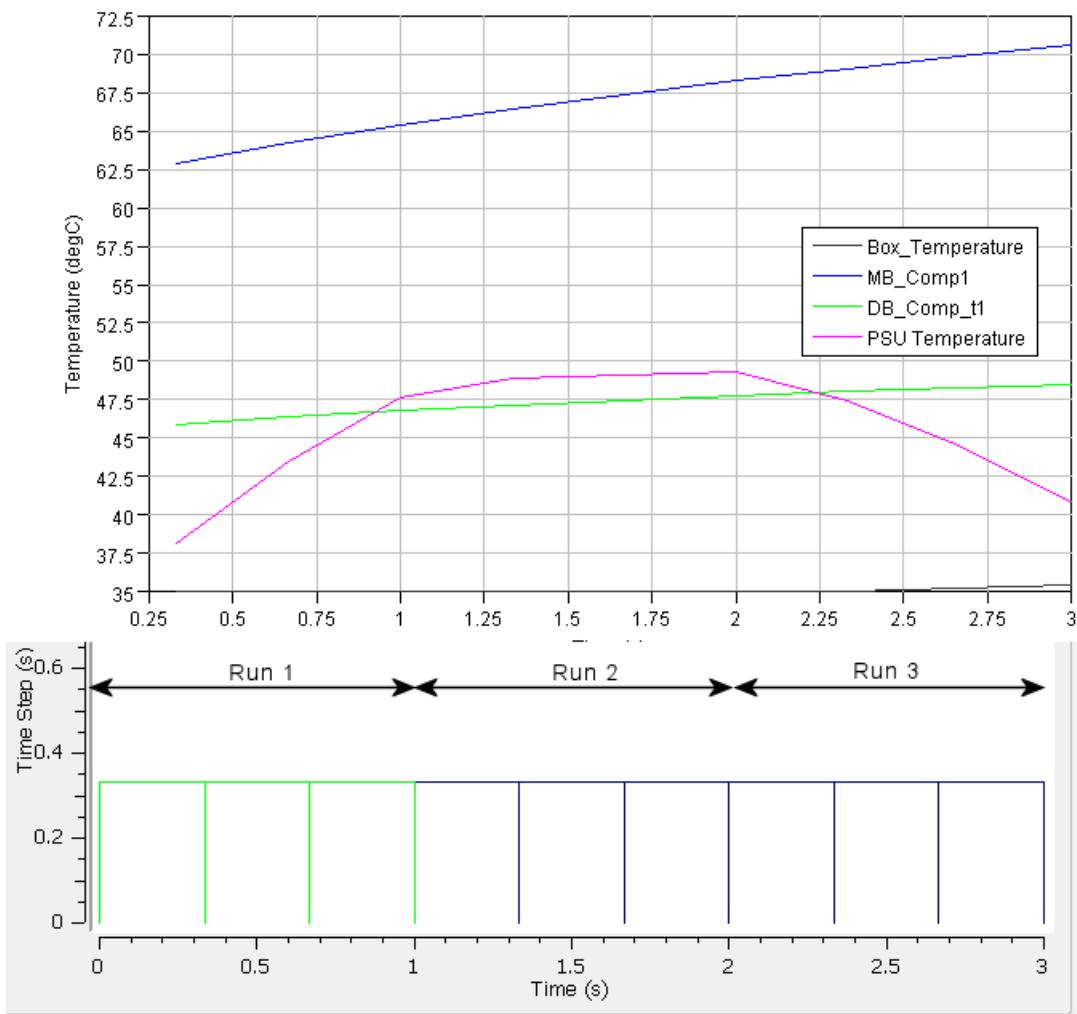
**Figure 9-4. Time Step Distribution Plot****Figure 9-5. Time Steps Along a 1D Time Axis**

Figure 9-6 shows a Monitor Points versus Time Plot and the corresponding Time Step Distribution Plot. In this example, the 3-second total transient time was divided into solution periods, each of 1 second, and the solver was restarted at the end of the first and second seconds. Each solution period was divided into three equal time steps, therefore, values were calculated for times 0.33, 0.66, 1.0, 1.33, 1.66, 2, 2.33, 2.66, and 3 seconds.

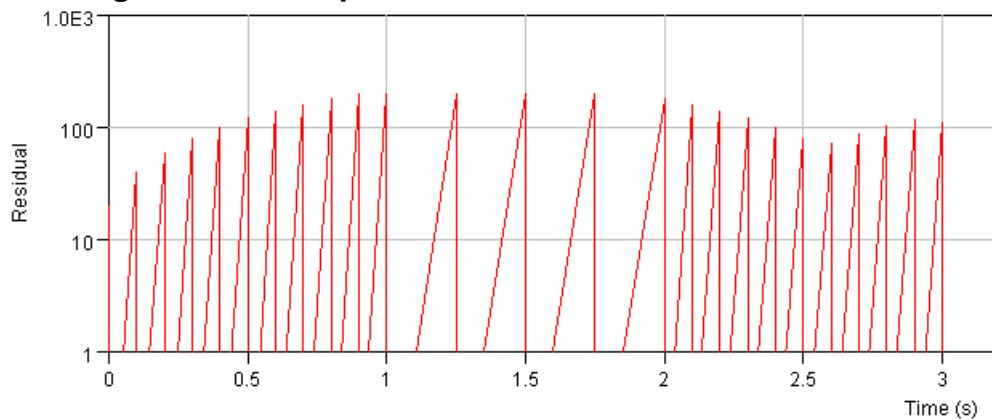
**Figure 9-6. Monitor Point Values at Each Time Step**

## Transient Analysis Residuals

For maximum accuracy, convergence must be achieved for each time step. Solution errors remaining from incomplete convergences are propagated (and possibly enlarged) from one time step to the next.

**Figure 9-7** is a detail of a residuals plot showing that residual errors are not being propagated from one time step to another.

**Figure 9-7. Example of Non-Accumulation of Residuals**

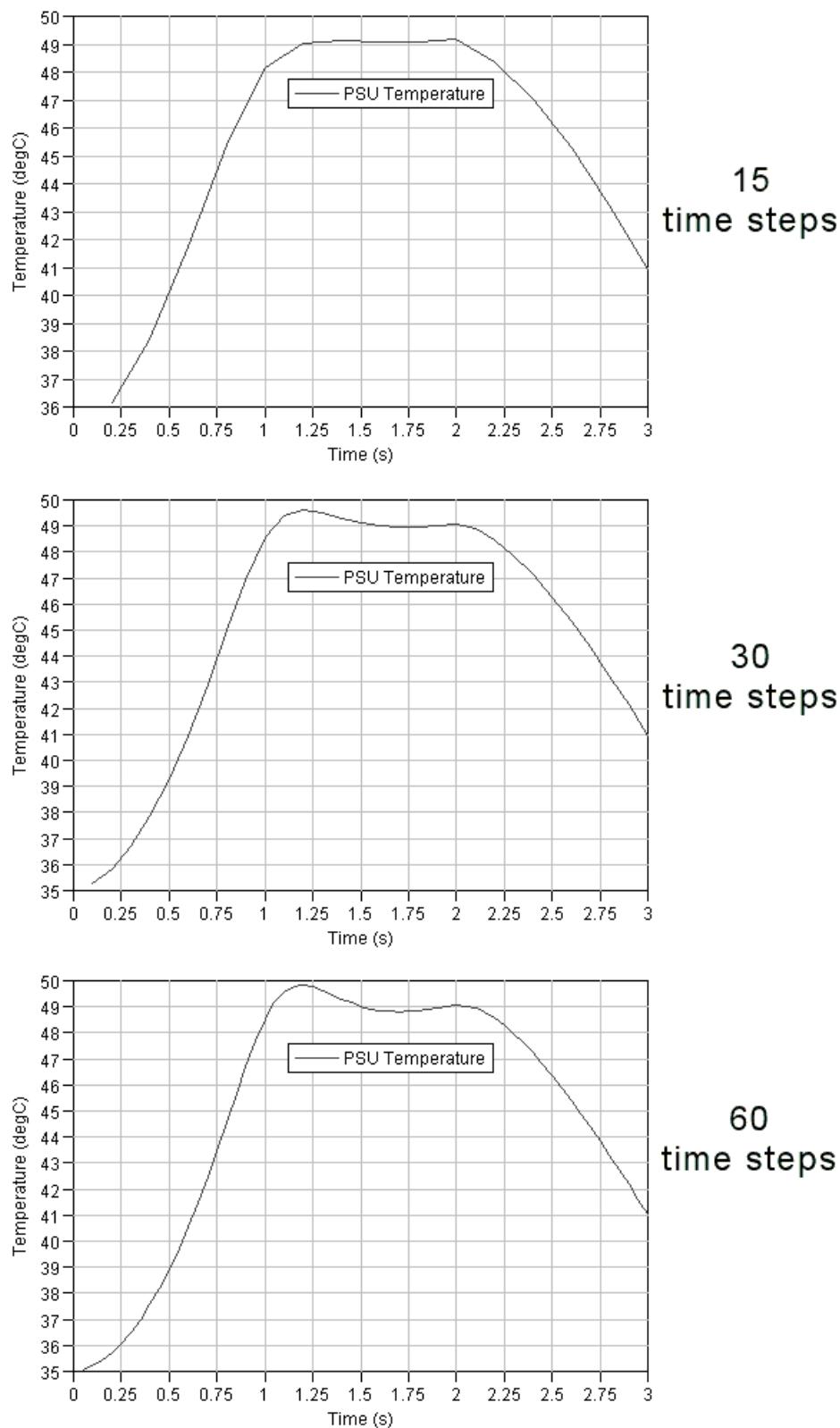


## Transient Analysis Resolution

As you use the transient analysis tool you will gain experience on how to tailor the time steps to get a good resolution without excessive computational time.

Figure 9-8 shows the effect on the accuracy of a transient analysis by changing the number of time steps. In this example, a ramp and fall transient function was applied to a PSU, and a monitor point added to the PSU.

**Figure 9-8. Transient Analysis Resolution With 15, 30 and 60 Time Steps**

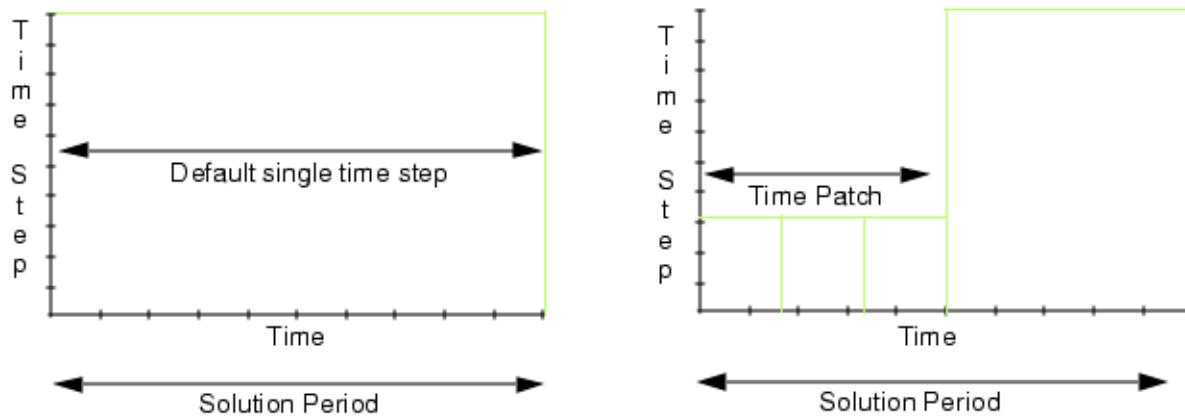


## Time Patches

By default, there is one time step which corresponds to the entire solution period. Time Steps are added by adding Time Patches. A Time Patch is made up of one or more Time Steps.

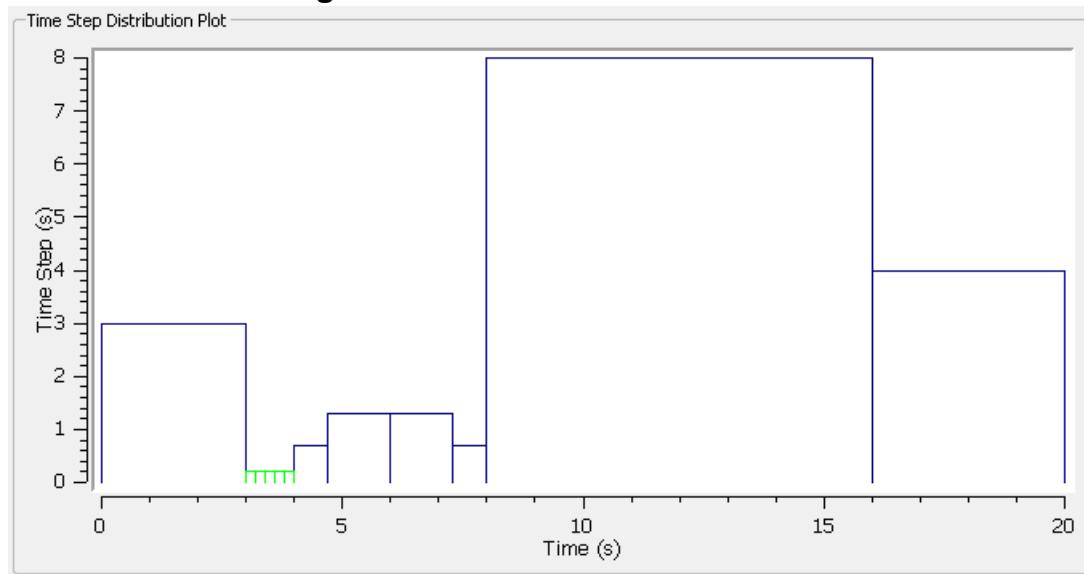
Time Patches allow for the entire transient to be simulated in a sequence of time intervals in each of which a distinct calculation is performed. The initial conditions for interval n is the solution obtained at the end of interval n-1. The start time of an interval and its elapsed time (that is, duration of the interval) define the integration time. [Figure 9-9](#) shows how the addition of a time patch has introduced more time steps.

**Figure 9-9. Additional Time Steps**



[Figure 9-10](#) shows an example of a highly refined time grid.

**Figure 9-10. Local Grid Refinement**

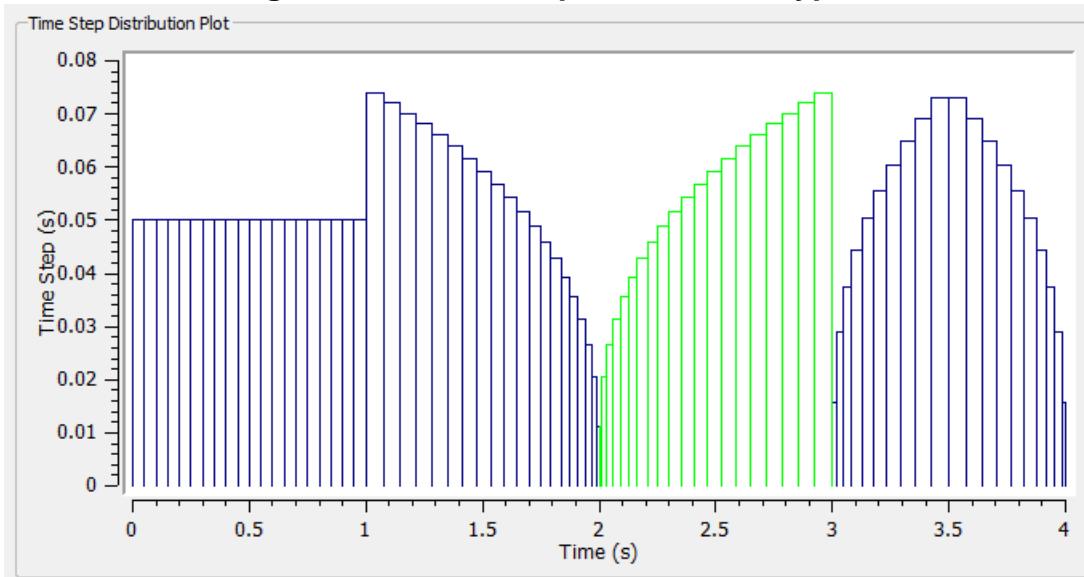


# Time Step Distribution Types

There are four possible distribution profiles of time steps within a time patch. These profiles enable you to vary the length of a time step. Often small time steps are required near the start of the transient and longer time steps towards the end of the transient, for example, in a heat-up or cool-down scenario where the greatest change in temperature will most-likely occur at the beginning of the time period, the Increasing Power distribution type is ideal for this.

- Uniform distribution type — the time steps are of equal duration, as shown from 0-to-1 s in [Figure 9-11](#).
- Increasing Power distribution type — the time steps become longer with time, as shown (highlighted) from 2-to-3 s in [Figure 9-11](#).
- Decreasing Power distribution type — the time steps become shorter with time, as shown from 1-to-2 s in [Figure 9-11](#).
- Symmetrical Power distribution type — the time steps progressively increase and then progressively decrease around the mid-time, as shown from 3-to-4 s in [Figure 9-11](#). An even number of time steps is enforced for this distribution type.

**Figure 9-11. Time Step Distribution Types**



An index sets the rate at which the time step duration increases or decreases when one of the Power distributions is selected such that:

$$T_{end(n)} = (n/N)^{index} * patch\_duration$$

where  $N$  is the total number of steps in a patch,  $n$  is the nth step in the patch, and  $T_{end(n)}$  is the time at the end of the nth step.

The duration of the nth step is:

$$\text{time\_step\_duration}_{(n)} = T_{\text{end}(n)} - T_{\text{end}(n-1)}$$

## Related Topics

[Creating Time Patches](#)

# Transient Keypoints and Keypoint Tolerance

Transient keypoints locate the solution time steps. By default, there is one time step, with a keypoint at the start and at end of the solution period.

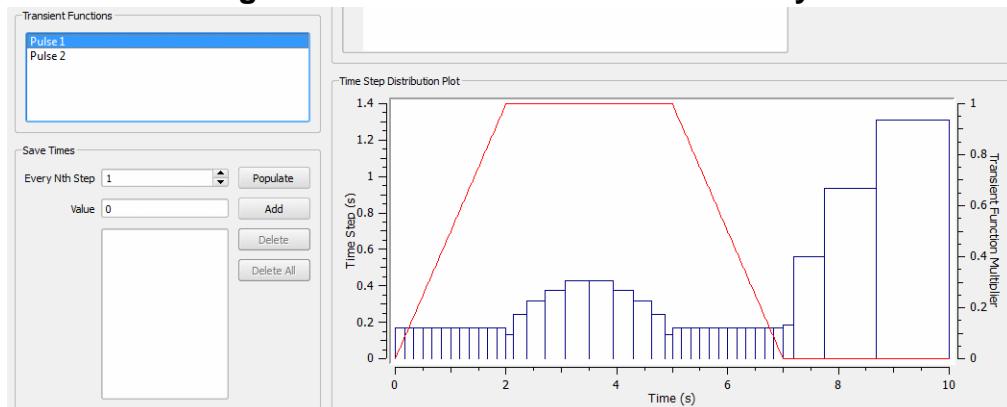
A grid defined using patches is imposed onto the system independently of any existing time grid, and this can result in the creation of small time steps. The Keypoint Tolerance enforces a limit on the smallest time step that can be created, suppressing unnecessary time steps that might otherwise be created.

# Transient Functions (Time-Varying Attributes)

You can set time-varying Thermal, Source, or Ambient attributes by attaching a Transient attribute that modifies the host attribute by a multiplication factor.

These Transient Functions can be overlaid on the Time Step Distribution Plot. When selected, these functions are displayed in red, and the multiplication factor is identifiable by reading the right-hand axis.

**Figure 9-12. Transient Function Overlay**



Such overlays provide help for setting up time steps. In this example, a combination of Uniform, Symmetrical Power and Increasing Power distribution types have been used to more closely monitor changes due to the transient function that is being applied.

## Related Topics

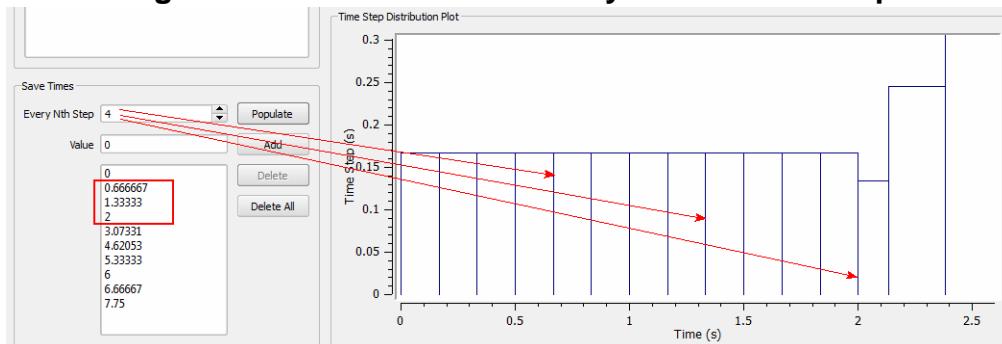
[Transient Attributes \[Simcenter Flotherm Project Attributes Reference Guide\]](#)

# Save Times

3D field data of transient analyses can be recorded at times other than the end of solution and will then be available for post-processing in Analyze mode.

The data is saved at “Save Times”, which can be specified in terms of time steps, at a specific time, or a combination of both.

**Figure 9-13. Save Times at Every Fourth Time Step**



After a solution has completed, the results are available in Analyze mode, but derived properties, such as flow rates and heat fluxes that are available for steady-state analyses, are not available for transient analyses.

## Related Topics

[Viewing Plots of Transient Save Time Data](#)

[Viewing Transient Save Time Data in Tables](#)

# Transient Analysis Procedures

---

Setting up, solving, and viewing results of a transient analysis.

<b>Obtaining Initial Data From Steady-State Analysis.....</b>	<b>346</b>
<b>Setting Up a Transient Analysis Time Grid .....</b>	<b>346</b>
<b>Creating Time Patches .....</b>	<b>347</b>
<b>Setting Save Times.....</b>	<b>348</b>
<b>Solving a Transient Analysis.....</b>	<b>349</b>
<b>Viewing Plots of Transient Save Time Data .....</b>	<b>350</b>
<b>Viewing Transient Save Time Data in Tables.....</b>	<b>350</b>

## Obtaining Initial Data From Steady-State Analysis

It is advisable to run a steady-state analysis before running a transient analysis to obtain a good level of convergence. Errors can accumulate over a transient solution, and a non-convergent case will take significantly longer to solve when run in transient mode.

### Procedure

1. Assign power levels, ambient conditions, flow rates, and so on as you would expect the model to be at Time = 0.  
  
For example, in a heat-up scenario all ambient conditions would be at room temperature and all powers at 0 W.
2. Save this case and give it a relevant name, for example, “Simulation at Time = 0”.
3. Solve as a steady-state analysis.
4. Save this case under a new name, for example, “Simulation over Time”.

## Setting Up a Transient Analysis Time Grid

The transient analysis grid is a balance between accuracy and computational time.

### Procedure

1. Either choose **Model Setup > Transient Solution** or, in the **Model Setup** tab, check Transient Solution and click the **Click to Edit** button.  
  
The Transient Solution dialog box is opened.
2. In the Overall Transient Time section:
  - a. Set the Total End Time to set the duration during which you would like to obtain data.

Later, the simulation can be extended and re-solved if more time is needed.

- b. Set the Period Duration to the same time interval.
3. Divide the time grid into time steps, as shown in “[Creating Time Patches](#)” on page 347:

## Related Topics

[Time Steps and the Time Distribution Plot](#)

[Transient Solution Dialog Box](#)

# Creating Time Patches

Time steps are created by adding and defining time patches. A time patch is made up of one or more time steps.

## Procedure

1. Choose **Model Setup > Transient Solution** to open the Transient Solution dialog box.
2. Use the Time Patches pane to create and configure time patches.

If you want to...	Do the following:
Use transient functions.	<p>Click <b>AutoCreate</b> to automatically create time patches based on those functions currently attached to the project.</p> <p>If there are none, then a single time patch is created. If there are transient functions, then time patches are created for each period (if the function is periodic) or for each subfunction (if the function is subdivided).</p> <p>The time patches are also divided into time steps dependent on the settings in the Number of Steps Control and Time Step Distribution frames.</p>
Adjust start/end times or edit time patch names.	Add time patches by clicking the plus (+) button, adjust start/end times if necessary, and edit time patch names by selecting them.
Modify a time patch.	Select the patch.  <b>Note:</b> Selected time patches are colored green in the Time Step Distribution Plot. <p>The parameters on the right of the patch list are those applicable to the currently selected patch.</p>

If you want to...	Do the following:
Select multiple time patches.	<p>Select the time patches, and press Ctrl+select or Shift+select.</p> <p> <b>Note:</b> When you select more than one time patch, data common to all of the selected time patches is shown, and changes made to common fields affect all selected time patches.</p> <p>Data fields that are not common are unavailable, and the values particular to the time patches are maintained.</p>

- When you have finished, click **OK** to close the Transient Solution dialog box.

## Related Topics

[Transient Solution Dialog Box](#)

[Time Patches](#)

[Time Step Distribution Types](#)

## Setting Save Times

When setting up a transient solution, you can optionally decide to save 3D data at specific times (Save Times) during the transient solver run.

### Procedure

- Choose **Model Setup > Transient Solution** to open the Transient Solution dialog box.
- In the Save Times pane.

If you want to...	Do the following:
Create a set of Save Times based on the value entered in Every Nth Step.	<p>Click <b>Populate</b>.</p> <p> <b>Note:</b> If you have entered a number greater than the total number of steps, only the time of the first step is saved.</p>
Add the time entered in the Value field to the list.	<p>Click <b>Add</b>.</p> <p>The value snaps to that of the nearest time step. For example, if the overall transient is 100 s long with time steps set at intervals of 20 s, attempting to add a Save Time at 38 s will create one at 40 s.</p> <p> <b>Note:</b> You cannot save data at any other times than the ends of time steps.</p>

## Results

The list of Save Times is updated as you use the **Populate**, **Add**, **Delete**, and **Delete All** buttons.

## Related Topics

[Save Times](#)

[Viewing Plots of Transient Save Time Data](#)

[Viewing Transient Save Time Data in Tables](#)

# Solving a Transient Analysis

When have set up the time step distribution, you can start a transient analysis. In practice you may need to repeat the analysis, adjusting the transient grid to obtain improving results.

## Prerequisites

- Before running a transient analysis it is good practice to start with a previous solution for data at Time = 0.
- Ensure that monitor points are present in your model in all of the areas of interest.
- Ensure that Transient Solution is switched on and that the overall parameters and distribution of time steps are set up.
- The density and specific heat capacity of each material used is important to allow heat to be stored within the cuboids and prisms. If you have defined any of your own materials check to see that you have provided this data, which is not needed for steady-state simulations. Similarly, an object will have to be uncollapsed in order to have any thermal mass, and to enable it to store heat over time.

## Procedure

1. Choose **Model Setup > Transient Solution** and make sure that Transient Solution is set.  
If the previous solution was a steady-state solution then the **Solution Type Changed** dialog box will be displayed. Select the relevant option from the dialog box.
2. Open the **Solver Control** tab.
3. In the Initial Variables section, select **Initial Value** of Selected Solution Set.
4. Click **Select** and select the project solution set created by a steady-state analysis. Click **OK** to close the Solution Set dialog box.
5. Choose **Solve > Re-Initialize** to read in the data from the selected solution set as a start condition.
6. Choose **Solve > Solve**.

## Results

The transient solution will need to solve for each of the time-steps defined. After two time steps have completed, the monitor point graph will appear showing the response at each defined point over time. The solution will normally take much longer than a steady-state simulation.

## Related Topics

[Obtaining Initial Data From Steady-State Analysis](#)

[Setting Up a Transient Analysis Time Grid](#)

[Solution Type Changed Dialog Box](#)

[Transient Solution Dialog Box](#)

# Viewing Plots of Transient Save Time Data

How to load Save Time transient data for viewing as plots when in Analyze mode.

## Prerequisites

One or more Save Times must have been defined before the transient solution.

## Procedure

1. Create a plot in Analyze mode.
2. Use the Transient Time Step Selector, located above the Results Tree, to select a Save Time.

See “[Transient Time Step Selector](#)” on page 423.

## Results

As you change the save time, the plot is updated with data corresponding to that time. The title in the GDA changes to show the new save time.

## Related Topics

[Save Times](#)

# Viewing Transient Save Time Data in Tables

How to load Save Time transient data for viewing tables in Analyze mode.

## Prerequisites

One or more Save Times must have been defined before the transient solution.

## Procedure

1. View a results table in Analyze mode.

2. Use the Transient Time Step Selector, located above the Results Tree, to select a Save Time.

See “[Transient Time Step Selector](#)” on page 423.

## Results

The table data changes to correspond with the save time.

## Related Topics

[Save Times](#)

# Transient Solution Dialog Box

To access:

- Choose **Model Setup > Transient Solution**.
- In the **Model Setup** tab, select Transient Solution, then click **Click to Edit**.

Use this dialog box to configure a transient analysis.

## Objects

Object	Description
Overall Transient	
The top section (above the horizontal line) is summary information. Use the lower section to set the total duration of the simulation, and the elapse period of the next solution run.	
On	Switches a Transient Analysis on. If switched off then the solver will perform a steady-state analysis.
Number of Steps	Displays the total number of time steps for the entire solution.
Smallest Step Size	Displays the duration of the smallest time step.
Smallest Step Location	Displays the time that the smallest time step occurs.
Total Start Time	The start time of the simulation.
Total End Time	The end time of the simulation.
Period Start Time	The start time of the <i>previous</i> solution and is automatically updated at the start of each solution.
Period Duration	The duration of the <i>next</i> solution period.
Keypoint Tolerance	The minimum time-span between the time steps. Transient keypoints locate the start and end of time steps forming the transient grid. Transient keypoints with a separation below this tolerance value will be combined.
Time Patches	
Time Patches Table	A table containing the names and start and end times of the grid patches currently defined. Selection of a row or number of rows populates the fields to the right of the table.
Steps Type	After adding a first step the following options are available: <ul style="list-style-type: none"><li>• Additional Steps to define the number of additional steps (for example, 3 if the total in the patch is to be 4).</li><li>• Minimum Number to define the minimum number of steps in the selected patch.</li><li>• Maximum Size to define the maximum step size in the patch.</li></ul>

<b>Object</b>	<b>Description</b>
Steps	(Additional Steps, Minimum Number) Either defines the additional number of time steps in the time patch or the minimum number of time steps in the time patch.
Step Size	(Maximum Size) Defines the maximum time step size.
Distribution Type	(Additional Steps, Minimum Number) Choose from Uniform, Increasing Power, Decreasing Power and Symmetrical Power.
Index	(a Power Distribution Type) Determines the rate of increase/decrease of power distribution types.
Transient Functions	A list of transient functions currently in use.
Save Times	
Every Nth Step	A positive integer that defines the data to be saved at each Nth time step. The default is 1. When specifying Save Times in this way, the data is always saved after the first time step and then at each Nth step thereafter. For example, for a set of eight time steps, if Every Nth Step is set to 2, data would be saved after the 1st, 3rd, 5th, and 7th time steps.
Value	Specifies a new Save Time to be added to the list.
List of Save Times	A list of times that data will be saved. Note, that if the times specified do not fall on a time step during the transient solution, then the data will be saved at the closest possible time step value.
Time Step Distribution Plot	<p>A graphical representation of the time grid. There is a limit of showing up to 10,000 lines.</p> <p>The plot enables rubber band zooming.</p> <ul style="list-style-type: none"> <li>• To undo a zoom, click the Middle mouse button.</li> <li>• To redo an undone zoom, click Shift+Middle mouse button</li> <li>• To reset the plot to its original zoomed-out state, click the Right mouse button</li> </ul>

## Usage Notes

If the project has already solved, swapping the solution types causes the solution data to be re-initialized. As a precaution, on confirming the settings, a dialog box enables you to save the new settings to a different project.

## Related Topics

[Total Transient Time and the Transient Solution Period](#)

[Transient Keypoints and Keypoint Tolerance](#)

[Creating Time Patches](#)

[Time Step Distribution Types](#)

[Transient Functions \(Time-Varying Attributes\)](#)

[Setting Save Times](#)

[Time Steps and the Time Distribution Plot](#)

# Chapter 10

## Solving the Project

---

The use of the solver in interactive and batch modes.

<b>Solver Overview</b> .....	<b>356</b>
<b>Monitor Points</b> .....	<b>356</b>
<b>Pre-Solve Checks</b> .....	<b>357</b>
<b>Exchange Factors</b> .....	<b>358</b>
<b>Interactive Solve Procedures</b> .....	<b>359</b>
Obtaining Maximum Solver Performance .....	359
Sanity Checking .....	359
Solving in Interactive Mode .....	360
Re-Initializing Variable Fields .....	361
Interrupting and Resuming an Interactive Solution .....	361
Aborting an Interactive Solution.....	362
Setting Solution Process Priority Using the Task Manager .....	362
<b>Batch Mode Solve</b> .....	<b>363</b>
<b>Files Used Before, During, and After Batch Solves</b> .....	<b>364</b>
Batch Files .....	364
logit File .....	364
floerror.log File .....	365
Output CSV Files .....	366
<b>Batch Solve Procedures</b> .....	<b>367</b>
Solving in Batch Mode .....	367
Using batchSolve to Solve in Batch Mode on Linux Systems .....	368
Environment Setup on Linux Systems .....	368
Monitoring the Solution in Batch Mode .....	368
Interrupting a Batch Solution .....	369
Setting up a Batch Mode Completion Email Notification .....	369
Setting Solution Process Priority Using an Environment Variable .....	370
<b>Transient Analysis Cases</b> .....	<b>371</b>
<b>Solve Dialog Boxes</b> .....	<b>373</b>
Solver Progress Dialog Box .....	374
Exchange Factors Calculation Progress Dialog Box .....	375

## Solver Overview

The Simcenter Flotherm solution activates the CFD algorithms which provide an integration of the fluid flow and heat transfer equations within the solution domain.

The solution may be monitored in profile plots of the convergence of residual errors and/or variable values at monitored points.

The solution can be run as an interactive process or in batch mode.

### Related Topics

[Solution Monitoring and Profile Plots](#)

## Monitor Points

Before starting the solution, set monitor points at locations where you want to monitor the variation of the solution variables.

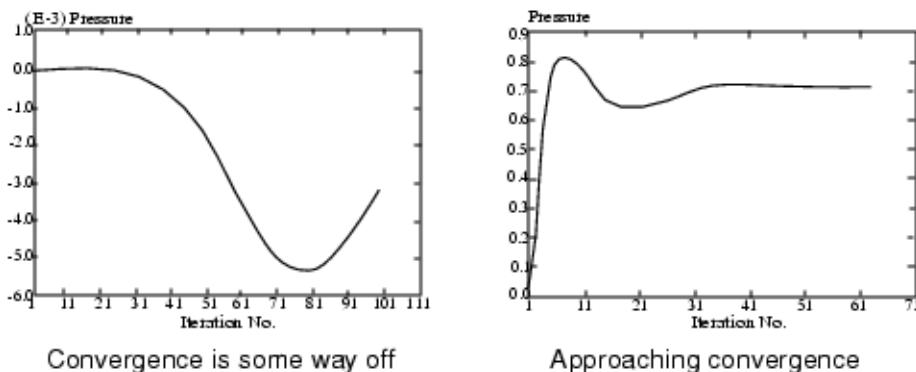
The accuracy of position is dependent on the grid, the value of the solution variable is that calculated for the cell containing the monitor point.

Although not geometry, monitor points are treated as part of the geometry structure, see [Monitor Points](#) in the *Simcenter Flotherm SmartParts Reference Guide*.

### Monitoring Variables in Steady State Solutions

In steady-state simulations, you can plot field values at the monitor point locations as a function of outer iteration, see “[Creating New Plots](#)” on page 396. This provides you with an alternative method of investigating convergence to that provided by viewing the plot of residuals against iteration. For example, if the pressure at a monitor location is shown to change with iteration count, then convergence may yet be some way off; but if the pressure is approaching a constant value, then convergence at this location has nearly been achieved, see [Figure 10-1](#).

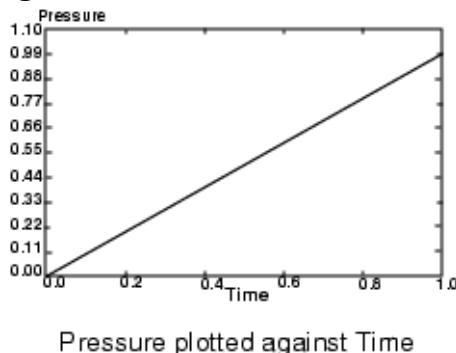
**Figure 10-1. Approaching Convergence**



## Monitoring Variables in Transient Solutions

In transient simulations, you can plot the field values at the monitor point locations as a function of time, see “[Creating New Plots](#)” on page 396.

**Figure 10-2. Transient Solutions**



The fields stored in program memory are those of the last time-step solved. Consequently, contour and vector plots can only show you what the fields look like at the last time-step. To see how things change with time, you have to profile the monitor values.

Be careful to set up monitor points at all locations for which you need data. Data from previous time intervals will only be available if a monitor point was present at that time.

## Displaying Transient Results

Before solution, if you set up a list of times using the Save Times frame of the [Transient Solution Dialog Box](#), then after solution you can display the field data at different snapshots in time when in Analyze mode. By default, only the field data at the end of the solution is shown.

# Pre-Solve Checks

Program checks are carried out before starting a solution.

- Model changes

Solve checks for any unsaved project changes. If any changes are detected, these must be saved prior to solving. For new projects, the Save Project dialog box is opened to name and save the project, if you cancel this dialog box, the solve is abandoned. Any changes found in projects already saved are written automatically over the already stored version.

- Data consistency and data errors

If problems are detected, messages are displayed. Those flagged as errors must be corrected before continuing with the solution. Warning and informational messages may also appear, but the solution will continue.

- Exchange factor calculations

Radiation models require the automatic calculation of the exchange factors before solving the equations if certain conditions have changed, see “[Exchange Factors](#)” on page 358.

- Solar

Models accounting for solar radiation have to determine where the solar radiation falls. This is done before the main solve calculation if:

- The project is to be solved for the first time, or
- The project is considered to have changed in some way, for example, a geometry change.

## Exchange Factors

Exchange Factors are applicable to radiation calculations and are the fraction of uniform diffuse radiant energy leaving one surface that is absorbed by a second surface.

Exchange Factors are generated by the Exchange Factor Generator (EFG).

The calculation of exchange factors is only available when the radiation model has been switched on (**Model Setup** tab), then the **Exchange Factors** icon changes color from gray to green.

The exchange factors can be generated by clicking the **Exchange Factors** icon, however, they are also generated automatically when:

- Solving for the first time without having previously calculated the exchange factors,
- Solving when the grid, the number of radiative surfaces, or the turbulence model has changed since the exchange factors were last calculated, or
- Solving when any attachment to any object has been modified (for example, material property, surface attribute, thermal attribute). In this case, you are prompted to re-calculate the exchange factors because the emissivity of a surface may have changed.

---

### Note

 The exchange factor process always starts for radiation-enabled models, but does not calculate the exchange factors if they are up to date. In such cases, the process informs the user by issuing message I/8010.

---

If you require the view factor values, then, before starting the calculation of the exchange factors, request the generation of the View Factor Log.

For further information, see [Radiative Exchange](#) and [View Factor](#) in the *Simcenter Flotherm Background Theory Reference Guide*.

# Interactive Solve Procedures

Tasks used when using the solver interactively.

<b>Obtaining Maximum Solver Performance</b> .....	<b>359</b>
<b>Sanity Checking</b> .....	<b>359</b>
<b>Solving in Interactive Mode</b> .....	<b>360</b>
<b>Re-Initializing Variable Fields</b> .....	<b>361</b>
<b>Interrupting and Resuming an Interactive Solution</b> .....	<b>361</b>
<b>Aborting an Interactive Solution</b> .....	<b>362</b>
<b>Setting Solution Process Priority Using the Task Manager</b> .....	<b>362</b>

## Obtaining Maximum Solver Performance

Hyper-Threading, under-use of parallel processing, and the double-precision solver can all have an adverse effect on solver performance.

### Procedure

1. Deactivate Hyper-Threading. Hyper-Threading is usually set in the BIOS. Consult your System Administrator if unsure how to enter the BIOS. Note: if Hyper-Threading is active, then the Task Manager will show twice as many cores as exist on your CPU.
2. Set Number of Processors to Use in the User Preferences dialog box to fully utilize the number of processors on your machine.
3. By default, the Double Precision Solver check box in the **Solver Control** tab is unchecked as it is not necessary in most cases. Ensure this is unchecked to optimize solver performance.

### Related Topics

[Double-Precision Solver](#)

## Sanity Checking

As the model is refined, it is good practice to regularly run sanity checks before running the solver.

### Procedure

Choose **Solve > Sanity Check**.

## Results

Any Error, Warning, or Information messages are output to the Message Window, followed by a I/9031 message that gives total numbers of the three different message types. Error messages stop the solver from running and therefore must be addressed.

## Related Topics

[Messages](#)

# Solving in Interactive Mode

Real-time monitoring of the solver.

## Restrictions and Limitations

- Re-running a simulation causes all existing results for the model to be deleted before the solver begins running. To save the results from the previous run, re-name the project before running the simulation again.

## Procedure

To start the solver, either click the **Solve** icon or choose **Solve > Solve**.

## Results

Once the solution of equations is activated, the following actions occur:

- Progress is traced dynamically by the plotting of the residual error values against iteration in the Profiles window.
- If there are monitor points in the project, a plot of monitored variables for temperature, or pressure if temperature is not available, is also displayed.
- Any warnings or information messages from the solver are displayed in the Message Window.
- If Radiation is switched and exchange factors require calculation, then the Exchange Factors Calculation Progress dialog box is displayed.
- Progress of the solver is shown by the Solver Progress dialog box.

Solution continues until either:

- it is manually interrupted or,
- convergence or the number of overall outer iterations is reached, whichever is the sooner.

When the solver stops, the solution status is displayed in the Message Window, and the Project Manager enters Analyze mode, see “[Viewing Results](#)” on page 419.

If you want to revert back to the previous solution results, choose **Solve > Revert**.

## Related Topics

- [Solution Monitoring and Profile Plots](#)
- [Interrupting and Resuming an Interactive Solution](#)
- [Re-Initializing Variable Fields](#)
- [Solver Progress Dialog Box](#)
- [Exchange Factors Calculation Progress Dialog Box](#)
- [Obtaining Maximum Solver Performance](#)

# Re-Initializing Variable Fields

Re-initializing restores the fields stored in program memory, with the exception of radiation and solar field results, to defaults or any settings that have been made in the Initial Variables section of the **Solver Control** tab.

## Procedure

Choose **Solve > Re-Initialize**.

## Results

The program checks the project data for consistency and errors and produces a summary report of any problems that need correcting before solution can take place.

If the project contains valid solution data, you will be warned that initialization deletes the existing solution data, and given the option to continue.

When the initialization completes, a messages are displayed in the [Message Window](#).

# Interrupting and Resuming an Interactive Solution

The solver can be stopped at any time to see results, and then resumed from the last iteration.

## Procedure

1. To view results before the solver has stopped, click the **Interrupt Solve** icon.  
The solution pauses at the end of the current outer-iteration, and control is returned to the Project Manager in Analyze mode.
2. View the intermediate results.
3. To resume the solver calculations, click the **Solve** icon.  
The solver starts from the point it was paused.

## Aborting an Interactive Solution

You can stop the solver without recording anything from the current solve process so that you can revert back to the previously recorded solution results.

### Procedure

1. While an interactive solve is in progress, choose **Solve > Abort Solver Action**.
2. Provided that you want to continue aborting the solver, click **OK** in the confirmation dialog box.

### Results

- No solver results are saved.
- An error message is output to record that a user-initiated solver abort has occurred.
- If required, choose **Solve > Revert** to revert back to the previously recorded solution results.
- If you do not choose **Solve > Revert**, but then choose **Solve > Solve**, you will start the solver from its initial variable values, that is, it is equivalent to choosing **Solve > Re-Initialize and Solve**.

## Setting Solution Process Priority Using the Task Manager

On Microsoft Windows systems, you can change the solution process priority during a solve.

### Procedure

1. Display the Task Manager Processes panel by right-clicking in the task bar and choosing Start Task Manager from the popup menu, then click on the **Processes** tab, if not on view.

When Simcenter Flotherm is solving, a process, called solexe, is listed as a Windows Task Manager process.

2. Right-click solexe, select Select Priority and choose the priority level of Normal, Below Normal or Low.

A warning message will be issued. Make sure you choose only one of the above levels.

### Related Topics

[Setting Solution Process Priority Using an Environment Variable](#)

# Batch Mode Solve

Use batch mode to solve large projects that require long processing times, or to solve PDML project files.

While a batch job is running, you can examine the *logit* file to check the solution progress and interrupt it if necessary. You can also set up an email notification of completion.

## Extended Batch Support for PDML With Tables Output

Simcenter Flotherm .pdml project files can be solved from the command line.

The regions, monitor points and cuboid flux results data normally displayed in the Tables application window when run interactively, may be output to a CSV file by specifying the output directory. The command line syntax is as follows:

```
flotherm -b <full.pathname>/<project>.pdml -o <output_directory>
```

This command creates the appropriate project subdirectory in the directory from the user configuration.

## Related Topics

[Files Used Before, During, and After Batch Solves](#)

[Batch Solve Procedures](#)

# Files Used Before, During, and After Batch Solves

---

Batch, log and output files.

<b>Batch Files</b> .....	<b>364</b>
<b>logit File</b> .....	<b>364</b>
<b>floerror.log File</b> .....	<b>365</b>
<b>Output CSV Files</b> .....	<b>366</b>

## Batch Files

Batch files contain sequences of flotherm command lines.

File, *flobatch.bat*, is provided in folder <install\_dir>\flosuite\_v<version>\flotherm\WinXP\bin.

This file is set up to accept the name of the project to be solved and, if so desired, the project to be used for the initialization, as command line input. The syntax is:

flobatch <project\_to\_solve> [<project\_to\_initialize\_from>]

Save the batch file with the extension .bat into any directory.

Names are case-sensitive on UNIX systems.

For example (Windows), to solve two projects, My Project1 and My Project2, add the following lines:

```
call "C:\<path>\flotherm" -b \"My Project1\"  
call "C:\<path>\flotherm" -b \"My Project2\"
```

where <path> is:

<install\_dir>\flosuite\_v<version>\flotherm\WinXP\bin

## Related Topics

[flotherm](#)

## logit File

The *logit* file logs the linear equation solver performance for each variable.

[Figure 10-3](#) is an extract taken from the top of an example file.

**Figure 10-3. Extract From logit File**

```

Mentor Graphics Corporation CFD Solver V10.0 Build 10.00.0
Copyright 1999-2005 Mentor Graphics Corporation. All Rights Reserved.
Using Double Precision Solver. Termination residual shown
Using Parallel Solver. in the Variable Solution Control
Number of processors: 2 section of the Solver Control tab
Number of domains 5
domain 0 no. in x =60 no. in y =22 no. in z =61 ← Grid regions
domain 1 no. in x =38 no. in y =35 no. in z =40 (Main + localized)
domain 2 no. in x =30 no. in y =38 no. in z =46
domain 3 no. in x =60 no. in y =15 no. in z =61
domain 4 no. in x =38 no. in y =13 no. in z =56
Total solar radiation heat source (W) 0.000000e+000
SUM(ABS(RES))/STOPIT & MONITOR VALUES (B)EFORE/ (A)FTER SOLUTION OF
LINEARIZED EQUATIONS AT PASS= 1 TIME STEP= 1
      VAR          Nits      R/S(B)      R/S (A)      M1 (A)      M2 (A)      M3 (A)      M4 (A)
Temperature        1  2.0000e+002  7.3184e-006  2.5012e+001
X-Velocity        2  2.5466e-008  1.9133e-024  0.0000e+000
Y-Velocity        2  5.5107e-010  3.4542e-026  6.5294e-113
Z-Velocity        2  2.4360e-008  1.7890e-024  0.0000e+000
Pressure          1  3.7461e-005  4.5743e-007  0.0000e+000

```

The solution values are recorded for each iteration (pass). After many iterations, if the solution process as a whole is convergent, the residuals presented to the solver will be small, and the corrections obtained will be correspondingly small, so there will be little change in the residual values indicated for a particular variable.

## Related Topics

[Monitoring the Solution in Batch Mode](#)

[Setting up a Batch Mode Completion Email Notification](#)

## floerror.log File

A text file of error, warning, and information messages generated when a batch solve is run, equivalent to the content of the Message Window when an interactive solve runs.

The file path is `<install_dir>\flosuite_v<version>\flotherm\WinXP\bin\floerror.log`.

The file can be sent in a batch mode completion email.

## Related Topics

[Setting up a Batch Mode Completion Email Notification](#)

## Output CSV Files

Solution results from batch mode are available in CSV files which can be loaded into spreadsheets.

The CSV files contain a row of column headers separated by commas, followed by the rows of table cells separated by commas.

There are two batch output options of the flotherm command that generate tables data, -o and -O:

- Results Tables Output CSV Files only

The CSV files that are output when the lower case o option (flotherm -o <output\_directory>) is used are listed in [Table A-1](#).

- All Tables CSV Output Files

The CSV files that are output when the uppercase O option (flotherm -O <output\_directory>) is used are those listed in [Table A-1](#) and the geometry and attribute settings files listed in [Table A-1](#).

### Related Topics

[flotherm](#)

# Batch Solve Procedures

---

Tasks used when running the solver in the background.

<b>Solving in Batch Mode</b> .....	<b>367</b>
<b>Using batchSolve to Solve in Batch Mode on Linux Systems</b> .....	<b>368</b>
<b>Environment Setup on Linux Systems</b> .....	<b>368</b>
<b>Monitoring the Solution in Batch Mode</b> .....	<b>368</b>
<b>Interrupting a Batch Solution</b> .....	<b>369</b>
<b>Setting up a Batch Mode Completion Email Notification</b> .....	<b>369</b>
<b>Setting Solution Process Priority Using an Environment Variable</b> .....	<b>370</b>

## Solving in Batch Mode

Batch mode solving is useful when runs take a long time, for example, overnight.

### Prerequisites

- Create projects interactively in Simcenter Flotherm.
- Specify the number of maximum number of iterations you want solved. See “[Solver Control Tab](#)” on page 200.
- Optionally, specify the number of cores/CPUs to be used while batch solving by setting these with the FLO\_NUMBER\_OF\_THREADS environment variable.

---

#### Note

 If solving one case at a time, the solver will be optimized by setting the KMP\_AFFINITY environment variable, which, by default, is commented out in the *flotherm.bat* file.

---

- Initialize the project.

### Procedure

1. To start a command window from the task bar, choose **Start > All apps > MentorMA > Simcenter Flotherm <version> > Simcenter Flotherm <version> Environment Shell**.

If you do this, the environment setup (flotherm -env) is run for you, otherwise, you will have to run this manually if you need to run Mentor Graphics tools which require the Simcenter Flotherm environment setup.

2. Use the flotherm command with the -b option.

See “[flotherm](#)” on page 606.

## Related Topics

[Environment Variables](#)

# Using **batchSolve** to Solve in Batch Mode on Linux Systems

Use **batchSolve** to batch solve a project XML or PDML file on Linux systems.

## Prerequisites

- You must have a valid license file, and the MGLS\_LICENSE\_FILE environment variable in `<install_dir>/flosuite_<version>/flotherm/LINUX/bin/setup` must point to that file.
- The DISPLAY environment variable (same location as above) must point to a valid accessible X Windows display.

## Procedure

1. Change to the following directory:

`<install_dir>/flosuite_<version>/flotherm/LINUX/bin/`

2. Enter the following command:

`batchSolve <file> [options]`

Where `<file>` is a valid FloXML or PDML file. Refer to “[batchSolve](#)” on page 612 for the batchSolve command options.

# Environment Setup on Linux Systems

Use the following procedure if you need to run Mentor Graphics tools which require the Simcenter Flotherm environment setup but Simcenter Flotherm has not been run.

## Procedure

Run **source setup** in the following directory:

`<install_dir>/flosuite_<version>/flotherm/LINUX/bin`

**source setup** sets up the path, and environment variables required to run Simcenter Flotherm and its utilities, but does not run Simcenter Flotherm.

# Monitoring the Solution in Batch Mode

Examine the *logit* file if you need to check your project when solving in batch mode.

## Procedure

1. Navigate to the solution project log subdirectory:  
`.../flouser/project_set_dir/DataSets/BaseSolution/PDTemp`
2. Open the *logit* file using a text editor such as Notepad. The file is an ASCII text record of the residual data as it is generated.  
If the *logit* file is increasing in size, then the project is still solving.

## Related Topics

[logit File](#)

# Interrupting a Batch Solution

Stopping a batch solver run to examine results obtained so far.

## Procedure

1. Create an empty file called *stopnow*.

---

**Note**

 This file must not have a *.txt* extension.

---

2. Place the file in the following folder:

`<solution_dir>/<project_set_dir>/DataSets/BaseSolution/`

3. Wait for the solution to interrupt, which will take two iterations.

## Results

You should now be able to load the project into an interactive session.

# Setting up a Batch Mode Completion Email Notification

By default, the email includes the solver *floerror.log* and *logit* files as attachments.

## Restrictions and Limitations

- This set up is for Batch Mode only. If an email notification is set up during Interactive Mode then the *floerror.log* file attached to the email will be out of date.

## Prerequisites

- Windows software includes an SMTP utility program which requires the name of the SMTP mail server. The *flomail.bat* script includes a variable, MAIL\_HOST, which must be configured to the correct value. The default value is mailhost. You must set this

variable to the correct name. If you do not know your mailhost name, contact your System Administrator.

## Procedure

Enter the following at the Windows command line:

```
set FLO_EMAIL=<user_name>@<domain>
```

## Results

When the solution stops, the software sends an email to the named recipient.

The email includes the solver *floerror.log* and *logit* files as attachments.

## Related Topics

[floerror.log File](#)

[logit File](#)

# Setting Solution Process Priority Using an Environment Variable

On Microsoft® Windows systems, before running a batch solve, you can change the solution process priority prior to the solve.

## Procedure

Before starting Simcenter Flotherm in the command prompt window, type the following:

```
set FLO_SOLUTION_PRIORITY = <n>
```

where <n> is an integer value:

1 = Low

2 = Normal (the default)

3 = High

4 = Real Time

If zero or a non-valid number is assigned, the setting will default to Normal. Numbers greater than 4 will default to Real Time.

---

### Note

 Advanced users can add the setting of FLO\_SOLUTION\_PRIORITY to their *flotherm.bat* start-up file.

---

## Related Topics

[Solving in Batch Mode](#)

[Setting Solution Process Priority Using the Task Manager](#)

# Transient Analysis Cases

Transients can be solved in one complete solution, or by a number of runs with consecutive integration periods.

Transients can be solved:

- Interactively.
- As batch jobs.
- Using the Command Center.

Care must be taken when solving a single transient in a number of stages, *as each transient stage must run with consecutive transient times*.

Simcenter Flotherm will generally handle the correct times but you should be aware of what is expected and how Simcenter Flotherm maintains the correct transient times.

Should the transient solution period start time become equal to or greater than the total transient end time, an error message (E/1029) is generated and the run aborted.

## Solving Transients Interactively

When running a transient case, the full transient is defined by the transient grid and the total transient time. But the case can be run in separate stages defined by the transient solution period.

Consider a case with total transient start and end times of 0 to 20 seconds. The case could, for example, be run in two separate runs of 10 seconds. When running interactively, to ensure the time values are consistent between stages, Simcenter Flotherm automatically updates the transient solution period start time to match the end of the previous stage.

When initializing from another project, the transient time starts from the current case total transient time, not from the initializing project.

## Solving Transients in Batch Mode

In Batch mode, when running a transient in separate stages, each stage must be provided with a case to start from. For example, to run one transient project over four different stages the batch jobs can be set and run as follows:

```
flotherm -b transientcase
flotherm -b transientcase -i transientcase
flotherm -b transientcase -i transientcase
flotherm -b transientcase -i transientcase
```

This will produce identical results to the same case run interactively four times. Using the `-i` option to the same name is effectively the same as a continuation in the interactive mode and ensures that the transient solution period start time is taken from the end of the previous stage at each run.

If running the same case but as individual projects (for example, to enable different integration periods to be set or to keep the results for each stage) the following should produce the same end result, except that the history for each run will only be available at each individual stage and not as one concatenated plot:

```
flotherm -b transientcase
flotherm -b transientcase1 -i transientcase
flotherm -b transientcase2 -i transientcase1
flotherm -b transientcase3 -i transientcase2
```

As with the interactive mode, when initializing from another project, the transient time is started from the total transient time and does not take the time from the initializing project.

## Solving Transients Using the Command Center

The case where a single transient is solved in a number of consecutive stages (“[Solving Transients in Batch Mode](#)” on page 372) can also be solved in the Command Center by setting up scenarios that are initialized using the All From Previous option. This will result in setting the transient solution period start time to the end time of the previous scenario. When using the All From Base option each scenario will take the transient solution period start time from the Base Project.

For more information on the Command Center settings, see [Scenario Table](#) in the *Simcenter Flotherm Command Center User Guide*.

## Related Topics

[Transient Analysis](#)

## **Solve Dialog Boxes**

---

The following dialog boxes are associated with the solver.

<b>Solver Progress Dialog Box.....</b>	<b>374</b>
<b>Exchange Factors Calculation Progress Dialog Box .....</b>	<b>375</b>

## Solver Progress Dialog Box

To access: Displayed after starting a Solve.

This dialog box shows solver progress data and closes at the end of a solve.

### Objects

Field	Description
Thermal/Airflow Solver	Read only. A percentage estimate of how far the solver has progressed.
Start Time	Read only. The time the solver was started.
Elapsed Time	Read only. The elapsed time during which the solver has been running.
Elapsed CPU	Read only. The recorded actual processing time of all utilized cores/CPUs

### Related Topics

[Solving in Interactive Mode](#)

## Exchange Factors Calculation Progress Dialog Box

To access: Displayed after starting a Solve or an Exchange Factors Calculation run when Radiation is switched on.

This dialog box shows progress data and closes at the end of the run.

### Objects

Field	Description
Radiation Exchange Factor Generator	Read only. A percentage estimate of how far the radiation exchange factor generator has progressed.
Start Time	Read only. The time the generator was started.
Elapsed Time	Read only. The elapsed time during which the generator has been running.
Elapsed CPU	Read only. The recorded actual processing time of all utilized cores/CPUs

### Related Topics

[Exchange Factors](#)

[Solving in Interactive Mode](#)



# Chapter 11

## Solution Monitoring and Profile Plots

---

Plots are generated during solution and can be created after solution.

The Viewing Profile Plots video shows how plots can be viewed within a frame, monitor point plot synchronization, and how you can copy plot data to a spreadsheet.



<b>Solution Monitoring</b> .....	<b>377</b>
<b>Profiles Window</b> .....	<b>379</b>
<b>Profile Plot Features</b> .....	<b>380</b>
<b>Available Plots</b> .....	<b>384</b>
Residuals v Iteration Plot (Steady State) .....	385
Residuals v Iteration Plot (Transient) .....	387
Residual Iterations v Time Plot (Transient) .....	388
Residuals v Time Plot (Transient) .....	389
Monitor Points v Iteration Plot (Steady State) .....	390
Monitor Points v Time Plot (Transient) .....	392
Results v Distance Plot .....	394
<b>Plot Procedures</b> .....	<b>396</b>
Creating New Plots .....	396
Configuring Monitor Point Plots .....	396
Tiling Plots .....	397
Exporting a Plot Profile .....	398
<b>Plot Property Sheet</b> .....	<b>400</b>
Plot Property Sheet Settings Tab .....	401

## Solution Monitoring

Plots are created as a solution progresses. These plots (or profiles) are viewed using the Profiles window. Additional plots are available after solution.

Plots can be grouped as follows:

- Residuals Plots

These are plots of solver residual errors against iteration number or time and monitor the solution progress.

- Monitor Point Plots

These plots show variable values against iteration number or time. These plots are only available when there are monitor points in the model.

- Distance Plots

These plots show how a variable value changes over a specified distance (defined by its end points in 3D space). These plots are only available when 3D results are available.

The following plots are presented during a solution:

- For steady-state solutions:

- Residuals v Iteration
  - Monitor Points v Iteration

- For transient solutions:

- Residuals v Iteration
  - Residual Iterations v Time
  - Monitor Points v Time

After a solution, you can create:

- For steady-state solutions:

- Residuals v Iteration
  - Monitor Points v Iteration
  - Results v Distance

- For transient solutions:

- Residuals v Iteration
  - Residuals v Time
  - Residual Iterations v Time
  - Monitor Points v Time
  - Results v Distance

## Profiles Window

To access: **Window > Launch Profiles**. If not already active, the window is opened when the solver starts, and shows the solution progress.

Use the Profiles window to display 2D plots of solution results for solution monitoring, displaying results for analysis, and comparing results from different solution runs.

### Description

Profile plots are displayed within the window as separate panels.

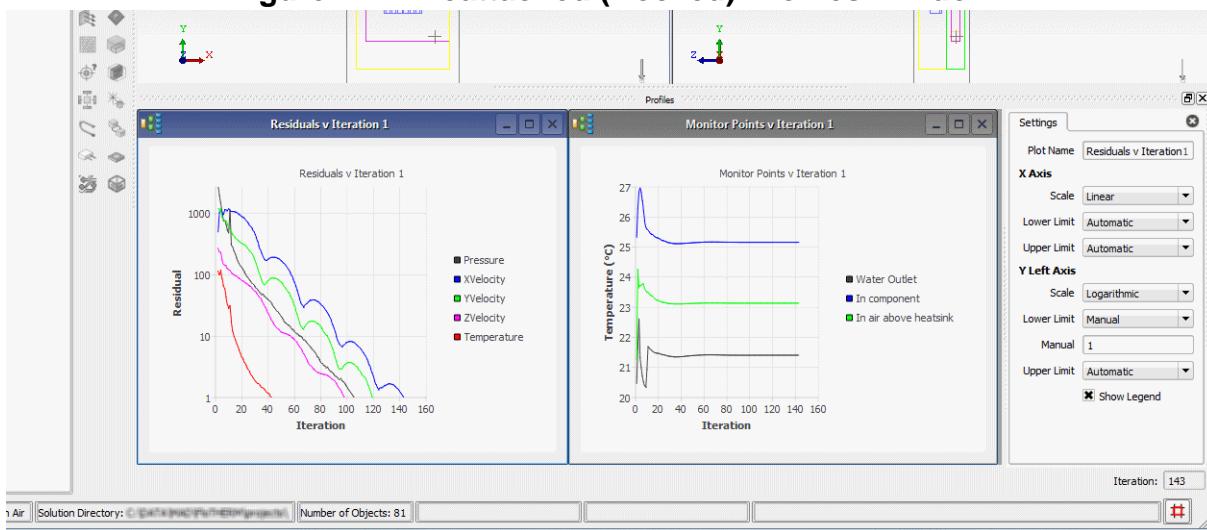
The window, which is normally detached, can be reattached as a pane within the Project Manager window

Provided the project is saved before being closed, the plot configuration is maintained across sessions.

**Figure 11-1. Detached Profiles Window**



**Figure 11-2. Reattached (Docked) Profiles Window**



## Objects

- Time (Transient Analysis Solutions)  
A read-only value of the time of the transient analysis.
- Step (Transient Analysis Solutions)  
A read-only value of the time step number.
- Iteration  
A read-only value of the iteration. For transient analysis solutions, this applies to the time step.

### Note

 These read-only values are updated during the solution. For transient analysis solutions, the final values show the end time, the total number of time steps, and the number of iterations taken to solve the last time step.

## Usage Notes

You can tile plots within the window.

## Related Topics

[Detaching/Reattaching the Profiles Window](#)

[Tiling Plots](#)

## Profile Plot Features

Each profile plot is displayed within a separate window.

## Axes

The y-axis is the calculated dependent variable unit class (for example, Temperature, Pressure, Velocity), and the x-axis is the independent variable (Time, Iteration, Distance). All plots with the same unit class are drawn on the plot.

## Axes Scales and Ranges

The default axes scales and ranges (the upper and lower limits) make use of the plot area available, however, you can re-define these values using the **Settings** tab of the Plot property sheet.

The units of the values displayed are the global units defined in the [Global Units Dialog Box](#).

When an axis scale is logarithmic, the scale limits are in powers of 10.

## Title

The plot window title is displayed in the title bar of the window, and in the plot.

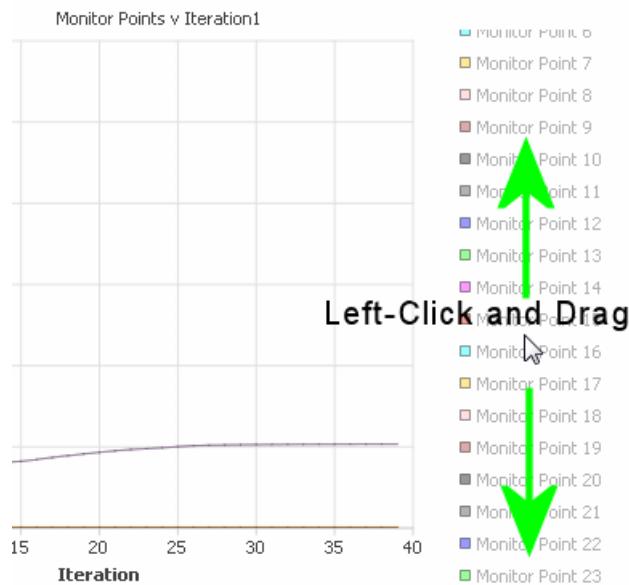
## Legends

A legend, showing the color of each plot line is displayed by default to the right of the plot area.

You can increase the area of a plot by hiding the legend. Uncheck the Show Legend check box in the Plot property sheet **Settings** tab.

If a legend contains a large number of plot lines, for example, if you have many monitor points, then left-click and drag the mouse to scroll up and down through the list.

**Figure 11-3. Scrolling Through Large Numbers of Monitor Points in a Profiles Plot**



## Plot Lines

All plot lines are highlighted by default.

To highlight a single line, click near to the plot line (this may not be possible when viewing an extreme zoom-in), or click the respective entry in the legend. Highlighting of lines is consistent across all existing open plots

To highlight more lines, hold down the Ctrl key and click more plot lines or legend entries.

To dim a highlighted line, hold down the Ctrl key and click the plot line or the respective entry in the legend.

You can also highlight a plot line associated with a monitor point by selecting the monitor point in the data tree or drawing board.

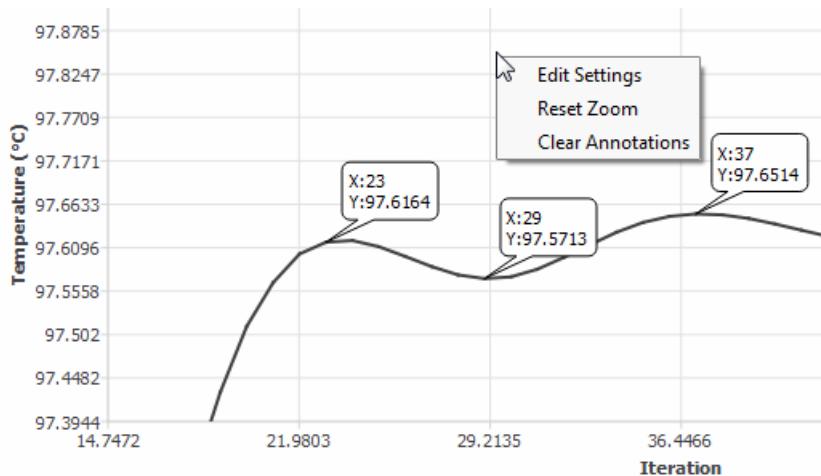
## Annotations

Plot values are show as annotations when you hover over a plot line. The annotation shows the x and y values of the nearest point on the line.

To make an annotation stick, double-click on the plot line.

To clear a single annotation, double-click within the annotation. To clear all annotations, right-click in the plot area and choose **Clear Annotations**.

**Figure 11-4. Profile Plot Annotations and Context-Sensitive Menu**



## Zooming

You can zoom in and out of a profile plot using the mouse scroll bar.

To define a zoom-in area, left-click, drag then release.

To return to full view, right-click then choose **Reset Zoom**.

## Related Topics

[Plot Procedures](#)

[Plot Property Sheet Settings Tab](#)

## Available Plots

---

The profile plots that are generated depend on the solution type: steady-state or transient. A monitor point profile plot is generated if there is one or more monitor points. You can create a (distance) plot of results between two points in the solution domain.

<b>Residuals v Iteration Plot (Steady State)</b> . . . . .	<b>385</b>
<b>Residuals v Iteration Plot (Transient)</b> . . . . .	<b>387</b>
<b>Residual Iterations v Time Plot (Transient)</b> . . . . .	<b>388</b>
<b>Residuals v Time Plot (Transient)</b> . . . . .	<b>389</b>
<b>Monitor Points v Iteration Plot (Steady State)</b> . . . . .	<b>390</b>
<b>Monitor Points v Time Plot (Transient)</b> . . . . .	<b>392</b>
<b>Results v Distance Plot</b> . . . . .	<b>394</b>

## Residuals v Iteration Plot (Steady State)

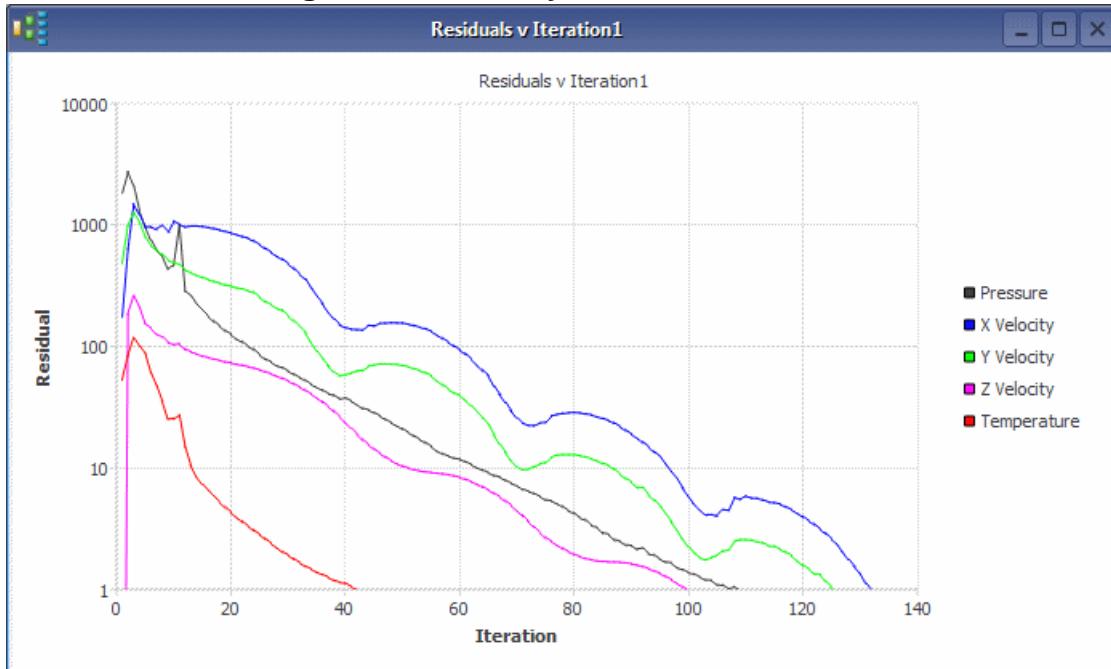
To access: **Profiles > Create Plot > Residuals v Iteration**. A default plot is created during a steady state solution.

Use this plot to view the convergence of steady solutions.

### Description

Solution variables are calculated iteratively until a predetermined level of accuracy is attained, solution convergence.

**Figure 11-5. Steady State Solution Plot**



The behavior of the Profiles window as it is used to monitor convergence can be characterized by two main functions:

- The display of the solution plot of residual error (deviation from the required accuracy limit) against iteration number, and
- The ability to change the configuration of the plots as the solution proceeds

Convergence may be displayed either for a completed solution, or in real-time as it is calculated by the solver program. Real-time display enables you to monitor the solution as it progresses and, therefore, decide whether to stop it or not.

The default configuration is:

- All dependent variables plotted on the same plot
- A vertical logarithmic scaled axis for the residuals.

- A horizontal axis displays the iteration numbers on a fixed scale.
- Profiles drawn in solid lines with a different color for each variable.

## Objects

- Context-Sensitive Menu

Choose **Edit Settings** to open the **Settings** tab of the plot property sheet.

## Related Topics

[Profile Plot Features](#)

[Plot Property Sheet Settings Tab](#)

## Residuals v Iteration Plot (Transient)

To access: **Profiles > Create Plot > Residuals v Iteration**. A default plot is created during a transient solution.

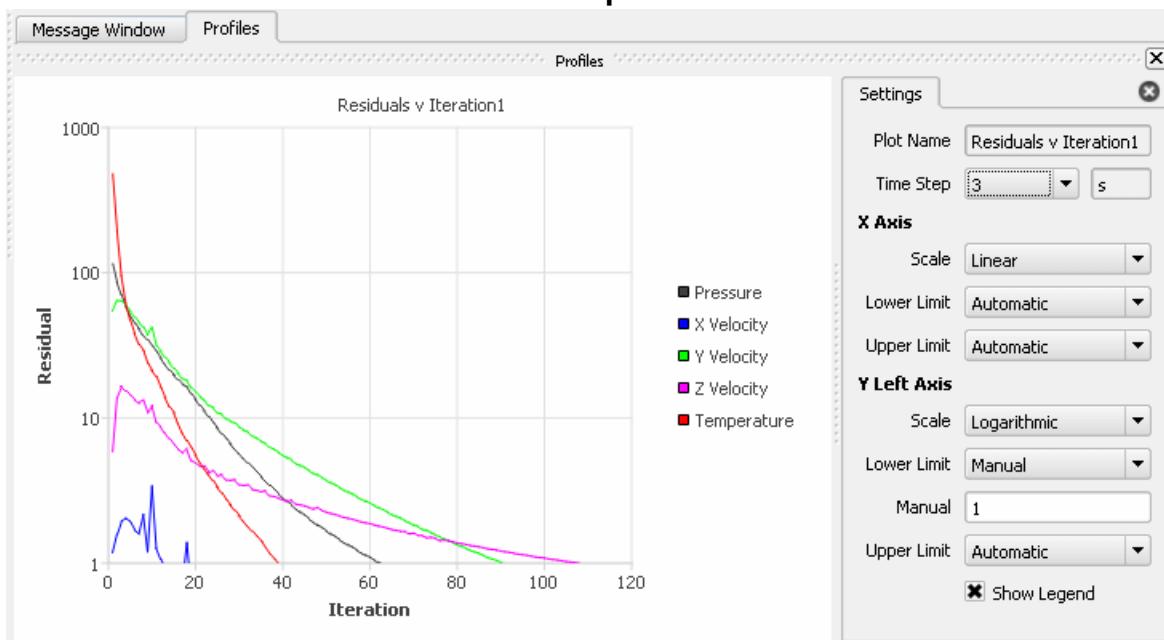
Use this plot to view the convergence during a particular time step of a transient solution.

### Description

Iterations for a selected time step are displayed.

Selection of the time step is from the **Settings** tab of the plot property sheet.

**Figure 11-6. Docked Transient Solution Plot of Residuals v Iteration for a Time Step**



### Objects

- Context-Sensitive Menu

Choose **Edit Settings** to open the **Settings** tab of the plot property sheet.

### Related Topics

[Profile Plot Features](#)

[Plot Property Sheet Settings Tab](#)

## Residual Iterations v Time Plot (Transient)

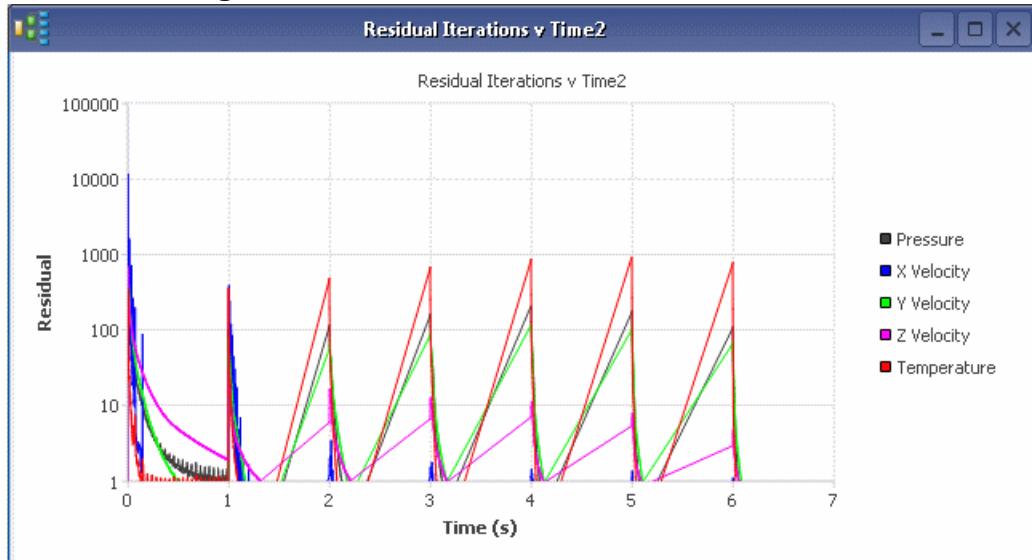
To access: **Profiles > Create Plot > Residual Iterations v Time**. A default plot is created during a transient solution.

Use this plot to view convergence at every time-step.

### Description

Convergence may be displayed either for a completed solution, or in real-time as it is calculated by the solver program. Real-time display gives the user the ability to monitor the solution as it progresses and, therefore, decide whether to stop it or not.

**Figure 11-7. Residuals Iterations v Time Plot**



The default configuration is:

- All dependent variables plotted on the same plot.
- A vertical logarithmic scaled axis for the residuals.
- A horizontal axis displays the time into the simulation on a fixed scale.
- Profiles drawn in solid lines with a different color for each variable.

### Objects

- Context-Sensitive Menu

Choose **Edit Settings** to open the Settings tab of the plot property sheet.

### Related Topics

[Profile Plot Features](#)

[Plot Property Sheet Settings Tab](#)

## Residuals v Time Plot (Transient)

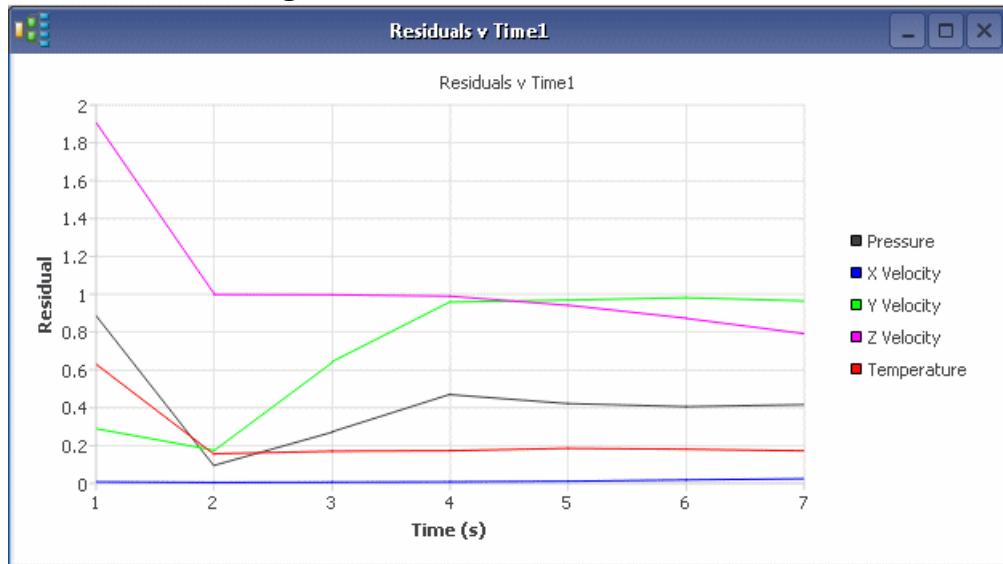
To access: **Profiles > Create Plot > Residuals v Time**

Use this plot to view the final residual versus time for the full simulation.

### Description

The residual at the last iteration at each time step is plotted.

**Figure 11-8. Residuals v Time Plot**



The default configuration is:

- All dependent variables are plotted on the same plot.
- A vertical linear axis is displayed for the residuals.
- A horizontal axis for the time of the full transient solution.
- The profiles are drawn in solid lines with a different color for each variable.

### Objects

- Context-Sensitive Menu

Choose **Edit Settings** to open the **Settings** tab of the plot property sheet.

### Related Topics

[Profile Plot Features](#)

[Plot Property Sheet Settings Tab](#)

## Monitor Points v Iteration Plot (Steady State)

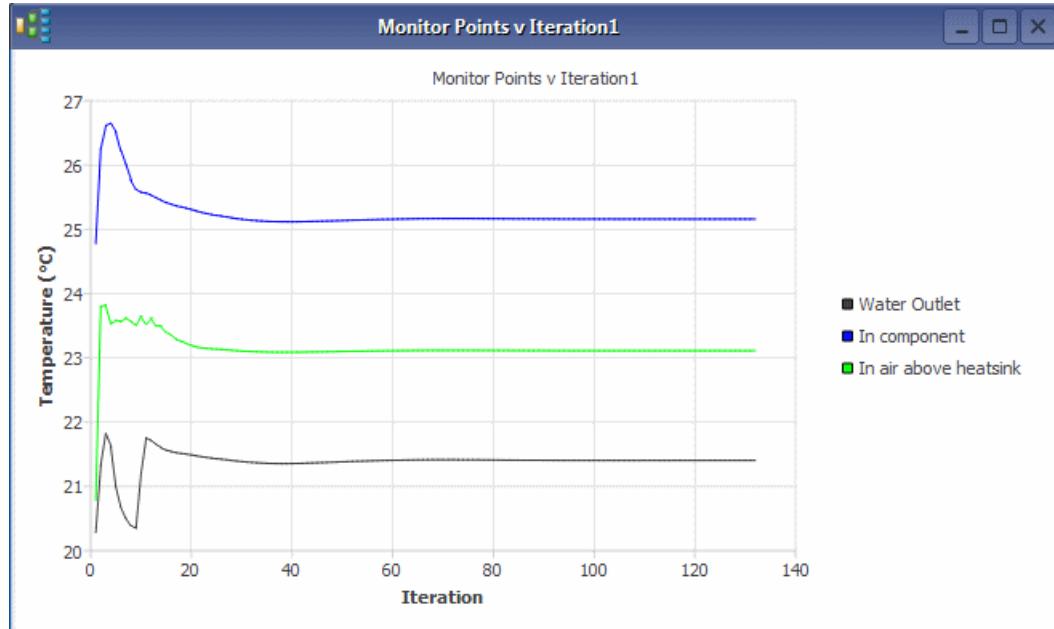
To access: **Profiles > Create Plot > Monitor Points v Iteration**. A default plot is created during a steady state solution when monitor points exist in the model.

Use this plot to view the variation of a variable at specific points against iteration number, either as they are calculated during solution, or after the solution is complete. Monitor points must be created prior to solution.

### Description

The variables are plotted on a fixed vertical linear scale, which means a separate plot for each variable unit class, for example, Temperature or Pressure. Also, as the horizontal axis displays the iteration number in a fixed linear scale, a separate plot is required for monitor points activated at different iteration numbers. [Figure 11-9](#) shows plots of Temperature against Iteration Number.

**Figure 11-9. Monitor Points v Iteration Plot**



The default configuration is:

- Variables are plotted on a fixed vertical linear scale.
- Iteration numbers are displayed along the horizontal axis.
- The profiles are drawn in solid lines with a different color for each monitor point with a limit of up to ten colors. After all the colors have been used, the color allocation sequence is repeated, starting from the first set color.

## Objects

- Context-Sensitive Menu

Choose **Edit Settings** to open the **Settings** tab of the plot property sheet.

## Usage Notes

A plot profile is highlighted (shown as a thick line) when the monitor point (in the data tree or drawing board), plot line or legend is selected (clicked).

A plot profile is unhighlighted (shown as a thin line) when you CTRL+click on the monitor point, line, or legend.

A plot profile can be removed using the Monitor Points section of the **Settings** tab, see “[Configuring Monitor Point Plots](#)” on page 396.

A plot profile is removed if a monitor point is deactivated or its parent assembly is deactivated or ignored.

## Related Topics

[Profile Plot Features](#)

[Plot Property Sheet Settings Tab](#)

## Monitor Points v Time Plot (Transient)

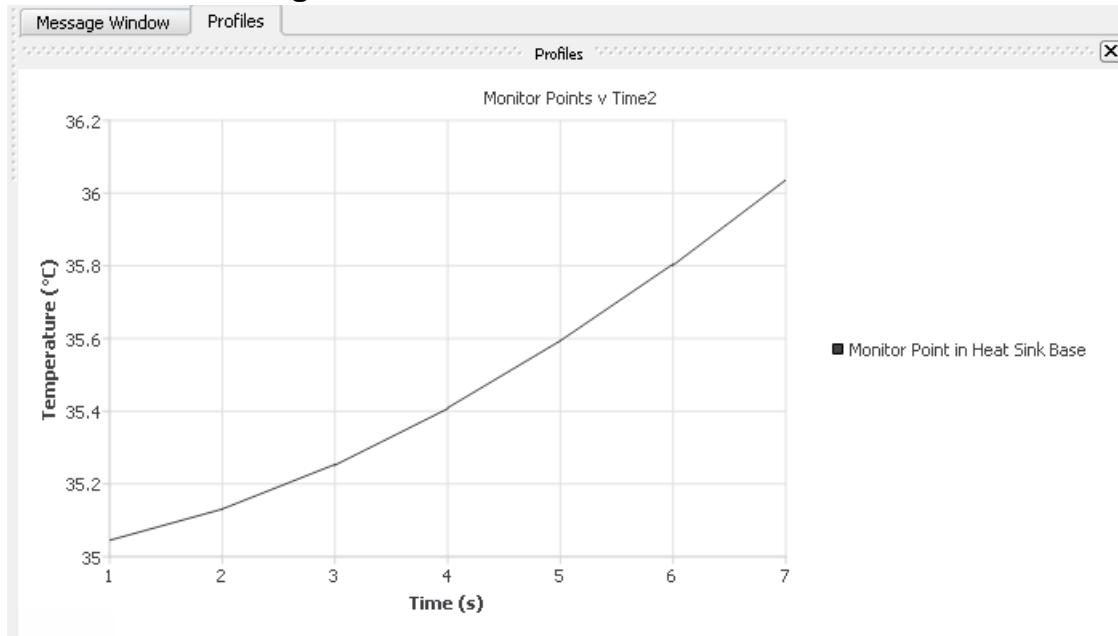
To access: **Profiles > Create Plot > Monitor Points v Time**. A default plot is created during a transient solution when monitor points exist in the model.

Use this plot to view the variation of variables at specific points against time, either as they are calculated during solution, or after the solution is complete.

### Description

The variables are plotted on a fixed vertical linear scale, which means a separate plot is needed for each variable unit class unless a 2nd Y-axis is selected in the Plot Parameters dialog box. The horizontal axis displays the time.

**Figure 11-10. Monitor Points v Time Plot**



The default configuration is:

- Variables are plotted on a fixed vertical linear scale with only one variable per plot. Simcenter Flotherm tries to plot Temperature, if available. If Temperature is not available, then it plots the first variable listed in the Plot Parameters dialog box.
- Time along the horizontal axis.
- The profiles are drawn in solid lines with a different color for each monitor point.

### Objects

- Context-Sensitive Menu

Choose **Edit Settings** to open the **Settings** tab of the plot property sheet.

## Usage Notes

A plot profile is highlighted (shown as a thick line) when the monitor point (in the data tree or drawing board), plot line or legend is selected (clicked).

A plot profile is unhighlighted (shown as a thin line) when you CTRL+click on the monitor point, line, or legend.

A plot profile can be removed using the Monitor Points section of the **Settings** tab, see “[Configuring Monitor Point Plots](#)” on page 396.

A plot profile is removed if a monitor point is deactivated or its parent assembly is deactivated or ignored.

## Related Topics

[Profile Plot Features](#)

[Plot Property Sheet Settings Tab](#)

## Results v Distance Plot

To access: **Profiles > Create Plot > Results v Distance**

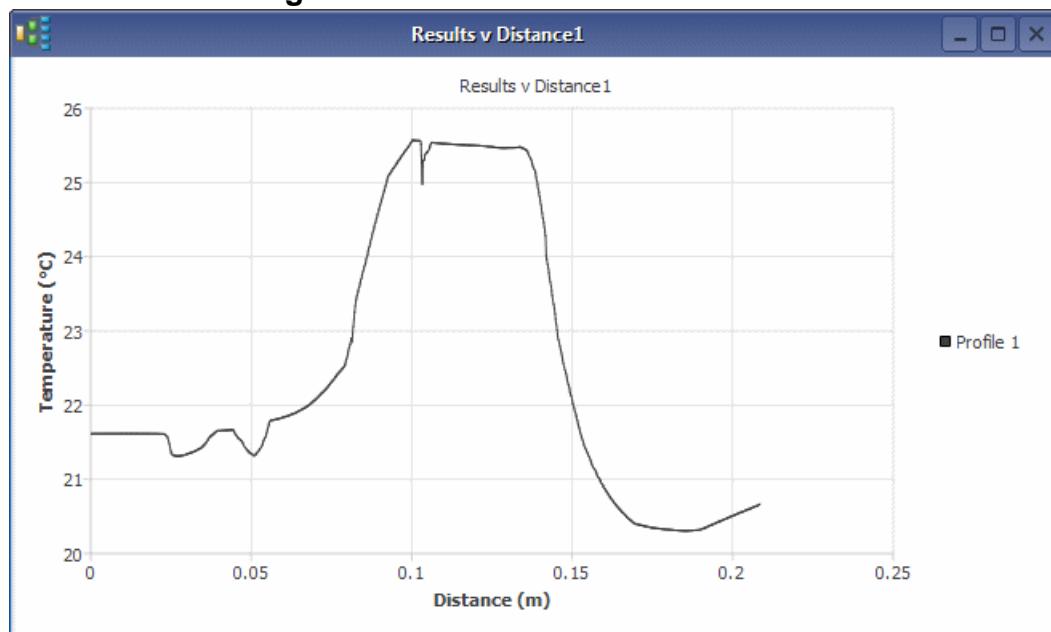
Use this plot to view how a variable value changes with distance along a straight line through a model between two definable points.

### Description

The variable is plotted along the vertical axis.

The horizontal axis is the distance along the profile line, whose start and end points are defined by X, Y, Z coordinates entered on the **Settings** tab of the plot property sheet.

**Figure 11-11. Results v Distance Plot**



The default configuration is:

- Vertical linear scale for the field value
- Horizontal linear distance scale
- Solid lines for profiles, with a different color for each line plotted
- All profiles for the same variable are plotted on the same plot

Deactivated monitor points do not appear in the plot.

### Objects

- Context-Sensitive Menu

Choose **Edit Settings** to open the **Settings** tab of the plot property sheet.

## Related Topics

- [Profile Plot Features](#)
- [Plot Property Sheet Settings Tab](#)

## Plot Procedures

---

You can create, configure, tile, and export profile plots.

See also, “[Profile Plot Features](#)” on page 380.

<b>Creating New Plots</b> .....	<b>396</b>
<b>Configuring Monitor Point Plots</b> .....	<b>396</b>
<b>Tiling Plots</b> .....	<b>397</b>
<b>Exporting a Plot Profile</b> .....	<b>398</b>

## Creating New Plots

Results can be plotted in different formats depending on the solution type.

---

### Note

 The panels display the information type determined at their creation. Monitored variable solution plots only display information for the monitor points present when the panel was created. If you want monitored variable solution plots to include data for monitor points *added since the creation* of the plot panel, then delete the monitor solution panel and re-create it.

---

### Prerequisites

- For Monitor Point and Distance plots, a steady-state or transient solution must have been run and results available.

### Procedure

Choose **Profiles > Create Plot** then choose a plot type.

See “[Available Plots](#)” on page 384.

### Results

A plot of the type chosen, and with the default settings, is added to the Profiles window. If the plot has previously been created, then another plot is created but has the suffix number in its name incremented by 1. You can use different settings for multiple copies of the same plot.

Plot window size and location are determined automatically by the program when you create new profile plots. By default, the plot panels are tiled within the display area and their size is adjusted according to the number of plots displayed.

## Configuring Monitor Point Plots

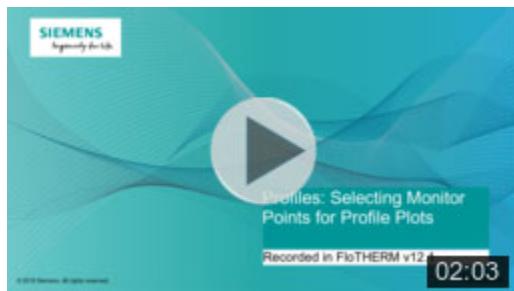
By default, all monitor points are plotted in a newly-created monitor point plot chart. You can reduce the number of plots shown.

## Prerequisites

- You have created a monitor point profiles chart, see “[Creating New Plots](#)” on page 396.

## Video

Learn how to modify a monitor points chart to only include plots for specific monitor points, and how to use the Select Associated Geometry check box to synchronize monitor point selection with a chart.



## Procedure

1. Select the monitor points that you want to be displayed in the profiles plot.  
Selection can be done in the plot itself or in the data tree or GDA.
2. Right-click in the plot and choose **Edit Settings**.  
The **Settings** tab is opened.
3. Click **Use Selection**.  
This button is in the Monitor Points section of the tab.
4. Optionally, check the Select Associated Geometry check box (this is on the far-right of the Monitor Points section) if you want the monitor points associated with the chart to be selected whenever the chart is selected.

## Results

Plots of unselected monitor points are removed from the chart.

The names of the selected monitor points appear in the read-only field to the right of the **Use Selection** button.

## Related Topics

[Plot Property Sheet Settings Tab](#)

# Tiling Plots

Tile plots to make use of the available space within the Profiles window.

## Restrictions and Limitations

- Tiled plots are not resized if the Profiles window is resized.
- If tiling removes slider bars, then the bars are replaced by blank areas, but repeated tiling will occupy these areas.

## Procedure

1. Open the Profiles window.
2. Click the **Tile Plots** icon on the Profiles toolbar, or choose **Profiles > Tile Plots**.

## Results

Provided plots have not been minimized, they are tiled within the Plots window so that each plot is visible.

# Exporting a Plot Profile

Profile data can be copied and pasted into Microsoft® Office Excel for examination.

## Procedure

1. Right-click on the plot and choose **Edit Settings** to open the **Settings** tab.
2. Click **Copy Data**.  
The data is copied to the clipboard.
3. Open an Excel spreadsheet, select a cell and then paste the contents of the clipboard.

## Results

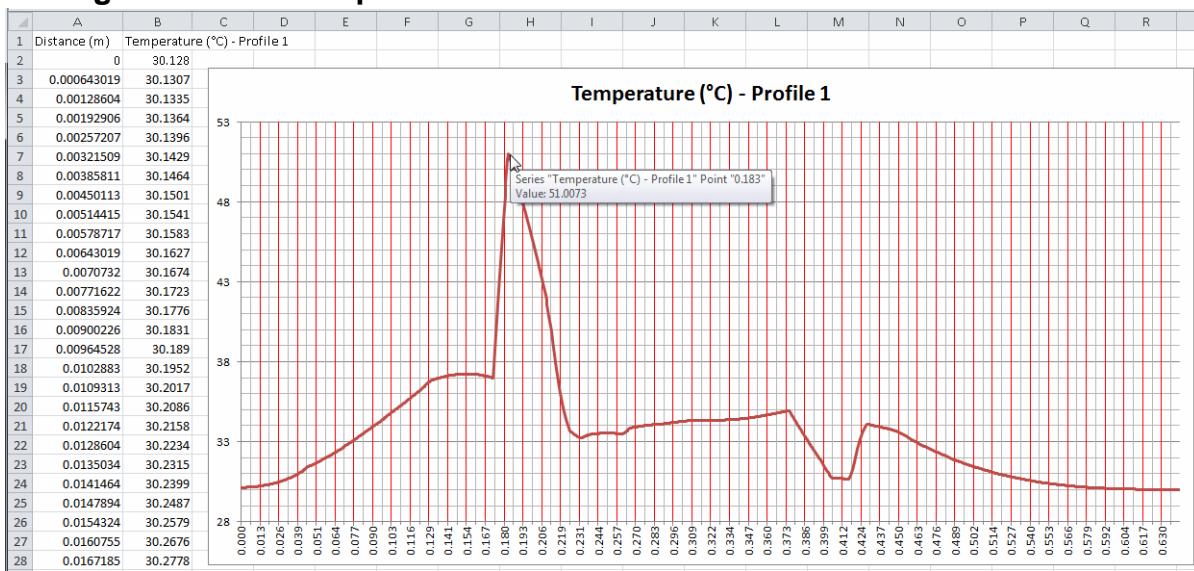
The first row contains titles, subsequent rows contain data. The first column corresponds to the x-axis, other columns contain the y-axis values for each variable plotted.

You can now use the Chart tools in Excel to create and format charts from the imported data.

## Examples

[Figure 11-12](#) shows data in Excel (columns A and B) that has been copied from a Results v Distance plot, and a chart created from the data and formatted using Excel tools.

**Figure 11-12. Example Results v Distance Plot Data Processed in Excel**



## Plot Property Sheet

To access: Right-click in a plot and choose **Edit Settings**.

Use this property sheet to control the scaling and ranges of a profile plots.

---

### Tip

 You can export profile plot data to a spreadsheet and use spreadsheet functionality to make stylistic changes to plots.

---

### Objects

- Settings tab.  
See “[Plot Property Sheet Settings Tab](#)” on page 401.

## Plot Property Sheet Settings Tab

To access: Right-click in a plot, choose **Edit Settings** then select the **Settings** tab.

Use this property sheet tab to control the scaling and range of a profile plot axis, and to copy the plot data.

### Objects

Field	Description
Plot Name	The name that appears in the title bar of the plot window.
Variable	(Monitor Points and Results v Distance plots) Sets the variable to be plotted from a dropdown list of available variables.
Grid Start X, Y, Z	(Results v Distance plots) The start point of the profile line. The x-axis (Distance) of the plot represents the length of the profile line.  The default start point is defined by the coordinates at the cell center in the <i>low XYZ</i> corner of the domain.
Grid End X, Y, Z	(Results v Distance plots) The end point of the profile line.  The default end point is defined by the coordinates at the cell center in the <i>high XYZ</i> corner of the domain.
Time Step	(Residuals v Iteration plots in Transient cases) Select the time step for which all the iterations are to be displayed from the dropdown list. The time step can be selected independently for each profile.
<b>X Axis/Y Left Axis</b>	
Scale	Sets the axis scaling as either Linear or Logarithmic.
Lower Limit	Sets the start of the axis range: <ul style="list-style-type: none"> <li>• Automatic — instructs the program to display data from the lowest value.</li> <li>• Manual — activates the value range input field to allow you to set the lowest value you want to see included in the plot.</li> <li>• % of Range — Activates the value range input field used to set the lowest value you want to see included in the plot as a percentage of the overall data range.</li> </ul>

Field	Description
Upper Limit	Sets the end of the axis range: <ul style="list-style-type: none"><li>Automatic — instructs the program to set the end at the highest value.</li><li>Manual — activates the value range input field to allow you to set the highest value you want to see included in the plot.</li><li>% of Range — activates the value range input field used to set the highest value you want to see included in the plot as a percentage of the overall data range.</li></ul>
Show Legend	Controls visibility of the legend in the plot.
Use Selection	(Monitor Points plots only) Click this button to show data only for the selected monitor points. The plots of unselected monitor points are removed from the chart.  To reinstate all the plots, choose <b>Edit &gt; Undo Modify Monitor Point Plot.Associated Geometry</b> .
Selected monitor points	(Monitor Points plots only) A read-only field, which is updated to show the names of the monitor points selected by the <b>Use Selection</b> button.
Select Associated Geometry check box	(Monitor Points plots only)  <b>Note:</b> The setting applies to <i>all</i> monitor point charts. Check this check box if you want the monitor points associated with a monitor point chart to be selected whenever the chart is selected.  If the check box is left unchecked then selecting a monitor point chart does not automatically select the monitor points in the data tree or GDA.
Copy Data	Click this button to copy the plot data to the clipboard, see “ <a href="#">Exporting a Plot Profile</a> ” on page 398.

## Related Topics

[Available Plots](#)

[Configuring Monitor Point Plots](#)

# Chapter 12

## Advanced Solution Controls

---

Normally, the default settings are sufficient to achieve convergence, however, if convergence problems are encountered, there are solution controls available to help.

<b>The Solution Process .....</b>	<b>403</b>
<b>Program-Calculated Termination Residuals .....</b>	<b>405</b>
<b>Possible Solution Scenarios.....</b>	<b>407</b>
<b>Controlling the Solution .....</b>	<b>408</b>
<b>Techniques for Controlling the Solution .....</b>	<b>410</b>
False Time Step Relaxation .....	410
Multi Grid Damping .....	412
Inner Iterations .....	413
Fan Relaxation .....	414
Monitor Point Convergence for Temperature.....	414
Error Field.....	416
User-Specified Termination Residuals .....	417
Double-Precision Solver.....	418

## The Solution Process

The Simcenter Flotherm solution activates the CFD algorithms which provide an integration of the fluid flow and heat transfer equations within the solution domain.

These calculations are of an iterative nature for each grid cell in the solution domain and continue until:

- A prescribed maximum number of iterations have been performed, or
- A predetermined level of residual error is attained, at which time the solution is said to be converged. In this case convergence applies to the complete system and can be verified using defined monitor points.

As the solution proceeds, you can monitor its progress in the plot of the residual errors against iteration number for each variable using the Profiles window, see “[Solution Monitoring and Profile Plots](#)” on page 377.

## Residual Errors

Residual errors represent a measure of:

- the mass imbalance in the system (associated with pressure,  $P$ )
- the momentum imbalance in the system (associated with the velocities  $U, V$  and  $W$ )
- the energy imbalance in the system (associated with temperature  $T$ )
- the electrical current imbalance in the system (associated with electrical potential,  $f$ )

For a full description of the residual error in terms of the system equations, see [Finite Volume Equations](#) in the *Simcenter Flotherm Background Theory Reference Guide*.

## Termination Residuals

The termination residuals determine the accuracy of the equations to be solved. However, it is important not to confuse this accuracy with the accuracy with which the solution models physical reality. For example, if you solve the equations on a very coarse grid, you can get a very accurate solution of the equations, but the solution obtained cannot be expected to accurately simulate physical reality.

When the program finds that the residual error for each variable is less than its termination residual, it stops the iterative solution on the grounds that it has satisfied the specified convergence criteria.

### How Termination Residuals Determine When the Solution Stops

The residual error for temperature,  $R_T$ , is defined as the sum for all grid cells of the absolute values of the error,  $r_T$ , of the temperature equation for each grid cell, namely:

$$R_T = \sum |r_T|$$

The grid cell error,  $r_T$ , is the extent to which the temperature equation in a cell is not satisfied:

$$r_T = (C_0 T_0 + C_1 T_1 + C_2 T_2 + C_3 T_3 + C_4 T_4 + C_5 T_5 + C_6 T_6 + S) - ((C_0 + C_1 + C_2 + C_3 + C_4 + C_5 + C_6) \times T)$$

When  $r_T$  equals zero, this equation is exactly satisfied.

Let  $E_T$ ,  $E_u$ ,  $E_v$ ,  $E_w$ , and  $E_p$  denote the termination residuals set in the dialog box. The program takes the specified convergence level as being reached when:

$$R_T / E_T < 1 \text{ and } R_u / E_u < 1 \text{ and } R_v / E_v < 1 \text{ and } R_w / E_w < 1 \text{ and } R_p / E_p < 1$$

When these conditions are satisfied, the outer iterations cease and the solution process will stop.

The residual errors, divided by the termination residuals, that is,  $R_T / E_T$  and so on, are plotted on the vertical axis of the plot of residual versus iteration.

## Program-Calculated Termination Residuals

The termination residuals can be set to be automatically calculated by the program.

These program-calculated values provide a safe margin for the majority of systems analyzed and are based on mass, momentum, and energy inputs into the system as described below. It is possible to set your own values, see “[User-Specified Termination Residuals](#)” on page 417.

- Mass Continuity

The termination residual for mass continuity,  $E_p$ , is taken as one half of one per cent of a characteristic flow rate,  $\dot{M}$ , within the enclosure:

$$E_p = 0.005\dot{M}$$

$E_p$  has units of kg/s.

For an enclosure with forced inflow or outflow (for example, fans),  $\dot{M}$  is taken as the total inlet flow rate or the total outlet flow rate, whichever is the largest. In the absence of forced flow (that is, the pure free convection case)  $\dot{M}$  is calculated as the product of the density, the Estimated Free Convection Velocity and the area of the domain normal to the vertical.

- Velocity Components

The termination residual for the velocity components,  $E_{vel}$ , is taken as one half of a per cent of the characteristic flow rate,  $\dot{M}$ , multiplied by the associated characteristic velocity,  $V$ :

$$E_{vel} = 0.005\dot{M}V$$

$E_{vel}$  has units of Newtons.

- Temperature

The termination residual for the temperature  $E_T$ , is calculated as one-half of a per cent of a characteristic heat source,  $Q$ :

$$E_T = 0.005 Q$$

If heat sources are specified within the enclosure,  $Q$  is taken as the sum of the total heat sources or the total heat sinks. If no heat sources are present,  $Q$  is taken as:

$$E_T = 0.005\dot{M}C_p\Delta T_{typ}$$

where:

$\dot{M}$  = the characteristic flow rate

$C_p$  = the specific heat of the fluid, and

$\Delta T_{typ} = 20^\circ\text{C}$

- Turbulence Parameters (k and epsilon)

In most calculations the production of turbulence within the solution domain is greater than that in the incoming flows. To compensate for this, the termination criteria are less stringently set to prevent unnecessary calculation as follows:

$$E_k = 5k\dot{M}$$

$$E_\varepsilon = 50\varepsilon\dot{M}$$

---

#### Note

 The default values for these turbulence parameters are likely to produce less accurate estimates since it is more difficult to know at the start of the calculation what internal generation there may be. You are advised always to use monitoring points in key areas of interest to ensure that convergence has been achieved or that iterations are not being undertaken unnecessarily. Further advice can be found in the Termination Residuals subsection under “[The Solution Process](#)” on page 403.

---

- Potential (Joule Heating On)

The termination residual for the potential  $E_V$ , is calculated as one-half of a per cent of the total current sources,  $I$ :

$$E_V = 0.005 I$$

When a model contains only Fixed Value Potential electrical sources, the current is derived from an estimated Resistance. In such cases, you may prefer to change the default value and use monitor points to check convergence.

These termination residuals work extremely well in guaranteeing convergence for the majority of problems. However, they should be considered as conservative estimates, and, for certain cases of problem, can be too restrictive a criteria to base convergence on.

## Heat Sources and Joule Heating

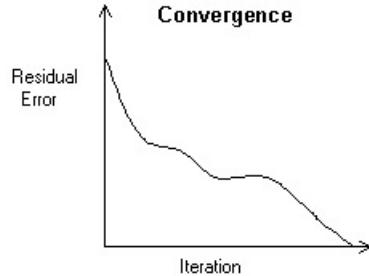
In cases where there is a combination of heat sources and Joule Heating contributions, the temperature termination residual ignores any Joule Heating contribution. This is to avoid temperature residuals from converging too quickly.

# Possible Solution Scenarios

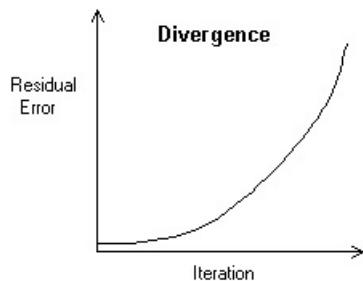
When the solution process is initiated, the most-likely scenario is that the solution will converge, however, there are other possibilities.

[Figure 12-1](#) shows a convergent solution. [Figure 12-2](#), [Figure 12-3](#), [Figure 12-4](#), [Figure 12-5](#) and [Figure 12-6](#) show alternative outcomes.

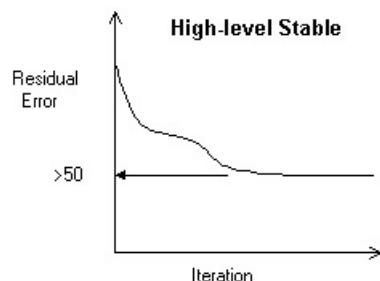
**Figure 12-1. Convergent Solution**



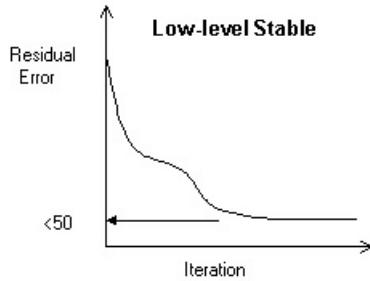
**Figure 12-2. Divergent Solution**



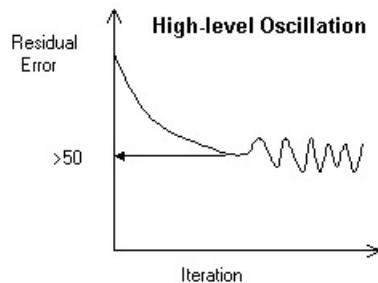
**Figure 12-3. High-Level Stable Residual Error**



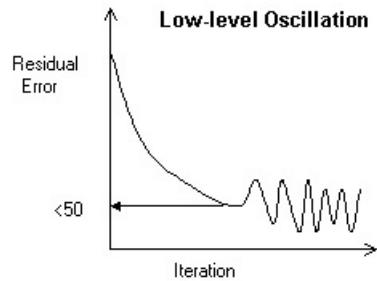
**Figure 12-4. Low-Level Stable Residual Error**



**Figure 12-5. High-Level Oscillation Residual Error**



**Figure 12-6. Low-Level Oscillation Residual Error**



## Related Topics

[Controlling the Solution](#)

[Techniques for Controlling the Solution](#)

# Controlling the Solution

If your solution fails to converge or converges extremely slowly, then you can reset the solution control panels.

First consider the following rules for assessing a solution convergence problem as the problem may well lie in the project setup.

## Rules for Assessing Convergence Problems

1. If a solution diverges, it is almost sure to be because of the problem definition. Be immediately suspicious of the setup and check all defined objects and attributes before proceeding to alter any solution control parameters.
2. If a solution fails to converge successfully, then it is important to check the grid. If there are poor aspect ratio grid cells and large jumps in grid size between adjacent grid cells, then this is the likely cause of the problem.
3. If you are happy with the setup and the grid, then, and only then, should the solution control parameters be adjusted.
4. *Do not waste time forcing low-level stable or low-level oscillation convergence profiles down to a residual error level of 1.* Use the monitor points and error field to sensibly assess whether the solution is converged to a defined level of accuracy, and then stop the solution.

However, if absolute residual convergence is required, then use the [Double-Precision Solver](#), or use monitor point convergence criteria, see [Monitor Point Convergence for Temperature](#), to automatically stop the solution when monitor point values have stabilized.

If you do need to change the control parameters, then the following section provides an overview of how to resolve and manage the solution process.

# Techniques for Controlling the Solution

---

There are various advanced techniques, both automatic and manual, which can be used to optimize the solution process. In discussing their use, it is important to know that it is only possible to give general guidelines rather than hard and fast rules on how they should be altered for particular situations.

In Simcenter Flotherm, extremely complex and highly non-linear systems involving multiple modes of heat transfer are being modeled and it is impossible to automatically generate appropriate solution control parameters that will guarantee convergence under all circumstances. The automatic settings have been designed to give a reasonable convergence profile for the majority of applications, but may need to be adjusted in more complex situations.

<b>False Time Step Relaxation</b> .....	<b>410</b>
<b>Multi Grid Damping</b> .....	<b>412</b>
<b>Inner Iterations</b> .....	<b>413</b>
<b>Fan Relaxation</b> .....	<b>414</b>
<b>Monitor Point Convergence for Temperature</b> .....	<b>414</b>
<b>Error Field</b> .....	<b>416</b>
<b>User-Specified Termination Residuals</b> .....	<b>417</b>
<b>Double-Precision Solver</b> .....	<b>418</b>

## False Time Step Relaxation

False time step under-relaxation is the principal means that you will use to control convergence of the iterative algorithm.

For a given variable, this under-relaxation provides an inertial effect which restrains the amount by which the value in each cell is permitted to change at each outer iteration.

The inertial effect is proportional to the mass in the cell and inversely proportional to the false time step, set for the variable. Consequently, the smaller the false time step, the larger the inertial effect, and so the less the variable is permitted to change.

By default, the program calculates an automatic false time-step which is used for all variables, the one exception being pressure, to which the method is not applied.

The program-calculated value is calculated from the expression:

$$\frac{\text{half average domain dimension}}{\text{maximum fan velocity}}$$

When there is no forced convection set (that is, no external or internal fans, or fixed flow set) the program-calculated value is determined from the expression:

$$\frac{\text{one tenth of average domain dimension}}{\text{Estimated Free Convection Velocity(EFCV)}}$$

Application of the false time step ( $dt_f$ ) relaxation involves adding a false transient term to the finite volume equations.

In general, the false time step should be set to a characteristic time scale for the problem in question. The automatic settings will set this in a sensible way, and any adjustments should normally be by factors (2-to-50 is typical) which increase or decrease the automatic value. For the profiles identified in “[Possible Solution Scenarios](#)” on page 407, the actions described in [Table 12-1](#) will work for the majority of cases.

**Table 12-1. False Time Step Relaxation**

Solution Profile	Example	Possible Action
Convergence	<a href="#">Figure 12-1</a> on page 407	Increase $dt_f$ (reducing damping) to speed up convergence for velocities and temperature.
Divergence	<a href="#">Figure 12-2</a> on page 407	Check problem setup and grid. Possibly decrease $dt_f$ (increased damping) to force extreme cases to converge.
High-Level Stable	<a href="#">Figure 12-3</a> on page 407	Increase $dt_f$ (reduced damping) to try to force convergence for velocities and temperature. In this case, reducing $dt_f$ (increased damping) can also sometimes be beneficial, depending on the complexity of the system being analyzed.
Low-Level Stable	<a href="#">Figure 12-4</a> on page 408	No action. Base decision on convergence on monitor point and error field assessment.
High-Level Oscillation	<a href="#">Figure 12-5</a> on page 408	Reduce $dt_f$ (increased damping) to try to stabilize the solution.
Low-Level Oscillation	<a href="#">Figure 12-6</a> on page 408	No action. Base decision on convergence on monitor point and error field assessment.

- Velocities

Usually some amount of false time step under-relaxation is essential for the velocities. The program-calculated value is often sufficient to procure a convergent algorithm.

For some flows, the program-calculated false time step on the velocities may be insufficient to get convergence. You can reset under-relaxations for  $u$ ,  $v$  and  $w$  to equal the smallest cell size divided by the largest anticipated velocity.

If convergence problems persist, consult “[Troubleshooting](#)” on page 559 (or contact Mentor Graphics customer support) for advice rather than increasing the under-relaxation still further.

- Temperature

Temperature under-relaxation is typically needed for buoyant flows in which the temperature field feeds back into the momentum equations via the buoyancy force.

The program ‘plays safe’ by providing an automatic setting for the false time step which is equivalent to that determined for the velocities.

However, in flows where the temperature solution does not have a significant feedback in to the velocity calculation (that is, small temperature differences so gravitational effects are minimal), the program-calculated false time-step for temperature may over restrain the pass-to-pass changes of temperature. In this case, you can either increase the rate of convergence by setting a value 10 to 100 times larger than the automatic calculated value (use the slider bar) or remove the automatic under-relaxation entirely.

You can do this by entering a user specified false-time step of  $10^{15}$  s.

- Very Low Reynolds Number Flows

In very slow flows which are dominated by diffusion (that is, the Reynolds number is very much less than unity), the fluid velocity is not the appropriate quantity from which to infer the false time step. In the case of very low Reynolds numbers (for example, less than unity), you should set the false time step from the following expression:

$$\frac{(density) \times (typical\ dimension)^2}{(viscosity)}$$

## Related Topics

[Solver Control Tab - Variable Solution Control Section](#)

[Solver Control Tab](#)

## Multi Grid Damping

The Multi Grid option is designed to choose an iteration procedure which uses multi grid acceleration to solve the linear equations for temperature.

For problems with conjugate heat transfer, it can improve convergence and significantly reduce overall computation time.

## Related Topics

[Solver Control Tab](#)

## Inner Iterations

Inner iterations solve the equations for temperature, velocities, and pressure at each outer iteration (or solution pass).

As an example for temperature, the task is to solve the following equation for temperature ( $T$ ) in each grid cell:

$$T = \frac{(C_0 T_0 + C_1 T_1 + C_2 T_2 + C_3 T_3 + C_4 T_4 + C_5 T_5 + C_6 T_6 + S)}{(C_0 + C_1 + C_2 + C_3 + C_4 + C_5 + C_6)}$$

Here  $T_1, T_2, \dots, T_6$  are the temperatures in the neighboring grid cells.

A single inner iteration calculates  $T$  from this equation at each grid cell. Repeated iterations are needed to solve the equation set because some of the neighbor values of temperature are the previous iteration values. The iterations on temperature continue until the specified number is completed or they will end early if the termination criterion is satisfied, that is, if the normalized residual falls below unity.

$$R_T \div E_T < 1$$

On the next outer iteration, it is necessary to solve the temperature equations again because the coefficients  $C_1, C_2, \dots, C_6$  will all have changed.

You can set the number of inner iterations on any variable in the list, that is, temperature, velocities, and pressure. The default numbers of inner iterations are as follows:

- Pressure, 50 inner iterations

In cases where the continuity equation is having difficulty then setting to 100 or more can help.

- X,Y and Z Velocities, 1 inner iteration

These sometimes benefit from 5 or 10 inner iterations.

- Temperature, 100 inner iterations

Often more than 100 inner iterations are beneficial, especially when the default false time-step for temperature is reset to a large value so that little or no under-relaxation is used for temperature.

- Potential (when Joule Heating is switched on), 1000 iterations.

### Related Topics

[Solver Control Tab - Variable Solution Control Section](#)

## Fan Relaxation

Fan Relaxation is used specifically to control the variation in the flow calculated at fans for each outer iteration.

Select values between 0.5 and 0.9. Fan Relaxation is particularly useful in cases where fans are specified with a non-linear fan curve and the operating point is near a low gradient section of the curve.

### Related Topics

[Solver Control Tab](#)

## Monitor Point Convergence for Temperature

For all solutions, monitor points can be used to determine whether the solved variables have reached a steady state. However, they are particularly useful for determining convergence in the low-level stable and low-level oscillation profiles.

Under these circumstances, the variation in all variables should be assessed and if small enough, then a considered judgment should be made that the solution is effectively converged.

In many situations encountered in electronics, there may well be a natural instability in the flow field that can manifest itself in a low-level oscillation profile. The monitor points under these circumstances can be used for making assessment of the level of instability involved.

To use the Monitor Point Convergence for Temperature option effectively, you must make sure that monitor points are placed at every point where the temperature is required to be monitored for convergence.

Convergence is achieved when:

$$(standard\_deviation \times 3) < required\_accuracy$$

where:

*required\_accuracy* is specified in the [Solver Control Tab](#), default 0.5 degC.

$$\text{Standard Deviation} = \sqrt{\frac{1}{N} \sum_{i=1}^N (x_i - \bar{x})^2}$$

where:

$x_i$  is the temperature at the  $i$ th iteration.

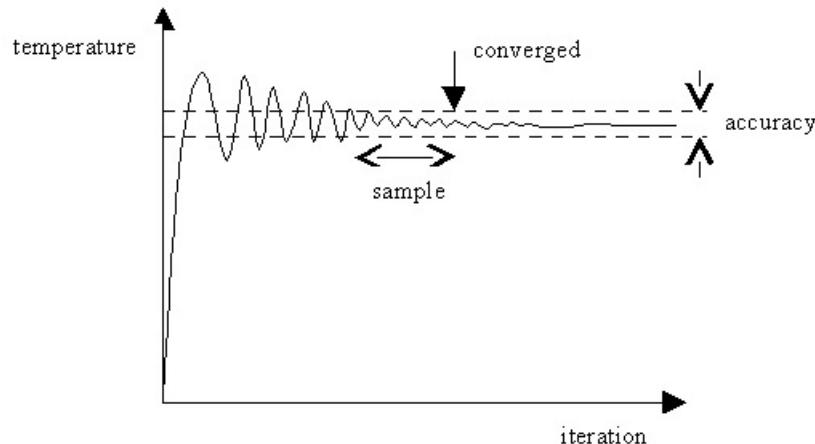
$\bar{x}$  is the mean temperature for the total number of iterations.

$N$  is the total Number of Iterations, as specified in the [Solver Control Tab](#), default 30.

By adjusting the Required Accuracy and the Number of Iterations (sample size), the moment the monitor point is judged to be converged can be changed.

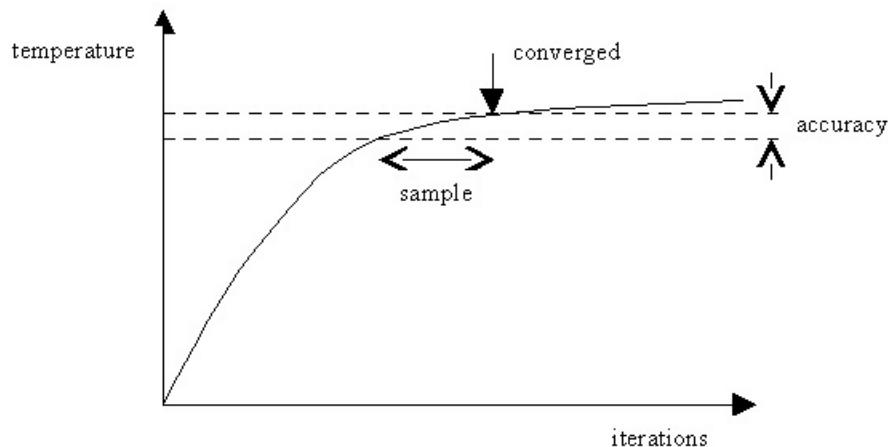
If the temperature is converging, [Figure 12-7](#) illustrates the moment of accepted convergence.

**Figure 12-7. Required Accuracy**



If the temperature is changing by a constant amount, the acceptable gradient is defined by the Number of Iterations and the Required Accuracy as shown in [Figure 12-8](#).

**Figure 12-8. Number of Iterations**



## Related Topics

[Solver Control Tab - Monitor Point Solution Control Section](#)

## Error Field

In situations where the convergence is of the low-level stable or low-level oscillation type, it can be useful to examine the error field for particular variables.

The Error Field option in the [Solver Control Tab](#) stores the residual error of a variable. These errors (Field Error scalar field) can be displayed in Analyze mode for analysis.

Usage:

- In Analyze mode, check where the maximum and minimum values are located. Often, this points to a location of little interest or concern, but it may be exhibiting some kind of instability which is keeping the overall residual level high.
- The maximum and minimum values identify the regions that are causing the solution to oscillate or stabilize at a higher residual error. If this is an irrelevant area, then a check of the monitor points should reveal whether the solution has reached a steady-state, and the solution should be considered to be effectively converged.
- The error field can also be used to help identify problem areas in the grid and where it may need to be refined. It is possible that instabilities in the solution are actually being caused by poor grid structure. This will usually be related to an aspect ratio problem or a large step change in grid size between adjacent grid cells.

### Storing Error Field Values

The residual error for temperature ( $r_T$ ) in a grid cell is the extent to which the temperature equation is not satisfied:

$$r_T = (C_0T_0 + C_1T_1 + C_2T_2 + C_3T_3 + C_4T_4 + C_5T_5 + C_6T_6 + S) - ((C_0 + C_1 + C_2 + C_3 + C_4 + C_5 + C_6) \times T)$$

When  $r_T$  equals zero this equation is exactly satisfied.

The quantities stored in the error field are the cell residuals divided by the termination residual, for example, for temperature in each cell the quantity stored in the error field is  $r_T \div E_T$ . It is this normalized error distribution which may be displayed.

For the pressure variable,  $r_p$  is the continuity error in a grid cell, that is, it is the difference between the mass flowing in and the mass flowing out of the cell. Display of the continuity error field will help to identify locations of the largest mass imbalances, it can sometimes be the case that all the error is located in one grid cell. Again, what is stored in the error field and what is, therefore, displayed is  $r_p \div E_p$ , the residual errors normalized by the termination residual for pressure.

**Note**

- ❑ The error field is normalized by the appropriate termination criterion, so that the sum of the absolute error field values is the value plotted on the residual plots in Profiles. For an example of the residual plots, see “[Monitor Points v Iteration Plot \(Steady State\)](#)” on page 390.
- 

## Related Topics

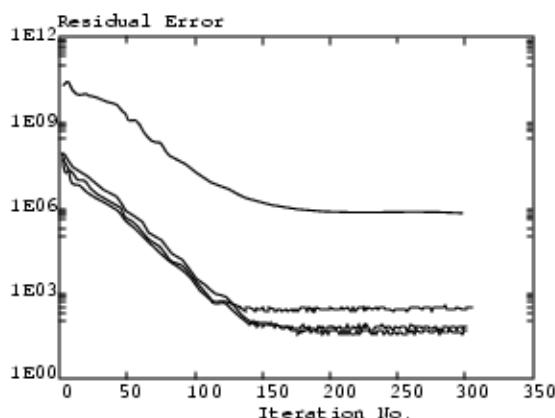
[Solver Control Tab](#)

## User-Specified Termination Residuals

You can choose not to use the program-calculated values and set your own values for the termination residuals.

For example, you may want a tighter convergence criterion than that provided automatically. If you set the termination residuals to very small values (for example,  $10^{-10}$ ), you will force the program to perform the specified number of iterations as it is unlikely ever to satisfy such a tight convergence criterion. In this event (that is, that the program does all the iterations), it does not mean that the solution is not converged, convergence may have been reached but the residuals have got stuck at levels below which they cannot be reduced because of computational precision limits.

**Figure 12-9. Residual Error**



**Tip**

- ❑ Rather than enter a user-specified value, a quick way of changing the termination residual is to use the slider bar to increase or decrease the program-calculated value.
- 

**Note**

- ❑ Care is needed with the termination residual settings for K.E. Turb. as the defaults are based solely on the inflow values and do not account for any internal generation.
-

## Related Topics

- [The Solution Process](#)
- [Program-Calculated Termination Residuals](#)
- [Solver Control Tab - Variable Solution Control Section](#)

# Double-Precision Solver

If the residual errors flatten off at an unacceptably high level (for example, greater than 100), stop the solution, activate the double-precision solver, then continue the solution.

Activation of the double-precision solver will reduce residuals, commonly temperature, when there are large areas of extremely high conductivity (and therefore almost uniform temperature) present in the model. Often, this does not result in a change of the temperature of such cuboids.

An alternative method of determining convergence is to utilize the monitor point convergence criteria on the [Solver Control Tab](#), see “[Monitor Point Convergence for Temperature](#)” on page 414.

The double-precision solver has an additional memory overhead which made need to be taken into consideration. If the solver needs more memory than is physically available, then it will use paged memory, which will increase the solve time

There are two methods of activating the double-precision solver:

- Environment Variable Method

Set the FLO\_DOUBLEP environment variable (see “[Environment Variables](#)” on page 615) to any value. Any loaded project will then be solved using the double-precision solver. The setting of this environment variable is not displayed in the GUI.

- Graphical User Interface Method

Select the Use Double Precision Solver option in the [Solver Control Tab](#). This setting is saved with the project, therefore, the project will use the double-precision solver each time the project is loaded.

# Chapter 13

## Viewing Results

---

The Project Manager in Analyze mode.

<b>Analyze Mode</b> .....	<b>419</b>
<b>Results Tree</b> .....	<b>422</b>
<b>Transient Time Step Selector</b> .....	<b>423</b>
<b>Results Tree Property Sheet Functionality</b> .....	<b>424</b>
<b>Analyze Mode Mouse Pointer</b> .....	<b>426</b>
<b>Analyze Mode Operations</b> .....	<b>428</b>
Changing Mouse Mode in Analyze Mode .....	428
Changing the Appearance of the GDA in Analyze Mode .....	428
Manipulating the View in Analyze Mode.....	430
Saving the Current Analyze State .....	430
Loading a Previously Saved Analyze State .....	430
Reinitializing the Analyze State .....	431
Working With Plots .....	432
Working With Annotations.....	442
Working with Viewpoints.....	446
Working With Animations .....	448
Selecting Geometry Associated With a Plot or Annotation .....	450
Exporting Cell by Cell Results Data.....	450
<b>Results Property Sheets</b> .....	<b>454</b>
Scalar Field Variable Property Sheet .....	455
Vector Field Property Sheet .....	457
Isosurface Property Sheet.....	459
Plane Plot Property Sheet .....	460
Surface Plot Property Sheet .....	463
Particle Source Property Sheet .....	464
Annotation Property Sheet .....	469
Animation Property Sheet.....	473
Saved Tables Property Sheet .....	474
Viewpoint Property Sheet.....	475

## Analyze Mode

When the Project Manager is in Analyze mode, the GDA views can be overlaid with graphical representations of results.

The Introduction to the Analyze Mode User Interface video shows aspects of the user interface when the Project Manager is in Analyze mode:

- Switching between Create and Analyze mode
- The Results tree
- The Tables window
- Analyze mode icons
- The **Analyze Mode** tab of the User Preferences dialog box
- Saving and loading state files
- Exporting legacy tables
- The Animation toolbar
- Annotate mode



Analyze mode is entered automatically when a solve has completed or been interrupted. If the solve has been interrupted, then the results from the incomplete solve are loaded.

To enter Analyze mode manually, press F10 or click the **Analyze** icon.

---

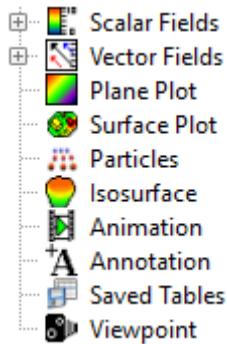
**Tip**  
 Press F10 to toggle between Create and Analyze mode.

---

Analyze mode icons provide quick access to commonly used functions.



The Results tree, on the right-hand side of the GDA, is always displayed when in Analyze mode.



## The GDA in Analyze Mode

The GDA provides views of the geometry and results in 3D world scenes, as well as the ability to create animations of the airflow.

The GDA can be used to view the following:

- **Maximum and Minimum Values** — Maximum and minimum values may be displayed for any stored variable.
- **Surface Temperature Values** — Surface temperature value plots may be displayed over selected bodies or the complete model.
- **Values over a Visualization Plane** — A visualization plane may be positioned to intersect a point of interest in the solution domain. Contours and vectors of any stored variable may be displayed over the plane.
- **Isosurfaces** — Points at a given temperature may be joined to create a constant temperature isosurface. This feature is particularly useful for high-lighting areas above a certain critical temperature level.
- **Particle Source Traces** — Particles can be released within the domain to visualize the flow or movement of vector variables. This feature is extremely effective for understanding complex re-circulating flows or complex heat flows.

If the Display Geometry Without Expanding Tree option is selected in the [User Preferences Dialog Box - Analyze Mode Tab](#) the geometry of all assemblies is displayed, regardless of whether they are expanded or collapsed in the data tree.

## Viewing in Analyze Mode

There are three view modes:

- **Manipulate Mode** — The default mode. Use when changing the view.

- **Select Mode** — Use when selecting objects.
- **Annotate Mode** — Use to create an annotation at geometry or on a plot.

## Transient Analysis Results

When transient analysis results are loaded, the Transient Time Step selector is displayed, see “[Transient Time Step Selector](#)” on page 423.

## Restrictions in Analyze Mode

- Unavailable features in Analyze mode:
  - Object resizing: selected objects do not have grab handles.
  - The **Cut**, **Copy** and **Paste** options of the **Edit** Menu.
  - The **Promote**, **Demote**, **Decompose**, **Activate**, **De-activate**, **Rotate Clockwise/Counterclockwise**, **Snap Toggle**, **Align**, **Move**, and **Mirror** options of the **Geometry** menu.
  - The **Inflate Grid**, **Deflate Grid**, and **Toggle Localize Grid** options of the **Grid** menu.
  - The Workplane toolbar.
  - The Project Attributes and Library trees.
  - MCAD Bridge.
  - EDA Bridge.
- Available but read-only features in Analyze mode:
  - Model tab. Hiding and changing the transparency of geometry and set lightweight assemblies remain active.
  - System Grid property sheet.
  - Model Setup tab, including the Transient Solution and Solar Configuration dialog boxes.
  - Solver Control tab.

# Results Tree

Use the Results Tree to explore results data and overlay the GDA with plots.

## Fields

The Scalar Fields and Vector Fields nodes expand to display lists of the scalar and vector field variables that can be plotted. The lists depend on the contents of the solution set, which in turn depends on the type of solution and options taken to store additional variables.

## Plots

Different types of plots can be created: values over an plane, surface temperatures, particle source traces, and isosurfaces. Instances of each type are listed in separate nodes: Plane Plot, Surface Plot, Particles, and Isosurface.

## Animations

Animations are created for each plane plot and particle plot, and transient variations.

## Annotations

Annotations can be added for reference or to enhance illustrations for hardcopy or inclusion in reports. Annotations can show maximum or minimum scalar field values in the model, values on plane or particle plots, or values in the centers of geometry.

## Viewpoints

Views of the model can be saved and returned to.

## Saved Tables

Saved tables are configurations of tables associated with selected geometry.

## Related Topics

[Results Property Sheets](#)

[Results Tree Property Sheet Functionality](#)

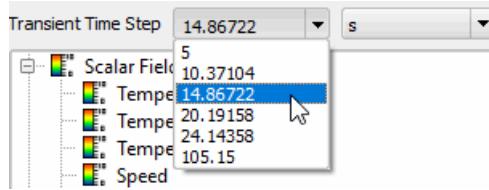
# Transient Time Step Selector

Loading results saved at transient analysis Save Times.

After a transient solution, the results that are loaded are those saved at the end of the transient period.

If save times have been defined, then the results at each saved time can be reloaded using the Transient Time Step selector.

The selector is only displayed when a transient analysis solution has been run and is located above the Results tree, as shown in [Figure 13-1](#).

**Figure 13-1. Results Transient Time Step Selector**

The dropdown list enables you to select any of the save times. The results for the selected save time are then loaded.

---

**Tip**

**i** Once the dropdown list has been activated with the mouse, you can use the up and down arrow keyboard keys to navigate the list. This provides a useful way to interact with the results.

---

The time units default to seconds (s), but can be changed using the units dropdown list.

The last selected save time is preselected when the project is reloaded.

## Related Topics

[Save Times](#)

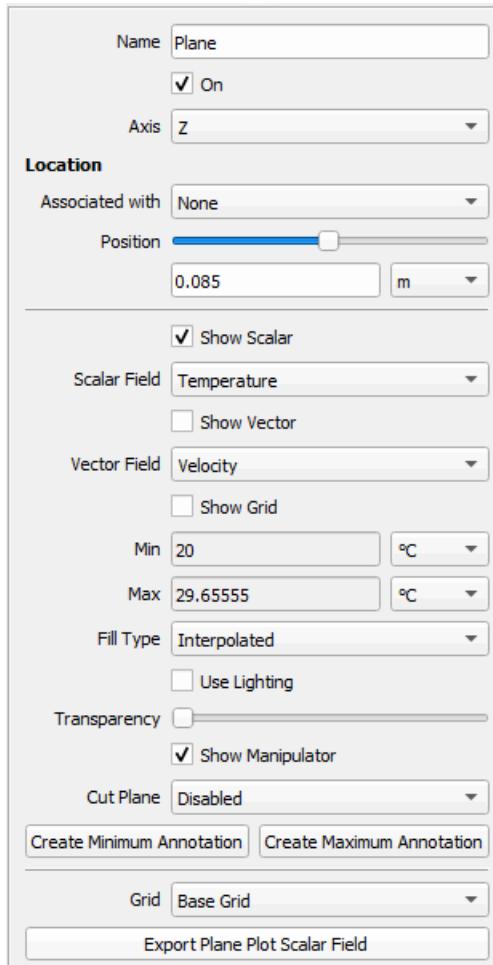
# Results Tree Property Sheet Functionality

Editing single and multiple properties.

[Figure 13-2](#) shows an example of a property sheet displayed under the results tree.

Property changes may require over-writing, selection from a dropdown list, or moving a slider bar, depending upon the data type.

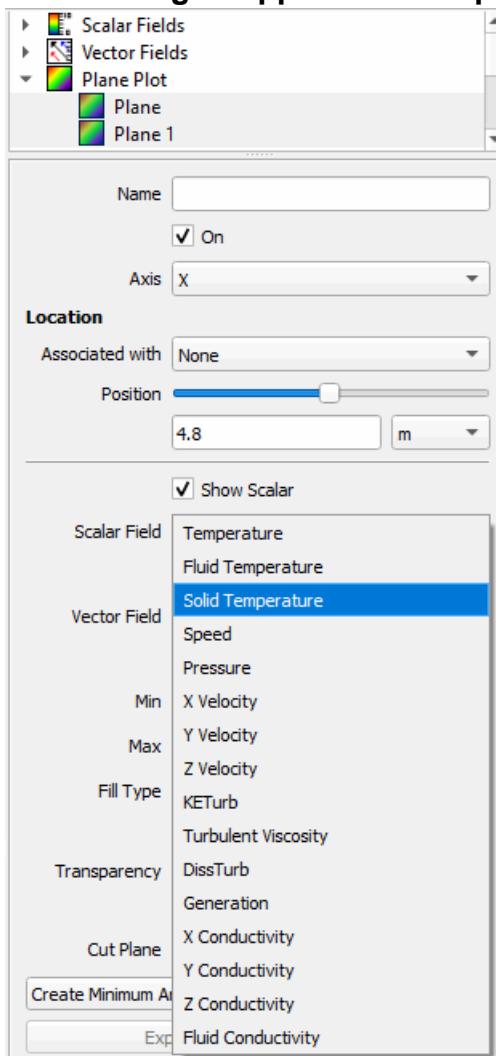
**Figure 13-2. Plane Plot Property Sheet**



## Multiple Selection of Items in Property Sheets

Changes can be applied to a group of items. When multiple results tree nodes or GDA objects are selected, the property sheet data entry fields are populated with those fields common to all. Editing the common fields updates all the selected objects. The example in [Figure 13-3](#) shows how two plane plots can be redefined to show different scalar values.

**Figure 13-3. Changes Applied to Multiple Plots**



## Related Topics

[Results Property Sheets](#)

## Analyze Mode Mouse Pointer

The mouse pointer changes, depending on what you are pointing to or the task you are performing.

**Table 13-1. Analyze Mode Mouse Pointers**

	Mode	Behavior
	Select	Selects geometry. Selected geometry can be hidden or made transparent.

**Table 13-1. Analyze Mode Mouse Pointers (cont.)**

Mode	Behavior
	Manipulate Changes the view when the mouse is dragged.
	Pan Pans the view in Manipulate Mode when the mouse is dragged with the middle button pressed.
	Zoom Shown when combining Ctrl with holding the mouse wheel and dragging to zoom in or out of the view.  <b>Note:</b> This icon is not shown when turning the mouse wheel to zoom in or out.
	Table Cell Selection Selects table cells in rows/columns.

# Analyze Mode Operations

---

Post-processing operations on results data.

<b>Changing Mouse Mode in Analyze Mode .....</b>	<b>428</b>
<b>Changing the Appearance of the GDA in Analyze Mode .....</b>	<b>428</b>
<b>Manipulating the View in Analyze Mode .....</b>	<b>430</b>
<b>Saving the Current Analyze State .....</b>	<b>430</b>
<b>Loading a Previously Saved Analyze State .....</b>	<b>430</b>
<b>Reinitializing the Analyze State .....</b>	<b>431</b>
<b>Working With Plots.....</b>	<b>432</b>
<b>Working With Annotations.....</b>	<b>442</b>
<b>Working with Viewpoints .....</b>	<b>446</b>
<b>Working With Animations .....</b>	<b>448</b>
<b>Selecting Geometry Associated With a Plot or Annotation.....</b>	<b>450</b>
<b>Exporting Cell by Cell Results Data .....</b>	<b>450</b>

## Changing Mouse Mode in Analyze Mode

There are three possible mouse modes when viewing results: Select, Manipulate, and Annotate.

### Prerequisites

- The Project Manager must be in Analyze mode.

### Procedure

1. To change to Select or Manipulate mode, refer to “[Changing Mouse Mode in Create Mode](#)” on page 259.
2. To change to Annotate mode, click the **Annotate Mode** icon  or choose **Viewer > Annotate Mode**.

### Related Topics

[Working With Annotations](#)

## Changing the Appearance of the GDA in Analyze Mode

The ambient light level, and title and legend sizes are controlled by user preferences settings.

## Procedure

1. Press F11 to open the User Preferences dialog box.
2. Click on the **Analyze Mode** tab.

You have a choice of options:

If you want to...	Do the following:
Hide the title.	The title (project name) is shown by default in each view. To hide the title, uncheck the Show Title check box.
Change the size of the title.	Either enter a value in, or use the stepper arrows of, the Title Font Size field.
Change the size of the legend font.	Either enter a value in, or use the stepper arrows of, the Legend Font Size field.
Change the background color.	Optionally, check the Gradient Background check box. Click on the Background Color swatch (or Top/Bottom Background Color swatch) and use the color picker (Modify Color dialog box) to select a color.
Change the number of significant figures displayed in the legend.	Increase or decrease the Number of Significant Figures field.
Change the lighting to improve 3D visibility.	Adjust the Ambient Light Level slider bar. The lighting effect is most noticeable when the bar is moved fully to the left.

3. Click **OK**.

Changes apply to the current project and any subsequently opened projects, and are saved across sessions.

4. To restore the defaults, open the **Analyze Mode** tab of the User Preferences dialog box, click **Restore Defaults** then click **OK**.

---

### Note

 The scope of this action is restricted to the settings in the currently opened tab of the dialog box.

---

## Related Topics

[User Preferences Dialog Box - Analyze Mode Tab](#)

## Manipulating the View in Analyze Mode

Changing the view in Analyze mode is similar to changing the view when in Create mode.

---

### Note

 The GDA background color in Analyze mode is set using the **Analyze Mode** tab of the User Preferences dialog box.

---

### Related Topics

[Viewing Geometry](#)

[User Preferences Dialog Box - Analyze Mode Tab](#)

## Saving the Current Analyze State

You can save post-processing configurations of plots, annotations, and so on for subsequent loading. This allows you to switch between different post-processing configurations of results.

### Procedure

1. Choose **Analyze > Save State As**.

The Save State dialog box is opened at `<solution_dir>/<project_dir>/PDProject`.

---

### Note

 Saving the state file to this directory will make sure it is included in any exported project PACK files.

---

2. Click **Save**.

### Results

The results state file contains scalar field and vector field settings, and definitions of any plots, annotations, animations, saved tables, and viewpoints.

### Related Topics

[Loading a Previously Saved Analyze State](#)

[Reinitializing the Analyze State](#)

## Loading a Previously Saved Analyze State

Return to a previous configuration of results.

### Prerequisites

- A saved *xml* results state file.

## Procedure

1. Choose **Analyze > Load State**.

The Load State dialog box is opened at *<solution\_dir>/<project\_dir>/PDProject*.

The file may have been saved to a different directory.

2. Select the file and click **Open**.

## Results

Any scalar field and vector field settings are reinstated, and any plot, annotation, animation, saved table, and viewpoint definitions are reloaded.

## Related Topics

[Saving the Current Analyze State](#)

[Reinitializing the Analyze State](#)

# Reinitializing the Analyze State

Removing post-processing items.

## Restrictions and Limitations

- There is no undo.



### Caution

If you want to keep your work you are advised to save the current analyze state.

---

## Procedure

Choose **Analyze > Reinitialize State**.

## Results

Scalar field and vector field settings are returned to the default settings.

All plots, annotations, animations, saved tables, and viewpoints are removed.

## Related Topics

[Saving the Current Analyze State](#)

[Loading a Previously Saved Analyze State](#)

## Working With Plots

How to define and manipulate plane and isosurface plots of results.

<b>Creating a Plane Plot.....</b>	<b>432</b>
<b>Associating an Existing Plane Plot with Geometry.....</b>	<b>434</b>
<b>Associating an Existing Plane Plot with Results.....</b>	<b>434</b>
<b>Moving a Plane Plot.....</b>	<b>434</b>
<b>Cutting a Plane Plot.....</b>	<b>435</b>
<b>Creating a Surface Plot.....</b>	<b>436</b>
<b>Creating an Isosurface.....</b>	<b>437</b>
<b>Creating and Moving Particle Sources .....</b>	<b>438</b>
<b>Moving a Plot Legend .....</b>	<b>440</b>

### Creating a Plane Plot

How to display results as contours and vectors. Plots are 2D GUI objects which extend to the full size of the domain.

#### Prerequisites

- The Project Manager is in Analyze mode.

#### Video

The Plots Associated With Geometry video demonstrates that a plot associated with geometry moves with the geometry.



#### Procedure

1. Optionally, select one or more objects.
2. Click the **Create Plane Plot** icon  or right-click the Plane Plot node of the Results tree and choose **Create Plane Plot**.

## Results

The first plane plot created is named Plane. Subsequent plots are named Plane <n>.

If more than one object was selected, then a plane plot is created for each selected object.

The plot cuts midway through the smallest dimension of the object.

This can be useful when investigating the temperature variation within a specific PCB layer, where manually locating the plane with the layer object would otherwise be difficult to achieve.

If no object was selected, then the plot cuts midway through the smallest dimension of the overall domain.

You can display scalar and/or vector values in plane plots.

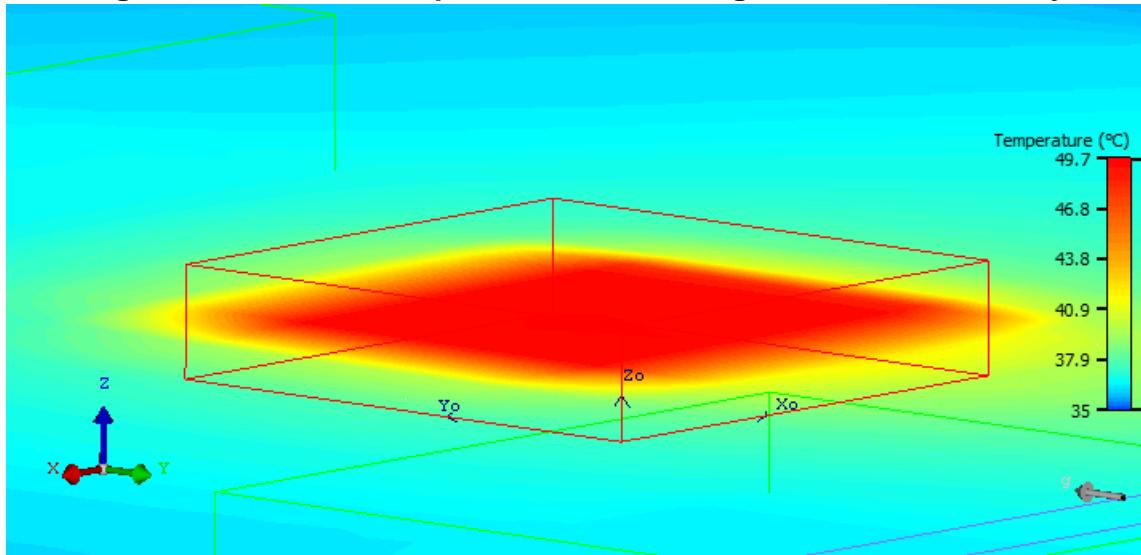
Scalar values are shown as contour lines, color-filled grid cells or interpolated (smoothed) color-filled grid cells.

Vector values are show as arrows originating from the centers of cells. The direction of the arrow indicates the local flow direction, unless normalized to the plane or perpendicular to the plane. The length and color of the arrow indicate the magnitude.

## Examples

Figure 13-4 shows an example of a plane plot at its default position, half-way through the smallest dimension of the selected geometry.

**Figure 13-4. Plane Temperature Plot Through Selected Geometry**



## Related Topics

[Plane Plot Property Sheet](#)

## Associating an Existing Plane Plot with Geometry

Moving a plot to selected geometry. Once the association is made, the plot will follow the object around for future model variations, including Command Center variations.

### Procedure

1. Select the plane plot in the Results tree.
2. Select the object and click the **Update** button in the plane plot property sheet.

### Results

The plot axis remains the same, and the plot will move to the midpoint of the newly-selected object.

The selected geometry is listed in the read-only text field next to the **Update** button.

## Associating an Existing Plane Plot with Results

Moving a plot to the location of the minimum or maximum scalar value. Once the association is made, the plot will follow the maximum or minimum value around for future model variations, including Command Center variations.

### Procedure

1. Select the plane plot in the Results tree.
2. In the plane plot property page, select Field from the “Associated with” dropdown list.
3. In the left-hand side Field dropdown list, select Max or Min.
4. In the right-hand side Field dropdown list, select a scalar variable.

### Results

The plot moves to the location of the minimum or maximum scalar variable value.

## Moving a Plane Plot

You can move a plane plot within the limits of the solution domain to a specific location, or incrementally, one cell at a time.

### Procedure

1. Select the plane plot.

---

**Note**

 If the plot is associated with geometry or a scalar field then the association will be lost when the plot is moved. However, the action can be undone.

---

2. You have a choice:

If you want to...	Do the following:
Move the plot by dragging the plot along an axis.	<ol style="list-style-type: none"><li>1. If not already activated, activate the manipulator by checking the Show Manipulator check box in the plot property sheet. The manipulator is located at a corner of the plot and is made up of three sets of arrow heads, pointing in both directions along each axis.</li><li>2. Make sure you are in Select mode, click on the arrow head pointing in the required direction and drag. Although the manipulator can be dragged along any axis, the plane plot will only move along one axis.</li></ol>
Move the plot to a specific location.	Either enter the value in the Position field in the plot property sheet, or drag the Position slide bar until the value is displayed in the field.
Move the plot one cell at a time	Press the right-arrow or left-arrow key.

## Cutting a Plane Plot

You can show the geometry clipped above or below a plane plot.

### Procedure

1. Select the plane plot.
2. In the property sheet, at the Cut Plane property, select Above or Below.

### Results

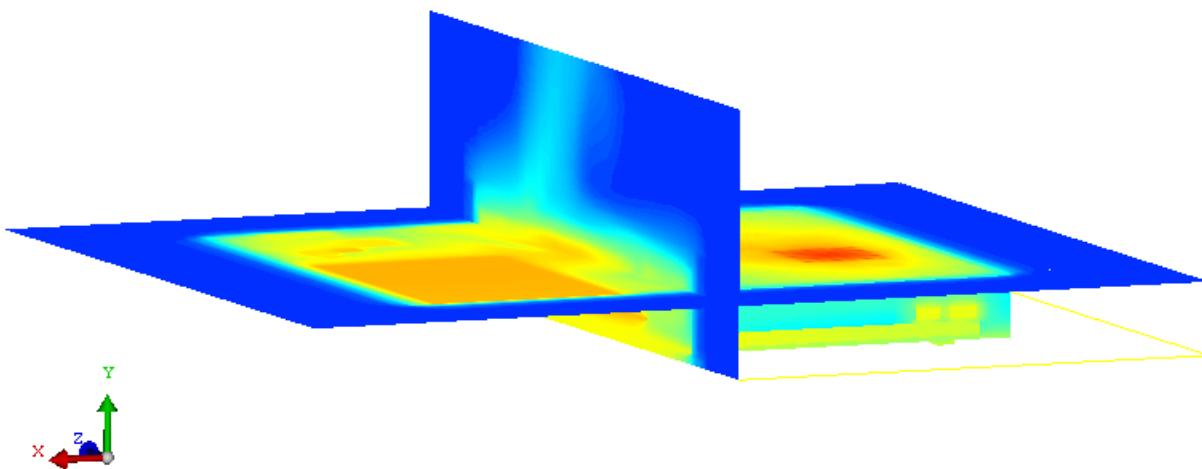
The model is cut at the plane plot such that it is not rendered above or below the plot.

You can cut more than one plane plot.

### Examples

In [Figure 13-5](#), geometry with a surface plot has been cut above an X plane plot and a Y plane plot.

**Figure 13-5. Cut Plane Plots**



## Creating a Surface Plot

Surface plots show surface temperatures over the geometry and are useful for spotting hot-spots in the model.

### Restrictions and Limitations

- Not available for Flow Only solutions.

### Prerequisites

- Surface Temperature values must have been generated when the project was solved.  
This is controlled by selecting Yes for Surface Temperatures in the Stored Variables section of the **Model Setup** tab.
- The Project Manager is in Analyze mode.

### Procedure

1. Select one or more objects.

If you select an assembly, then the surface plot will cover the surfaces of the objects within that assembly and any sub-assemblies.

If you select nothing, then the surface plot will cover the external surfaces of the objects in the Root Assembly.

2. Click the **Create Surface Plot** icon 
3. Use the property sheet to define the representation of the surface plot.  
If you want the plot to cover a different object or set of objects then select the object(s) then click **Update**.

The selected geometry is listed in the read-only text field next to the **Update** button.

## Results

The first surface plot created is named Surface Plot. Subsequent plots are named Surface Plot  $< n >$ .

## Related Topics

[Surface Plot Property Sheet](#)

# Creating an Isosurface

An isosurface plot is a surface joining all points which have the same value for a given variable.

## Prerequisites

- The Project Manager is in Analyze mode.

## Procedure

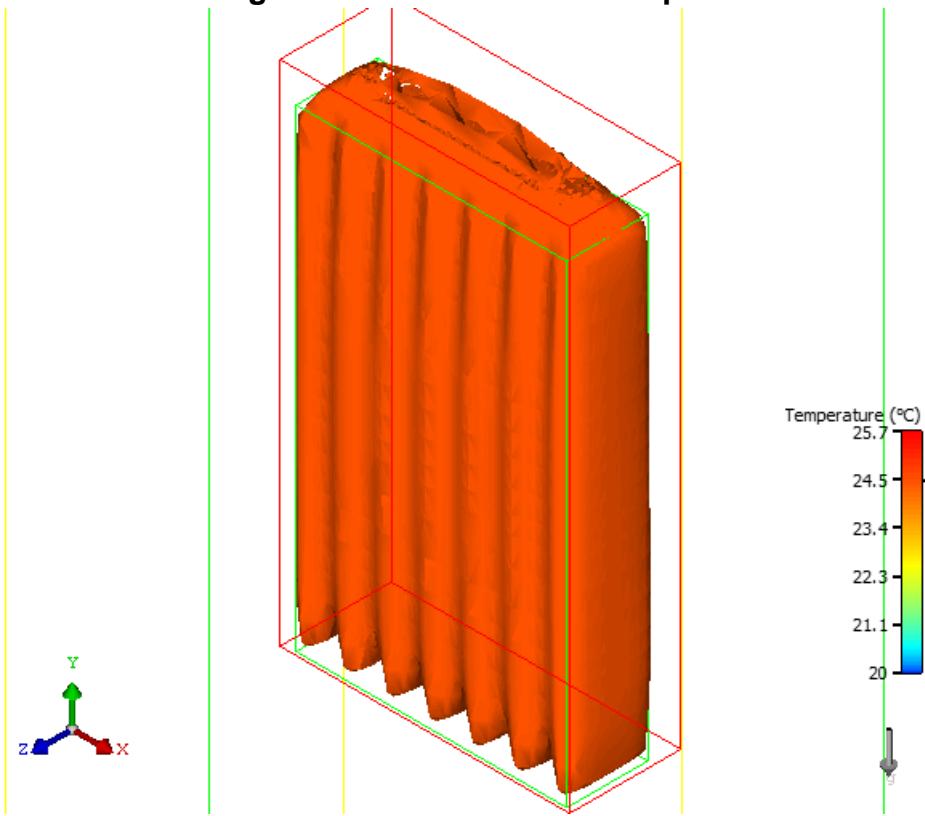
1. Click the **Create Isosurface** icon 
2. You can customize the plot by changing values and settings in the plot property sheet.

## Results

The first isosurface plot created is named Isosurface. Subsequent plots are named Isosurface  $< n >$ .

## Examples

**Figure 13-6. Isosurface Example**



## Related Topics

[Isosurface Property Sheet](#)

## Creating and Moving Particle Sources

Use particles to plot flow through fluids.

### Prerequisites

- The Project Manager is in Analyze mode.

### Procedure

1. Optionally, select the object or objects from where you want particles to be sourced.  
If you select nothing, then the particles sources will be located in the center of the root assembly.
2. Click the **Create Particles** icon ( ) or right-click the Particles node of the Results tree and choose **Create Particles**.

3. In the property sheet, optionally select a different Source Plane Orientation from the default.
4. To move the particle source, select a Manipulator Type other than None (the default):
  - The Translation manipulator is a set of axial arrows. To move the sources along an axis, select Translation, then drag an arrow.
  - The Scale manipulator is a set of four white corner handles on the bounding box of the selected source. The handles are not visible when viewing along an axis. To resize the source shape, select Scale, then drag the handles.
5. To relocate the particle sources to a different object or objects, then select the object or objects and click **Update**.

The selected geometry is listed in the read-only text field next to the **Update** button.

## Results

A Particle Source is added to the Particles node of the Results tree. The first particle source is named Particle Source. Subsequent plots are named Particle Source <n>.

To copy or delete a particle source, right click on the particle source in the Results tree and choose **Copy** or **Delete**.

For each object that was selected, sources are located at the center of the bounding box of the object. The plane over which the sources are distributed is the same as that of a plane plot created using the same selection.

If you move an object associated with particle sources, then the sources will move.

There are two types of particle streaming:

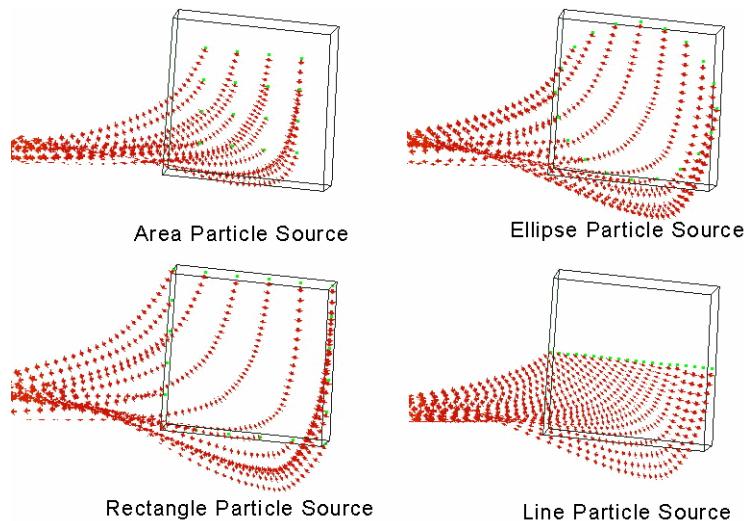
- Velocity vector flows can be used to visualize fluid motion.
- Heat Flux vector flows (if available) can be used to visualize the movement of heat through both solids and fluids.

For Heat Flux values to be available, the model must have been solved with the Heat Fluxes option enabled in the Stored Variables section of the **Model Setup** tab.

**Figure 13-7. Particle Smear and Width**

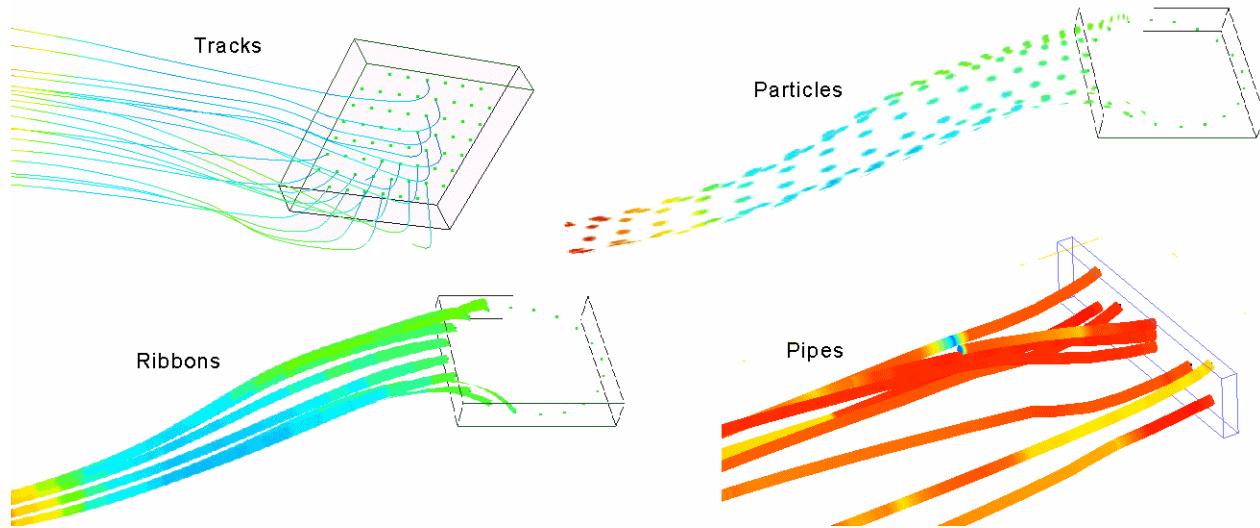


The sources can be distributed over difference shapes.



Three types of streamlines are available.

**Figure 13-8. Track, Particle, Ribbon and Pipe Streamlines**



## Related Topics

[Particle Source Property Sheet](#)

## Moving a Plot Legend

When plots are shown, legends can be moved so that they do not obscure the view of the plot.

## Restrictions and Limitations

- There is no undo.

## Prerequisites

- You are in Select mode.
- The Show Legend check box in the Scalar Field Variable property sheet is checked.

## Procedure

1. Select the legend by clicking inside or close to it.

A red bounding box surrounds the legend.

The respective Scalar Fields node in the Results tree is selected and the Scalar Field Variable property sheet is opened.

2. Drag the legend to a new position.

The legend remains selected until another item is selected or the GDA is clicked in an area away from the legend.

## Working With Annotations

Adding annotations to global maximum or minimum scalar values, geometry, or maximum or minimum scalar values on a plot.

The following icons are used to distinguish between different types of annotation:



Scalar annotation



Geometry annotation



Plot annotation

<b>Adding an Annotation to Geometry or a Plot.....</b>	<b>442</b>
<b>Adding a Global Maximum or Minimum Annotation.....</b>	<b>443</b>
<b>Adding an Annotation to the Center of a Geometry Object.....</b>	<b>443</b>
<b>Adding a Maximum or Minimum Annotation to a Plot.....</b>	<b>444</b>
<b>Adding a Note to an Annotation.....</b>	<b>444</b>
<b>Moving an Annotation.....</b>	<b>445</b>

## Adding an Annotation to Geometry or a Plot

Annotations, showing scalar values, can be added to locations in the model provided they are on geometry or plots.

### Prerequisites

- The Project Manager is in Analyze mode.

### Procedure

1. Click the **Annotate Mode** icon or choose **Viewer > Annotate Mode** to enter Annotate mode.

The cursor icon changes to cross-hairs.

As you move the cursor across the GDA, an annotation updates a variable value. For geometry, the variable defaults to temperature. For plots, the variable is the variable that is plotted.

2. Click the mouse to create an annotation.

### Results

If the clicked location was on a plot, then a plot annotation is created.

If the clicked location was on geometry, then a geometry annotation is created.

The name of the annotation is the name of the plot or object.

When surface plots are displayed, if you click on an edge of the surface plot, then the added annotation will be a plot annotation. If you prefer to add a geometry annotation, then click away from an edge, or remove displayed edges by unchecking the Plot Edges check box in the surface plot property sheet.

## Adding a Global Maximum or Minimum Annotation

You can identify the location in the model of a scalar maximum or minimum value by adding an annotation.

### Procedure

1. Expand the Scalar Fields branch of the Results tree.
2. Right-click a scalar field and choose **Create Annotation At Global Minimum** or **Create Annotation At Global Maximum**.

### Results

The annotation icon is 

The annotation name is:

*<scalar field name>* : Global [Minimum | Maximum].

## Adding an Annotation to the Center of a Geometry Object

Adding an annotation to one or more selected objects to show scalar values at those objects.

### Prerequisites

- The Project Manager is in Analyze mode.

### Procedure

1. Select one or more objects.
2. Right-click the Annotation node of the Results tree and choose **Create Annotation**.

Alternatively, click the **Create Annotation** icon 

### Results

An annotation is created for each object selected at the center of the object.

The annotation icon is 

The default name of the annotation is the name of the object.

The type of object is shown in parentheses.

You can add more than one annotation to an object.

## Adding a Maximum or Minimum Annotation to a Plot

You can identify the location on a plot of a scalar maximum or minimum value by adding an annotation.

### Procedure

1. Expand a plots branch of the Results tree.
2. Right-click a plot and choose **Create Minimum Annotation** or **Create Maximum Annotation**.

Alternatively, click the **Create Minimum Annotation** or **Create Maximum Annotation** button in the plot property sheet.

### Results

The annotation icon is

The default name is the name of the plot.

If you select another plot using the Plot dropdown list in the property sheet then the Name will change to that of the selected plot and the annotation will move to that plot.

The annotation name as presented on the annotation is:

*<plot name>* : [Minimum | Maximum]

Where *<plot name>* is the Name defined in the **Definition** tab of the annotation property sheet.

The default Scalar Field of the annotation is the Scalar Field of the plot.

## Adding a Note to an Annotation

Short user notes can be added to an annotation.

### Restrictions and Limitations

- The note can be up to 32 characters long.

### Procedure

1. Select the annotation.
2. Type text in the Note field of the **Definition** tab of the annotation property sheet.

### Results

The text is inserted at the top of the annotation, above the annotation name.

## Moving an Annotation

You can move the annotation anchor point, or only move the label. The anchor point is the position where the value is sampled

### Restrictions and Limitations

- The position of the anchor point is constrained to the solution domain.
- The position of the label is constrained to within  $\pm 100$  pixels of the anchor point.

### Procedure

1. Select the annotation in the Results tree.
2. Open the **Location** tab of the annotation property sheet.
3. You have a choice:

If you want to...	Do the following:
Move the anchor point and label.	Use the Position X, Y and Z slider bars, or enter coordinate values.
Only move the label.	Use the X and Y Offset slider bars.

### Results

Values shown in the annotation label are updated when you move the anchor point. The label moves with the anchor point.

## Working with Viewpoints

Creating, and jumping to, a viewpoint.

**Creating a Viewpoint** ..... **446**

**Jumping to a Viewpoint** ..... **446**

### Creating a Viewpoint

Save the current view of the GUI for later recall.

#### Prerequisites

- The Project Manager is in Analyze mode.

#### Procedure

1. Change the view of the GUI to one that you want to save.
2. Click the **Save Current Viewpoint** icon  or right-click the Viewpoint node of the Results tree and choose **Save Current Viewpoint**.

#### Results

The first viewpoint created is named Viewpoint. Subsequent viewpoints are named Viewpoint <n>.

#### Related Topics

[Viewpoint Property Sheet](#)

[Jumping to a Viewpoint](#)

### Jumping to a Viewpoint

Recalling a saved view of the GUI.

#### Procedure

Double-click the viewpoint in the Results tree.

Alternatively, select the viewpoint then click the **Jump to Viewpoint** button in the property sheet.

#### Results

The view of the GUI changes to that of the viewpoint.

#### Related Topics

[Viewpoint Property Sheet](#)

## Creating a Viewpoint

# Working With Animations

How to create animations and movies from results.

Animating Plots .....	448
Saving Animations .....	449

## Animating Plots

Movement of a plane plot or particles in the GUI.

### Restrictions and Limitations

- For animation preference settings, see “[User Preferences Dialog Box - Analyze Mode Tab](#)” on page 133.
- Transient solution animations can only step between the data recorded at the saved Time Steps.

### Prerequisites

- When a plane plot or particle source is created, a corresponding animation is created under the Animation node in the Results tree.

### Procedure

1. Expand the Animation node and select the animation.
2. Check the On check box in the Animation property sheet.
3. For steady-state solution plane plots, set the Minimum Value and Maximum Value distances between which the plot will move.
4. For transient solution plane plots:
  - a. Set the Minimum Value and Maximum Value time steps between which the plot will move.
  - b. Check the Transient is Relative check box to run the animation in proportion to the data saved times, or uncheck the check box to run the animation at equal time intervals.
5. To start the animation, click the **Play Forwards** icon in the Animation toolbar.

Use the Animation toolbar icons (**Play Backwards**, **Step Backwards**, **Stop**, **Step Forwards**, **Play Forwards**) to control the animation.



**Tip**

The Vector Scale Type settings in the Vector Field property sheet (see “[Vector Field Property Sheet](#)” on page 457) can improve the appearance of vectors in animated particle plots, but you must create a new particle plot *after* changing the Vector Scale Type to see the changes.

---

## Results

Plane plots move between two limits, either repeating a sweep or ‘swinging’ forward and backward.

Particles move through the solution domain.

## Related Topics

[Saving Animations](#)

# Saving Animations

Creating a video files or saving multiple static images to record an animation.

## Prerequisites

- For animation preference settings, see “[User Preferences Dialog Box - Analyze Mode Tab](#)” on page 133.

## Procedure

- Ensure the animation is active, by checking the On check box in the animation property sheet(s).  
More than one animation can be recorded at the same time.
- Click the **Output Animation** icon The Output Animation dialog box is opened.
- Navigate to a folder and select the file type to be saved.
  - To create a single animation movie file, select a file type of MP4 (\*.mp4).
  - To create multiple static image files, select a file type of JPEG (\*.jpg \*.jpeg) or PNG (\*.png).
- Click **Save**.

## Results

The animation runs through one cycle and stops.

File(s) are created in the selected folder.

Multiple static image files can be combined into a single animation file using external animation software.

## Related Topics

[Animating Plots](#)

# Selecting Geometry Associated With a Plot or Annotation

Plots and annotations created when geometry is selected are associated with the selected geometry. Subsequently the geometry may become deselected. This task describes how to reselect the geometry.

## Restrictions and Limitations

- Applies only to plots and annotations that are associated with geometry.

## Procedure

1. Right-click a plot or annotation node in the Results Tree.
2. Choose **Select Associated Geometry**.

## Results

The associated geometry is selected in the data tree and in the GDA.

# Exporting Cell by Cell Results Data

You can export solution results for any calculated field variable for the solution domain or any assembly or object as a csv file. The results at the center of every grid cell are reported.

Similarly, solution results data over a plane for any calculated field variable can be exported using Export Plane Plot Scalar Field button in the “[Plane Plot Property Sheet](#)” on page 460.

## Procedure

1. Select Analyze.
2. In the Model tab, right-click over the item that you want to export the results of and choose **Export Cell by Cell Results...**  
The Export Cell by Cell Results dialog opens.
3. Use the Variable dropdown menu to choose one of the calculated field variables available in the solution.
4. Use the Grid dropdown menu to choose a grid.

You can export the results of two types of solution grid:

- Base Grid – for the grid created from keypoints, constraints, and system grid.
- <Object> – for the localized grids covering the selected object for which the cell by cell results are exported.

5. Click **OK**, navigate to a folder, optionally enter a filename, and click **Save**.

The default filename is:

*<object name>\_<variable name>\_<grid name>.csv*

## Results

- A .csv file is saved with all the grid cells from the selected grid, for the selected variable, within the bounding box of the selected object. If there are any localized grid settings within the solution domain, then averaged values will be displayed for the Base Grid selection. These averaged values are those mapped onto the Base Grid from the localized grid solution and are enclosed in square brackets.

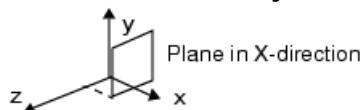
For Transient solutions, results for the timestep selected in the Transient Time Step dropdown menu are exported.

- Results data exported for 2D objects or Over a Plane Plot

Plane plot or 2D objects data is aligned in one of three ways:

- A plane or a 2D object aligned in X direction, where Y is horizontal and Z is vertical.
- A plane or a 2D object aligned in Y direction, where X is horizontal and Z is vertical.
- A plane or a 2D object aligned in Z direction, where X is horizontal and Y is vertical.

**Figure 13-9. Example of a Plane or 2D Object Aligned in the X Direction**



Two grid directions are used in the exported the results.

Figure 13-10 shows the temperature in each grid cell in the Z-plane located at Z=0.0197.

- The cell highlighted is labeled by the grid indices IY=5 and IX=4.
- The coordinates of the cell center are X=0.053, Y=0.116, and Z=0.0197.
- The temperature in this cell is 102°C.

**Figure 13-10. Example csv Results File for a Plane Plot in the X Direction**

	A	B	C	D	E	F
1		IX=1 X=0.005	IX=2 X=0.015	IX=3 X=0.0333333	IX=4 X=0.0533333	IX=5 X=0.07
2	IZ=3 Z=0.0196667					
3	IY=6 Y=0.128	2.04E+01	2.20E+01	5.28E+01	9.75E+01	2.79E+01
4	IY=5 Y=0.116	2.02E+01	2.14E+01	5.57E+01	1.02E+02	2.81E+01
5	IY=4 Y=0.1	2.01E+01	2.07E+01	[8.632942e+01]	[2.603741e+02]	[2.923049e+01]
6	IY=3 Y=0.075625	2.01E+01	2.05E+01	[9.071912e+01]	[2.845568e+02]	[2.582421e+01]
7	IY=2 Y=0.0459375	2.00E+01	2.00E+01	2.01E+01	2.03E+01	2.00E+01
8	IY=1 Y=0.0153125	2.00E+01	2.00E+01	2.00E+01	2.00E+01	2.00E+01

**Note**

 A similar csv file is exported for 2D objects.

- Results data exported for a volume object

When exporting results data for a volume object, all three grid directions are included in the csv file. In [Figure 13-11](#), the cell highlighted: table cell IX=3, IY=5, IZ=2 (location X=0.033, Y=0.116, Z=0.012) has a temperature of 24.1°C.

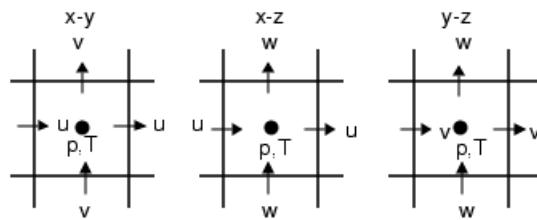
**Figure 13-11. Example csv Results File for a Volume Object - Cell Results for Temperature**

	A	B	C	D	E	F	G	H
1		IX=1 X=0.005	IX=2 X=0.015	IX=3 X=0.0333333	IX=4 X=0.0533333	IX=5 X=0.07	IX=6 X=0.085	IX=7 X=0.095
2	IZ=1 Z=0.004							
3	IY=6 Y=0.128	2.02E+01	2.03E+01	2.12E+01	2.19E+01	2.06E+01	2.02E+01	2.01E+01
4	IY=5 Y=0.116	2.01E+01	2.01E+01	2.05E+01	2.09E+01	2.02E+01	2.01E+01	2.00E+01
5	IY=4 Y=0.1	2.00E+01	2.01E+01	2.05E+01	2.09E+01	2.02E+01	2.01E+01	2.00E+01
6	IY=3 Y=0.075625	2.00E+01	2.01E+01	2.04E+01	2.08E+01	2.01E+01	2.00E+01	2.00E+01
7	IY=2 Y=0.0459375	2.00E+01	2.00E+01	2.00E+01	2.00E+01	2.00E+01	2.00E+01	2.00E+01
8	IY=1 Y=0.0153125	2.00E+01	2.00E+01	2.00E+01	2.00E+01	2.00E+01	2.00E+01	2.00E+01
9	IZ=2 Z=0.012							
10	IY=6 Y=0.128	2.03E+01	2.08E+01	2.49E+01	2.99E+01	2.18E+01	2.04E+01	2.02E+01
11	IY=5 Y=0.116	2.01E+01	2.04E+01	2.41E+01	2.89E+01	2.13E+01	2.02E+01	2.01E+01
12	IY=4 Y=0.1	2.01E+01	2.03E+01	2.39E+01	2.92E+01	2.10E+01	2.02E+01	2.01E+01
13	IY=3 Y=0.075625	2.01E+01	2.02E+01	2.28E+01	2.71E+01	2.06E+01	2.01E+01	2.00E+01
14	IY=2 Y=0.0459375	2.00E+01	2.00E+01	2.01E+01	2.01E+01	2.00E+01	2.00E+01	2.00E+01
15	IY=1 Y=0.0153125	2.00E+01	2.00E+01	2.00E+01	2.00E+01	2.00E+01	2.00E+01	2.00E+01
16	IZ=3 Z=0.0196667							

- Velocity Components and Stagger Option

When **Tables Velocity View Stagger** is active (the default), the velocities displayed are the discretized values of the components  $u$ ,  $v$  and  $w$  of the velocity vector field located at the faces of the grid cells. This arrangement is depicted schematically in the X-Y, X-Z and Y-Z planes for a single cell as shown in [Figure 13-12](#).

**Figure 13-12. View Stagger**



When deactivated, the velocity is interpolated to the cell center, as for the other variables.

# Results Property Sheets

Selecting a node of the Results tree causes the relevant property sheet to be displayed under the tree.

<b>Scalar Field Variable Property Sheet .....</b>	<b>455</b>
<b>Vector Field Property Sheet .....</b>	<b>457</b>
<b>Isosurface Property Sheet.....</b>	<b>459</b>
<b>Plane Plot Property Sheet.....</b>	<b>460</b>
<b>Surface Plot Property Sheet .....</b>	<b>463</b>
<b>Particle Source Property Sheet.....</b>	<b>464</b>
Particle Source Property Sheet - Definition Tab .....	465
Particle Source Property Sheet - Appearance Tab .....	467
<b>Annotation Property Sheet .....</b>	<b>469</b>
Annotation Property Sheet - Definition Tab.....	470
Annotation Property Sheet - Location Tab .....	472
<b>Animation Property Sheet.....</b>	<b>473</b>
<b>Saved Tables Property Sheet .....</b>	<b>474</b>
<b>Viewpoint Property Sheet .....</b>	<b>475</b>

# Scalar Field Variable Property Sheet

To access: Expand the Scalar Fields node of the Results tree and select a scalar variable.

Use this property sheet to modify the display of the results in plots.

## Objects

Object	Description
Total Range Min Value	The minimum variable value. (Command Center) The minimum variable value across all scenarios that have Full results stored. For Transient solutions, the minimum value is from the last time steps of all scenarios.
Total Range Max Value	The maximum variable value. (Command Center) The maximum variable value across all scenarios that have Full results stored. For Transient solutions, the maximum value is from the last time steps of all scenarios.
Transient Min Value	(Transient solution) The minimum variable value at the current time step.
Transient Max Value	(Transient solution) The maximum variable value at the current time step.
Scenario Min Value	(Command Center) The minimum variable value within the currently-selected scenario. For Transient solutions, the value is from the last time step of the scenario.
Scenario Max Value	(Command Center) The maximum variable value within the currently-selected scenario. For Transient solutions, the value is from the last time step of the scenario.
Range	Select the range of data plotted: <ul style="list-style-type: none"> <li>• Total Range – the range of the solution data.</li> <li>• Auto Range – the range of all visible plots. This defaults to the Total Range if no plots are visible.</li> <li>• User Range – a user defined range, defined by values in the Current Min and Current Max data entry fields.</li> <li>• (Transient solution) Transient Time Step Range – the range of the solution data at the current time step.</li> <li>• (Command Center) Current Scenario Range – the range of the solution data for the currently-selected scenario.</li> </ul>
Legend Min Value	The minimum variable value of the range selected. Read-only unless User Range.

<b>Object</b>	<b>Description</b>
Legend Max Value	The maximum variable value of the range selected. Read-only unless User Range.
Clip To >Min and <Max	Only show results within the range between the Current Min and Current Max values.
Log Scale	(Current Min and Current Max must be positive values.) Make the legend scale logarithmic.
Show Legend	Display the plot legend.
Legend Title	(Show Legend) If empty (the default), then the legend title inherits the scalar variable name, for example “Temperature”. Otherwise, the legend title is the name entered here.
Legend Number of Divisions	(Show Legend) The number of legend intervals (ticks placed against the legend). The default is 6 and the maximum is 21. You have the choice of defining the number of divisions by using a slider bar or by direct entry into a field.
Legend Orientation	Choose between: <ul style="list-style-type: none"><li>• Vertical – always display the legend as a vertical bar.</li><li>• Horizontal – always display the legend as a horizontal bar.</li></ul>
Color Map	Sets the color map for the results values. For example, different color maps are useful for distinguishing between different scalar fields.  Examples of use are Iron for thermal imaging of chips, and solid Black/White for speed vectors. By default, the Spectrum of colors is used.
<b>Create Minimum Annotation</b> button	Click to add an annotation at the location of the minimum scalar field value.
<b>Create Maximum Annotation</b> button	Click to add an annotation at the location of the maximum scalar field value.

# Vector Field Property Sheet

To access: Expand the Vector Fields node of the Results tree and select a vector variable.

Use this property sheet to modify the display of the results in plots.

## Objects

Object	Description
Total Range Min Value	The minimum variable value. (Command Center) The minimum variable value across all scenarios that have Full results stored. For Transient solutions, the minimum value is from the last time steps of all scenarios.
Total Range Max Value	The maximum variable value. (Command Center) The maximum variable value across all scenarios that have Full results stored. For Transient solutions, the maximum value is from the last time steps of all scenarios.
Transient Min Value	(Transient solution) The minimum value at the current time step.
Transient Max Value	(Transient solution) The maximum value at the current time step.
Scenario Min Value	(Command Center) The minimum variable value within the currently-selected scenario. For Transient solutions, the value is from the last time step of the scenario.
Scenario Max Value	(Command Center) The maximum variable value within the currently-selected scenario. For Transient solutions, the value is from the last time step of the scenario.
Range	Select the range of data plotted: <ul style="list-style-type: none"> <li>• Total Range – the range of the solution data.</li> <li>• Auto Range – the range of all visible plots. This defaults to the Total Range if no plots are visible.</li> <li>• User Range – a user defined range, defined by values in the Current Min and Current Max data entry fields.</li> <li>• (Transient solution) Transient Time Step Range – the range of the solution data at the current time step.</li> <li>• (Command Center) Current Scenario Range – the range of the solution data for the currently-selected scenario.</li> </ul>
Scale Factor	Vary the length of vector arrows using the slider bar.
Head Size	Vary the size of vector arrow heads using the slider bar.
Current Min Value	Specify a minimum value if clipping to within a range.

Object	Description
Current Max Value	Specify a maximum value if clipping to within a range.
Clip To >Min and <Max	Only show results within the range between the Current Min and Current Max values.
Width	Vary the width of vector arrow heads using the slider bar.
Vector Scale Type	Specify how the length of a vector arrow varies with the vector value: <ul style="list-style-type: none"><li>• Linear – the length varies linearly with the value.</li><li>• Logarithmic – the length varies logarithmically with the value.</li><li>• None – all arrows are the same length.</li></ul>  <b>Note:</b> The setting affects animated particle plots, see “Animating Plots” on page 448.
Use Thinning	Allow for reducing the density of the arrows.
Thinning Ratio	(Use Thinning) Reduce the density of arrows using the slider bar.
Component	Control which components of the vector are shown by arrow: <ul style="list-style-type: none"><li>• All – show the entire vector.</li><li>• In Plane – only show the in-plane component of the vector.</li><li>• Perpendicular – only show the component of the vector that is perpendicular (normal) to the plane.</li></ul>

# Isosurface Property Sheet

To access: Expand the Isosurface node of the Results tree and select an isosurface.

Use this property sheet to customize the display of the selected plot.

## Objects

Object	Description
Name	Editable name field identifying the plot.
On	Toggles on/off the display of the plot. The plot icon in the Results tree is gray when the plot is not displayed.
Scalar Field	Sets the field variable to be plotted from the dropdown menu.
Color By	Selects the variable to be plotted over the isosurface plot
Iso. Value	Sets the value to be plotted.
Show Grid	Displays the solution grid over the selected plot. Alternatively, press G when a plane plot is selected to toggle the display of the solution grid. If the Show Tooltip Cell check box is checked, the results and their location are displayed in hover text as the mouse moves over the grid.
Min	Indicates minimum value plotted.
Max	Indicates maximum value plotted.
Transparency	Moving the slider bar changes the transparency of the plot allowing the geometry structure to be seen through it. The slider bar ranges from 0 (opaque coloring) to 1 (invisible).
Use Lighting	Adds lighting to give a 3D appearance.
Smooth Lighting	Smooths the edges of the plot.
<b>Create Minimum Annotation</b> button	Click to add an annotation to the plot at the location of the minimum Color By variable value.
<b>Create Maximum Annotation</b> button	Click to add an annotation to the plot at the location of the maximum Color By variable value.

## Related Topics

[Creating an Isosurface](#)

## Plane Plot Property Sheet

To access: Expand the Plane Plot node of the Results tree and select a plane plot.

Use this property sheet to activate or deactivate the display of scalar values or vector fields and change its specification.

### Objects

Object	Description
Name	Editable name field identifying the plot.
On	Toggles on/off the display of the plot. The plot icon in the Results tree is dimmed when the plot is not displayed.
Axis	Locates the plane plot across the chosen axis X, Y or Z.
Location	
Associated with	<ul style="list-style-type: none"><li>None – The plot is at an arbitrary position.</li><li>Geometry – The plot is positioned at the center of a geometry object.</li><li>Field – The plot is positioned at the location of a minimum or maximum value of a scalar field.</li></ul>
Geometry	(Geometry) The name of the associated geometry.
Update	(Geometry) Click to align the plot with the center of the currently-selected object, see “ <a href="#">Creating a Plane Plot</a> ” on page 432.
Min/Max dropdown list	(Field) <ul style="list-style-type: none"><li>Min – Aligns the plot with the location of the minimum value of the selected scalar field.</li><li>Max – Aligns the plot with the location of the maximum value of the selected scalar field.</li></ul>
Fields dropdown list	(Field) A list of selectable scalar fields.
Position	Locates the plot along the chosen axis by either using the slider bar (range determined by the solution set) or entering a value directly.
Show Scalar	Plots scalar values.
Scalar Field	Sets the field variable to be plotted from the dropdown menu.
Show Vector	Plots the vector field variable selected from the Vector Field dropdown list.
Vector Field	Determines which vectors (if any) are to be displayed over the solution plot. Vector arrows originate from the center of each cell. The direction of the arrow indicates the direction, and both its length and color give a measure of the magnitude.

Object	Description
Show Grid	Displays the solution grid over the selected plot. Alternatively, press G when a plane plot is selected to toggle the display of the solution grid on or off.
Min	(Show Scalar) Minimum scalar value plotted.
Max	(Show Scalar) Maximum scalar value to be plotted.
Vector Min	(Show Vector) Minimum vector value plotted.
Vector Max	(Show Vector) Maximum vector value plotted.
Fill Type	Contouring formatting for the scalar values can be: <ul style="list-style-type: none"> <li>Cell Fill — Each solution grid cell is completely painted with the color with no interpolation between cell values.</li> <li>Interpolated — Color fills between contour levels, color graded between the intervals</li> <li>Contour Lines — Displays contour lines for locations of the same value</li> </ul>
Number of Contours	(Contour Lines) Sets the number of contour lines.
Use Lighting	Adds lighting to give a 3D appearance.
Transparency	Moving the slider bar changes the transparency of the plot allowing the geometry structure to be seen through it. The slider bar ranges from 0 (opaque coloring) to 1 (invisible).
Show Manipulator	Displays the Translator manipulator, which enables you to drag the plane along an axis.
Cut Plane	Choose from: <ul style="list-style-type: none"> <li>Disabled — Geometry is rendered on both sides of the plot.</li> <li>Above — Geometry is not rendered above the plot, that is, on the high side of the axis along which the plot moves.</li> <li>Below — Geometry is not rendered below the plot, that is, on the low side of the axis along which the plot moves.</li> </ul>
<b>Create Minimum Annotation</b> button	Click to add an annotation to the plot at the location of the minimum scalar field value.
<b>Create Maximum Annotation</b> button	Click to add an annotation to the plot at the location of the maximum scalar field value.
Grid	Use to specify the base grid or any other grid in the model to be exported during a plane plot scalar field export.

Object	Description
<b>Export Plane Plot Scalar Field</b> button	<p>Click to export the plane plot scalar field.</p> <p>The Export Plane Plot dialog opens. You can specify the location and name of the .csv file saved.</p> <p> <b>Note:</b> You can also export cell by cell results. See “<a href="#">Exporting Cell by Cell Results Data</a>” on page 450 where you can also see information on the contents of csv files created by the Export Plane Plot Scalar Field process.</p>

## Related Topics

[Working With Plots](#)

# Surface Plot Property Sheet

To access: Expand the Surface Plot node of the Results tree and select a surface plot.

Use this property sheet to define how the results are displayed over the geometry.

To display surface plots, the Temperature Surface Plot option must be selected and the surface plot temperatures must be available from the solution (by selecting Yes for Surface Temperatures in the Stored Variables section of the **Model Setup** tab).

## Objects

Object	Description
Name	Editable name field identifying the plot.
On	Toggles on/off the display of the plot. The plot icon in the Results tree is dimmed when the plot is not displayed, or when surface plots are unavailable.
Geometry	The name of the associated geometry.
<b>Update</b>	Click to overlay the currently-selected geometry with the surface plot, see “ <a href="#">Creating a Surface Plot</a> ” on page 436.
Plot Edges	Displays edges of geometry overlaid by the plot. The check box is checked for newly-created plots. When unchecked, no geometry edges are shown.
Show Grid	Displays the solution grid over the model.
Min	The minimum temperature on the plot.
Max	The maximum temperature on the plot.
Fill Type	Contouring formatting for the scalar values can be: <ul style="list-style-type: none"><li>Cell Fill — Each solution grid cell is completely painted with the color with no interpolation between cell values.</li><li>Interpolated — Color fills between contour levels, color graded between the intervals</li></ul>
Use Lighting	Adds lighting to give a 3-dimensional appearance.
<b>Create Minimum Annotation</b> button	Click to add an annotation to the plot at the location of the minimum surface temperature value.
<b>Create Maximum Annotation</b> button	Click to add an annotation to the plot at the location of the maximum surface temperature value.

## Related Topics

[Creating a Surface Plot](#)

## Particle Source Property Sheet

To access: Expand the Particles node of the Results tree and select a particle source.

Use this property sheet to define particle sources, particles, and particle streamlines. A particle source is a point from where particles originate. A particle streamline is a representation of the path that particles take through the domain.

### Description

The property sheet has two tabs.

### Objects

- Definition Tab

Defines the particle sources and streamlines, see “[Particle Source Property Sheet - Definition Tab](#)” on page 465.

- Appearance Tab

Defines the appearance of particles, see “[Particle Source Property Sheet - Appearance Tab](#)” on page 467.

### Related Topics

[Creating and Moving Particle Sources](#)

## Particle Source Property Sheet - Definition Tab

To access: Expand the Particles node of the Results tree and select a particle source. The **Definition** tab opens by default.

Use this tab of the property sheet to define particle sources and particle streamlines.

### Objects

Object	Description
Name	The name may be changed by overtyping.
On	Show/hide particle streamlines. The particle source icon in the Results tree is dimmed when particle streamlines are not displayed.
Geometry	The name of the associated geometry.
<b>Update</b>	Click to move the particle sources to the currently-selected object.
Number of Streamlines	The number of sources or streamlines.
Visible Source	Show/hide particle sources.
Area Source	When unchecked, the sources are distributed around the edge of the source shape. When checked, the sources are distributed across the area of the source shape. If the source shape is a line, then this check box has no effect.
Source Shape	The shape of the area where sources are located. Choose from: <ul style="list-style-type: none"><li>• Ellipse — particles are released from an elliptical shaped source area.</li><li>• Rectangle — particles are released from a rectangular shaped source area.</li><li>• Line — particles are released along a line.</li></ul>
Manipulator Type	Choose one of the source manipulators to move the sources. You must be in Select mode for the manipulator to function. <ul style="list-style-type: none"><li>• None — No manipulator is active.</li><li>• Translation — Activates the Translation manipulator.</li><li>• Scale — Activates the Scale manipulator.</li></ul> See “ <a href="#">Creating and Moving Particle Sources</a> ” on page 438.
Source Plane Orientation	The plane in which particle sources are located: X, Y, or Z.

Object	Description
Particle Direction	Choose from: <ul style="list-style-type: none"><li>• Forward — Particles track away from their source.</li><li>• Reverse — Particles track backward towards their source. This is useful for locating particle sources.</li><li>• Both Directions — A combination of the above tracks: particles track away from and towards their source.</li></ul>
Scalar Field	The colors of particles are dependent on the values of the scalar variable selected here.
Vector Field	The sizes and directions of particles are dependent on the values of the vector variable selected here. If Heat Flux or Current Density is selected, the particles flow through solid objects, even when those objects are hidden.
Min	Minimum value of the Scalar Field variable.
Max	Maximum value of the Scalar Field variable.
<b>Create Minimum Annotation</b> button	Click to add an annotation to the plot at the location of the minimum scalar field value.
<b>Create Maximum Annotation</b> button	Click to add an annotation to the plot at the location of the maximum scalar field value.

## Particle Source Property Sheet - Appearance Tab

To access: Expand the Particles node of the Results tree, select a particle source then click the **Appearance** tab.

Use this tab of the property sheet to define particles.

### Objects

Object	Description
Particle Appearance	(Type = Particles or Ribbons) The shape of particles. Choose from: <ul style="list-style-type: none"> <li>• Solid</li> <li>• Ellipses</li> <li>• Arrows. The default.</li> <li>• Triangles</li> <li>• Smoke 1</li> <li>• Smoke 2</li> <li>• Smoke 3</li> </ul>
Type	Defines how the particle streamlines are rendered. Choose from: <ul style="list-style-type: none"> <li>• Tracks — Continuous lines.</li> <li>• Particles — Separate shapes, as defined by the Particle Appearance.</li> <li>• Ribbons — Solid bands. Particle shapes, as defined by Particle Appearance, are superimposed over the ribbons.</li> <li>• Pipe — Continuous cylindrical tubes.</li> </ul>
Width	(Type = Particles, Ribbons, or Pipe) The width of the particles. The maximum width is limited so that particles do not overlap.
Smear	(Type = Particles or Ribbons) The length of the particles. As each particle is displayed for a fixed time, faster moving particles will appear longer than slower moving particles.  <b>Note:</b> Large values of smear, particularly associated with small Time-Step values, may cause particles to overlap, producing a continuous ribbon.
Lifetime	The life expectancy of particles, in seconds. The maximum time that a particle exists as it travels along its streamline.
Timestep	(Type = Particles or Ribbons) The interval between the display of the particles on the streamline. Specify a low value when particles are flowing through solid objects.

Object	Description
Distribution	(Type = Particles or Ribbons) The fraction of the time step used to spread the release times of the particles: <ul style="list-style-type: none"><li>• At the leftmost position, all particles are released at the same time from the source locations. This can create the appearance of a sheet of particles moving through the model.</li><li>• At the rightmost position, the release times are randomly distributed over the whole of the first time step. This creates a more random pattern of particles, being less dependent on the source area shape.</li></ul>
Use Lighting	(Type = Particles, Ribbons, or Pipe) Adds lighting to give a 3D appearance. Lighting is particularly effective when viewing pipes.
Transparency	Changes the transparency of the streamlines, allowing underlying geometry to be seen. The slider bar ranges, left-to-right, from fully opaque to invisible.

# Annotation Property Sheet

To access: Expand the Annotation node of the Results tree and select an annotation.

Use this property sheet to change the appearance, content, anchor point and position of the selected annotation.

## Description

The property sheet has two tabs.

## Objects

- Definition Tab

Defines the appearance and content of the annotation, see “[Annotation Property Sheet - Definition Tab](#)” on page 470.

- Location Tab

Defines the anchor point and position of the annotation, see “[Annotation Property Sheet - Location Tab](#)” on page 472.

## Related Topics

[Working With Annotations](#)

## Annotation Property Sheet - Definition Tab

To access: Expand the Annotation node of the Results tree and select an annotation. The **Definition** tab opens by default.

Use this tab to change the appearance and content of the selected annotation. The options available depend on the type of annotation: Geometry, Plot, or Scalar.

### Objects

Object	Description
Name	(Geometry and Plot annotations) Editable name field identifying the annotation. The default name is the name of the object or plot selected when the annotation was created.
On	Toggles on/off the display of the annotation. In addition, plot annotations are turned on or off when the corresponding plot is turned on or off.
Geometry	(Geometry annotations) The name of the geometry item to which the annotation is connected.
Update	(Geometry annotations) Click to connect the annotation to the center of the currently-selected object.
Field	(Scalar annotations) Min/Max option: <ul style="list-style-type: none"><li>• Min – The annotation is located at the position of the global minimum value.</li><li>• Max – The annotation is located at the position of the global maximum value.</li></ul> Scalar Field dropdown list – Use to change the annotation for a different scalar. This also changes the name of the annotation.
Plot	(Plot annotations) Min/Max option: <ul style="list-style-type: none"><li>• Min – The annotation is located at the position of the minimum value on the plot.</li><li>• Max – The annotation is located at the position of the maximum value on the plot.</li></ul> Plot dropdown list – Use to change the annotation to a different plot. This also changes the name of the annotation.
Scalar Field	A dropdown list of selectable scalar variables. The value displayed in the annotation.
Font Size	The font size for the selected annotation. To change the font size, either enter a value between 4 and 24, or move the slider bar.

Object	Description
Color	The color of the border, text, and leader line of the selected annotation. By default, the color is black. Click the color bar to open the Modify Color dialog box and use this to select or create a different color.
Note	Use this field to add text to the annotation.
Show Name	Show/hide the name of the annotated object.
Show Type	(Geometry annotations) Show/hide the type of geometry. This will be displayed in parentheses by default.
Show Value	Show/hide the results value at the annotated location. The number of significant figures is defined in the <b>Analyze Mode</b> tab of the User Preferences dialog box.
Show Position	Show/hide the absolute coordinates (X, Y, Z) of the annotated location.
Show Background	Toggles between a white (checked) and transparent (unchecked) background for the annotation box.

## Annotation Property Sheet - Location Tab

To access: Expand the Annotation node of the Results tree, select an annotation then click the **Location** tab.

Use this tab to change the anchor point and position of the selected annotation.

### Objects

Object	Description
Position X, Y, Z	The position where the value is sampled, called the anchor point. By default, this is at the center of the object. The position of the anchor point is restricted by the size of the object.
X, Y Offset	The X and Y distances that the annotation is offset from the anchor point. By default this is 8 pixels in each direction. The ranges are from -100 to +100 pixels.

# Animation Property Sheet

To access: Expand the Animation node of the Results tree and select an animation.

Use this property sheet to activate and scope a plot animation.

## Objects

Object	Description
On	Activates the animation. Check this check box to include the animation when the Animation toolbar icons are used.
Plot	The type of plot that is animated: Plane or Particle Source.
Transient is Relative	(Transient solution) If set, the animation frame display sequence is in proportion to the data saved times. If not set, the saved data frames are displayed at equal time intervals.
Minimum Value	(Plane plot animation) Sets the start location for animations of steady solution results. (Transient solution) Sets the start Time Step for animations of transient solution results.
Maximum Value	(Plane plot animation) Sets the end location for animations of steady solution results. (Transient solution) Sets end Time Step for animations of transient solution results.

## Related Topics

[Working With Animations](#)

[Animating Plots](#)

## Saved Tables Property Sheet

To access: Expand the Saved Tables node of the Results tree and select a saved tables.

Use this property sheet to open, rename, or redefine a saved tables. A saved tables is a set of one or more results tables associated with one or more data tree objects.

### Objects

Object	Description
Name	Editable name field identifying the saved tables.
Geometry	The name of the associated geometry.
<b>Update</b>	Redefines the saved tables to be those associated with the currently-selected geometry.

### Related Topics

[Saving the Currently Opened Tables for Re-Display](#)

# Viewpoint Property Sheet

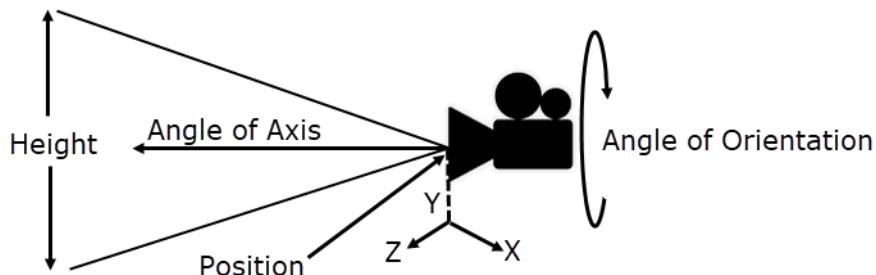
To access: Expand the Viewpoint node of the Results tree and select a viewpoint.

Use this property sheet to view the position and orientation of a viewpoint, and to jump to a viewpoint.

## Objects

Object	Description
Name	The name of the viewpoint. Default names are Viewpoint <n>.
Camera Type	Sets the type of view: <ul style="list-style-type: none"> <li>Perspective – adjusts for the effects of perspective</li> <li>Orthographic – no rules of perspective are applied</li> </ul>
Height	The height of the camera view.
Position X/Y/Z	The camera location.
Angle of Axis X/Y/Z	The camera lens direction.
Angle of Orientation	The rotational angle of the camera about the axis of the lens. A value of 180 deg turns the camera upside down. A value of 360 deg is equivalent to a value of 0 deg.
<b>Jump to Viewpoint button.</b>	Display the model from the viewpoint. Alternatively, you can double-click the viewpoint in the Results tree.

## Usage Notes



## Related Topics

[Creating a Viewpoint](#)



# Chapter 14

## Reporting Project Data and Results in Tables

---

Tables of results, displayed when the Project Manager is in Analyze mode.

<b>Results Tables.....</b>	<b>477</b>
Temperature and Heat Flow Values in Solid Conductor Fluxes Results Tables.....	478
Heat Transfer Coefficient Values in Solid Conductor Fluxes Results Tables .....	480
Reported Junction and Case Temperatures of EDA Components .....	480
<b>Tables Operations .....</b>	<b>482</b>
Viewing Tables .....	482
Searching Results Tables .....	483
Expanding Results Tables to Show SmartPart Primitive Data.....	485
Selecting Table Cell Data .....	486
Resizing Table Columns.....	486
Hiding Results Table Columns.....	487
Sorting Tables .....	487
Copying Selected Results Table Cell Data.....	488
Saving the Currently Opened Tables for Re-Display.....	488
Exporting a Results Table.....	490
Exporting a Legacy Results Table .....	490
<b>Export Legacy Tables Dialog Box .....</b>	<b>492</b>
<b>Results Tables Data .....</b>	<b>493</b>
Geometry Results Tables .....	494
Legacy Tables .....	532

## Results Tables

The types of results tables available in Analyze mode after solving.

### Geometry Data

Selected geometry tables list the calculated results for the selected geometry.

Generally there is one table per geometry type, but some geometry types have more than one table. For example, there are three tables associated with primitives:

- Solid Conductor Fluxes.

- Solid Conductor Temperatures.
- Solid Conductors Summary.

and there are three tables associated with regions:

- Region Mean Flows.
- Regions Summary.
- Region Variables.

When run in batch mode, the tables data can be output to CSV files, see “[Batch Mode Solve](#)” on page 363.

## Legacy Tables

The following results data, that was rendered as tables in earlier releases, can be exported in CSV format files:

- The Geometry Model table.
- Grid
- Attributes

## Related Topics

[Geometry Results Tables](#)

[Legacy Tables](#)

# Temperature and Heat Flow Values in Solid Conductor Fluxes Results Tables

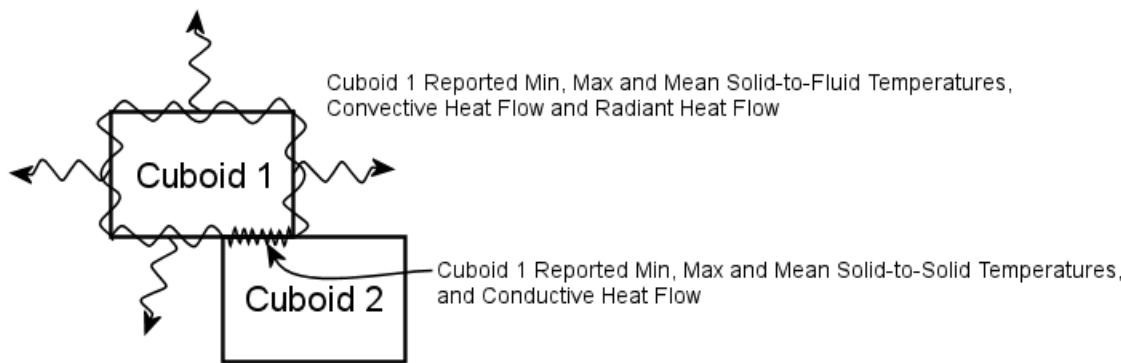
Cases where cuboids are in contact with cuboids, overlap other cuboids, or are collapsed.

## Solid-to-Solid and Solid-to-Fluid Calculations for Cuboids in Contact With Each Other

[Figure 14-1](#) illustrates the Solid Conductor Fluxes table variables as they apply to the faces of Cuboid 1, when it is in contact with Cuboid 2.

Solid-to-Solid (S-S) data is reported for where Cuboid 1 is in contact with Cuboid 2. Solid-to-Fluid (S-F) data is reported for where Cuboid 1 is exposed to fluid.

**Figure 14-1. Two Cuboids in Contact With Each Other**



The displayed surface temperatures are temperatures calculated on the outside of any surface resistance. This means that the temperatures of contact cuboids will be the same on both cuboids at the area of contact, that is, the Minimum, Maximum, and Mean Solid-to-Solid (S-S) Temperatures.

### Overlapping Cuboids

Where cuboids overlap, say a corner of one cuboid overlaps the corner of another, the object that is overwritten (that is, the object higher up the Project Manager data tree) no longer has six faces, but a number of faces as a result of the section removed from it because of the overlap. As such, the face-based flux and temperature information for the cuboid will not be correct.

### Collapsed Cuboids

When a collapsed cuboid is conducting, the Solid Conductor Fluxes tables also report the heat flux through the cuboid. In certain circumstances, this can produce reported heat fluxes that appear to be incorrect.

For example, consider a conducting collapsed cuboid with an attached thermal of 10 W placed between two conducting cuboids. The High and Low-side fluxes for the thermal do not add up to 10 W because the heat is placed into the solid cells on either side of the collapsed cuboid and it is not therefore seen by the software as passing through the cuboid. The tables are in fact correctly reporting the fluxes resulting from the temperature differences between the adjacent solid cells on either side of the collapsed cuboid. The same would also be true if air, rather than solid, adjoined the collapsed cuboid. In this case the flux magnitudes would probably be significantly lower.

### Related Topics

[Solid Conductor Fluxes Results Table](#)

# Heat Transfer Coefficient Values in Solid Conductor Fluxes Results Tables

Calculation of HTC values presented in results tables.

The Convective Heat Transfer Coefficient values in Solid Conductor Fluxes tables are calculated from the following formula:

$$htc = Q/(Area \times (T_{ref} - T_{surf}))$$

where:

$Q$  is the net convective heat flux column.

$Area$  is the surface area exposed to the air.

$T_{surf}$  is the mean surface temperature column.

$T_{ref}$  is the reference temperature.

The Conductive Heat Transfer Coefficient value calculated for any solid-surface interfaces is calculated in a similar way, but where:

$Q$  is the net conductive heat flux column.

$Area$  is the unexposed solid-solid area.

$T_{surf}$  is the mean surface temperature on the solid-solid interface.

$T_{ref}$  is the reference temperature.

## Related Topics

[Solid Conductor Fluxes Results Table](#)

# Reported Junction and Case Temperatures of EDA Components

The values of Junction Temperature and Case Temperature that are reported in EDA Components results tables depend on the construction of the EDA Component.

When Lightweight is switched off, an EDA Component SmartPart extends to its constituent objects, refer to [Modeling EDA Components](#) in the *Simcenter Flotherm SmartParts Reference Guide*.

- For EDA Components containing a Compact Component:

The junction and case temperatures reported in the EDA Components results tables are the same as the Hottest Junction Temperature and Case Temperature values reported in the Compact Component results table when Lightweight is switched off.

- For EDA Components containing a Network Assembly:

The junction and case temperatures reported in the EDA Components results tables are the same as the relevant Node Temperatures from the Network Assemblies results table when Lightweight is switched off.

- For EDA Components that are a combination of primitives, that is, Cylindrical, Simple and Detailed components:

- a. “junction” or “sensor” are searched for in monitor point names.

If multiple monitor points are found, then the one with the maximum temperature is reported as the Junction Temperature of the EDA Component.

- b. “case” is searched for in monitor point names.

If multiple monitor points are found, then the one with the maximum temperature is reported as the Case Temperature of the EDA Component.

Searches are case-insensitive and in monitor points anywhere within the component assembly.

## Related Topics

[EDA Components Results Table](#)

[Compact Component Results Table](#)

[Network Assemblies Results Table](#)

# Tables Operations

Viewing, configuring, copying, and exporting results table data.

The Getting Started With Results Tables video is a short introduction to tables, demonstrating:

- Selecting objects (monitor points) for tables
- Detaching the Tables window
- Sorting tables
- Hiding columns
- Copying and pasting cells



<b>Viewing Tables .....</b>	<b>482</b>
<b>Searching Results Tables.....</b>	<b>483</b>
<b>Expanding Results Tables to Show SmartPart Primitive Data.....</b>	<b>485</b>
<b>Selecting Table Cell Data .....</b>	<b>486</b>
<b>Resizing Table Columns .....</b>	<b>486</b>
<b>Hiding Results Table Columns .....</b>	<b>487</b>
<b>Sorting Tables .....</b>	<b>487</b>
<b>Copying Selected Results Table Cell Data.....</b>	<b>488</b>
<b>Saving the Currently Opened Tables for Re-Display.....</b>	<b>488</b>
<b>Exporting a Results Table.....</b>	<b>490</b>
<b>Exporting a Legacy Results Table .....</b>	<b>490</b>

## Viewing Tables

Tables are displayed in a pane within the Project Manager window, or in a dedicated window. The number of significant figures of the values is defined in the **Analyze Mode** tab of the User Preferences dialog box.

## Prerequisites

- The Project Manager is in Analyze mode.

By default, results tables are displayed in the Tables pane below the GDA.

## Procedure

- To detach the Tables pane from the Project Manager, click on the header, drag the pane away from the Project Manager window, then release the mouse button.  
Tables are displayed in a separate window.
- To reattach a detached Tables window, drag the window to below the GDA until a blank pane appears, then release the mouse button.
- To re-open a Tables window or pane that has been closed, choose **Window > Launch Tables**.

# Searching Results Tables

Using the extended Find to find results data in tables.

## Prerequisites

- Simcenter Flotherm must be in Analyze mode.

## Procedure

- Click the **Find** icon at the top-right-hand corner of the Tables window.



The **Extended Criteria** tab of the Find dialog box opens by default.

- You have a choice:

If you want to...	Do the following:
Search all tables based on one or more criteria using the following variables: Surface Temperature, Temperature, Bn, and Volume Flow Rate.	1. Select <b>Common Results Data</b> .
Only search tables associated with a particular SmartPart type.	1. Select <b>Geometry Results Data</b> . 2. Select a SmartPart type.  EDA Boards and EDA Components are the only EDA SmartParts that have results tables.

3. Select a variable from the Criteria dropdown list and click the + button.  
The variable is added to the Criteria list.
4. Define a criterion for the variable.  
A criterion can be:
  - An equality or inequality between the variable value and a user-specified value. The user-specified value is usually numeric, but for some variables may be an enumerated type, for example, a fan being in or out of range.
  - The maximum or minimum value of the variable.
5. Continue adding criteria, as necessary, to refine the search results.
6. Either select **Match All** for all listed criteria to be satisfied, or select **Match Any** for only one of the criteria to be satisfied.
7. Decide on the actions to be taken following a search:
  - To highlight all of the objects in the data tree whose table data matches the criteria, then select **Select All** from the first (left-hand) Select dropdown list.
  - To only highlight one data tree object at a time, then select **Cycle Select** from the first Select dropdown list.
  - To search the whole model, then select **Model** from the Search dropdown list.
  - To restrict searches to the currently-selected objects, select **Current Selection** from the Search dropdown list.
  - To remove objects from the data tree whose results tables that do not match the search criteria, check the Filter check box.

---

**Note**

 Children of objects whose results tables match the search criteria will remain in the data tree and GDA.

---

8. To begin the search, click **Find**.

## Results

The **Results Count** field is updated to show the number of objects whose tables contents match the search criteria.

If you selected **Select All**, all of the matched objects will be highlighted.

If you selected **Cycle Select**, use the forward and back buttons (>> and <<) to cycle around the matched objects.

The results tables associated with a selected object are shown in the Tables window.

## Related Topics

[Find Dialog Box - Extended Criteria Tab](#)

# Expanding Results Tables to Show SmartPart Primitive Data

Displaying results data for each primitive (cuboid, prism, tet, or inverted tet) which forms part of a SmartPart.

## Restrictions and Limitations

- Only applicable to the following results tables:
  - Solid Conductor Fluxes.
  - Solid Conductor Temperatures.
  - Solid Conductors Summary.
- Only applicable to SmartParts that can be decomposed to solid conductors.  
The tables are opened when relevant SmartParts are selected in Analyze mode.

## Procedure

1. Right-click anywhere in the table to open the context-sensitive menu.
2. Check the SmartPart Details check box.

## Results

A row of data is displayed for each primitive.

## Examples

[Figure 14-2](#) shows an expanded Solid Conductors Summary table for a Heat Sink SmartPart where rows have been added for each cuboid (fins and base).

**Figure 14-2. Solid Conductors Summary Results Table Expanded to Show SmartPart Details**

	Max S-F Surface Temperature (°C)	Min S-S Surface Temperature (°C)	Max S-S Surface Temperature (°C)	Conducted Heat In (W)	Conducted Heat Out (W)	Net Conducted Heat (W)
# HeatSink	25.6	25.5	25.6	0.428	0.201	0.227
HSInnerFin 1:1	25.5	25.5	25.5	0.0307	0	0.0307
HSInnerFin 2:1	25.6	25.5	25.6	0.0175	0.00147	0.016
HSInnerFin 3:1	25.6	25.5	25.6	0.022	0.006	0.016
HSInnerFin 4:1	25.6	25.5	25.6	0.0259	0.00994	0.016
HSInnerFin 5:1	25.6	25.5	25.6	0.0219	0.00602	0.0158
HSInnerFin 6:1	25.6	25.5	25.6	0.0174	0.00142	0.016
HSInnerFin 7:1	25.5	25.5	25.5	0.0306	0	0.0306
HSBase	25.6	25.5	25.6	0.262	0.176	0.0858

## Selecting Table Cell Data

For operations on the table data, such as data export and row hiding, selection of rows, columns, or cells is required.

### Procedure

To select a cell, click in the cell.

Other selection options are described below:

If you want to...	Do the following:
Select a row.	Click a row header cell.
Select a column.	Click a column header.
Select the whole table.	Click the top-left corner of the table.

#### Note

 You can multi-select using Ctrl+click and Shift+click.

## Resizing Table Columns

You may need to resize a column width to show the title or data.

### Procedure

1. Hover over the right-hand boundary of the column until the resize icon (double line with arrows) is displayed.

Pressure (Pa)	X Velocity (m/s)
0.008105308	-0.01948309

- Double-click the resize icon to resize the table to the width of the header title, or drag it to the right/left to increase/decrease the column width.

Pressure (Pa)	X Velocity (m/s)
0.008105308	-0.01948309

## Hiding Results Table Columns

Tidy up large results tables by hiding unwanted columns.

### Procedure

You have a choice:

If you want to...	Do the following:
Hide selected columns.	<ol style="list-style-type: none"><li>Select the column(s) that you want to hide.</li><li>Right-click in the table column header and choose <b>Hide Selected Columns</b>.</li></ol>
Hide unselected columns.	<ol style="list-style-type: none"><li>Select the column(s) that you do not want to hide.</li><li>Right-click in the table column header and choose <b>Hide Unselected Columns</b>.</li></ol>

### Results

The columns are removed from the table.

To reveal hidden columns, right-click anywhere within the table and choose **Reveal Hidden Columns**.

## Sorting Tables

Table sorting is not enabled by default.

### Procedure

- Select the header of the column by which the table is to be sorted.
- Right-click and choose **Enable Sorting**.

The sort descending icon  is displayed to the right of the header title indicating that the table has been sorted with this column in descending order (high to low).

3. To sort the table by a different column, click the header of that column.

The sort descending icon moves to the newly-selected column and the table is now sorted with this column in descending order.

4. To sort in ascending order (low to high), click the header of the column that is the current sort-by column.

The sort ascending icon  is displayed.

5. To restore the default order, right-click in any column header and choose **Disable Sorting**.

## Copying Selected Results Table Cell Data

Copying table cell data to the buffer for pasting.

### Procedure

1. Select the cells to be copied.
2. For steady-state analyses, press Ctrl+C , or right-click and choose **Copy**.
3. For transient analyses, you have a choice:

If you want to...	Do the following:
Copy the selected data for the currently-selected time step.	Press Ctrl+C , or right-click and choose <b>Copy</b> .
Copy the selected data for all <i>saved</i> time steps, or, in the case of Monitor Points, copy the selected data for <i>all</i> time steps.	Choose <b>Copy (with Transient Data)</b> .

### Results

The data is now in the buffer and can be pasted into a spreadsheet or text document.

If saving data for multiple time steps, data for each time step is written on a separate row.

## Saving the Currently Opened Tables for Re-Display

You can save a set of results tables for re-display.

## Procedure

1. In Analyze mode, select one or more objects in the data tree or GDA.  
The results tables associated with the selected objects are opened.
2. Optionally, configure the table(s).  
The following settings are preserved when the tables are re-displayed.
  - If there are multiple tables, select the one that you want to be in the foreground.
  - Sort the tables by preferred column, see “[Sorting Tables](#)” on page 487.
  - Hide any columns that you want to be hidden, see “[Hiding Results Table Columns](#)” on page 487.
  - Show SmartPart details, if that is an option, see “[Expanding Results Tables to Show SmartPart Primitive Data](#)” on page 485.
3. Do one of the following:
  - Click the **Save Current Table** icon at the top-right-hand corner of the Tables window.



- Click the **Save Current Table** Analyze mode icon.
- Right-click the Saved Tables node in the Results tree and choose **Save Current Table**.

## Results

A Saved Tables results tree item is created.

The first saved tables is named Saved Tables. Subsequent saved tables are named Saved Tables <n>.

Clicking a saved tables item reselects the objects that were selected when the saved tables was created, and the associated tables for those objects are displayed.

The selected objects are listed in the Geometry field of the Saved Tables property sheet.

To modify a saved tables to show tables for a different set of objects, then select the objects and click **Update**.

## Related Topics

[Saved Tables Property Sheet](#)

## Exporting a Results Table

Creating a comma-separated text file containing results table data.

### Procedure

1. You have a choice:

If you want to...	Do the following:
Only export the currently selected table.	Right-click anywhere over the table cell area and choose <b>Export Table</b> .
Export all tabbed tables in the current group.	Right-click anywhere over the table cell area and choose <b>Export All Tables</b> .
For transient analyses, export the table data for all <i>saved</i> time steps, or, in the case of Monitor Points, export the table data for <i>all</i> time steps.	Right-click anywhere over the table cell area and choose <b>Export Table (with Transient Data)</b> .

The Export Table dialog box is opened.

2. Navigate to a folder, optionally enter a filename, and click **Save**.

The default filename is:

*<project name> - <object name> - <table name>.csv*

Where *<object name>* is only present if a single object was selected, and *<table name>* is a list of table names separated by commas if more than one table is exported.

Reserved characters that might appear in the filename are replaced by underscores to comply with Windows filename conventions.

### Results

If you chose to export all tables, the tables data are all contained within one worksheet. Each table is separated by a blank row.

If the length of the file pathname exceeds the Windows limit, then the file will not be exported and an E/6002 error message will be output.

## Exporting a Legacy Results Table

Legacy results tables are not available as tables in the product window, but their data can be exported in CSV files.

## Procedure

1. Choose **Analyze > Export Legacy Tables** to open the Export Legacy Tables dialog box.
2. Select the table(s) to be exported and click **OK**.  
The Select Export Folder dialog box is opened.
3. Select a folder and click **Select Folder**.

## Results

One or more CSV files are created in the selected folder.

The default CSV filenames are:

- Geometry Model table - *geometry\_model.csv*.
  - Grid tables:
    - *baseGrid\_<axis>-Grid\_table.csv*
    - *<localized object>\_<axis>-Grid\_table.csv*
- where:
- *<localized object>* is the name of an object that has had a localized grid applied.
- *<axis>* is X, Y, or Z.
- (Transient solutions) *time\_grid\_table.csv*
  - Attributes table - *object\_attribute.csv*.

## Related Topics

[Legacy Tables](#)

[Export Legacy Tables Dialog Box](#)

## Export Legacy Tables Dialog Box

To access: **Analyze > Export Legacy Tables**

Use this dialog box to select which legacy results tables to export as CSV files.

### Objects

Object	Description
Geometry Model	Creates a file named <i>geometry_model.csv</i> . See “ <a href="#">Geometry Model Results Table CSV File</a> ” on page 533.
Grid	Creates base grid files named <i>baseGrid_&lt;axis&gt;-Grid_table.csv</i> . If there are objects with localized grid, also creates files named <i>&lt;localized object&gt;_&lt;axis&gt;-Grid_table.csv</i> . See “ <a href="#">Base Grid and Localized Grid Results Tables CSV Files</a> ” on page 535. For transient solutions, creates a file named <i>time_grid_table.csv</i> . See “ <a href="#">Transient Grid Results Table CSV File</a> ” on page 536.
Attributes	Creates a file named <i>object_attribute.csv</i> . See “ <a href="#">Object/Attributes Results Table CSV File</a> ” on page 537.

### Related Topics

[Exporting a Legacy Results Table](#)

# Results Tables Data

---

Descriptions of each Results Table.

<b>Geometry Results Tables.....</b>	<b>494</b>
Assemblies Results Table .....	495
Collapsed Resistances Results Table .....	497
Compact Component Results Table .....	498
Component Fluxes Results Table .....	500
Controllers Results Table .....	502
Coolers Results Table .....	503
Cutouts/Overall Results Table .....	504
Cutouts/Overall Results Summary Results Table.....	505
Dies Results Table .....	506
EDA Boards Results Table .....	507
EDA Components Results Table .....	508
EDA Heat Sinks Results Table .....	510
Enclosures Results Table .....	511
Fans Results Table .....	512
Fixed Flows Results Table .....	513
Heat Pipes Results Table .....	514
Heat Sinks Results Table .....	515
Monitor Points Results Table .....	516
Network Assemblies Results Table .....	517
Perforated Plates Results Table .....	518
Power Maps Results Table .....	519
Racks Results Table .....	520
Recirculation Devices Results Table .....	521
Region Mean Flows Results Table .....	522
Regions Summary Results Table .....	523
Region Variables Values Results Tables .....	524
Solid Conductor Fluxes Results Table .....	525
Solid Conductor Temperatures Results Table .....	527
Solid Conductors Summary Results Table .....	528
Sources Results Table .....	530
TEC Results Table .....	531
<b>Legacy Tables.....</b>	<b>532</b>
Geometry Model Results Table CSV File .....	533
Base Grid and Localized Grid Results Tables CSV Files .....	535
Transient Grid Results Table CSV File .....	536
Object/Attributes Results Table CSV File .....	537

## Geometry Results Tables

Results tables for selected model geometry, including assemblies, SmartParts, primitives, and monitor points. Data for each selected object is shown on a separate row or group of rows.

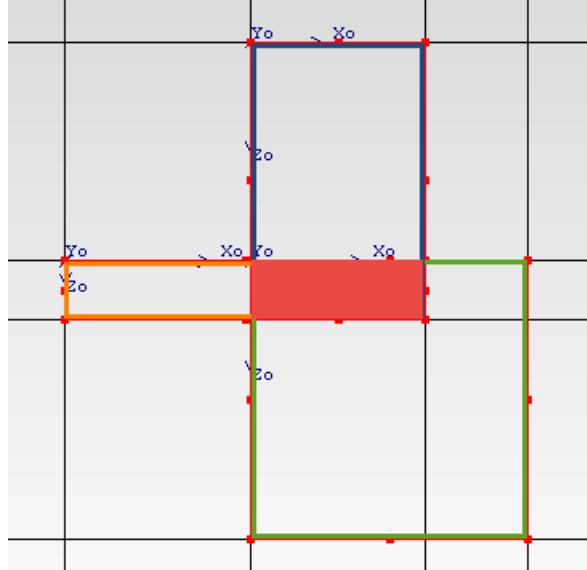
<b>Assemblies Results Table</b> .....	<b>495</b>
<b>Collapsed Resistances Results Table</b> .....	<b>497</b>
<b>Compact Component Results Table</b> .....	<b>498</b>
<b>Component Fluxes Results Table</b> .....	<b>500</b>
<b>Controllers Results Table</b> .....	<b>502</b>
<b>Coolers Results Table</b> .....	<b>503</b>
<b>Cutouts/Overall Results Table</b> .....	<b>504</b>
<b>Cutouts/Overall Results Summary Results Table</b> .....	<b>505</b>
<b>Dies Results Table</b> .....	<b>506</b>
<b>EDA Boards Results Table</b> .....	<b>507</b>
<b>EDA Components Results Table</b> .....	<b>508</b>
<b>EDA Heat Sinks Results Table</b> .....	<b>510</b>
<b>Enclosures Results Table</b> .....	<b>511</b>
<b>Fans Results Table</b> .....	<b>512</b>
<b>Fixed Flows Results Table</b> .....	<b>513</b>
<b>Heat Pipes Results Table</b> .....	<b>514</b>
<b>Heat Sinks Results Table</b> .....	<b>515</b>
<b>Monitor Points Results Table</b> .....	<b>516</b>
<b>Network Assemblies Results Table</b> .....	<b>517</b>
<b>Perforated Plates Results Table</b> .....	<b>518</b>
<b>Power Maps Results Table</b> .....	<b>519</b>
<b>Racks Results Table</b> .....	<b>520</b>
<b>Recirculation Devices Results Table</b> .....	<b>521</b>
<b>Region Mean Flows Results Table</b> .....	<b>522</b>
<b>Regions Summary Results Table</b> .....	<b>523</b>
<b>Region Variables Values Results Tables</b> .....	<b>524</b>
<b>Solid Conductor Fluxes Results Table</b> .....	<b>525</b>
<b>Solid Conductor Temperatures Results Table</b> .....	<b>527</b>
<b>Solid Conductors Summary Results Table</b> .....	<b>528</b>
<b>Sources Results Table</b> .....	<b>530</b>
<b>TEC Results Table</b> .....	<b>531</b>

## Assemblies Results Table

To access: In Analyze mode, select an Assembly in the model.

This table shows thermal values within the extent of the assembly.

### Objects

Column	Description
Maximum Solid Temperature	Maximum solid temperature within the assembly.
Minimum Solid Temperature	Minimum solid temperature within the assembly.
Mean Solid Temperature	<p>Mean solid temperature within the assembly.</p> <p>If the assembly has overlapping solid objects, then the temperature of cells where objects overlap will be taken into account for each object.</p> <p>For example, the figure below shows three cuboids, with blue, green and orange outlines, that have a common overlapped cell, shown in red. When calculating the mean temperature of the assembly, the cell temperature will be accounted for three times, once for each cuboid. This could cause a slight error in the mean solid temperature calculation.</p> 
Heat Applied	The total heat applied by all objects within the assembly, excluding Joule heat.
Joule Heat	(Joule Heating On) The total Joule heat generated within the assembly.
Max S-F Surface Temperature	Maximum solid-to-fluid surface temperature within the assembly.

Column	Description
Min S-F Surface Temperature	Minimum solid-to-fluid surface temperature within the assembly.
Max S-S Surface Temperature	Maximum solid-to-solid surface temperature within the assembly.
Min S-S Surface Temperature	Minimum solid-to-solid surface temperature within the assembly.
Coefficient of Thermal Spreading	<p>See description in “<a href="#">Enclosures Results Table</a>” on page 511.</p> <p>In the case of assemblies, the calculation is based on all solid temperatures within the assembly.</p> <p>The usage of this value is to provide an approximate calculated CTS value for electronic casings that cannot be adequately described with an Enclosure SmartPart.</p> <p> <b>Note:</b> Take care to ensure that objects in the assembly can be described as “thin-walled”, in which case the calculated CTS value will be very close to the true value.</p>

## Collapsed Resistances Results Table

To access: In Analyze mode, select one or more collapsed Resistances.

The table displays the fluid flow data across the selected collapsed resistances.

### Objects

Column	Description
Volume Flow In	Volume flow rate into the collapsed resistance.
Mass Flow In	Mass flow rate into the collapsed resistance.
Heat Flow In	Heat flow rate into the collapsed resistance.
Temperature In	Temperature into the collapsed resistance.
Volume Flow Out	Volume flow rate out of the collapsed resistance.
Mass Flow Out	Mass flow rate out of the collapsed resistance.
Heat Flow Out	Heat flow rate out of the collapsed resistance.
Temperature Out	Temperature out of the collapsed resistance.
Net Volume Flow	Net volume flow rate across the collapsed resistance.
Net Mass Flow	Net mass flow rate across the collapsed resistance.
Net Heat Flow	Net heat flow rate across the collapsed resistance.
Mean Temperature	Mean temperature across the collapsed resistance.
Pressure Drop	Pressure drop across the collapsed resistance.

### Usage Notes

When a collapsed resistance is situated on the domain boundary, “Out” is flow leaving the domain and “In” is flow entering the domain. If there is incoming flow then the temperature reported in the table is the Default Ambient Temperature, as defined in the **Model Setup** tab (Create mode).

## Compact Component Results Table

To access: In Analyze mode, select a Compact Component SmartPart in the model.

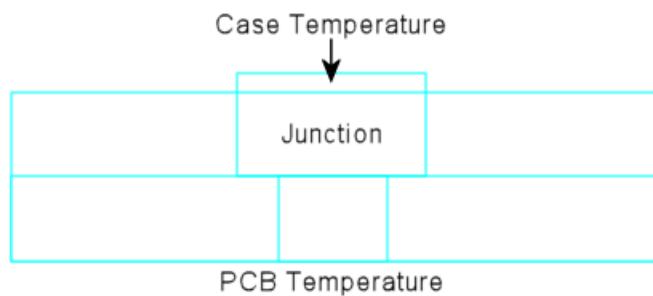
This table shows temperature values for selected Compact Components.

### Objects

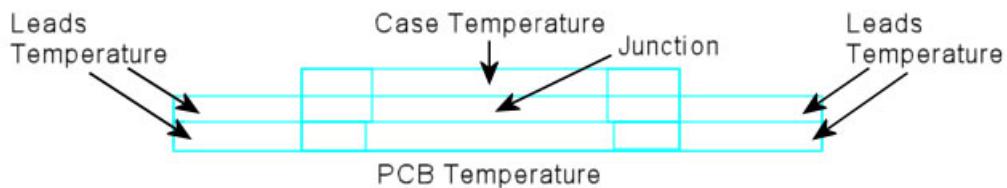
Column	Description
Hottest Junction Temperature	The temperature of the hottest junction.
Case Temperature	The case temperature, refer to <a href="#">Figure 14-3</a> to <a href="#">Figure 14-5</a> .
PCB Temperature	The PCB temperature, refer to <a href="#">Figure 14-3</a> to <a href="#">Figure 14-5</a> .  For peripheral packages, the temperature value in the board immediately adjacent to the central bottom inner node in the network.
Leads Temperature	(Not appropriate for area array packages) The leads temperature, refer to <a href="#">Figure 14-4</a> .  For peripheral packages, there is a choice of 1, 2 or 4 lead nodes. For 2 or 4 leads, the temperature value is the numerical average of the 2 or 4 nodal values from the network.
Hottest Junction Number	The number of the hottest junction.
Junction $<n>$ Temperature	Additional columns contain values for each junction temperature.

### Usage Notes

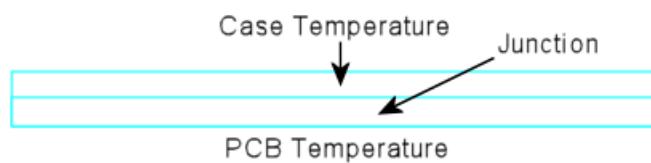
**Figure 14-3. Compact Component Temperatures for an Area Array Package Type Modeled Using a General Network of Resistors**



**Figure 14-4. Compact Component Temperatures for a Peripheral Package Type Modeled Using a General Network of Resistors**



**Figure 14-5. Compact Component Temperatures for an Area Array Package Type Modeled With a Two-Resistor Representation**



## Related Topics

[Component Fluxes Results Table](#)

## Component Fluxes Results Table

To access: In Analyze mode, select a Compact Component SmartPart in the model.

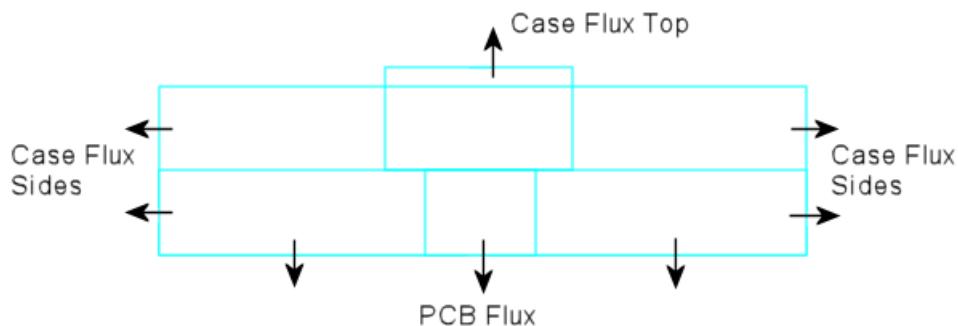
This table shows heat values for selected Compact Components.

### Objects

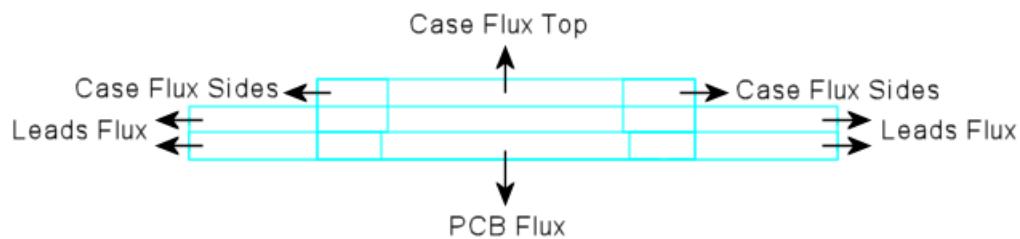
Column	Description
Case Flux Top	The heat from the top of the case, refer to <a href="#">Figure 14-6</a> to <a href="#">Figure 14-8</a> .
Case Flux Sides	The heat from the sides of the case, refer to <a href="#">Figure 14-6</a> and <a href="#">Figure 14-7</a> .
PCB Flux	The heat into the PCB, refer to <a href="#">Figure 14-6</a> to <a href="#">Figure 14-8</a> .
Leads Flux	The heat from the leads, refer to <a href="#">Figure 14-7</a> .
Component Power	The total heat.

### Usage Notes

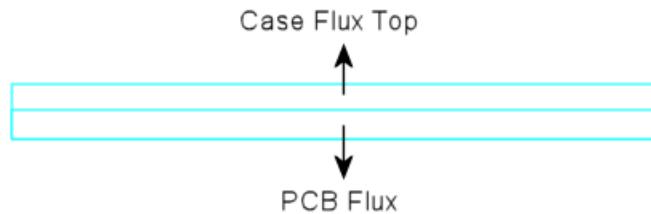
**Figure 14-6. Compact Component Heat Flow for an Area Array Package Type Modeled Using a General Network of Resistors**



**Figure 14-7. Compact Component Heat Flow for a Peripheral Package Type Modeled Using a General Network of Resistors**



**Figure 14-8. Compact Component Heat Flow for an Area Array Package Type Modeled With a Two-Resistor Representation**



## Related Topics

[Compact Component Results Table](#)

## Controllers Results Table

To access: In Analyze mode, select a Controller SmartPart in the model.

This table displays the data for selected Controllers.

### Objects

Object	Description
Controller Frequency	The controller frequency at the currently-selected time step.
Controller Power	The controller power at the currently-selected time step.

## Coolers Results Table

To access: In Analyze mode, select a Cooler SmartPart in the model.

This table displays the flow information for selected coolers. There is one row of information for each parent, followed by a row for each child extract and supply.

### Objects

Column	Description
Total Mass Flow Rate	The mass flow rate of air passing through the cooler.
Supply Volume Flow Rate, Extract Volume Flow Rate	The volume flow rate of air passing through the supply or extract.
Supply Heat Flow Rate, Extract Heat Flow Rate	See descriptions in “ <a href="#">Recirculation Devices Results Table</a> ” on page 521.
Supply Mean Temperature, Extract Mean Temperature	The mean temperature of the air passing through the supply or extract.
Extract Max Temperature	The maximum temperature of the air passing through the extract.
Heat Added	The heat added by the device.
Static Pressure	The effective static pressure the device is operating against.
Cooler Load	The amount of heat extracted by the cooler.

## Cutouts/Overall Results Table

To access: In Analyzer mode, select the domain (System node) and/or one or more Cutouts. This table displays the fluid and heat flow information for the overall boundary and selected cutouts.

### Objects

Column	Description
Face	The face to which the row of data applies.
Volume Flow In	The volume flow into the domain or cutout face.
Mass Flow In	The mass flow into the domain or cutout face.
Radiation In	(Radiation modeling on) The radiated heat into the domain or cutout face.
Heat Flow In	The heat into the domain or cutout face.
Face Temperature In	The temperature into the face of the domain or cutout.
Volume Flow Out	The volume flow out of the domain or cutout face.
Mass Flow Out	The mass flow out of the domain or cutout face.
Radiation Out	(Radiation modeling on) The radiated heat out of the domain or cutout face.
Heat Flow Out	The heat out of the domain or cutout face.
Face Temperature Out	The temperature out of the face of the domain or cutout.
Net Volume Flow	The net volume flow across the domain face or cutout.
Net Mass Flow	The net mass flow across the domain face or cutout.
Net Radiation	(Radiation modeling on) The net radiated heat across the domain face or cutout.
Net Heat Flow	The net heat flow across the domain face or cutout.
Maximum Solid Temperature	Maximum solid temperature of any solid objects abutting a cutout.
Mean Solid Temperature	Mean solid temperature of any solid objects abutting a cutout.

### Usage Notes

---

#### Caution

---

-  Care should be taken when switching from 3D to 2D when ambients are attached to the Z-Low, Z-High or Default All boundaries of the Overall Solution Domain. The software calculates the heat transfer through the Z as well as the X and Y boundaries.
-

## Cutouts/Overall Results Summary Results Table

To access: In Analyze mode, select the domain (System node) and/or one or more Cutouts.

This table displays the values for volume, mass, and heat flow. There is one row for the domain, and one row for each cutout.

### Objects

Column	Description
Volume Flow In	Volume flow rate into the domain or cutout.
Volume Flow Out	Volume flow rate out of the domain or cutout.
Net Volume Flow	Net volume flow across the domain or cutout.
Mass Flow In	Mass flow rate into the domain or cutout.
Mass Flow Out	Mass flow rate out of the domain or cutout.
Net Mass Flow	Net mass flow across the domain or cutout.
Radiation In	(Radiation modeling on) Radiated heat into the domain or cutout.
Radiation Out	(Radiation modeling on) Radiated heat out of the domain or cutout.
Net Radiation	(Radiation modeling on) Net radiated heat across the domain or cutout.
Heat Flow In	Heat flow rate into the domain or cutout.
Heat Flow Out	Heat flow rate out of the domain or cutout.
Net Heat Flow	Net heat flow across the domain or cutout.

## Dies Results Table

To access: In Analyze mode, select one or more Dies.

This table displays the power dissipation and maximum temperature for selected dies. If a die has a non-uniform heat dissipation, then additional rows are added for each die source.

### Objects

Column	Description
Power Dissipation	The power dissipated from the die.
Maximum Die Temperature	The maximum temperature on the die.

## EDA Boards Results Table

To access: In Analyze mode, select an EDA Board SmartPart in the model.

This table displays the data for selected EDA boards.

---

### Note

---

 Conducted, Convected, Radiated and Total Heat values are reported in the Solid Conductors Summary results table.

---

## Objects

Column	Description
Number of Failing Components	The number of components on the board that have “Failed”. See Grade in “ <a href="#">EDA Components Results Table</a> ” on page 508.
Number of Marginal Components	The number of components on the board that are “Marginal”. See Grade in “ <a href="#">EDA Components Results Table</a> ” on page 508.
Maximum Component Junction Temperature	The maximum component junction temperature found when analyzing components on the board.
Maximum Component Case Temperature	The maximum component case temperature found when analyzing components on the board.
Maximum Component Mean Temperature	The maximum component mean temperature found when analyzing components on the board.
Maximum Board Temperature	The maximum temperature on the board including components.
Total Power	The total power of all components on the board.

## EDA Components Results Table

To access: In Analyze mode, select an EDA Component SmartPart in the model.

This table displays the data for selected EDA components.

---

### Note

 Conducted, Convected, Radiated and Total Heat values are reported in the Solid Conductors Summary results table.

---

### Objects

Column	Description
EDA Board Name	Identifies the board to which the component is attached.
Reference Designator	Identifies the component.
Package Name	Identifies the package to which the component belongs.
Part Number	Identifies the component part number.
Component Type	Cylindrical, Simple, 2-Resistor, Detailed Powered, Detailed Unpowered, DELPHI Resistor, T3Ster.
Grade	<ul style="list-style-type: none"><li>Fail – The junction or case temperature is greater than the maximum specified plus the set margin.</li><li>Marginal – The junction or case temperature are within a margin of the maximum specified. The margin is specified by the Marginal Temperature Range value in the <a href="#">User Preferences Dialog Box - Analyze Mode Tab</a>.</li><li>Pass – The junction or case temperature is below the maximum specified less the set margin.</li></ul> <p>See Usage Notes for an example.</p>
Thermal Margin	The difference between the calculated junction temperature and the maximum specified: $T_{j\ Max} - \text{Junction Temperature}$
Power	The power output of the component.
Junction Temperature	The calculated junction and case temperatures, see <a href="#">“Reported Junction and Case Temperatures of EDA Components”</a> on page 480.
Case Temperature	

<b>Column</b>	<b>Description</b>
Mean Temperature	The calculated mean temperature.  Detailed EDA Components comprise overlapping solid objects. Refer to the Mean Solid Temperature description in “ <a href="#">Assemblies Results Table</a> ” on page 495 for an explanation of how mean temperature is calculated when solid objects overlap.
T <sub>j</sub> Max	The maximum junction temperature specified for the component.
T <sub>c</sub> Max	The maximum case temperature specified for the component.

## Usage Notes

### Grade Reporting

For example, if T<sub>j</sub> Max = 140°C, T<sub>c</sub> Max = 90°C, and a 3°C margin has been specified, then:

- T<sub>j</sub> < 137°C or T<sub>c</sub> < 87°C is a Pass.
- 137°C ≤ T<sub>j</sub> ≤ 143°C or 87°C ≤ T<sub>c</sub> ≤ 93°C is a Marginal.
- T<sub>j</sub> > 143°C or T<sub>c</sub> > 93°C is a Fail.

When there are two different grades, then the worst-case is reported, for example, in the above case:

- If T<sub>c</sub> = 86°C (Pass) and T<sub>j</sub> = 137°C (Marginal), then Marginal is reported.
- If T<sub>j</sub> = 137°C (Marginal) and T<sub>c</sub> = 94°C (Fail), then Fail is reported.

## EDA Heat Sinks Results Table

To access: In Analyze mode, select an EDA Heat Sink SmartPart in the model.

The table displays thermal details for selected EDA heat sinks.

### Objects

Column	Description
EDA Component Name	The name of the EDA Component to which the heat sink is attached.
Thermal Resistance	The calculated thermal resistance of the heat sink, refer to “Heat Sinks Results Table” on page 515.
Heat Flow	Conducted heat into the heat sink base.
Convective Heat Flow	Convected heat flow from the heat sink.
Radiative Heat Flow	(Radiation modeling on) Radiated heat flow from the heat sink.
Maximum Temperature	Maximum temperature of the heat sink.
Minimum Temperature	Minimum temperature of the heat sink.
Mean Solid Temperature	Mean temperature of the heat sink.
Solid Mass	Mass of the heat sink.

## Enclosures Results Table

To access: In Analyze mode, select an Enclosure SmartPart in the model.

This table displays the data for selected Enclosures.

### Objects

Object	Description
Minimum External Surface Temperature	The minimum external surface temperature on the enclosure.
Maximum External Surface Temperature	The maximum external surface temperature on the enclosure.
Mean External Surface Temperature	The mean external surface temperature on the enclosure.
Coefficient of Thermal Spreading	<p>The Coefficient of Thermal Spreading (CTS):</p> $CTS = (T_{mean} - T_{ambient}) / (T_{max} - T_{ambient})$ <p>Where:</p> <ul style="list-style-type: none"><li>• <math>T_{mean}</math> is the average temperature for all external faces of the enclosure.</li><li>• <math>T_{max}</math> is the maximum temperature for all external faces of the enclosure.</li><li>• <math>T_{ambient}</math> is the default ambient temperature defined in the <b>Model Setup</b> tab.</li></ul>

## Fans Results Table

To access: In Analyze mode, select a Fan SmartPart in the model.

This table displays the data for selected Fans.

### Objects

Column	Description
Context	Whether it is an internal or external fan, where: an internal fan is completely within the domain and an external fan is on the domain boundary or a cutout.
Volume Flow	The volume flow rate of air passing through the fan.
Mass Flow	The mass flow rate of air passing through the fan.
Heat Flow	The enthalpy of air passing through the fan.
Flow Direction	Positive indicates a direction towards the higher coordinates. Negative indicates a direction towards the lower coordinates.
Mean Temperature	The mean temperature of the air passing through the fan.
Static Pressure	The effective static pressure against which the fan is operating.
Swirl Speed	The swirl speed of the fan.
Range Flag	Indicates whether the fan is operating within its normal operating range.
Derating Factor	The factor, by which, the fan operating RPM is to be reduced.
Derated Fan Power	The value for fan rpm derating.
Fan Efficiency	The ratio between power transferred to the airflow and the power consumed by the fan, as determined from the operating point of the fan (that is, the working point on the fan curve).   <b>Note:</b> If the fan model has been set to a Fixed Volume flow, this figure is set to 0 and the Dissipated Fan Power is consequently equal to the Derated Fan Power.
Dissipated Fan Power	The fan power dissipated as heat.
Derated Fan Noise	The value for fan rpm derating.

## Fixed Flows Results Table

To access: In Analyze mode, select a Fixed Flow SmartPart in the model.

This table displays data for selected Fixed Flows.

### Objects

Column	Description
Volume Flow	The volume flow rate through the device.
Mass Flow	The mass flow rate through the device.
Heat Flow	The heat flow (enthalpy based on minimum domain temperature, $\dot{m}C_p(T - T_{min})$ ) passing through the device.
Flow Direction	Indicates the direction as towards either the higher coordinates (Positive) or the lower coordinates (Negative) and into (Inward) or out from (Outward) the solution domain. For example, Negative Inwards indicates the flow is into the domain and towards the lower coordinates.

## Heat Pipes Results Table

To access: In Analyze mode, select one or more Heat Pipes.

The table displays thermal data for selected heat pipes.

### Objects

Column	Description
Minimum Heat Pipe Temperature	The minimum temperature of the heat pipe.
Maximum Heat Pipe Temperature	The maximum temperature of the heat pipe.
Convective Heat Flux	The convective heat flow from the heat pipe.
Conductive Heat Flux	The conductive heat flow from the heat pipe.
Radiative Heat Flux	(Radiation modeling on) The radiative heat flow from the heat pipe.
Total Heat Flow	Always a positive number, regardless of how many network cuboids are used in its design.
Validity	Indicates whether the heat pipe is moving more heat than it can handle. The maximum heat flow value is defined in the <b>Construction</b> tab of the Heat Pipe property sheet.

## Heat Sinks Results Table

To access: In Analyze mode, select one or more Heat Sinks.

The table displays thermal details for selected heat sinks.

### Objects

Column	Description
Thermal Resistance	The calculated thermal resistance of the heat sink: $(T_{max} - T_{amb})/Q_{base}$ where: <ul style="list-style-type: none"><li>• <math>T_{max}</math> is the maximum surface temperature.</li><li>• <math>Q_{base}</math> is the heat flow entering the heat sink base.</li></ul>
Heat Flow	Conducted heat into the heat sink base.
Convective Heat Flow	Convected heat flow from the heat sink.
Radiative Heat Flow	(Radiation modeling on) Radiated heat flow from the heat sink.
Maximum Temperature	Maximum temperature of the heat sink.
Minimum Temperature	Minimum temperature of the heat sink.
Mean Solid Temperature	Mean temperature of the heat sink.
Solid Mass	Mass of the heat sink.

## Monitor Points Results Table

To access: In Analyze mode, select one or more Monitor Points.

The Monitor Points table contains variables values for the end of a steady-state solution. In the case of transient solutions, the variables values shown are those for the selected Transient Time Step.

---

### Note

---

 Monitor point data versus iteration or time is available from Profile plots by using Copy Data, see “[Exporting a Plot Profile](#)” on page 398.

---

## Objects

Column	Description
$<variable>$	The variable value (Temperature, Pressure, X Velocity, and so on) at the monitor point position.
Controller Frequency	(Transient solution) The frequency power curve being used at the time. Only applies to monitor points attached to Controller SmartParts.
Controller Power	(Transient solution) The power being output at the time. Only applies to monitor points attached to Controller SmartParts.

## Network Assemblies Results Table

To access: In Analyze mode, select a Network Assembly SmartPart in the model.

This table displays the details of selected network assemblies and their component network nodes.

---

### Note

---

 The heat gains apply only to nodes that interact with the rest of the Simcenter Flotherm model, that is, peripheral nodes (nodes that have collapsed network cuboids). Heat flow is not shown from internal nodes.

---

### Objects

Column	Description
	The names of the network assemblies and their component network nodes are listed in the first column.
Power	The power source at the component network node.
Node Temperature	The temperature at the component network node.
Convected Heat Gain	A negative value means that heat is lost from the node to the rest of the model.
Conducted Heat Gain	A negative value means that heat is lost from the node to the rest of the model.
Radiated Heat Gain	(Radiation modeling on) A negative value means that heat is lost from the node to the rest of the model.
Total Heat Gain	The sum of the different types of heat gain.

## Perforated Plates Results Table

To access: In Analyze mode, select one or more Perforated Plates.

The table displays thermal data for selected perforated plates.

### Objects

Column	Description
Volume Flow In	Volume flow rate into the perforated plate.
Mass Flow In	Mass flow rate into the perforated plate.
Heat Flow In	Heat flow rate into the perforated plate.
Temperature In	Temperature into the perforated plate.
Volume Flow Out	Volume flow rate out of the perforated plate.
Mass Flow Out	Mass flow rate out of the perforated plate.
Heat Flow Out	Heat flow rate out of the perforated plate.
Temperature Out	Temperature out of the perforated plate.
Net Volume Flow	Net volume flow across the perforated plate.
Net Mass Flow	Net mass flow across the perforated plate.
Net Heat Flow	Net heat flow across the perforated plate.
Mean Temperature	Mean temperature across the perforated plate.
Pressure Drop	Pressure drop across the perforated plate.

### Usage Notes

When a perforated plate is situated on the domain boundary, “Out” is flow leaving the domain and “In” is flow entering the domain. If there is incoming flow then the temperature reported in the table is the Default Ambient Temperature, as defined in the **Model Setup** tab (Create mode).

## Power Maps Results Table

To access: In Analyze mode, select one or more Power Maps.

The table displays the temperatures and applied power values.

### Objects

Column	Description
Maximum Power Map Temperature	Maximum temperature of the power map.
Minimum Power Map Temperature	Minimum temperature of the power map.
Mean Power Map Temperature	Mean temperature of the power map.
Applied Power Map Power	Applied power to the power map.
Joule Heat	(Joule Heating on) Heat from Joule heating.

## Racks Results Table

To access: In Analyze mode, select a Rack SmartPart in the model.

This table displays the flow information for all selected racks. There is one row of information for each parent, followed by a row for each child extract and supply.

### Objects

Column	Description
Total Mass Flow Rate	The mass flow rate of air passing through the rack.
Supply Volume Flow Rate, Extract Volume Flow Rate	The volume flow rate of air passing through the supply or extract.
Supply Heat Flow Rate, Extract Heat Flow Rate	See descriptions in “ <a href="#">Recirculation Devices Results Table</a> ” on page 521.
Supply Mean Temperature, Extract Mean Temperature	The mean temperature of the air passing through the supply or extract.
Extract Max Temperature	The maximum temperature of the air passing through the extract.
Heat Added	The heat added by the device.
Static Pressure	The effective static pressure the device is operating against.
Capture Index (Hot)	The Hot Aisle Capture Index, see “ <a href="#">Concept of the Capture Index</a> ” on page 159.
Capture Index (Cold)	The Cold Aisle Capture Index, see “ <a href="#">Concept of the Capture Index</a> ” on page 159.

## Recirculation Devices Results Table

To access: In Analyze mode, select a Recirculation Device SmartPart in the model.

This table displays the flow information for selected recirculation devices. There is one row of information for each parent, followed by a row for each child extract and supply.

### Objects

Column	Description
Total Mass Flow Rate	The mass flow rate of air passing through the recirculation device.
Supply Volume Flow Rate, Extract Volume Flow Rate	The volume flow rate of air passing through the supply or extract.
Supply Heat Flow Rate, Extract Heat Flow Rate	Enthalpy based on minimum domain temperature, that is: $\dot{m}C_p(T - T_{min})$ passing through the device.
Supply Mean Temperature, Extract Mean Temperature	The mean temperature of the air passing through the supply or extract.
Extract Max Temperature	The maximum temperature of the air passing through the extract.
Heat Added	The heat added by the device.
Static Pressure	The effective static pressure against which the device is operating.
Capture Index (Hot)	The Hot Aisle Capture Index, see “ <a href="#">Concept of the Capture Index</a> ” on page 159.
Capture Index (Cold)	The Cold Aisle Capture Index, see “ <a href="#">Concept of the Capture Index</a> ” on page 159.

## Region Mean Flows Results Table

To access: In Analyze mode, select a Region in the model.

This table reports calculated mass and heat flow rates, presenting data for one face of a region per row.

### Objects

Column	Description
Face	The face to which the row of data applies: X/Y/Z High/Low. Plane (collapsed) regions have one row of data. If a plane region is collapsed at Mid Face then the data is reported as X, Y, or Z High.
Volume Flow High	Volume flow at the high coordinate side of the region.
Mass Flow High	Mass flow at the high coordinate side of the region.
Heat Flow High	Heat flow at the high coordinate side of the region.
Face Temperature High	Temperature at the high coordinate side of the region.
Volume Flow Low	Volume flow at the low coordinate side of the region.
Mass Flow Low	Mass flow at the low coordinate side of the region.
Heat Flow Low	Heat flow at the low coordinate side of the region.
Face Temperature Low	Temperature at the low coordinate side of the region.
Net Volume Flow	Net volume flow across the region.
Net Mass Flow	Net mass flow across the region.
Net Heat Flow	Net heat flow across the region.
High Current	(Joule Heating On) Electrical current at the high coordinate side of the region.
Low Current	(Joule Heating On) Electrical current at the low coordinate side of the region.

## Regions Summary Results Table

To access: In Analyze mode, select one or more Regions in the model.

This table presents the inward, outward, and net values for the volume, mass, and heat flow for each region, one row per region.

### Objects

Column	Description
Volume Flow In	Volume flow rate into the region.
Volume Flow Out	Volume flow rate out of the region.
Net Volume Flow	Net volume flow rate for the region.
Mass Flow In	Mass flow rate into the region.
Mass Flow Out	Mass flow rate out of the region.
Net Mass Flow	Net mass flow rate for the region.
Heat Flow In	Heat flow rate into the region.
Heat Flow Out	Heat flow rate out of the region.
Net Heat Flow	Net heat flow rate for the region.
Volume	The region volume. Not applicable to plane regions.

## Region Variables Values Results Tables

To access: In Analyze mode, select one or more Regions in the model.

The table presents one region per row and reports the minimum, maximum, mean, standard deviation values for each calculated variable. Not applicable to plane regions.

### Objects

Column	Description
Maximum Region <variable>	Maximum value for the region.
Minimum Region <variable>	Minimum value for the region.
Mean Region <variable>	Mean value for the region.
Standard Deviation Region <variable>	Standard Deviation value for the region.

### Usage Notes

The columns are repeated for each variable.

Irrelevant cells are ignored. For example, solid cells are ignored when calculating the maximum, minimum, and standard deviation values of Fluid Temperature.

## Solid Conductor Fluxes Results Table

To access: In Analyze mode, select one or more solid conductors (cuboids, prisms, tets, or inverted tets), or SmartParts built from primitives.

The table displays the thermal information for the selected solid conductors in the project. There is one row for each surface that has heat transfer (not insulated).

---

**Note**

 For interpretation of the results when cuboids are collapsed or overlap, see “[Temperature and Heat Flow Values in Solid Conductor Fluxes Results Tables](#)” on page 478.

---

### Objects

Column	Description
Face	The X, Y, Z-High and low side surface of the object that has heat transfer.
Min S-F Surface Temperature	Minimum solid-to-fluid surface temperature.
Max S-F Surface Temperature	Maximum solid-to-fluid surface temperature.
Mean S-F Surface Temperature	Mean solid-to-fluid surface temperature.
Min S-S Surface Temperature	Minimum solid-to-solid surface temperature.
Max S-S Surface Temperature	Maximum solid-to-solid surface temperature.
Mean S-S Surface Temperature	Mean solid-to-solid surface temperature.
S-F Surface Area	Solid-to-Fluid surface area.
S-S Surface Area	Solid-to-Solid surface area.
Conducted Heat In	Inward conductive heat.
Conducted Heat Out	Outward conductive heat.
Net Conducted Heat Out	Net conductive heat.
Convected Heat In	Inward convective heat.
Convected Heat Out	Outward convective heat.
Net Convected Heat	Net convective heat.
Radiated Heat In	(Radiation modeling on) Inward radiative heat.
Radiated Heat Out	(Radiation modeling on) Outward radiative heat.
Net Radiated Heat	(Radiation modeling on) Net radiative heat.
Convective and Conductive Heat Transfer Coefficients	These are calculated as described in “ <a href="#">Heat Transfer Coefficient Values in Solid Conductor Fluxes Results Tables</a> ” on page 480.

Column	Description
Current In	(Joule Heating on) Inward electric current.
Current Out	(Joule Heating on) Outward electric current.
Net Current	(Joule Heating on) Net electric current.

## Usage Notes

You can optionally display data for the constituent primitives of SmartParts, see “[Expanding Results Tables to Show SmartPart Primitive Data](#)” on page 485.

The information supplied does not cover thermal data for solids that connect directly to Compact Components. This information can be obtained from the Compact Component and Component Fluxes tables.

## Solid Conductor Temperatures Results Table

To access: In Analyze mode, select one or more solid conductors (cuboids, prisms, tets, or inverted tets), or SmartParts built from primitives.

The table displays the surface temperature and size for all selected solid conductors in the project that have heat transfer (that is, that are not insulated).

### Objects

Column	Description
Minimum Temperature	Maximum surface temperature of the conductor.
Maximum Temperature	Minimum surface temperature of the conductor.
Mean Solid Temperature	Mean solid temperature of the conductor.
Standard Deviation	Standard deviation of the mean solid temperature.
Volume	Volume of the conductor.

### Usage Notes

You can optionally display data for the constituent primitives of SmartParts, see “[Expanding Results Tables to Show SmartPart Primitive Data](#)” on page 485.

## Solid Conductors Summary Results Table

To access: In Analyze mode, select one or more solid conductors (cuboids, prisms, tets, or inverted tets), or SmartParts built from primitives.

This table displays the total flux budget, that is, the proportion of heat leaving the entire solid conductor by convection, conduction, and radiation for selected solid conductors.

### Description

This table is useful for reporting maximum values for entire assemblies. For example, when importing a voxelized MCAD assembly, containing say a thousand cuboids, the maximum temperature and total amount of heat loss can be displayed.

### Objects

Column	Description
Type	The type of SmartPart.
Min S-F Surface Temperature	Minimum solid-to-fluid surface temperature.
Max S-F Surface Temperature	Maximum solid-to-fluid surface temperature.
Min S-S Surface Temperature	Minimum solid-to-solid surface temperature.
Max S-S Surface Temperature	Maximum solid-to-solid surface temperature.
Conducted Heat In	Inward conductive heat.
Conducted Heat Out	Outward conductive heat.
Net Conducted Heat	Net conductive heat.
Convected Heat In	Inward convective heat.
Convected Heat Out	Outward convective heat.
Net Convected Heat	Net convective heat.
Radiated Heat In	(Radiation modeling on) Inward radiative heat.
Radiated Heat Out	(Radiation modeling on) Outward radiative heat.
Net Radiated Heat	(Radiation modeling on) Net radiative heat.
Total Heat Net	The net heat flow resulting from all three modes: conduction, convection, and radiation.
Joule Heat	(Joule Heating on) Heat from Joule heating.
Current In	(Joule Heating on) Inward electric current.
Current Out	(Joule Heating on) Outward electric current.
Net Current	(Joule Heating on) Net electric current.

## Usage Notes

You can optionally display data for the constituent primitives of SmartParts, see “[Expanding Results Tables to Show SmartPart Primitive Data](#)” on page 485.

## Sources Results Table

To access: In Analyze mode, select one or more Sources.

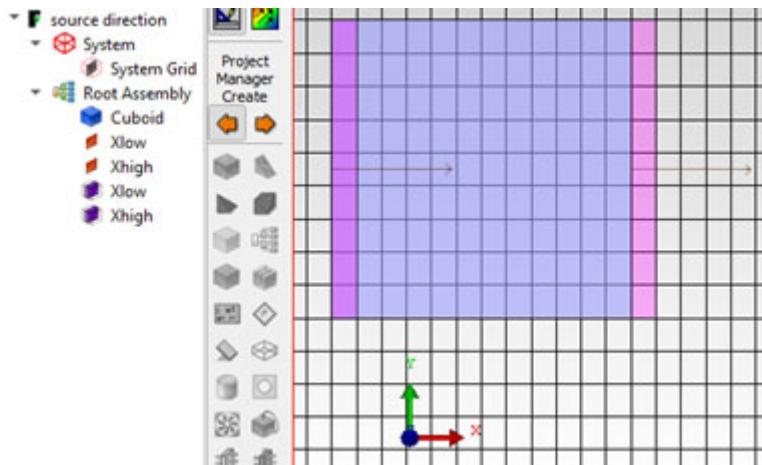
The table displays the power dissipation from all selected sources.

### Objects

Column	Description
Maximum Temperature	Maximum temperature of the source.
Minimum Temperature	Minimum temperature of the source.
Mean Temperature	Mean temperature of the source.
Power Dissipation	The power dissipation from the source.  <b>Note:</b> ‘-’ is displayed when a Source/Area Source attribute has been attached to an uncollapsed source, or a Source/Volume attribute has been attached to a collapsed source.

- Collapsed Sources

Collapsed sources report the maximum, minimum, and mean temperatures of the first row of grid cells in the direction of which the heat is being applied.



In the example above there are two collapsed sources.

- The Xlow collapsed source reports temperatures from the first row of cells inside the cuboid.
- The Xhigh collapsed source reports temperatures from the first row of fluid cells.

## TEC Results Table

To access: In Analyze mode, select a TEC SmartPart in the model.

This table displays thermal data for selected TECs.

### Objects

Column	Description
Heat Pumped	The amount of heat that the TEC pumps.
Heat Added	The additional heat that the TEC generates.
Hot and Cold Side Temperature Difference	$\langle \text{mean hot-side temperature} \rangle - \langle \text{mean cold-side temperature} \rangle$
Maximum Hot Side Temperature	The maximum temperature at the hot side of the TEC.

## Legacy Tables

Legacy tables are only available in the format of CSV files.

<b>Geometry Model Results Table CSV File .....</b>	<b>533</b>
<b>Base Grid and Localized Grid Results Tables CSV Files .....</b>	<b>535</b>
<b>Transient Grid Results Table CSV File.....</b>	<b>536</b>
<b>Object/Attributes Results Table CSV File.....</b>	<b>537</b>

## Geometry Model Results Table CSV File

A CSV file, exported as a legacy table.

A file, with the default name *geometry\_model.csv*, containing data for all the modeling objects in the project.

### Format

The first line of the file contains table column heading titles, separated by commas.

Each subsequent line contains the following data, separated by commas, for each geometry object in the model.

### Parameters

- *geometry\_model*

The name of the geometry object.

- *type*

The geometry construction type, for example, cuboid, enclosure, and so on.

- *level*

The data hierarchy level of the object.

0

The root assembly.

1

A child of the root assembly.

2

A grandchild of the root assembly. And so on.

- *x\_position, y\_position, z\_position*

The geometry position with respect to the root origin (absolute 0, 0, 0).

- *x\_size, y\_size, z\_size*

The geometry size in the three coordinate directions.

- *collapsed*

The collapsed state [YES | NO]. Assemblies are always NO because assemblies cannot be collapsed. This field is not appropriate for SmartParts.

- *heat/area specified, heat/volume specified, heat specified, heat applied*

The “specified” values for heat are taken directly from the project settings, split according to whether they are described as per unit area, per unit volume or total. The Heat Applied is determined after the project is solved and is always given as a total value in Watts.

For Fans, the Heat Applied value is the Dissipated Fan Power, see “[Fans Results Table](#)” on page 512.

- *joule\_heat\_applied*  
(Joule Heating On) The calculated heat due to Joule heating.
  - *current, voltage*  
(Joule Heating On) The values of electric current and electric voltage at a Potential source.
  - *localized*  
The localized grid status of the object [ON | OFF]
  - *derating\_factor*  
The factor, by which, the fan operating RPM is to be reduced.
  - *cold\_aisle\_group, cold\_aisle\_sub\_group, hot\_aisle\_group, hot\_aisle\_sub\_group*  
Names any aisle group attachments.
  - *cooler\_group*  
Names any cooler group attachments.

## Examples

Geometry Model,Type,Level,X Position (m),Y Position (m),Z Position (m),X  
 Size (m),Y Size (m),Z Size (m),Collapsed,Heat/Area Specified (W/m^2),Heat/  
 Volume Specified (mW/mm^3),Heat Specified (W),Heat Applied (W),Joule Heat  
 Applied (W),Current (A),Voltage (V),Localized,Derating Factor,Cold Aisle  
 Group,Cold Aisle Sub Group,Hot Aisle Group,Hot Aisle Sub Group,Cooler  
 Group,  
 Root Assembly,Assembly,0,0,0,0,0.010000000000000009,0.10000000000000001,0  
 .1499999999999999,NO,,.,.,.,.,.,.  
 Large Plate,Cuboid,1,0.02999999999999999,0.10000000000000001,0.100000000  
 00000001,0.00499999999999975,0.10000000000000001,0.1499999999999999,NO  
 .,.,.,.,.,.,.,.,.,.,.  
 Heated Block,Cuboid,1,0.03500000000000003,0.12,0.14000000000000001,0.005  
 00000000000044,0.0399999999999998,0.04000000000000008,NO,,,8,8,,.,.,.  
 .,.,.,.,.,.,.,.,.,.,.  
 Heated Block,Monitor Point,1,0.03750000000000006,0.1399999999999999,0.1  
 600000000000003,-,-,-NO,.,.,.,.,.,.,.

## Related Topics

## Exporting a Legacy Results Table

## Base Grid and Localized Grid Results Tables CSV Files

CSV files, exported as legacy tables.

The base grid files have default names *baseGrid\_<axis>-Grid\_table.csv*. There is a separate file for each direction (*<axis>* = X, Y, or Z). Each base grid file presents a single point in the grid per row for the selected direction.

If there are objects with a localized grid, additional localized grid files are created, named *<localized object>\_<axis>-Grid\_table.csv*.

### Format

The first line of the file contains table column heading titles, separated by commas.

Each subsequent line presents data for each point in the grid.

If there are a number of parts that give rise to keypoints at a single location, only the first encountered keypoint is recorded.

### Parameters

- *grid point identity*  
*<axis>-Direction <number>*
- *location*  
The location of the grid point in the X, Y or Z direction.
- *type*  
KEYPOINT or GRID
- *state*  
Active, De-Active, or blank when the grid line is not a keypoint.
- *object*  
The name of the object to which the grid line is attached.  
“unset” indicates the gridline is not created by a keypoint on an object.  
“:GRIDSPACE” is appended to the object name when the object has a localized grid attached.

### Related Topics

[Exporting a Legacy Results Table](#)

## Transient Grid Results Table CSV File

CSV file, exported as legacy table.

For transient solutions only. A file, with the default name *time\_grid\_table.csv*, containing the time-step grid points.

### Format

The first line of the file contains table column heading titles, separated by commas.

Each subsequent line presents data for each time point.

If there are a number of parts or patches that give rise to keypoints at a single location, only the first encountered keypoint is recorded.

### Parameters

- *time point identity*  
Time <number>
- *time*  
The point in time.
- *type*  
KEYPOINT or GRID

### Related Topics

[Exporting a Legacy Results Table](#)

## Object/Attributes Results Table CSV File

A CSV file, exported as a legacy table.

A file, with the default name *object\_attribute.csv*, that lists the project geometry and any attached attributes.

### Format

The first line of the file contains table column heading titles, separated by commas.

Each subsequent line contains the data, separated by commas, for each geometry object in the model.

Additional lines (rows) are created if different attributes are attached to individual faces.

Objects with grid constraints have additional rows for the grid directions.

Transient attributes are indirectly attached to objects; one or more transients may be attached to a source attribute, and a single transient may be attached to a thermal attribute.

### Parameters

- *geometry\_model*

The name of the geometry object occupies the first column.

- *attached\_attribute*

The names of the attributes attached to the object occupy the remaining columns.

### Related Topics

[Exporting a Legacy Results Table](#)



# Chapter 15

## Additional Auxiliary Variables

---

In addition to the variables calculated during the solution, Simcenter Flotherm can calculate auxiliary variables after solution. These additional calculated variables are stored as part of the solution data.

<b>Available Auxiliary Variables.....</b>	<b>540</b>
Flow Angle .....	540
Total Pressure .....	540
<b>Calculating and Displaying Additional Auxiliary Variables.....</b>	<b>542</b>
Calculating Flow Angle .....	542
Calculating Total Pressure .....	542
Displaying Auxiliary Variables Values .....	543
<b>Flow Angle Dialog Box .....</b>	<b>544</b>

## Available Auxiliary Variables

Simcenter Flotherm derives auxiliary variables from the base variables (for example, temperature, pressure, and velocity) and standard derived variables (for example, speed, heat and mass flux, KE Turb, KE Diss).

<b>Flow Angle .....</b>	<b>540</b>
<b>Total Pressure .....</b>	<b>540</b>

## Flow Angle

The flow angle is used to evaluate the performance of a laminar (uni-directional) flow scheme.

The direction (axis) is the design flow direction, and the results are viewed as the deviation from the design. Typically, flow angles of less than 15° might be considered as good.

The required Solution Data are X,Y, Z-Velocities and Speed.

### Flow Angle Formula

The flow angle is calculated from:

$$\text{Cos}(Flow\ Angle) = X, Y \text{ or } Z\text{velocity}/Speed$$

For example, the flow angle for the X-axis is:

$$\text{Cos}(Flow\ Angle) = X\text{velocity}/Speed$$

The *Flow Angle* units can be radians or degrees, depending on the setting in the Global Units dialog box.

### Related Topics

[Calculating Flow Angle](#)

## Total Pressure

The Total Pressure is important because the pressure associated with the velocity head ( $1/2 \rho V^2$ ) can drive a current of air through against a pressure gradient, or when facing an obstruction can be converted to static pressure and therefore drive the flow.

This, for example, is why in a floor or ceiling void, maximum flow occurs through the active tiles on the opposite side of the void to the inlet where the air is slowed down by the opposite wall and velocity pressure is converted to static pressure.

The required Solution Data are Pressure, Speed, and Density.

## Total Pressure Formula

$$P_{Total} = P_{Static} + 1/2 \rho V^2$$

where:

$\rho$  = fluid density.

$V$  = speed.

$P_{Static}$  = static pressure.

## Related Topics

[Calculating Total Pressure](#)

# Calculating and Displaying Additional Auxiliary Variables

Calculating additional auxiliary variables places additional burden on program memory.

<b>Calculating Flow Angle .....</b>	<b>542</b>
<b>Calculating Total Pressure .....</b>	<b>542</b>
<b>Displaying Auxiliary Variables Values .....</b>	<b>543</b>

## Calculating Flow Angle

Flow Angle is calculated as an additional auxiliary variable following a solve. Calculate the flow angle if it is important to determine whether the airflow is traveling in a specific direction.

### Procedure

1. Run a 3D Flow and Heat Transfer solution to obtain base variables of X, Y, Z-velocity and speed.
2. In the Auxiliary Variables section of the **Model Setup** tab, click the **Calculate** button next to Flow Angle.  
The Flow Angle dialog box is displayed.
3. Choose the axis from which the flow angle is to be measured.

### Results

FlowAngle values are available in Analyze mode.

### Related Topics

[Flow Angle Dialog Box](#)

[Flow Angle](#)

## Calculating Total Pressure

Total Pressure is calculated as an additional auxiliary variable following a solve. Calculate the total pressure when you require pressure data that includes the pressure associated with the velocity head.

### Procedure

1. Run a 3D Flow and Heat Transfer solution containing base variables of speed, temperature, and density.
2. In the Auxiliary Variables section of the **Model Setup** tab, click the **Calculate** button next to Total Pressure.

## Results

Total Pressure values are available as TotalPressure scalar values in Analyze mode.

## Related Topics

[Total Pressure](#)

# Displaying Auxiliary Variables Values

After calculating auxiliary variables, the results can be displayed in Analyze mode.

## Procedure

You have a choice:

If you want to...	Do the following:
View the minimum and maximum values of additional auxiliary variables.	<ol style="list-style-type: none"><li>Select the variable in the Scalar Fields node of the Results Tree.</li></ol> <p> <b>Note:</b> You can create annotations for the minimum and maximum global values.</p>
Display additional auxiliary variables in plots.	<ol style="list-style-type: none"><li>Create a plane or particle plot.</li><li>Select the calculated auxiliary variable from the Scalar Field dropdown list.</li></ol>

## Related Topics

[Creating a Plane Plot](#)

## Flow Angle Dialog Box

To access: In the **Model Setup** tab, click **Calculate** for the Flow Angle auxiliary variable.

Use this dialog box to set the axis for the flow angle and confirm the additional variable calculation request.

### Objects

Field	Description
Axis	The axis from which the flow angle is measured. If there is no existing flow angle, then the initial setting is +Y. If there is an existing flow angle, then the initial setting is the axis that was used to calculate the flow angle previously.

### Related Topics

[Flow Angle](#)

[Calculating Flow Angle](#)

# Chapter 16

## Frequently Asked Questions

---

Answers to FAQs about Simcenter Flotherm.

<b>Project Manager FAQs .....</b>	<b>545</b>
<b>Drawing Board FAQs .....</b>	<b>547</b>
<b>Solution Grid FAQs.....</b>	<b>547</b>
<b>Solution Variables FAQs.....</b>	<b>547</b>
<b>Building Geometry FAQs .....</b>	<b>549</b>
<b>Fan FAQs .....</b>	<b>549</b>
<b>Heat Sink FAQs .....</b>	<b>550</b>
<b>Recirculation Device FAQs.....</b>	<b>550</b>
<b>Support Site FAQs.....</b>	<b>551</b>
<b>Solver FAQs .....</b>	<b>551</b>
<b>Modeling FAQs .....</b>	<b>552</b>
<b>FoXML FAQs .....</b>	<b>553</b>

## Project Manager FAQs

Answers to frequently asked questions (FAQs) about the Program Manager and the data tree.

1. Q: How can I import or export PDML files into Simcenter Flotherm now that Simcenter Flotherm GATE no longer exists?

A: Use the popup Assembly Menu functionality in the Project Manager to import or export PDML files. Right-click the assembly to receive the geometry and choose **Import > PDML** or **Export > PDML** from the Assembly menu. Browse to the directory where the file is loaded, select the file and choose **Open**. The PDML geometry is copied into the assembly or written to the file.

To import or export a project PDML, use the Project Menu popup in the Project Manager. Right-click on the project name at the top of the data tree, and choose **Import Project > PDML** or **Export Project > PDML**. Browse to the directory where the file is loaded, select the file and choose**Open**. The project PDML is copied into the assembly or written to a file.
2. Q: How can I access the new Library Manager?

A: The Library Manager is not displayed by default, to save display space. The quickest way to display the Library Manager is to click the **Open Library Manager** icon or double-click F7. See “[Libraries](#)” on page 285.

3. Q: Can I drag geometry files from the desktop into Simcenter Flotherm?

A: On Windows systems, the easiest way to import a PDML file is to place the PDML file on your desktop and drag it to a Library Manager node from where it can be dragged into the geometry node tree.

For information on the Library, see “[Libraries](#)” on page 285.

4. Q: How can I quickly remove unwanted attributes from my model?

A: Ctrl+Shift+F flushes all the unattached attributes from the project. Alternatively, you can use the **Flush All** option after right-clicking an attribute type node in the Project Attributes tab.

5. Q: How can I change the side details of an enclosure?

A: Use the Enclosure property sheet to set the thickness and modeling levels of the enclosure walls. To add a hole to an enclosure wall, select the wall in the data tree and click the **Hole** icon in the New Object Palette.

See [Enclosures](#) in the *Simcenter Flotherm SmartParts Reference Guide*.

6. Q: How can I export projects?

A: To export a project that you can send via e-mail to colleagues or to Mentor Graphics customer support, right-click on the project name at the top of the Project Manager tree and choose **Export Project > PDML**. Browse to the directory where you want to save the file, name it and choose **Open**. The file is saved to the directory.

7. Q: How can I read archived packed Simcenter Flotherm version 3.x projects?

A: Simcenter Flotherm V3.2 pack files can be unpacked into this version of Simcenter Flotherm. Since this version of Simcenter Flotherm no longer has an External menu item in the Project Manager, the Pack/Unpack features have been moved to the Project/Load menu item.

However, you will not be able to unpack in Simcenter Flotherm V3.2 any pack files that were created in this version of Simcenter Flotherm.

8. Q: How do I remove unused attributes from the project?

A: Press Ctrl+Shift+F.

9. Q: How do I add a hole to a Cuboid?

A: Select the cuboid in the Project Manager data tree, and click on the **Hole** icon in the New Object Palette.

The hole can be moved and resized using the Hole Menu or the drawing board. See [Holes](#) in the *Simcenter Flotherm SmartParts Reference Guide*.

Similarly, holes can be added to individual sides of enclosures. See [Creating Holes](#) in the *Simcenter Flotherm SmartParts Reference Guide*.

## Drawing Board FAQs

Answers to frequently asked questions (FAQs) about the drawing board.

1. Q: How can I use the workplane to view the grid at different locations in the solution domain?  
A: See “[Controls for Viewing the Spatial Solution Grid](#)” on page 260.
2. Q: Can I graphically move an object without accidentally resizing it?  
A: Yes, use the middle mouse button and drag. This will disable object resize.
3. Q: Found objects are not appearing in the drawing board, how can I see them?  
A: The picture has probably been scaled so that the items are not in view. In this case, use the Project Manager to find objects because the data tree updates after an **Edit > Find** operation.

## Solution Grid FAQs

Answers to frequently asked questions (FAQs) about the solution grid.

1. Q: What is the easiest way to localize the grid on an object or assembly?  
A: Select the object or assembly and press l on the keyboard.
2. Q: When an object is saved to the library, will any associated grid be maintained?  
A: All attached attributes will be saved with the object. Therefore, any grid constraints attached to the object when it is saved in the library will also be retrieved when the object is later loaded from the library.
3. Q: Can I use a region to define a localized grid space?  
A: Yes, use regions over areas of special interest where the grid is not dependent on geometry. See “[Grid Constraints](#)” on page 307.

## Solution Variables FAQs

Answers to frequently asked questions (FAQs) about solution variables.

1. Q: How and when to have heat and mass fluxes stored for a given solution?

A: The heat and mass fluxes are stored if requested in the [Model Setup Tab](#).

Store heat fluxes when flow and heat transfer is present and you want to analyze the direction of the heat flow.

Store mass fluxes when a flow is present and you want to analyze its direction.

Note: As  $\text{Mass Flux} = \text{Velocity} \times \text{Density}$ , the velocity vector field will give the same information except in cases where the density varies significantly.

2. Q: Does storing mass and heat fluxes influence the convergence and solution time?

A: Requesting the storage of heat and mass fluxes stores up to an additional six variables:

- Heat fluxes (heat flow per unit area) across cell faces normal to the X-direction
- Heat fluxes across cell faces normal to the Y-direction
- In 3D simulations, the heat fluxes across cell faces normal to the Z-direction
- Mass fluxes (flow rates per unit area) across cell faces normal to the X-direction
- Mass fluxes across cell faces normal to the Y-direction
- In 3D simulations, the mass fluxes across cell faces normal to the Z-direction

As these are only derived from the solved variables, the solution time should only marginally increase. See “[Heat Flux](#)” on page 151 and “[Mass Flux](#)” on page 150 for further background on the variables solved.

3. Q: What are auxiliary variables, and how are they calculated?

A: Auxiliary variables are additional variables that can be derived from the current solution data.

Simcenter Flotherm is capable of calculating flow angle and total pressure providing the solution contains the necessary base variables.

The base variables required by the auxiliary variables are illustrated in the following table:

**Table 16-1. Base Variables Required by the Auxiliary Variables**

Auxiliary Variable	Solution Data Required
Flow Angle	X,Y, Z-Velocity, and Speed
Total Pressure	Pressure, Speed, and Density

See “[Available Auxiliary Variables](#)” on page 540 for instructions on how to request the calculation of auxiliary variables.

# Building Geometry FAQs

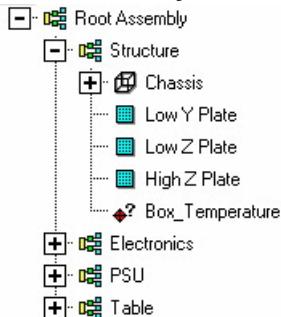
Answers to frequently asked questions (FAQs) about geometry.

1. Q: What are assemblies and how do they work?

A: Assemblies are collections of geometry shapes grouped together to create functional units. Grouping geometry enables a structural approach to creating geometry and makes it easier to debug complex shapes.

The complete geometry structure is held in the top (Root) assembly, which can be divided into sub-assemblies as shown in [Figure 16-1](#).

**Figure 16-1. Root Assembly and Sub-Assemblies**



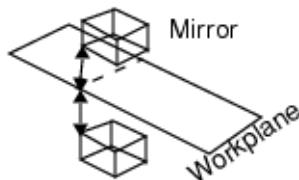
Each assembly is located via a reference origin and can contain primitives, SmartParts, and other assemblies, as well as regions and monitor points. The assembly geometry is located relative to assembly origin, so moving an assembly will move all the constituent parts by the same amount.

See “[Creating, Importing, and Exporting Geometry](#)” on page 211 for a description of the geometry model.

2. Q: Can I mirror Geometry?

A: Objects can be mirrored around the workplane, see [Figure 16-2](#) and “[Mirroring an Object](#)” on page 239.

**Figure 16-2. Mirroring Around the Workplane**



# Fan FAQs

Answers to frequently asked questions (FAQs) about fans.

1. Q: How is a fan curve affected by blockages, since the published fan curve is based on an open room?

A: Fan curves are measured with flow upstream and downstream of the fan unobstructed. Introducing an obstruction close to the fan disturbs the airflow, degrading the flow through the fan. For an axial fan, introducing an obstruction downstream of the fan tends to reduce the maximum flow rate the fan can achieve, but does not significantly affect the pressure increase developed by the fan. For an obstruction upstream of the fan both the maximum flow rate and the pressure increase developed by the fan are reduced.
2. Q: How does high back pressure affect the performance of an axial fan?

A: If a fan is operating at quite a high back pressure (that is, close to the fan static pressure) the flow though the fan will be quite low, as defined by the fan curve. Under these circumstances, air leaving the fan will be rotating at close to the rotational speed of the fan blades, so most of the effect of the fan is to ‘stir’ the air. This provides some localized cooling, but does not help to remove heat from the system. However, operating a fan continuously under these conditions is not recommended, as the lifetime of the fan may be much shorter than when it is used within its normal operating range close to the maximum volume flow rate.

## Heat Sink FAQs

Answers to frequently asked questions (FAQs) about heat sinks.

1. Q: How can I show how much heat is going into a heat sink?

A: Create a 2D region (by clicking the **Plane Region** icon) at the interface between the heat sink and the base object. See [Regions](#) in the *Simcenter Flotherm SmartParts Reference Guide* for help on building regions.  
  
After solution, select Analyze mode. The net flow of heat through the region is reported in the Net Heat Flow column of the Region Mean Flows results table.

## Recirculation Device FAQs

Answers to frequently asked questions (FAQs) about recirculation devices.

1. Q: How can I obtain rapid solution when using the Recirculation Device CRAC unit functionality to model Data Center CRAC units?

A: The operation of the new CRAC functionality ensures that the supply temperature will be maintained at a set point providing that the return air does not rise above a temperature determined by the cooling power of the unit.  
  
In a typical data center the supply will be in a floor void and the return in the main room with a large resistance between the two. Hence it will take a number of iterations for the

flow to develop and the cooler supply air to reach the room. In the meantime the return will remove air from the room which has been heated by the heat load in the space. The return air temperature can, therefore, rise significantly and will cause the supply temperature to move from the set point. In the extreme, it could take a large number of iterations to redress the balance. In this case it is beneficial to develop the flow regime by running an isotherm simulation (50 to 100 iterations should be sufficient) before activating the temperature calculation.

## Support Site FAQs

Answers to frequently asked questions (FAQs) about Support Center.

1. Q: How can I access the support site?

A: Online support is provided by Support Center:

<https://support.sw.siemens.com>

This site provides support pages for Simcenter Flotherm. To access Support Center, you will need to register by clicking **Register** on the Sign In page.

Once you have a password you will be able to call up Support Center from the product by choosing **Help > Support**.

To report a problem or make a request, choose **Help > Report a Problem On-Line**.

2. Q: What Simcenter Flotherm parts, macros, and libraries are available on the support website?

A: Support Center is continuously being updated with new information. Users should periodically check Support Center to see the latest available information.

## Solver FAQs

Answers to frequently asked questions (FAQs) about the solver and solving.

1. Q: How can I change the solution process priority (Windows Only)?

A: You can set a lower than normal priority for the solver by setting the **FLO\_SOLUTION\_PRIORITY** environment variable to 1.

**FLO\_SOLUTION\_PRIORITY** has four possible process priorities. In ascending order, they are Idle, Normal, High, and Real Time. The default is Normal, however you can change this by configuring the following environment variable:

set FLO\_SOLUTION\_PRIORITY = n

where n is an integer value from one to four as follows:

1 = Idle

2 = Normal (the default)

3 = High

4 = Real Time.

If zero or a non-valid number is assigned, the setting will default to Normal. Numbers greater than four will default to Real Time.

Alternatively, you can use the Windows Task Manager to change the processing priority. See “[Setting Solution Process Priority Using the Task Manager](#)” on page 362 for further information.

2. Q: I have built a model but the simulation time seems extraordinarily long, what is wrong?

A: The most likely problem is the existence of a very small grid cell in the model which is causing an extremely small time-step to be required. The time-step is controlled by the smallest grid cell in the model and any single small grid cell will cause a very small time-step to be required. A quick check of the Grid Summary Dialog will show the smallest cell size existing in the model. If the smallest cell size in any one direction is significantly smaller than the rest, then this cell may be inadvertently caused by a slightly mis-aligned geometry and may be the cause of the long simulation time.

3. Q: How can I use run Simcenter Flotherm using more than one processor?

A: Simcenter Flotherm can be run in a parallel mode to take advantage of the computational speed-up offered by multi-processor machines. To run Simcenter Flotherm in the parallel mode, select Use Parallel Solver in the User Preferences dialog box and specify the number of processors you want Simcenter Flotherm to use.

4. Q: How can I run batch files?

A: Refer to “[Solving in Batch Mode](#)” on page 367.

5. Q: What machine specification do I need to run Simcenter Flotherm?

A: The minimum and recommended hardware requirements are listed in the *Simcenter Flotherm Detailed Installation Instructions*.

## Modeling FAQs

Answers to frequently asked questions (FAQs) about modeling.

1. Q: What is the difference between global and ambient?

A: The external ambient temperature settings made in the Global System Settings section of the **Model Setup** tab is used for the following:

- o As a reference temperature for buoyancy calculations

- As the initial condition at the start of a solution
- As a default temperature of air convected into the model

The external radiant temperature is used as a default external temperature that is radiated to, from any object within the solution domain that is part of the radiation calculation and can ‘see through’ to the outside of the solution domain.

Ambient attributes attached to the sides of the solution domain or a solution domain cutout can be used to override the Global temperature values.

2. Q: When do I have to use a source?

A: A source attribute attached to a source primitive, as opposed to a thermal attribute attached to a cuboid, is useful for the following reasons:

- Definition of a source of heat in a portion of a cuboid.
- Ability to define heat dissipation in terms of  $\text{W}/\text{m}^3$  (for an uncollapsed source),  $\text{W}/\text{m}^2$  (for a collapsed source) or as a linear function of the temperature.

In addition to this, a source can be placed in air and values of pressure, velocity, and so on, defined for that portion of air space.

## FloXML FAQs

Answers to frequently asked questions (FAQs) about FloXML.

1. Q: What is FloXML?

A: FloXML is an ASCII markup language designed to store and transport data. FloXML does not have predefined tags as found, for example, in HTML, but has elements defined that are specific to an application. Simcenter Flotherm has its own set of elements (schema) that describes objects, attributes, and modeling settings used to define models.

2. Q: Why would I use FloXML?

A: FloXML files can be created by program and are importable into Simcenter Flotherm via the software or command line, and so they provide a means of automating the model building process. FloXML files can increase productivity by accelerating common Simcenter Flotherm tasks, such as:

- Library Creation  
Quickly create and maintain a library of commonly used parts.
- Supply Chain

Create and maintain models for internal or external distribution.

- Task Automation

Part or assembly characterization. One example would be the automation of the derivation of an appropriate advanced resistance object and attribute for a complex piece of geometry (power supply, or grille work).

- Customized SmartParts

- Data Translation

Automate the translation of disparate data sources into Simcenter Flotherm objects and attributes.

- Tools Integration

Create data flows from other tools to Simcenter Flotherm.

- Customized Front Ends

Create tailored front ends, that allow users without traditional mechanical/thermal engineering backgrounds to make use of Simcenter Flotherm.

- Model Set Up Wizards

Quickly perform the first ten modeling steps or decisions based on wizard inputs.

- Best Known Methods

An established set of Best Known Methods for a Simcenter Flotherm application can be programmatically ensured.

3. Q: How do I get started?

A: There are several validly-formed FloXML example files, and example spreadsheets that write valid FloXML files, installed with the software in the following folder:

`<install_dir>\flosuite_v<version>\flotherm\examples\XML`

Open the FloXML files in a text editor to understand the structure of an FloXML. Then, open each of the installed spreadsheet examples, and use each one to understand the functionality that is achieved with the spreadsheet macros.

4. Q: How can I confirm the FloXML files I've created are correct?

A: It is recommended, when developing your own FloXML writing applications, that you regularly verify the FloXML output by manually loading it into Simcenter Flotherm, running a sanity check, visually inspect data in property sheets, inspect images in the drawing board, and so on. You should continue doing this until you feel your application is safe to use as a basis for producing further models, without the need for manual verification.

5. Q: Why are some of my FloXML objects and attributes renamed upon import?

A: The FloXML reader requires that all objects and attributes have unique names within a single FloXML file. If objects or attributes with identical names are detected, then the name of the last object or attribute to be created is appended with a digit to make it unique. For example, if three cuboids are defined with the name, Block, then they will be imported as: Block, Block1, and Block2.

6. Q: What distinguishes a Project FloXML file from an Assembly FloXML file?

A: Project FloXML files contain the `<solution_domain>` element, Assembly FloXML files do not.

7. Q: How do I group geometry within an assembly?

A: Any geometry to be held within an assembly must be enclosed within `<geometry>` elements, for example:

```
<assembly>
  <name>EXAMPLE</name>
  <geometry>
    All geometry to be placed in the EXAMPLE assembly
    is listed within the geometry element
  </geometry>
</assembly>
```

8. Q: What units are used?

A: SI Units, that is, length (m), time (s), temperature (K) and so on.

9. Q: Are the coordinates local or absolute?

A: All coordinates are local.

10. Q: How do I use the `<orientation>` tags?

A: The `<orientation>` tags are used to describe vectors that relate the local x-, y- and z-coordinates for an object or assembly to the global coordinate system.

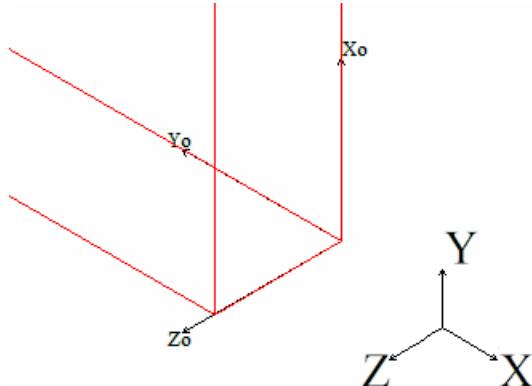
For example, the following FloXML:

```
<orientation>
  <local_x><i>0</i><j>1</j><k>0</k></local_x>
  <local_y><i>-1</i><j>0</j><k>0</k></local_y>
  <local_z><i>0</i><j>1</j><k>1</k></local_z>
</orientation>
```

defines the following relationships, see also [Figure 16-3](#):

- o Local X vector ( $X_0$ ),  $0\ i + 1\ j + 0\ k$ , points in the positive global Y-direction
- o Local Y vector ( $Y_0$ ),  $-1\ i + 0\ j + 0\ k$ , points in the negative global X-direction
- o Local Z vector ( $Z_0$ ),  $0\ i + 1\ j + 1\ k$ , points in the positive global Z-direction

**Figure 16-3. Example Usage of FloXML Orientation Tag**



11. Q: Where are the schema files?

A: The schema files are located in the following folder:

```
<install_dir>\flosuite_v<version>\common\WinXP\lib\flotherm\Plugin\XML_PLUGIN_SCHEMA
```

The parent schema file is *xmlSchema.xsd*, which then references the other schema files present in the folder.

12. Q: How can I interact with the schema files?

A: Third-party tools can provide a graphical display of FloXML schemas to better understand the structure and parent/child relationships of the defined FloXML elements, and to understand which FloXML elements and settings are optional.

13. Q: How do I interact with FloXML files?

A: You can do any of the following:

- Load Project and Assembly FloXML files into the Project Manager.
- Drag Assembly FloXML files into the Library Manager from Windows Explorer.
- Convert FloXML files into PDML from the command line, see “[Converting Files to PDML Format](#)” on page 248.

When the FloXML contains references to PDML, the references are ignored (the assembly that the PDML is imported to will be present). This is because flogate\_cl is a PDML converter, and there is no referencing currently specified in PDML.

- Solve Project FloXML files from the command line. For example:

```
> flotherm.bat -b c:\XML-Project.XML ...
```

Where the project to be solved is defined by a FloXML file. See “[Solving in Batch Mode](#)” on page 367.

14. Q: What can I do with the provided spreadsheet examples?

A: Spreadsheet examples, located in:

`<install_dir>\flosuite_v<version>\flotherm\examples\XML\Spreadsheets`

serve as worked examples, and as reference code for those familiar with working with VBA as the coding tool. See “[Creating FloXML From Spreadsheets](#)” on page 249 for further details.

15. Q: Do I have to use a spreadsheet to write FloXML?

A: No, any coding language or structure that can write an output text file can be used.

16. Q: What happens if I omit optional elements? What are the assumed values?

A: Any optional value that is not included in the FloXML file will be assigned a default value. The default values are the same as those found in Simcenter Flotherm for a new object, attribute, or model setting.

You are advised to include any and all FloXML elements if the data is critical to the analysis.

17. Q: Where is the option to export FloXML file from the Project Manager?

A: There is no such option.

18. Q: What objects, attributes, and settings are supported in this version?

A: Refer to “[FloXML Files](#)” on page 229.

19. Q: What is the minimum FloXML definition required for a planar flow resistance attribute?

A: A planar resistance attribute must have a `<resistance_z>` element present, in addition to a `<loss_coefficient>` element. The `<loss_coefficient>` element is *not* set as mandatory in the schema definition, however, it should be present to ensure backward compatibility with previous versions.

20. Q: The behavior of the FloXML size definition for an object seems to be dependent upon the object being defined. What is the relationship between object sizes, object types, and local coordinate systems?

A: The sizes defined in the FloXML schema for all primitive objects are *always* interpreted as being in the coordinate system of the holding assembly (the same as when using Simcenter Flotherm interactively), *not* the coordinate system of the object itself. The following are the primitive objects that work this way:

- Cuboid
- Resistance
- Source
- Prism

- Tet
- Inverted Tet
- Network Cuboids (for use in Network Assemblies and Heat Pipes)

For example, upon FloXML import, the Z-size of a cuboid will always be aligned with the local Z-axis of the holding assembly, regardless of the orientation defined for that cuboid.

All other objects have dimensions that are directly related to the local coordinate system of the object. That is, the X-size of a perforated plate will always be aligned with the local X-axis of the perforated plate.

# Chapter 17

## Troubleshooting

---

Symptom-related advice for troubleshooting problems. Troubleshooting information is also available on Support Center in the form of TechNotes.

<b>Problems When Loading, Saving, Importing, Exporting, Packing, and Unpacking ..</b>	<b>559</b>
<b>Floating Point (FP) Errors .....</b>	<b>560</b>
<b>Convergence Problems .....</b>	<b>561</b>
<b>Display Problems .....</b>	<b>563</b>
<b>Unexpected Results .....</b>	<b>564</b>
<b>Miscellaneous Problems .....</b>	<b>566</b>

## Problems When Loading, Saving, Importing, Exporting, Packing, and Unpacking

Problems when using project files.

Symptom	Cause	Solution
Project file is not listed when trying to load.	File does not belong to the current user.	Check the filter by username option. Change the username to the project owner or switch off filter by username.
	File list is empty.	Project catalog file needs to be reset. Press the <b>Catalog</b> button in the Load Project dialog box.
Errors when loading or saving a project.	Errors can occur because of file locking and lack of disk space.	<ul style="list-style-type: none"><li>When loading, the file may be locked by another user or locked by program after a system or program crash. If it is locked by program then unlock project by choosing <b>Project &gt; Load &gt; Unlock</b>.</li><li>If you have problems saving a project, check the available disk space using appropriate system commands in the solution directories.</li></ul>

Symptom	Cause	Solution
Project file will not import.		<ul style="list-style-type: none"> <li>Check available disk space using appropriate system commands in the solution directory.</li> <li>If the project being imported already exists in the database, then take no action, delete the existing case or save the existing case to a different name, as appropriate.</li> </ul>
Project file will not export.		<ul style="list-style-type: none"> <li>Check available disk space for directory in which project is being saved.</li> <li>If the file is locked by another user, then wait until it has been released.</li> <li>If the file is locked by program after system or program crash, unlock project by choosing <b>Project &gt; Load &gt; Unlock</b>.</li> <li>If the file is currently loaded, save the case if required, then choose <b>Project &gt; New</b> prior to export.</li> </ul>
Error when packing a file.	A file may not pack because of lack of space.	Check available disk space using appropriate system commands in the solution directory and the directory from which Simcenter Flotherm was started.
Error when unpacking a file.	A file may not unpack because of lack of space or the project may already exist.	<ul style="list-style-type: none"> <li>Check available disk space using appropriate system commands in solution directory.</li> <li>If the project already exists in the database, then take no action, delete the existing case or save the existing case to a different name, as appropriate.</li> </ul>

## Floating Point (FP) Errors

Possible floating point problems when re-initializing and solving.

Symptom	Cause	Solution
Floating Point error output during re-initialization.		Check that data settings are sensible, for example, ensure there are no zero densities.
Floating Point error output during a solve but before residuals are plotted in the Profiles application.	Ill-posed problem. For example, mass inflow boundary prescribed but no outflow or pressure boundaries set, expansivity or temperature field implies a very strong buoyancy force.	Check boundary data inputs. See also advice on convergence difficulties.
	Poor initial field values, for example, zero temperature when solving in K.	Check initial field settings.
Floating Point error output during a solve.	Divergence of linear equation solver. Too high over-relaxation.	Re-set over-relaxation values to 1.0. Values in excess of 1.7 should not be used.
	Non-physical density field predicted in compressible flow problem.	Apply physically realistic limits on density values in the Fluid attribute property sheet.

## Convergence Problems

Solver troubleshooting advice.

Symptom	Cause	Solution
Residuals continuously increasing	Badly-posed problem.	Re-think problem formulation, and re-check data inputs.
	Inadequate relaxation.	Reduce false time-steps, particularly for velocities ( <b>Solver Control</b> tab).

Symptom	Cause	Solution
Residuals do not decrease.	The termination residual is set inappropriately, for example, incorrect use of free-convection velocity.	Set termination residuals and false time-steps for all variables. Check convergence using monitor points and field error.
	Fluctuating flow patterns, for example, moving recirculation zone.	Increase relaxation.
	Very high loss coefficient. Note: to set velocities to 0, use a zero free area ratio.	Check values of loss coefficient.
Solution diverges rapidly.	A badly-posed problem, for example, enclosed regions which contain a net mass imbalance.	Check modeling assumptions, and re-think formulation. In particular, for pressure on ambients, these settings correspond to gauge pressures.
Values increasing and no convergence.	Internal sources with no loss from the solution domain, for example, heat source within adiabatic enclosure. Often has the effect of producing a level convergence plot for temperature and a linear increase in temperature values.	Check the problem is well posed. Check monitor point behavior of the affected variable. If the variable increases at a constant or increasing rate it is likely that a net imbalance in the sources is present.
Large residual error on a continuation, that is, after resuming an interrupted solve.	Introduction of a blockage.	Ignore if blockage added as residuals recover quickly.
	Introduction of a new pressure boundary.	Ignore, for program should recover by re-adjusting pressure field to give correct boundary flow.
	Grid change forcing interpolation on continuation.	Ignore.

Symptom	Cause	Solution
Plots show erratic behavior during solve.	Physically unstable flow conditions, for example, strong jets.	Reduce false time-steps on velocities. Conjugate gradient solver for Pressure activated in Solver Control.
	Pressure boundary conditions alternate between inflow and outflow. This can be a particular problem if the external temperature is significantly different from that in the calculation domain.	Provided that only outflow is expected in the final solution, change boundary condition on temperature to give a temperature close to the fluid exit temperature.
	Attempt to solve a turbulent case as laminar ( <b>Model Setup</b> tab).	Set to turbulent with appropriate turbulence parameters.
No convergence when flow is due to buoyancy.	Initial temperature field is not set to an ambient level, for example, if solving in K.	Initialize to a suitable value.
Residuals do not converge and level off.	‘Round Off’ error of a magnitude which is significant compared to residual errors.	Check magnitude of the termination residuals. Check that reference values are sensible (for example, datum pressure and temperature in the <b>Model Setup</b> tab). Set datum values for pressure and temperature so as to avoid large values being set at boundaries.
Unacceptable residual for temperature.	Insufficient inner iterations set for temperature.	Increase number of iterations to say 100 or more.
	Relaxation on temperature too high.	Deactivate automatic relaxation on temperature, and try to run with no relaxation (user specified value of 1.E15).

## Display Problems

Possible problems when trying to display solver progress and results.

Symptom	Cause	Solution
In Profiles, unable to plot iteration/time profile during a solve.	Not enough solution passes available	Re-try when number of iterations exceeds two.
In Profiles, no residuals plot for transient solution.	Duration of period plus start time exceed overall time of transient.	Check the duration time in the Transient Solution dialog box and reset the value so that start plus duration is less than the overall time.
In Analyze mode, no vectors or contours plotted when vector (or contour) option activated. Empty results plots.	Attempt to plot in a plane which is fully blocked.	Check value of velocities using hover text over the plot (Show Tooltip Cell check box in Plane Plot property sheet).
	No flow has been solved or project has not been re-initialized.	Check file selected in Initial Variables ( <b>Solver Control</b> tab).
No residual plot in Profiles application.	Problem converged, therefore requiring no further convergence.	Check monitor point values using Iteration Profile Plot in the Profiles application window to verify solution has converged.
	Number of solution iterations set to 1 or 0.	Check specified number of Outer Iterations ( <b>Solver Control</b> tab).
	Termination residuals set to very large values.	Check setting of termination residuals.
	No boundary conditions set.	Check boundaries.
No time profile plot.	Not enough time steps solved.	Add time steps using Transient Solution dialog box.
Velocity vectors not shown.	Velocity vectors can sometimes be too small to distinguish.	Increase the scale factor in the <a href="#">Vector Field Property Sheet</a> .

## Related Topics

[Transient Solution Dialog Box](#)

[Plane Plot Property Sheet](#)

[Solver Control Tab - Variable Solution Control Section](#)

# Unexpected Results

Post-solve problems.

Symptom	Cause	Solution
In Analyze mode, there is no heat transfer or heat transfer where there should be none.	Heat sources cut off from area of concern by non-conducting obstructions.	Draw boundaries and check positioning.
	Incorrect setting of material, thermal attributes or dimensions.	Check data settings.
In Analyze mode, there is no flow or flow where there should be none.	Airflow is blocked by obstructions.	Draw boundaries and check positioning.
In Analyze mode, the magnitude of flow rates is not as expected.	Incorrect use of volume inflow option, for example, flow is specified using incorrect units.	Check volume flow settings in Fan or Fixed Flow SmartParts.
	Incorrect inlet area.	Check inlets extend over entire depth of domain (2D cases).
	Incorrect domain dimensions.	Check settings in System property sheet.
	Velocity set in wrong direction.	Check fan direction by showing primitives for Fan or Fixed Flow SmartParts.
	Flow resistances set incorrectly.	Check magnitude of loss coefficient and free area ratio, ensure resistance attribute has been attached.
	Characteristics for fan curve incorrectly set.	Check flow setting and/or fan curve specification. Check fan operating range via Tables.
No variation of field values throughout model.	Inadvertent use of solve re-initialize after solution so that fields re-initialized to uniform values after solution.	Problem must be re-solved.
	No initial values read in using a Solution Set ( <b>Solver Control</b> tab) prior to a solve re-initialize, resulting in fields not being read in.	Select a Solution Set from which to initialize fields then re-initialize.

Symptom	Cause	Solution
Results unexpectedly show initial field values.	The data has been overwritten with the initial fields by the automatic saving of the results after a solve re-initialize.	Problem must be re-solved.

## Miscellaneous Problems

Problems that do not fall into other categories.

Symptom	Cause	Solution
Drawing Board area turns white.	This can occur when a window is dragged over the drawing board.	Right-click on your desktop, choose Personalize and select the Windows 7 theme (listed under Aero Themes). In addition, make sure you have the latest video drivers for your video card. If the problem persists, contact Mentor Graphics customer support.
Solver runs slowly.	Hyper-Threading is activated.	To obtain full solver performance on your machine, see “ <a href="#">Obtaining Maximum Solver Performance</a> ” on page 359.
	Parallel solver is not being used.	
	Unnecessary use of the double-precision solver.	

Symptom	Cause	Solution
Not enough disk or stack space to run the model.	Abnormal exit from solver. Solver died on signal.	Check disk availability in solution repository directory. Alternatively, limits on stack size have been set. Save and exit from Simcenter Flotherm, then check the current stack limits using the limit system command. Use unlimit to remove limits set before re-starting Simcenter Flotherm.
	Problem too complex and/or overspecified (for example, unnecessary use of k-epsilon model, or variable density option, and so on.).	Check model for possible simplification (for example, use constant density option, use constant turbulent viscosity model).
	Grid too fine.	Use fewer additional cells by merging or deactivating keypoints. Storage can also be reduced by replacing thick walls by thin walls where possible.
Grid poorly defined for the model.	Grid Constraint not attached.	Check and apply as appropriate.



# Chapter 18 Messages

---

An overview of Simcenter Flotherm error reporting and lists of non-self-explanatory messages.

<b>Message Types</b> .....	<b>569</b>
<b>Message Format</b> .....	<b>570</b>
<b>Error Window</b> .....	<b>571</b>
<b>Generated Messages</b> .....	<b>573</b>
System Error Messages.....	573
User System Messages (1000 Series).....	573
Drawing Board Messages (3000 Series).....	576
Project Manager Messages (4000 Series).....	577
Profiles Messages (5000 Series).....	580
Tables Messages (6000 Series).....	582
Converter Messages (7000 Series).....	583
Solver Messages (8000 Series).....	586
Translator Messages (9000 Series).....	587
EFG Messages (10000 Series) .....	595
Data Storage Messages (11000 Series).....	596
MCAD Messages (12000 Series) .....	597
FloSCRIPT Messages (15000 Series) .....	600
Interface Messages (17000 Series) .....	601
Logic Error Messages (19000 Series).....	601
Command Center Messages (20000 Series) .....	601

## Message Types

Error, Warning, and Information message numbering.

- Error Messages are prefixed with E/ and are displayed in red.
- Warning Messages are prefixed with W/ and are displayed in pale green.
- Information Messages are prefixed with I/ and are displayed in black.

**Table 18-1. Message Numbering**

Series	Application/Software Subsystem
1000	User System messages
2000	Analyze Mode Graphics

**Table 18-1. Message Numbering (cont.)**

Series	Application/Software Subsystem
3000	Create Mode Drawing Board
4000	Project Manager
5000	Profiles
6000	Analyze Mode Tables
7000	V1.4 Converter
8000	CFD Solver
9000	Solver Translator
10000	Radiation Preprocessor
11000	Storage Agent
12000	MCAD Window
14000	Solar Calculation
15000	FloSCRIPT
16000	Project Manager (extension to 4000 series)
17000	Interface
18000	CSV Import
19000	Logic
20000	Command Center

## Message Format

Message status and hyperlinking.

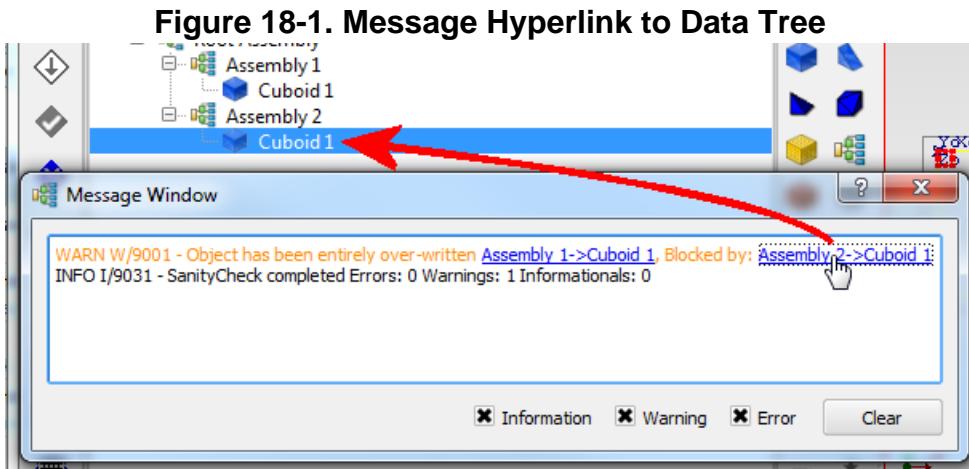
Each message has the following format:

*<status> <number> - <message> <object>*

where:

- *<status>* indicates three warning levels:
  - ERROR — data error interrupting solution,
  - WARN — flags set up problems such as incorrect location of boundaries, for example, a fan detected inside a block, or
  - INFO — purely informational, for example, object encountered outside the solution domain.

- <number> is a trouble-shooting number, see “[Message Types](#)” on page 569, and is the number to quote to Mentor Graphics customer support when you need help.
- <message> explains the problem.
- <object> identifies the problem area, and may be hyperlinked to the data tree, for example, see [Figure 18-1](#).



Multiple selections can be made by holding down the Ctrl key while clicking hyperlinks.

## Related Topics

[Message Window](#)

[Troubleshooting](#)

[Frequently Asked Questions](#)

[Technical Support](#)

## Error Window

In rare circumstances, an Error window is displayed after a fatal program error.

There are three option buttons:

- **OK** — Enables continuation, but it is advisable to save the project as repeated occurrences can cause software failure.
- **TraceBack** — Generates debug information that can be relayed to Mentor Graphics customer support.
- **Abort** — Causes the program to stop.

If the Error window is displayed, then exit and re-enter Simcenter Flotherm. If the problem persists, then contact Mentor Graphics customer support.

To determine which application window generated the Error window, use the middle mouse button and a popup list will appear.

## Generated Messages

Messages and explanations. Self-explanatory messages are excluded from these lists.	
<b>System Error Messages</b> .....	<b>573</b>
<b>User System Messages (1000 Series)</b> .....	<b>573</b>
<b>Drawing Board Messages (3000 Series)</b> .....	<b>576</b>
<b>Project Manager Messages (4000 Series)</b> .....	<b>577</b>
<b>Profiles Messages (5000 Series)</b> .....	<b>580</b>
<b>Tables Messages (6000 Series)</b> .....	<b>582</b>
<b>Converter Messages (7000 Series)</b> .....	<b>583</b>
<b>Solver Messages (8000 Series)</b> .....	<b>586</b>
<b>Translator Messages (9000 Series)</b> .....	<b>587</b>
<b>EFG Messages (10000 Series)</b> .....	<b>595</b>
<b>Data Storage Messages (11000 Series)</b> .....	<b>596</b>
<b>MCAD Messages (12000 Series)</b> .....	<b>597</b>
<b>FloSCRIPT Messages (15000 Series)</b> .....	<b>600</b>
<b>Interface Messages (17000 Series)</b> .....	<b>601</b>
<b>Logic Error Messages (19000 Series)</b> .....	<b>601</b>
<b>Command Center Messages (20000 Series)</b> .....	<b>601</b>

## System Error Messages

System error messages that are not self-explanatory.

- E/1 - invalid parameters/arguments

Invalid command line parameters have been entered. See “[Solving in Batch Mode](#)” on page 367.

The file might be damaged. (It might have been overwritten by another application or transferred incorrectly over a network). There might be a problem with your file system, disk, or network. Try opening the file again. If the problem persists, see your system administrator. If the file is damaged, try restoring the most recent backup file.

- E/10 - memory allocation error

Increase your swap space or exit some processes and try again.

## User System Messages (1000 Series)

User system error, warning, and information messages that are not self-explanatory.

## User System Error Messages

- E/1001 - Out of Memory  
Increase your swap space or exit some processes and try again.
- E/1014 - Cannot delete attached property  
The property is still attached to an object. It must be detached before you can delete it.
- E/1020 - Could not create Solution Data in repository  
Check solution data directory has write access.
- E/1025 - Unable to access Solution Data failed to save grid  
Check read permission on the solution data directory for the project. You need the same group permissions as when it was created.
- E/1026 - Operation disabled (bad access to solution)  
Check read permission on the solution data directory for the project. You need the same group permissions as when it was created.
- E/1027 - Unable to access Solution Data for initialization - de-select in Solver Control/ Initial Variables  
Select Solution Set is active, but no solution set has been specified. Select a different option in the Initial Value dropdown (**Solver Control** tab), or select a solution set. This applies to each variable.
- E/1029 - Attempt to continue beyond end of transient - Check Overall times  
The full transient simulation for the period set in the Overall Transient dialog box has already been completed. If you need to continue, then increase the End Time for the total transient in the Overall Transient dialog box.
- E/1030 - Pack operation failed  
Check that you have enough space and you have write access to the directory you are writing to.
- E/1031 - Unpack operation failed Failed to get the project name from the Pack file  
The file you are trying to unpack is not packed. Check the file pathname.

## User System Warning Messages

- W/1002 - Solution data not deleted from repository  
Solution data directory permissions changed while user is running the software.
- W/1003 - Solution data not found - re-creating without results

The project being loaded has probably been imported without the solution results. You will have to re-solve to get the results.

- W/1006 - No Results - Unable to use solution data for initialization

The solution set you have chosen has no solution results available for use as initial variables.

- W/1013 - Decomposed model may not be the same as internal model.

This message may appear when decomposing the following geometry:

- Blocks with Holes and Enclosures with Holes:

Grid constraint attributes that are attached to holes are not retained in the decomposed model.

- Heat Sinks:

The heat transfer coefficient associated with the surface exchange attribute created internally for compact heat sinks is of a type not available to Simcenter Flotherm users. This is a local value in each grid cell in the heat sink fin or pin volume and is determined from a correlation based on the local Reynolds number in each cell and the geometry of the heat sink. See [Compact Modeling](#) in the *Simcenter Flotherm SmartParts Reference Guide*.

When the compact heat sink is decomposed, the surface exchange representation will be defaulted back to a constant value of zero. If the project containing the decomposed heat sink is then re-solved, the results will be different.

- Sloping Blocks and Cylinders:

A thermal attribute can be attached to (thick) sloping blocks or cylinders with a total power distributed proportionally to the internal cuboid and prism volumes. A decomposed sloping block or cylinder has the same thermal representation as the original SmartPart if the attached thermal is Non-conducting, Conducting with Total Power or Fixed Temperature.

It is also possible, however, to attach an existing thermal attribute with Fixed Heat Flow. This setting is not available for the sloping block or cylinder SmartParts and is ignored.

If the SmartPart is decomposed to an assembly of cuboids and prisms, then the settings will be valid in principal. Users should check whether the setting for Fixed Heat Flow is specified correctly for the geometry defined.

- W/1014 - The maximum number of defined overlapped localized grid spaces has been exceeded. Overlapped localized grid spaces will not be created. Consult the documentation for additional information and modeling advice.

For models that have a very large number of overlapped localized grid spaces, the time required to create the decomposed localized grid spaces can be excessive.

A limit is imposed by the environment variable MAX\_NUMBER\_OF\_OVERLAPS. When this limit is reached, support for overlapped localized grid spaces is disabled, that is, the pre-V11 grid generation method is used. You can increase the environment variable value but should be aware that processing of overlapped localized grid spaces may take several minutes. See “[Environment Variables](#)” on page 615.

## System Information Messages

- I/1003 - Removing unused deactivated keypoints

You have moved an item of geometry with deactivated keypoints. The program is just noting the removal of the deactivation of the grid at the location that the object used to occupy.

- I/1005 - Name of library property changed - existing property retained.

This message occurs because the library attribute you are attempting to attach is recognized as already existing in the project, but with different setup values, so the attachment is prevented.

For example, this can occur if you attach a library attribute to a cuboid, change its name and settings, then try to re-attach the original attribute from the library.

Simcenter Flotherm identifies attributes by internally generated serial numbers, not by their names.

Be aware that you can have problems if you modify library attributes and attach them to geometry in an assembly which you later save to the library. Importing the assembly from the library to a new project in which the library attribute is already used will prevent the modified attribute being used and the assembly definition will not be as expected.

Do not modify library attributes. If you need to create a new attribute based on the settings of a library attribute, first copy the library attribute then modify and use this new attribute instead.

## Drawing Board Messages (3000 Series)

Drawing Board error, warning, and information messages that are not self-explanatory.

### Drawing Board Error Messages

- E/3001 - Buffer failure

Exit some windows or other processes to free some memory and try again.

- E/3002 - Not enough colors

Other processes are monopolizing the screen colors. Close some processes and try again.

## Drawing Board Warning Messages

- W/3004 - View up direction was normalized

The values you entered for the up-direction have been normalized. For example, if  $x'$ ,  $y'$ , and  $z'$  are the values you enter, then the resultant value for  $x$  is given by:

$$x = \frac{x'}{\sqrt{x'^2 + y'^2 + z'^2}}$$

- W/3005 - Cannot display snap grid as its too fine - try modifying display interval  
If there is more than one snap grid line per pixel, the grid is too fine for the individual lines to be seen. Increase the snap grid display interval in the **Drawing Board** tab of the User Preferences dialog box.

## Project Manager Messages (4000 Series)

Project Manager error, warning, and information messages that are not self-explanatory.

### Project Manager Error Messages

- E/4017 - Failed to keypoint localized grid

Try changing the location of the named object in the Project Manager tree hierarchy.

- E/4018 - Zoom in Creation Error - <*more info*>

Where <*more info*> is:

- Project not consistent with solution: backup directory exists.

Solve again.

- Project not consistent with solution: residual file does not exist.

Solve again.

- Project not consistent with solution: Tag file does not exist.

Solve again.

- 2D projects cannot be used to create environment

Set Dimensionality to 3-Dimensional in the **Model Setup** tab.

- Transient projects cannot be used to create environment

Change Solution Type to Steady State in the **Model Setup** tab.

- Project must have Temperature, Pressure, Velocities, and Heat Flux. Ensure these variables are solved and run for one more iteration.

In the **Model Setup** tab, ensure that Type of Solution is set to Flow and Heat Transfer and Store Heat Fluxes is checked on prior to solving again.
- Region is not valid if it is collapsed

Uncollapse the region or create a new 3D region instead.
- Region is not valid if it is a localized grid boundary or if it is at the solution domain  
Ensure there are at least two grid cells between the region and the offending boundary.
- E/4019 - Zoom In Creation Error, Intersecting Dissipating Object/s - Invalid object(s) intersecting region:

Occurs when the thermal attribute for the named object is set to Fixed Heat Flow. Ensure that the thermal attribute is set to either Fixed Temperature or Conduction.
- E/4020 - Zoom In Creation Error, Intersecting object(s) intersecting region  
Ensure that the named object is within the region. For PCBs, both the board and components must be checked.
- E/4021 - Zoom In Creation Error - Abutting Dissipating Object/s - Invalid object(s) abutting the region  
Occurs when the thermal attribute for the named object is set to Fixed Heat Flow. Ensure that the thermal attribute is set to either Fixed Temperature or Conduction.
- E/4022 - Zoom In Creation Error - Abutting Object/s - Invalid object(s) abutting the region  
Move the object or resize the region.
- E/4026 - Due to proximity to an adjacent localized space keypoint, localized grid creation failed for *<object>*

Possible causes:

  - Two localized grid keypoints are not aligned.
  - Two localized grid keypoints are so close together that the cells that link localized grid spaces together would have been of zero volume, causing the failure.

Localized grid spaces should either be resized or aligned to avoid this failure.
- E/4999 - Grid Constraint on object within localized grid space *<object with localized grid>* could not be applied because of internal gridding tolerances. Encountered with object *<object inside localized grid>*. See documentation for further advice.

Reports an object within a localized grid space that cannot have an attached grid constraint satisfied because of internal gridding tolerances.

When determining how to distribute grid cells across an object with an attached Grid Constraint attribute, Simcenter Flotherm utilizes an internal tolerance to determine if an existing grid line should be considered as part of the current grid constraint. Sometimes, this can cause a grid constraint to ignore an existing grid cell for the purposes of the grid constraint. If this occurs within a localized grid space, such that the object's own keypoint grid line is removed from consideration for the application of the grid constraint, then:

- the defined grid setup becomes invalid,
- this error message is output, and
- Simcenter Flotherm turns off all localized grid space settings to preserve model integrity.

You have a choice of remedial actions to consider:

- Detach the grid constraint from the object that is listed in the error message.
- Increase the minimum cell size defined in the localized grid space grid constraint (or if not present, the minimum cell size for the System Grid) so that small grid cells within the localized grid will be removed, thereby reducing the number of situations in which the internal tolerance can be activated.
- Inspect the other grid lines in the vicinity of the edges of the offending object, searching for extremely small grid cells, and nearby objects that may cause these small cells. The Grid Summary tool can be used effectively for this task. Then resize or move any such identified objects so that object edges are coincident.

## Project Manager Warning Messages

- W/4005 - Localized grid - Localized grids overlap - this is not allowed

Overlapping of two grid spaces is not permitted. The localization settings for the second object will be kept, but ignored, while the spaces overlap. The same is true for localized grid spaces that are nested. Nested abutting localized grid spaces are permitted. See “[Grid Localization](#)” on page 309.

- W/4007 - Keypoints of localized grid spaces cannot be deactivated

This warning is issued when the user tries to deactivate (using the Tables application window) a keypoint which is a grid line defining the boundary of a localized grid space. Deactivating the keypoints could lead to an inconsistent grid and so is inhibited.

- W/4008 - Zoom In Creation Warning - More than 10,000 cells exist on the region boundary

Issued to warn that a large project will be created for the zoom in region.

Reduce the complexity of the geometry in the region to prevent performance degradation.

## Project Manager Information Messages

- I/4001 - Creating a new User Configuration file

Simcenter Flotherm has been started up for the first time and creates a configuration file to match the installation setup and your own system. This configuration file will be used to setup your Simcenter Flotherm session each time it starts.

To change the configuration use the User Preferences dialog box. Changes made to the configuration are saved in the configuration file.

- I/4003 - Could not write to config directory, saving user preferences to home file system

When first starting Simcenter Flotherm, if the configuration directory could not be written to the Simcenter Flotherm installation directory, then directory /floconfig, containing all the program configuration files, will be created in your home directory.

## Profiles Messages (5000 Series)

Profiles application window error, warning, and information messages that are not self-explanatory.

### Profiles Error Messages

- E/5001 - Out of available memory

Exit some windows or other processes to free some memory and try again.

- E/5003 - No valid profiles available to plot

A distance plot has been requested without defining the lines through the model along which the variable values are to be plotted.

Create the profiles in the Distance Profile Plot and try again.

- E/5005 - Internal error 1

Contact Mentor Graphics customer support and describe the actions leading up to this error.

- E/5008 - Internal error 4

Contact Mentor Graphics customer support and describe the actions leading up to this error.

- E/5009 - Unable to display a logarithmic range at or above this value

The program is unable to display large values on a logarithmic scale. Change the scale to linear.

- E/5011 - Internal error 7

Contact Mentor Graphics customer support and describe the actions leading up to this error.

- E/5015 - Divide by Zero

A value of zero was entered in a numeric field used for a calculation involving division. Correct data entry field and try again.

## Profiles Warning Messages

- W/5001 - Profile location not within solution domain

The requested location of the distance profile is outside of the solution domain and is, therefore, ignored.

- W/5002 - Solution data not yet available for selected monitor point

No results are yet available for the requested monitor point (at least one iteration for steady state or one time step for a transient must be completed). Either re-solve or, if the case is currently running, wait until at least one iteration or time step has been completed.

- W/5005 - Not enough points to plot a profile

This can occur when trying to plot a time profile when only one time-step has been solved.

- W/5017 - Null monitor point or no point attached <Point Name>

There are two possible reasons for this message:

- a. You have attempted to create an iteration profile plot but forgot to select a monitor point, or
- b. A null monitor point has been created. Contact Mentor Graphics customer support and describe the actions leading up to this error.

- W/5018 - No data available to plot

No data may be available because of a number of reasons, such as, no memory available to load data, no solution data or no points in import file.

- W/5021 - Cannot plot negative or zero values on a logarithmic scale

Negative or zero values cannot be plotted on a logarithmic scale. Use linear scale.

- W/5023 - Read errors when importing file

Errors have been encountered when reading data from an imported file. Check the format of the file is correct.

- W/5024 - Cannot display profiles of different start points on same plot

Monitor points that start at different start times for time plots and different iterations for iteration plots cannot be plotted on the same plot. Use a different plot to display profiles of different start times or iterations.

- W/5027 - Incompatible file version

The software could not read the configuration from an incompatible version of a configuration file, so it will set up a configuration.

- W/5028 - File read error

An error resulted when attempting to read a configuration file, because of an unreadable format, so it will set up a configuration.

## Tables Messages (6000 Series)

Tables error, warning, and information messages that are not self-explanatory.

### Tables Error Messages

- E/6001 - Unable to allocate more memory

Exit some windows or other processes to free some memory and try again.

- E/6005 - Unable to allocate memory for derived properties

Exit some windows or other processes to free some memory and try again.

- E/6009 - Unable to allocate memory for table

Exit some windows or other processes to free some memory and try again.

### Tables Warning Messages

- W/6001 - No data available in solution data

Results table displayed, but no results available. Solve project. If you have further problems, then check disk space.

- W/6002 - No derived property data available for current object

Derived property data expected for object, but none was found. Possible cause - problems during translation. Perform sanity check.

## Tables Information Messages (6000 Series)

- I/6003 - Tables cannot display all junction temperatures

This message is issued when the project contains a compact component with more than ten junctions. Tables can only display ten junction temperatures.

# Converter Messages (7000 Series)

Input converter error, warning, and information messages that are not self-explanatory.

## Converter Error Messages

- E/7003 - Invalid file selected

File is not recognized by the V1.4 converter — choose a *.lib*, *.prb* or *.pfm* file.

- E/7004 - Unable to open import file

There are conversion problems - check the syntax of your file.

- E/7006 - Invalid file contents or incompatible version

The file is not Simcenter Flotherm v1.4 *.prb*, *.pfm* or *.lib* format.

- E/7009 - Excessive thickness for planar object - check value

The thickness of either a conducting internal plate or external wall has been set to an excessive value. This value is used to set the physical thickness of the object rather than just a thermal parameter, therefore problems can occur. Reset the thickness to a sensible value.

## Converter Warning Messages

- W/7001 - Recirc. supply out of sequence or previous invalid

Syntax error detected in the library file. Check the location statements. This can happen if the *.lib* file has been created manually resulting in the extract object being invalid or missing.

- W/7002 - Recirc. Extract out of sequence or previous invalid

Syntax error detected in the library file. Check the location statements. This can happen if the *.lib* file has been created manually resulting in the supply object being invalid or missing.

- W/7003 - No location number for recirculation

Check for missing location number in the library file.

- W/7004 - Cannot convert, unrecognized object type

This can happen if an incorrect object type name is entered during manual file creation.

- W/7006 - Unsupported turbulence model - Defaulted to laminar

This can result from importing a .prb file that did not originate in Simcenter Flotherm V1.4.

- W/7007 - Ambient not attached to system - different ambient already set

The Version 1.4 Converter has detected a conflict in trying to attach more than one ambient to a domain edge.

- W/7008 - Ambient not attached - requires user attachment

The Version 1.4 Converter does not attach ambient to internal cut out surfaces, so you will have to add them manually.

- W/7009 - Ambient not attached - Library files require user attachment

Imported library file ambients do not attach ambients in case of conflict with existing attachments.

- W/7010 - Attempt to convert an unknown variable - ignored

This can result from importing a .prb file that did not originate in Simcenter Flotherm V1.4.

- W/7012 - Excessive thickness for planar object - reduced to 1.E5

The thickness of either a non-conducting internal plate or external wall has been set to an excessive value. This value is used to set the physical thickness of the object rather than just a thermal parameter, therefore the thickness has been reset to a value that does not cause problems.

- W/7013 - Ambient Temperature not defined in lib file - Check object

An object converted from a library file has the default ambient selected as a boundary value. As the library file does not contain this value, a default has been assumed. The user should check the setting.

- W/7014 - Resistance Free Area Ratio <=0 - Set to 1.E-10

A free area ratio of 0 or less has been detected when importing a v1.4 project. The free area ratio is reset to a small value to avoid error messages from the resistance attribute property sheet.

- W/7015 - Object located outside of solution domain — Check object

An imported v1.4 object was found to be completely outside of the imported solution domain. The user should check whether this was expected as the translator will ignore all objects outside the solution domain.

## Converter Information Messages

- I/7001 - Internal Radiation boundaries not converted

No conversion of internal radiation is carried out. The currently installed version of Simcenter Flotherm uses a different radiation model.
- I/7002 - External Radiation boundaries not converted

No conversion of external radiation is carried out. The currently installed version of Simcenter Flotherm uses a different radiation model.
- I/7007 - Default Density used for Fixed Flow SmartPart

As no inflow temperature was specified for the object, a default value for density was assumed to convert the mass flow to a volumetric flow.
- I/7008 - Default Temperature used for Fixed Flow SmartPart

As no inflow temperature was specified for the object, a default value for temperature was assumed to convert the mass flow to a volumetric flow.
- I/7009 - Default Density used for Fan SmartPart

As no inflow temperature was specified for the object, a default value for density was assumed to convert the mass flow to a volumetric flow.
- I/7011 - Default Density used for Recirculation SmartPart

As no inflow temperature was specified for the object, a default value for density was assumed to convert the mass flow to a volumetric flow.
- I/7015 - Inward direction on domain edge wrong - corrected

A version 1.4 object was placed on the edge of the domain with the inward direction set incorrectly, the converter has corrected this value.
- I/7016 - User specified htc, ambient attachment on face replaced by:<*Ambient property name*>

The ambient on a particular domain face was replaced by an attachment which sets the heat transfer coefficient.
- I/7017 - Source converted for Transient - Check values for Steady State use: <...>

Source values were imported with Transient Function values which were converted with the assumption that the source will be used in Transient mode. If the sources are to be used in a steady state mode, then check the source values.
- I/7018 - Unsupported Standard Turbulence Model converted to Revised Model

The imported version 1.4 model contains a setting for the Standard Turbulence model. This model is no longer supported in version 3.2 and has been converted to the newer Revised Turbulence model.

- I/7019 - Fixed Heat Transfer Coefficient not supported for Prisms - ignored <*prism name*>

The option to specify a fixed heat transfer coefficient on external temperature has been removed as prisms now conduct heat. The program will default to use the standard wall functions for the prism surface.

## Solver Messages (8000 Series)

Solver error, warning, and information messages that are not self-explanatory.

### Solver Error Messages

For any E/8000 error messages that are not self-explanatory and are not listed here, consult Mentor Graphics customer support.

- E/8013 - A array size error

Incorrect data files loaded from disk which leads to a negative number on the calculated A array size. The problem may be corrected by recalculating the EFG result or re-initializing the project.

- E/8070 - Inconsistent exchange factor, re-run EFG

Geometry has changed so exchange factors are no longer correct. Re-run EFG (**Solve > Exchange Factors**) to obtain correct values.

- E/8153 - Not enough memory available for the specified number of subdivisions. EFG stopped

Either free up memory by terminating other processes running on the system or reduce the number of radiation surfaces being solved, and then try again.

- E/8160 - Insufficient memory to run the solver. Solver performed abnormal exit

Either free up memory by terminating other processes running on the system and try again or add more memory.

- E/8161 - Failed to get a Solver license.

Either a solution is already in progress or the machine is not licensed to run the solver.

- E/8162 - A multiple processor solve was requested but no parallel solver license was found. Set the Number of Processors to 1 in User Preferences to use a single processor license if available.

Choose **Edit > User Preferences** to open the User Preferences dialog box. The number of processors is specified on the **Project Manager** tab.

## Solver Information Messages

- I/8008 - Solver stopped: steady solution converged. NOT all monitor points are stable, check termination residual

Monitor Point Convergence for temperature is selected in the **Solver Control** tab. Solution has converged to the specified residual, however, not all monitor points have reached the required accuracy for the specified number of iterations. Check the termination residual selected.

## Translator Messages (9000 Series)

Translator error, warning, and information messages that are not self-explanatory.

### Translator Error Messages

- E/9001 - Failed to save Solver Control data to disk (Wrong privileges or file system full?)  
Check with your systems administrator.
- E/9002 - Failed to save Solver List data to disk (Wrong privileges or file system full?)  
Check with your systems administrator.
- E/9003 - Failed to save geometric variable data to disk (Wrong privileges or file system full?)  
Check with your systems administrator.
- E/9004 - Cannot access Translator-output directory (protections wrong)  
Check write-protection on the solution data directory.
- E/9005 - Object does not lie completely inside a single localized grid  
Some objects, when in a localized grid space, must be located wholly inside or on the edge of a single localized grid space. These objects are fans, recirculation devices, fixed flows, compact components, heat sinks, thin sloping blocks.
- E/9012 - Too few grid-cells to be solved  
For a 2D model you need at least two grid cells in the X and Y directions. For a 3D model you additionally need at least two grid cells in the Z direction.
- E/9013 - 3D option needs more than 1 cell in Z direction NZ = 1  
3D models need at least two grid cells in the Z direction. Generated if a model contains only 2D objects and Dimensionality is set to 3-Dimensional in Model Setup.
- E/9015 - Solar Calculation

Solve Solar Radiation has been activated in the Solar Radiation dialog box, and requires positional as well as intensity settings.

- E/9016 - Different Fluids Adjacent at Z directed cell face <*X, Y, Z coordinates*>  
Regions with attached fluids must be totally separated from one another by a solid interface.
- E/9017 - Too many fluids  
There is a limit of 100 regions with different attached fluids.
- E/9020 - Compact heat sink base is overwritten <*object name*>, Blocked by: <*object name*>  
Overwriting the compact heat sink base seriously degrades its efficiency. Move the heat sink or the object so they no longer overlap.
- E/9097 - Monitor point for transient attribute is no longer valid [<*monitor point*> | see transient attribute]  
A monitor point specified in the Multiplier vs. Temperature tab of transient attribute has either been deactivated (in which case it is named) or deleted (in which case you will have to examine the transient attributes for an unspecified Associate Monitor Point).
- E/9101 - Compact component junction-board interface overwritten. Change node tree precedence <*object*>, Blocked by: <*object*>  
A PCB is below a 2-Resistor Compact Component in the data tree hierarchy. Promote the PCB above the Compact Component.
- E/9106 - Potential Source is not electrically connected to other Potential Sources Source Joule Heating is switched on and geometry that has Potential Source attributes attached is not inter-connected by electrically conducting (that is non-dielectric) material. Check circuit geometry for continuity.
- E/9107 - Potential source is not electrically connected to a Fixed Value Potential Source Voltage  
Joule Heating is switched on and no voltage has been defined. Define a Fixed Value Potential Source attribute and attach to a Source SmartPart that is located on the circuit geometry.
- E/9108 - Failed to realize object  
Object ignored by the mesher and solver.  
This can occur for PCBs if the dielectric or conductor material has Orthotropic or Temperature Dependent conductivity and the % Conductor By Volume is less than 100. In such cases, E/19021 is also generated.

## Translator Warning Messages

- W/9001 - Object has been entirely over-written <*over-written object*>, Blocked by:<*over-writing object*>

Coincident objects, the last one in the hierarchy normally takes precedence. See “[Construction Precedence Rules](#)” on page 218. Where an object has been overwritten by a combination of objects, then <*over-writing object*> is output as <multiple objects>, and it is up to the user to identify the objects.
- W/9003 - Overlap of this object with another is disallowed therefore it is deactivated

See geometry hierarchy description “[Construction Precedence Rules](#)” on page 218.
- W/9012 - Fluid flow entirely blocked by solid, or not directed into solution domain - it is deactivated

The flow exit is obscured by a solid. You might just need to rotate it by 180°.
- W/9013 - Object is in contact with more than one External Boundary surface therefore it is deactivated

Generated when no unique ambient can be deduced by the software. Move object or reallocate ambient.
- W/9014 - Object set to conducting with no material attached therefore it will not conduct heat

A conducting object has no material attached, so the conduction and friction sources will be deactivated. However, the obstruction caused by the object will remain and unless you attach a material to the object, the results will be incorrect.
- W/9021 - Resistance is NOT collapsed but has planar resistance property attached therefore it is deactivated

Volume resistance with a planar property type is inconsistent. Either collapse the resistance or set the resistance attribute type to volume.
- W/9022 - Resistance is collapsed but has volume resistance property attached therefore it is deactivated

Planar resistance with a volume property type is consistent. Either un-collapse the resistance or set the resistance attribute to planar.
- W/9024 - Resistance in partial contact with external boundary surface therefore it is deactivated

Move resistance to ensure it is either fully attached to an external surface or stands on its own.
- W/9028 - Object is in contact with more than one external boundary surface therefore it is deactivated

No unique ambient can be deduced by the software.

- W/9029 - Flow direction is parallel to object plane therefore it is deactivated  
Flow direction must have a non-zero normal component.
- W/9030 - Object or Sloping surface of Object overwritten by blockage <*blocked object*>, Blocked by: <*blocking object*>  
See geometry hierarchy description in “[Construction Precedence Rules](#)” on page 218.
- W/9033 - Fan deactivated: direction must have non-zero normal component  
When the angled fan option is activated, the flow direction must have a non-zero normal component.
- W/9034 - Inconsistent source - source attribute must be fixed value when used with mass source  
When a mass source is active, all other variables must have fixed value source attributes.
- W/9040 - More than one symmetry boundary defined on X direction - ignored for radiation
- W/9041 - More than one symmetry boundary defined on Y direction - ignored for radiation
- W/9042 - More than one symmetry boundary defined on Z direction - ignored for radiation

These messages are generated when both domain faces in the referenced direction are set to symmetry with radiation switched on.

As two radiation symmetry faces on opposite ends of the solution domain does not make physical sense for thermal radiation calculation, the radiation symmetry on both faces is ignored. By using one symmetry face in each direction, the warning is removed.

- W/9047 - Thermostat not inside domain: Recirculation device deactivated  
The thermostat attached to the recirculation device is located outside of the overall solution domain, hence the recirculation device has not been included in the solution. Relocate the thermostat to be within the overall solution domain and use the drawing board application window to check the location of the thermostat before continuing with the solution.
- W/9049 - No source has been activated in attached source property  
A source attribute is attached to a source cuboid, but the source has not been activated for any variable.

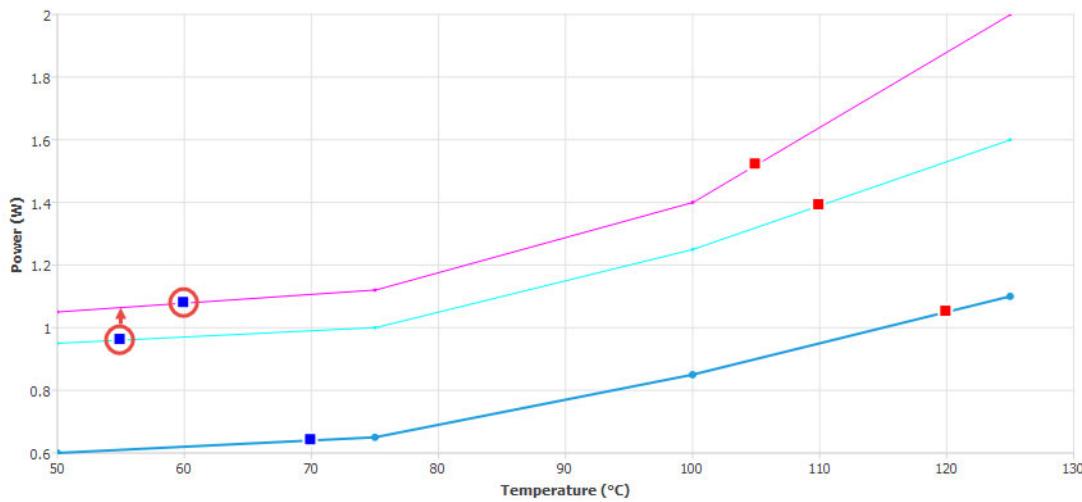
See [Source Attribute Property Sheet](#) in the *Simcenter Flotherm Project Attributes Reference Guide* for information on how to activate the source attributes.

- W/9092 - No active monitor point attached to controller object - controller will be ignored.

There is no monitor point child of a Controller SmartPart, or there is one but it is deactivated.

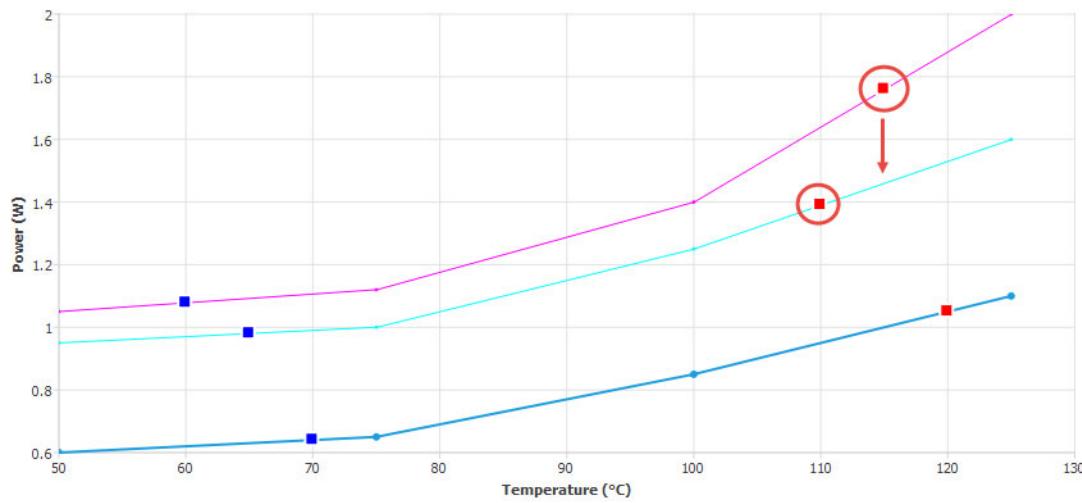
- W/9093 - Inconsistent frequency curves – Minimum temperature limit of a frequency curve doesn't fall within the operating temperature range of the next highest defined frequency.

Applies to power versus temperature curves for Control attributes. For example, see the power curve below.



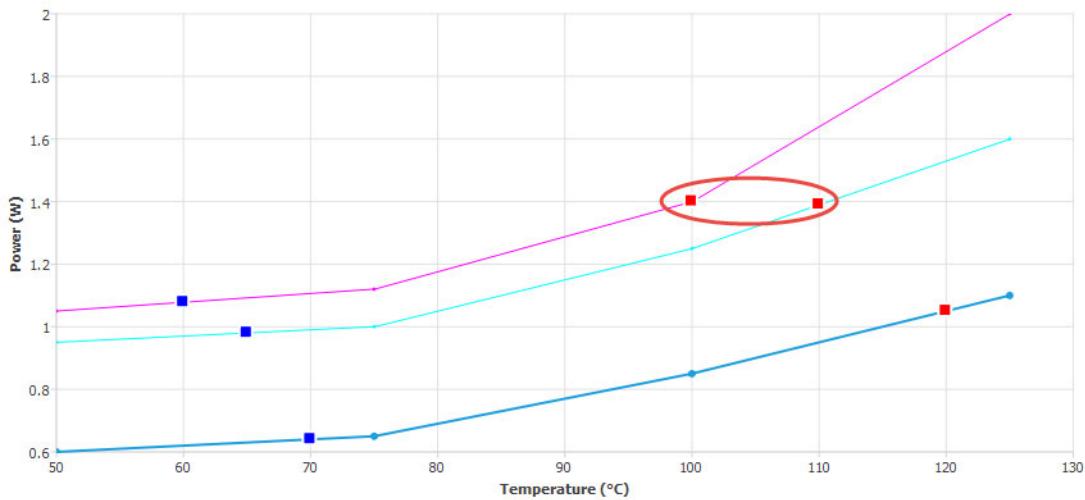
- W/9094 - Inconsistent frequency curves – Maximum temperature limit of a frequency curve doesn't fall within the operating temperature range of the next lowest defined frequency.

Applies to power versus temperature curves for Control attributes, see example, below.



- W/9095 - A lower frequency controller curve is partially or completely above a higher frequency curve.

Applies to power versus temperature curves for Control attributes, see example below. This power curve will also generate a W/9093 and a W/9094 message.

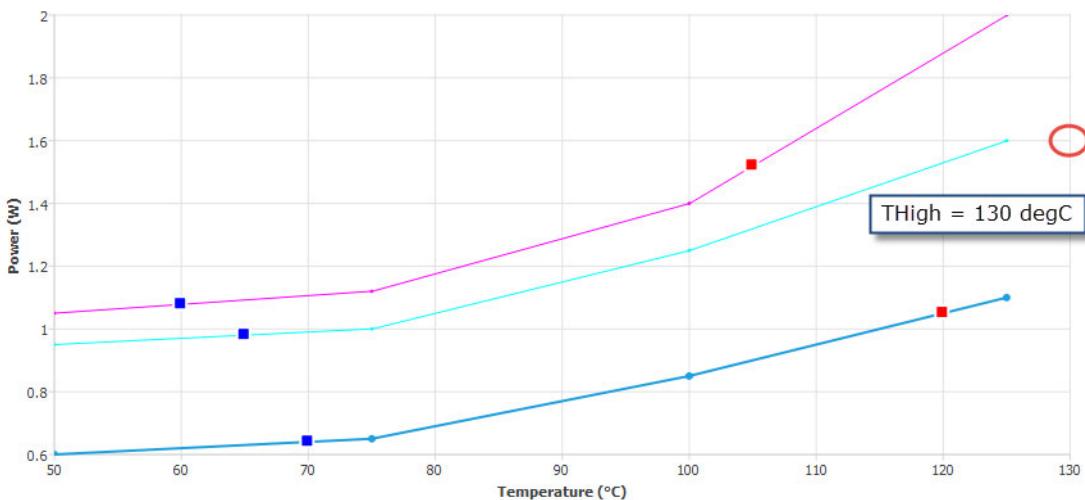


- W/9096 - No Control attribute attached to controller.

A Control attribute is attached to a Controller SmartPart from the **Construction** tab of the Controller property sheet.

- W/9097 - Controller attribute curves should span the full range between TLow and THigh to enable stepping between curves

Applies to power versus temperature curves for Control attributes. For example, for the middle curve below, THigh has been defined as 130 degC, but the maximum temperature defined for the curve is 125 degC.



## Translator Information Messages

- I/9007 - Collapsed plate thickness extends more than one cell, results unpredictable, consider modeling as thick plate

With Activate Plate Conduction checked in the **Solver Control** tab, the program has detected a collapsed cuboid with thickness in the collapsed direction greater than that of the grid cell in that direction.

Collapsed cuboids are considered sub-cell features. Consider modeling the plate using uncollapsed cuboids.
- I/9008 - Symmetry boundary cannot overwrite open boundary - boundary will remain open

An attempt has been made to overwrite an open domain boundary with a symmetry face of a cutout. An open boundary cannot be overwritten by a symmetry boundary, so it remains open.
- I/9010 - Symmetry Boundary overwritten with attached Ambient. All settings for heat transfer coefficient and radiant temperature will be applied.

By default, no heat or fluid flow passes through a symmetry plane. A symmetry boundary with an attached ambient no longer acts as a symmetry plane for heat transfer, allowing heat flow through the symmetry plane. Fluid flow will still be prevented.
- I/9011 - Open External Boundary surface without Ambient Property

Global ambient will be used by default - check that this is what is intended.
- I/9012 - Object has not been placed on a Solid or Domain Boundary

You are attempting to locate a fixed flow or recirculation device in an unsuitable position.
- I/9014 - PCB blocks > 1/2 cell so changed to complete blockage

Blockage caused by component will fill the near-board cell.
- I/9015 - Thermal Power is ignored for SmartParts

Thermal power can only be attached to individual cuboids.
- I/9016 - Fan curve - local minima ignored

If a fan curve has a local minima, then Simcenter Flotherm uses a modified fan curve which ignores the minima.

See Usage Notes under [Fan Curve Chart Dialog Box](#) in the *Simcenter Flotherm SmartParts Reference Guide*.
- I/9017 - Radiation property attached to surface of non-conducting block

Radiation properties attached to non-conducting blocks have no effect and are ignored.  
Radiation sources can only be applied to conducting objects.

- I/9021 - No Flow Boundaries, but user-set Convection Velocity is 0  
Set the EFCV in the **Solver Control** tab.
- I/9022 - Change in variables, cannot continue residual display  
Solution continues, but display restarts from current iteration number.
- I/9033 - Total number of Grid Cells are: <XXXXXX> + <YYYYYY> from embedded conduction solver = <ZZZZZZ>  
Details of the generated mesh reported for all models, where:
  - <XXXXXX> is the normal system plus the localized grid cell count.
  - <YYYYYY> is the contribution from the embedded conduction solver, if appropriate.
  - <ZZZZZZ> is the total grid size (<XXXXXX> + <YYYYYY>).
- I/9041 - At least 2 Components intersect  
Sanity check shows there are discrete PCB components overlapping. Check the PCB property sheet.
- I/9042 - A Component's X position is located off the board
- I/9043 - A Component's Y position is located off the board
- I/9044 - A Component's X boundary is located off the board
- I/9045 - A Component's Y boundary is located off the board  
Sanity check shows the boundary of the PCB component extends beyond the perimeter of the PCB.
- I/9050 - Resistance property attached to 2D fan is ignored  
In the case of 2D fans, the flow velocity is calculated from the fan curve and local pressures only because any attached resistance acts at the same location as the fan primitive, so the fan cannot detect a pressure drop. For 3D fans, however, the resistance and fan primitive are located at opposite ends of the fan, so a pressure drop can be detected.

Also see “Modeling the Effects of Finger Guards or Grilles” under [Applications of Fan SmartParts](#) in the *Simcenter Flotherm SmartParts Reference Guide*.

- I/9051 Compact Component interconnect resistance partially over-written  
Compact components use collapsed cuboids to represent component-board interconnect resistances such as balls/pins, and the standoff resistance. This message is issued when any of these collapsed cuboids are partially overwritten by another object, as this will

affect the compact component representation, and thereby have an influence on the results.

Any of these collapsed cuboids which are completely over-written will generate a "W/ 9001 - Object has been entirely over-written" message.

The most likely cause of this message is that the component is positioned in the project data tree above the board on which it sits. The remedy is to make sure that the board appears before any attached compact components in the data tree.

- I/9078 Model is transient but density or specific heat have values of 1.0 in the material attribute *<material attribute name>*

Although a solution will be attempted this warning is issued to make you aware that either the Density or Specific Heat of a material has been defined as having a value of "1.0". The message is only generated if a transient solution has been selected.

- I/9083 - Transient attribute may impose a negative transient multiplier. Fan will be deactivated during this transient condition *<object>*

This message appears when a fan is attached with a transient attribute *and* the transient multiplier for that attribute will be negative at some point during the total solution time. Applies for both the Multiplier vs Time and Multiplier vs Temperature options.

## **EFG Messages (10000 Series)**

EFG error, warning, and information messages that are not self-explanatory.

### **EFG Error Messages**

- E/10010 - Error accessing solution data

A file read/write error has been detected while attempting to calculate the exchange factors. Check solution data directory has read/write access and that there is enough disk space.

- E/10011 - Error loading zonal1.pts file

A file read error has been detected while attempting to calculate the exchange factors. Check read permissions.

- E/10013 - Insufficient memory

Increase your swap space or exit some processes and try again.

- E/10014 - Error writing log file

Check solution data directory has write access and that there is enough disk space.

- E/10015 - Error writing exchange factors

A file write error has been detected while attempting to calculate the exchange factors. Check solution data directory has write access and that there is enough disk space.

- E/10016 - Error loading view factors

A file read error has been detected while attempting to load the view factors file while attempting to calculate the exchange factors. The file might be corrupted or does not have read access. Also, check disk space.

- E/10021 - A multiple processor exchange factor solve was requested but no parallel solver license was found. Set the Number of Processors to 1 in User Preferences to use a single processor license if available.

Choose **Edit > User Preferences** to open the User Preferences dialog box. The number of processors is specified on the **Project Manager** tab.

## Data Storage Messages (11000 Series)

Data storage error, warning, and information messages that are not self-explanatory.

### Data Storage Error Messages

For E/11000 errors that are not self-explanatory and are not listed here, contact Mentor Graphics customer support.

- E/11013 - Failed to get a lock: If project not already in use Select [Project/Load/Unlock] to unlock

Unable to give access to the project for one of the following reasons: it is already being used by somebody else, the system crashed when the project was open, or you are trying to delete the currently loaded project.

- E/11015 - Failed to create file

Check that you have enough space and write access to the directory you are trying to write to.

- E/11016 - Failed to open file

Check file exists.

- E/11019 - Failed file not found

Check file path.

- E/11022 - Failed incompatible file version Future PDML version encountered

The file is incompatible with this version of the database.

- E/11044 - Failed not owner of project

You cannot delete or unlock a project you do not own.

## MCAD Messages (12000 Series)

MCAD error, warning, and information messages that are not self-explanatory.

### MCAD Error Messages

- E/12001 - Failed to open file  
Check the file exists and that you have the correct permissions to open the file.
- E/12002 - Failed to read in file  
The file format may be corrupted.
- E/12004 - Failed to write file  
Check that you have write permission to the directory.
- E/12007 - No topology found in IGES file trying to recover  
The structure definition is missing from the file. Regenerate the \*.igs file reading the advice given in *Generating IGES files from Pro/ENGINEER Release 20.0*, *Generating IGES files from IDEAS 7* and *Exporting IGES Files from IDEAS 5*.
- E/12010 - Unknown unit type, assuming meters  
The type of units used in the external file are not of the expected type and meters are used instead. If this is incorrect, then regenerate the file using standard units.
- E/12012 - ACIS model error  
There was an internal error in the ACIS Solid Modeler. If this is causing problems, contact Mentor Graphics customer support.
- E/12014 - Out of memory  
The MCAD application has run out of memory. Increase your swap space or exit some processes and try again.
- E/12015 - Unexpected end of file - aborting read  
Badly formatted file. Check file syntax.
- E/12016 - Failed to Heal Model  
Input data is not accurate enough.
- E/12017 - IGES read failed - will try to read with legacy IGES Reader  
The legacy reader is not so robust but does support assemblies of IGES files.

- E/12018 - Failed to convert parameter data to ACIS entities

Failed to read an IGES file. To debug the file, set the FLOMCADKEEPLOGFILES environment variable to TRUE and restart Simcenter Flotherm before attempting to import the MCAD file again. MCAD Bridge will create the log file in your system's TEMP directory, with the appropriate file type and date stem-name, for example:

*IGES\_READ-05-04-16-35-35.log*

- E/12019 - User Interrupt, Operation aborted

Regenerate the file and try importing again.

- E/12020 - Failed to remove feature(s) of face(s)

Could not grow the model back to a solid after attempting to remove faces or features — invalid operation attempted.

- E/12028 - Failed to rotate align model

Contact Mentor Graphics customer support with details of the problem.

- E/12029 - Error in Step Reader

Check if the imported design looks OK. If not suitable for use, check you are using a supported version of STEP.

See [Supported Import File Formats](#) in the *Simcenter Flotherm MCAD Bridge User Guide*.

- E/12030 - Fatal Error in Step Reader

STEP file might be corrupted. Regenerate STEP file and try importing again.

- E/12032 - Error in CATIA Reader

Check if the imported design looks OK. If not suitable for use, check you are using a supported version of CATIA.

See [Supported Import File Formats](#) in the *Simcenter Flotherm MCAD Bridge User Guide*.

- E/12033 - Fatal Error in CATIA Reader

CATIA file might be corrupted. Regenerate CATIA file and try importing again.

- E/12034 - Error in PRO/E Reader

Check if the imported design looks OK. If not suitable for use, check you are using a supported version of Pro/E.

See [Supported Import File Formats](#) in the *Simcenter Flotherm MCAD Bridge User Guide*.

- E/12035 - Failed to perform requested Slice

The resultant representation of the imported design might not be of a high enough quality. Try healing the model using **Tools > Heal**.

- E/12036 - Current Platform will not support the MCAD AW  
Current platform requires Open GL. See your system administrator.
- E/12037 - Failed to get a license for this translator  
Check your license type. If it is a floating license, it might be checked out. If you need a license, then contact customer support.
- E/12038 - PRO/E read failed - will try to read with InterOp Pro/E Reader  
By default, MCAD Bridge uses the TTF reader for Pro/Engineer files. If the read fails, then MCAD Bridge will retry, but using the InterOp PRO/E Reader.
- E/12040 - Voxelize mesh is too large  
There is an upper limit of ten million cells.

## MCAD Warning Messages

- W/12003 - Accuracy reported below 1e-6, might have problems with model  
Simcenter Flotherm works to an accuracy of 1e-6. Change accuracy in your MCAD package.  
See [Creating MCAD Files Using External Software](#) in the *Simcenter Flotherm MCAD Bridge User Guide*.
- W/12004 - No Geometry (Bodies or Primitives) in the assembly, clearing model  
Message appears during tidy-up process after the deletion of all the component parts of the MCAD assembly.
- W/12005 - Unexpected form for group entity  
Formatting error. While reading an IGES file, came across an unexpected entity name.
- W/12006 - First entity in group is not to a surface  
Formatting error found while reading IGES file.
- W/12007 - ACIS Model Warning  
There was an internal warning generated in the ACIS Solid Modeler. If this is causing problems, then contact Mentor Graphics customer support.
- W/12008 - Body appears to be a thin non-planar shell, this could cause problems in the dissector

This type of object is not easily modeled in Simcenter Flotherm as a collection of primitives. Instead of using the MCAD package to model the part, try modeling it using an Enclosure SmartPart.

- W/12009 - Degenerate triangle found in STL data - ignoring it

An illegal degenerate triangle (a line or a point) has been detected in the input .stl file and cannot be converted to a planar face, hence it is being ignored.

- W/12012 - Warning in Step Reader

Check if the imported design looks OK. If not suitable for use, check you are using a supported version of STEP.

See [Supported Import File Formats](#) in the *Simcenter Flotherm MCAD Bridge User Guide*.

- W/12013 - Warning in CATIA Reader

Check if the imported design looks OK. If not suitable for use, check you are using a supported version of CATIA.

See [Supported Import File Formats](#) in the *Simcenter Flotherm MCAD Bridge User Guide*.

- W/12014 - Warning in PRO/E Reader

Check if the imported design looks OK. If not suitable for use, check you are using a supported version of PRO/E.

See [Supported Import File Formats](#) in the *Simcenter Flotherm MCAD Bridge User Guide*.

- W/12015 - Current Platform will not support lighting model

Current platform requires Open GL. See your system administrator.

- W/12017 - Specified voxelization mesh resolution is insufficient to capture all conduction paths. Multiple disconnected cuboid groups have been formed for <MCAD part name>

Cuboid groups are disconnected or are only connected edge-to-edge. If you want cuboid faces to be in contact, undo the voxelize, refine the mesh and re-voxelize.

## FloSCRIPT Messages (15000 Series)

FloSCRIPT error, warning, and information messages that are not self-explanatory.

### FloSCRIPT Error Messages

- E/15007 - Failed to Load project

Check for the correct permissions to open the file.

## Interface Messages (17000 Series)

Interface error, warning, and information messages that are not self-explanatory.

### Interface Information Messages

- I/17003 - Solver Interrupted

All Simcenter Flotherm operations are inhibited until the solver saves the results obtained so far.

## Logic Error Messages (19000 Series)

Logic error messages that are not self-explanatory.

### Logic Error Messages

- E/19021 - PCB Material conductivity type must be Constant

You have assigned a material that has “Constant” conductivity to a PCB, but then changed that material’s conductivity to be Othotropic or Temperature Dependent.

- E/19022 - PCB Mass in incorrect range

The Board Mass is out of range, based on the volume of the board and the densities of the dielectric and conducting materials.

## Command Center Messages (20000 Series)

Command Center messages that are not self-explanatory.

### Command Center Warning Messages

- W/20102 - Collision Detection does not support Dependent Variables that depend on non Design Parameters: <*input variable*>

You have defined a Dependent Variable expression that is not solely a function of Design Parameter type input variables, for example, the expression may be a function of a Dependent Variable type input variable. Collision Detection is not supported for these scenarios.

- W/20103 - Collision Detection does not support Non-Linear Dependent Variables: <*input variable*>

Collision detection is only implemented for linear dependent variables of the strict form,  $mx + c$ , where  $c$  is the calculated “Constant Term” shown in the Dependent Variable dialog box. This warning message is generated when collision detection is not supported.

# Chapter 19

## Technical Support

---

Mentor Graphics offers a skills resource network to help you use Simcenter Flotherm as effectively as possible.

After software registration, the technical support includes:

- Support services via our dedicated website
- Training
- Modeling services

<b>Global Customer Support and Success .....</b>	<b>603</b>
<b>Training .....</b>	<b>603</b>

## Global Customer Support and Success

A support contract with Siemens Digital Industries Software is a valuable investment in your organization's success. With a support contract, you have 24/7 access to the comprehensive and personalized Support Center portal.

Support Center features an extensive knowledge base to quickly troubleshoot issues by product and version. You can also download the latest releases, access the most up-to-date documentation, and submit a support case through a streamlined process.

<https://support.sw.siemens.com>

If your site is under a current support contract, but you do not have a Support Center login, register here:

<https://support.sw.siemens.com/register>

## Training

Mentor Graphics provides training in all aspects of thermal management of electronics.

To inquire about training or enroll on classes:

- Visit our training pages at the Mentor Graphics website:  
<https://www.mentor.com/training-and-services>
- Contact your local account manager for details.



# Appendix A

## Command Reference

---

This appendix contains reference information for command line commands.

<b>flotherm</b> .....	<b>606</b>
<b>batchSolve</b> .....	<b>612</b>
<b>flogate_cl</b> .....	<b>613</b>

# **flotherm**

Runs on Windows systems.

Solves a Simcenter Flotherm project, or runs a FloSCRIPT file.

## Usage

`flotherm [[-b | -p] project] [options] | -env`

`flotherm [-b] -f floscript`

## Arguments

- `-b`

Usage with ***project***. Runs the command in batch mode with the specified project. If you want to use parallel processing in batch mode, you must specify the number of processors using the CCNUMBERTHREADS environmental variable, see “[Environment Variables](#)” on page 615.

Usage with ***floscript***. Runs the FloSCRIPT file without opening the user interface, unless the FloSCRIPT file contains EDA Bridge commands, in which case the EDA Bridge user interface will open.

- `-p`

Runs the command interactively and attempts to read in the specified project when the program is loaded.

- ***project***

One of the following:

The name of the project to be solved. If there are any spaces in the project name, then, following standard Microsoft® Windows conventions, the project name must be enclosed in single or double quotation marks. For example, `\\"my project\\"`.

The name of a project PDML file.

The name of a project FloXML file.

- `-i another_project`

Initializes the starting solution using the solution set of a different project from that currently loaded. The project solution set must exist in the default project directory.

- `-x`

Forces the software to only calculate the radiation exchange factors.

- `-s`

Forces the software to only solve the CFD equations.

- `-z pack_filename`

Exports the solved project as a compressed file for archiving or transfer to other systems.

- **-r *htmlfile***  
Exports all the Tables specification and results data to the named *html* file.
- **-c**  
Ensure that the base case is fully solved before using this option to solve all the unsolved Command Center scenarios for the project.  
To use parallel processing when solving scenarios and the base case for a Command Center project, you must specify the number of processors using the CCNUMBERTHREADS environmental variable (the Number Of Processors To Use setting in the User Preferences dialog box is ignored).  
Use the FLOVOLUNTEERMAXJOBS environmental variable to set the maximum number of scenarios to be solved simultaneously.
- **-o *output\_directory***  
Outputs the tables results data in CSV file format to the specified directory.

**Table A-1. Results Tables Output CSV Files (-o Option)**

Filename	Description
For all cuboids:	
<i>cubvol_Temperature.csv</i>	Cuboid names and temperature
When monitor points are present:	
<i>mon_Density.csv</i>	Density at monitor points
<i>mon_DissTurb.csv</i>	Dissipation of turbulence at monitor points
<i>mon_KETurb.csv</i>	Kinetic energy of turbulence at monitor points
<i>mon_Potential.csv</i>	Electrical Potential at monitor points (Joule Heating On)
<i>mon_Pressure.csv</i>	Pressure at monitor points
<i>mon_Temperature.csv</i>	Temperature at monitor points
<i>mon_TurbVis.csv</i>	Turbulent viscosity at monitor points
<i>mon_XVelocity.csv</i> , <i>mon_YVelocity.csv</i> and <i>mon_ZVelocity.csv</i>	Velocities at monitor points
When volume regions are present:	
<i>regvol_Bn.csv</i>	Bottleneck number in volume regions
<i>regvol_Density.csv</i>	Density in volume regions
<i>regvol_DissTurb.csv</i>	Dissipation of turbulence in volume regions
<i>regvol_FieldError.csv</i>	Error Field values
<i>regvol_Generation.csv</i>	Source of turbulence generation

**Table A-1. Results Tables Output CSV Files (-o Option) (cont.)**

Filename	Description
<i>regvol_KETurb.csv</i>	Kinetic energy of turbulence in volume regions
<i>regvol_MagGradT.csv</i>	Temperature gradient in volume regions
<i>regvol_MagHeatFlux.csv</i>	Heat flux in volume regions
<i>regvol_MagMassFlux.csv</i>	Mass flux in volume regions
<i>regvol_PowerDensity</i>	Power density in volume regions
<i>regvol_Pressure.csv</i>	Pressure in volume regions
<i>regvol_Sc.csv</i>	Shortcut number in volume regions
<i>regvol_SolarViz.csv</i>	Dimensionless SolarViz scalar field values
<i>regvol_Speed.csv</i>	Speed in volume regions
<i>regvol_Temperature.csv</i>	Temperature in volume regions
<i>regvol_TurbVis.csv</i>	Turbulent viscosity in volume regions
<i>regvol_xGradT.csv</i> , <i>regvol_yGradT.csv</i> and <i>regvol_zGradT.csv</i>	Directional temperature gradient
<i>regvol_XHeatFlux.csv</i> , <i>regvol_YHeatFlux.csv</i> and <i>regvol_ZHeatFlux.csv</i>	Directional heat flux
<i>regvol_XMassFlux.csv</i> , <i>regvol_YMassFlux.csv</i> and <i>regvol_ZMassFlux.csv</i>	Directional mass flux
<i>regvol_XVelocity.csv</i> , <i>regvol_YVelocity.csv</i> , and <i>regvol_ZVelocity.csv</i>	Velocities in volume regions
When plane regions are present:	
<i>planar_regions.csv</i>	Field results over plane regions
For all conductors:	
<i>solidcuboid_conduct.csv</i>	Solid conductor fluxes
For the solution domain:	
<i>Cutouts_Overall.csv</i>	Cutouts and overall solution domain

- *-O output directory*

Outputs the information for *all* the tables in CSV file format to the specified directory.

In addition to the files listed in [Table A-1](#), the files listed in [Table A-2](#) may be output.

**Table A-2. All Tables Output CSV Files (-O Option)**

<b>Filename</b>	<b>Description</b>
For geometry tables:	
<i>collapsed_resistances.csv</i>	Collapsed Resistances
<i>Component_Fluxes.csv</i>	Conductive, convective, and radiative heat fluxes for components and network assembly nodes
<i>Component_Nodes.csv</i>	Temperatures at component and network assembly nodes
<i>Cooler_smartparts.csv</i>	Cooler SmartParts
<i>Cutouts_Overall.csv</i>	Cutouts and overall solution domain
<i>Die_smartparts.csv</i>	Die SmartParts
<i>fans_smartparts.csv</i>	Fan SmartParts
<i>Fixed_Flows_smartparts.csv</i>	Fixed Flow SmartParts
<i>Heat_Pipe_smartparts.csv</i>	Heat Pipe SmartParts
<i>input_geometry.csv</i>	Geometry model
<i>Network_Assembly_smartparts.csv</i>	Network Assemblies
<i>Powermap_smartparts.csv</i>	Powermap SmartParts
<i>Rack_smartparts.csv</i>	Rack SmartParts
<i>Recirculation_smartparts.csv</i>	Recirculation Device SmartParts
<i>TEC_smartparts.csv</i>	TEC SmartParts
For attribute tables:	
<i>Ambient.csv</i>	ambient/object
<i>Fluid.csv</i>	fluid attributes
<i>GridConstraint.csv</i>	grid constraint/object
<i>Material.csv</i>	material/object
<i>Object_Properties.csv</i>	objects/properties
<i>RadiationProperty.csv</i>	radiation/object
<i>ResistanceProperty.csv</i>	resistance/object
<i>SourceProperty.csv</i>	source/object
<i>SurfaceExchange.csv</i>	surface exchange
<i>SurfaceFinish.csv</i>	surface finish/object
<i>Thermal.csv</i>	thermal/object

**Table A-2. All Tables Output CSV Files (-O Option) (cont.)**

Filename	Description
<i>TransientProperty.csv</i>	transient attributes

- **-env**  
Sets up the path and environment variables required to run Simcenter Flotherm and its utilities, but does not run Simcenter Flotherm itself. This replaces the setup command line utility available for earlier versions of Simcenter Flotherm.

---

**Note**

 This environment is set up for you when Simcenter Flotherm runs in interactive mode, or if a command window has been opened from the Start menu Environment Shell option.

---

- **floscript**

Runs the specified FloSCRIPT file at startup.

## Description

A typical installation puts flotherm in folder:

`<install_dir>\flosuite_v<version>\flotherm\WinXP\bin\`

There is a defined rule of precedence for concurrent command line options. The -p option overrides -b. If the -p *project* option fails, the default project is loaded. If the specified batch project fails to load, the program exits.

---

**Note**

 If both -x and -s are used, the program calculates the radiative Exchange Factor before the CFD solution.

---

Default: Omitting -x and -s forces the program to decide whether to perform the radiation Exchange Factor calculations prior to CFD solution. That is, if radiation is on and there is not a valid set of view factors, then the view factors will be calculated.

## Examples

- To start up Simcenter Flotherm interactively with project myproject already loaded:

**flotherm -p myproject**

Omitting the -p option results in Simcenter Flotherm being loaded with the default project installed with the software.

- To output the tables specification and results to an *html* file:

**flotherm -b \"my project\" -r \"my html file\"**

- To run Simcenter Flotherm from FloSCRIPT file, *MyLogFile\_saved.xml*:

```
flotherm -f \"<install_dir>\flosuite_v<version>\flotherm\WinXP\bin\LogFiles\MyLogFile_saved.xml\"
```

## Related Topics

[Files Used Before, During, and After Batch Solves](#)

[Batch Solve Procedures](#)

[Environment Variables](#)

[About FloSCRIPT](#)

# batchSolve

Runs on Linux platforms.

Solves a Simcenter Flotherm project, described by a FloXML or PDML file, in a batch run.

## Usage

batchSolve *file* [options]

## Arguments

- *file*

Name of a Project FloXML or PDML file to be solved.

- [options]

The same options as for the **flotherm** command, except for the following:

-b

This is not required because **batchSolve** always operates as a batch command.

-c

Ensure that the base case is fully solved before using this option to solve all the unsolved Command Center scenarios for the project.

The FLOVOLUNTEERMAXJOBS environmental variable is ignored; only one scenario is solved at a time.

-env

This is a Windows-only option. To set up the path and environment variables required to run Simcenter Flotherm and its utilities, run **source setup**, as described under “[Environment Setup on Linux Systems](#)” on page 368.

## Description

A typical installation puts **batchSolve** in directory:

*/opt/MentorMA/flosuite\_v<version>/flotherm/LINUX/bin*

## Related Topics

[Using batchSolve to Solve in Batch Mode on Linux Systems](#)

[flotherm](#)

[FloXML Files](#)

# flogate\_cl

Runs on Windows systems.

Converts files into ASCII or binary PDML.

## Usage

```
flogate_cl -i format -r file [-l library] -o format -w file
```

## Arguments

- If no arguments are included in the command line then help text is displayed.  
It lists all the command options.
- **-i *format***

The input file format, which can be any of the following:

PDML

T3Ster

IDF

IDF\_PCB

“IDF Library Link”

“V1.4 File”

“V2/V3 Project”

XML

A Project FloXML file conforming to the FloXML Schema and containing a Minimum Project FloXML Definition.

Format names are not case sensitive and any names containing spaces must be bound by quotation marks.

- **-r *file***

The name of file (or IDF board file) to be read.

- **-l *library***

The name of the IDF library (IDF files only).

- **-o *format***

The name of output format, which can be any of the following:

PDML

“ASCII PDML”

- **-w *file***

The name of the output file.

## Examples

To convert an ASCII PDML file to a binary PDML file:

```
flogate_cl -i "ASCII PDML" -r ascii_pdml.pdml -o PDML -w binary_pdml.pdml
```

This would allow an ASCII PDML file that has been written by an external source or macro to be converted into the binary form for license-free import into Simcenter Flotherm.

---

### Note

 A flogate license increment line is required to convert to and from ASCII PDML. Contact your local Mentor Graphics Mechanical Analysis office for further information.

---

## Related Topics

[FloXML Files](#)

# Appendix B

## Advanced Operations

A list of environment variables.

**Environment Variables .....** **615**

## Environment Variables

Simcenter Flotherm uses a number of environment variables to control the behavior of various aspects of the program.

---

### **Caution**

 These variables are only recommended for the advanced user, and are not needed for the basic running of the software and may result in undocumented appearance, features, and behavior.

---

**Defined** indicates that any setting is valid; setting the value to TRUE in these cases will suffice.

**Table B-1. Environment Variables**

Environment Variable Name	Value	Comment
ABUTTING_DISALLOWED	<b>Defined</b>	Fails the sanity check if objects or grid spaces are abutting.
CALIBRATION_EXPORT_DATA	<i>Directory path</i>	Defines the destination of exported Model Calibration transient data files. See <a href="#">Exporting Transient Data to T3Ster in the Simcenter Flotherm Command Center User Guide</a> .
CCNUMBERTHREADS	<i>Number</i>	Sets the maximum number of threads (number of processors) for the parallel solver when running in batch mode or the Command Center.
COMPUTERNAME	None	Gets the computer name.

**Table B-1. Environment Variables (cont.)**

Environment Variable Name	Value	Comment
DEFAULT_ODB_PROPERTY	Enabled   <i>&lt;property name&gt;</i>	Reads component properties from ODB++ files when importing to EDA Bridge, and displays these in an expandable <b>ODB Property</b> field in the component property sheet. If <i>&lt;property name&gt;</i> is specified then that name is the default that appears in the field.
FLO_CALIBRATION_LOAD_THREADS	<i>Number</i>	By default, half of the available hardware threads are used when T3Ster calibration data is loaded to the Command Center. You can set this environment variable to force the use of a specified number of threads.
FLOCENTRALDIR	<i>Directory path</i>	Specifies where the central library files are kept.
FLO_DOUBLEP	<i>Defined</i>	Sets the solver to use double precision instead of single precision.   <b>Note:</b> If you only want to use the double-precision solver on a per project basis then set the Use Double Precision Solver option in the <b>Solver Control</b> tab, see “ <a href="#">Double-Precision Solver</a> ” on page 418.
FLO_EMAIL	<i>Email address</i>	Requests solver status emails at end of a solve. used in conjunction with MAIL_HOST, which sets the server host.
FLO_FAN_NOISE_COEFF	<i>Number</i>	Overrides default coefficient (50) where Fan Noise (dB) = Nlog10 (derating factor).
FLO_FAN_POWER_EXP	<i>Number</i>	Specifies the power exponent in the calculation of derated fan power. Default value is 3. Derated Fan Power = Derating FactorN × Original Fan Power.

**Table B-1. Environment Variables (cont.)**

<b>Environment Variable Name</b>	<b>Value</b>	<b>Comment</b>
FLOFIXEDFLOWSOVERWRITE	<b>Defined</b>	Enforces hierarchical precedence when there are overlapping fixed flows, see “ <a href="#">Context Rules for Overlapping Fixed Flows</a> ” on page 225.
FLO_HELP_PATH	<b>File path and name</b>	Defines the base directory location for the on-line help.
FLOHOME	<b>Directory path</b>	Specifies where the solution directory is kept (flouserdir) for FloVolunteer. Default defined in batch startup scripts.
FLO_IDF_CONFIG_FILE	<b>File path and name</b>	Defines the idf config file, for importing idf board data, complete with full path.
FLO_LANGUAGE	[jp   zh]	Alternative language displayed in the application windows: <ul style="list-style-type: none"> <li>• jp – Japanese</li> <li>• zh – Simplified Chinese</li> </ul>
FLOLOOK	[WINDOWS   WINDOWSXP   <ANYTHING ELSE>]	Specify look and feel of GUI. Default is WINDOWS on Windows, MOTIF on non-Windows (equivalent to <ANYTHING ELSE>).
FLOMCADKEEPLOGFILES	<b>Defined</b>	Instructs MCAD to keep any log files written when reading or writing CAD data files.
FLOMCADOPTIONDONOTUSETETS	<b>Defined</b>	Disable use of Tets during dissection
FLOMCAD_SET_COLOR_ON_TRANSFER	<b>Defined</b>	Specifies that the MCAD part color is transferred into the project
FLO_NUMBER_OF_THREADS	<b>Number</b>	Specifies the number of threads to run in the parallel solver.

**Table B-1. Environment Variables (cont.)**

Environment Variable Name	Value	Comment
FLO_OVERCOMPLEX_CELL_LIMIT	<i>Number</i>	Specifies a limit to the dissection of geometry by MCAD, thereby limiting computing resource requirements. Default 5000. If models are taking too long to dissect/decompose, reduce this number to obtain yield faster, but less accurate, results.
FLOPCM_MELT_BAND_MINIMUM	<i>Number</i>	When defining the Melt Temperature Band in Material attribute property sheets (Phase Change active), a minimum value of 0.1 degC is used regardless of what is entered in the property sheet. Use this environment variable to specify a different value (in units of degC).
FLO_SERVICE_PATH	<i>Directory path</i>	Override the location of required services.
FLO_SOLUTION_PRIORITY	[1   2   3   4]	Sets the runtime priority of the solver with the following specification: 1 - Idle/low, 2 - Normal, 3 - High, 4 - Realtime.
FLOPRODUCTCONFIG	<i>File path and name</i>	Specifies where the configuration file for the product is kept.
FLOPRODUCTDIR	<i>Directory path</i>	Specifies where the Mentor Graphics Mechanical Analysis executable files are kept.
FLOQUEUELOCATION	<i>Directory path</i>	Specifies the directory where the FloVolunteer queue is kept.
FLOQUEUENETWORKLAG	<i>Number</i>	Number of seconds to wait after saving a queue file over a network to make sure the file has been written and closed correctly.

**Table B-1. Environment Variables (cont.)**

<b>Environment Variable Name</b>	<b>Value</b>	<b>Comment</b>
FLOTINYMODEL	<b>Defined</b>	Enables sub-micron grid scales to be created. Subsequent use of the double-precision solver is strongly recommended.
FLOUSERDIR	<b>Directory path</b>	Specifies the current solution data folder.
FLOUSERDIR_SAVE	<b>Directory path</b>	Specify the user directory for a Simcenter Flotherm Viewer Installation.
FLO_USE_V6_LINEAR_SOLVER	<b>Defined</b>	Force use of the v6 linear solver.
FLOVISHINTS	[QUICKDRAW   DISABLESTENCIL   DISABLESMOOTHSHADE   USE24BITRGB   USE12BITRGB   USE12BITMAP   USE8BITMAP   REPORTCONNECTION]	Various MCAD display options to do with render speed, shading, color depth, and the last option reports what video options are available.
FLOVOLUNTEERMAXJOBS	<b>Number</b>	Specifies maximum number of simultaneous jobs the Command Center should start. Ignored when solving on Linux® <sup>1</sup> platforms, that is, no simultaneous processing of scenarios is possible.
FLOVOLUNTEERPARALLELTHREADS	<b>Number</b>	Specifies the maximum number of threads (= number of processors) for the parallel solver when running the FloVolunteer.
HOME	<b>Directory path</b>	Specifies users home directory.
HOMEDRIVE	<b>Drive id</b>	Specifies drive of user's home directory.

**Table B-1. Environment Variables (cont.)**

Environment Variable Name	Value	Comment
HOMEPATH	<i>Directory path</i>	Specifies path to user's home directory on drive HOMEDRIVE.
IDFSMALLEDGETOL	<i>Floating point number</i>	Tolerance size used when importing eda data. Any edge sizes below this tolerance are removed.
IV_SEPARATOR_MAX_CACHES	0	If you encounter memory problems when creating surface plots in Analyze mode, set the variable value to 0. This has the downside of generally slowing the plotting.
MAIL_HOST	<i>Server name</i>	Specifies mail server host for sending automatic email solve reports.
MAX_NUMBER_OF_OVERLAPS	<i>Number</i> . Default 2500.	The maximum number of overlapped localized grid spaces that will be resolved before a Warning message (W/1014) is output. This is a limit imposed to prevent excessive time being taken resolving overlapped localized grids.
MCADEXPORTVRML	<i>Defined</i>	Enables the Export VRML option in MCAD.
MGLS_LICENSE_FILE	<i>File path and name</i>	Specifies location of license file.
MENTORMA_PLUGINS	<i>Directory path</i>	Specifies the location of the file read/write plugins.
NUMBER_OF_PROCESSORS	<i>Number</i>	Specify the number of processors installed on the machine. Queried when running sequential jobs from Command Center.
OUTPUTEFGLOG	1	Activates an extra ASCII log file ( <i>viewf.log</i> ) of data from the EFG.  Warning: can create large files.

**Table B-1. Environment Variables (cont.)**

<b>Environment Variable Name</b>	<b>Value</b>	<b>Comment</b>
PATH	<i>Directory path; Directory path;...</i>	Path is retained and temporarily appended to when starting the software. Reset to retained value on exit.
SO_MATERIAL_DIR	<i>Directory path</i>	Specifies the directory containing information for visualization of materials.
SO_TEXTURE_DIR	<i>Directory path</i>	Specifies the directory containing information for visualization of textures.
TEMP	<i>System defined</i>	Temporary directory for all sorts of temporary operations.
TMP	<i>System defined</i>	Temporary directory for all sorts of temporary operations.
TMPDIR	<i>System defined</i>	Temporary directory for all sorts of temporary operations.
USERPROFILE	<i>Directory path</i>	Specifies the documents directory for the current Windows user.
XMLREADER_SCHEMA_FILE	<i>Directory path</i>	Specifies the directory containing the *.xsd files that define the valid syntax for XML files to be imported. This environment variable is set automatically when Simcenter Flotherm is started.

1. Linux® is a registered trademark of Linus Torvalds in the U.S. and other countries.



# Glossary

---

## Absolute Coordinates

A location relative to the model origin, see “[Coordinate Systems](#)” on page 215.

## Ambient Attributes

The conditions that prevail outside the solution domain. Attached to the solution domain edge, cutouts, or flow devices (fixed flow, recirculation devices), ambients apply values for external pressure, temperature, turbulence, heat transfer coefficient, and external flow velocities.

## Analyze

The mode used to view results. In Analyze mode, the GDA can display plots, the Results Tree is available, and Tables of results are available. See also “[Create](#)” on page 625.

## Approach Velocity

The velocity of the fluid as it approaches a device.

## Area Factor

The area enhancement factor used to model special surface conditions.

For example, the friction and heat transfer effects of rippled surfaces with ribs in the direction of the flow can be specified by setting a factor to represent the equivalent flat surface area which would cause the same effect.

## Aspect Ratio

The ratio of two sides of a grid cell, see “[Grid Suitability](#)” on page 305.

## Assembly

A collection of items of geometry, that is, a group of primitives, SmartParts, and/or other sub-assemblies.

## Attachment

The process of applying a particular attribute to a geometry object. The **Attachments** tab in an object’s property sheet shows which attributes are currently attached to the object.

## Attribute

The quality ascribed to a particular object. Attributes modeled include: material, surface finish, resistance, ambient, thermal, source, transient function, radiation, and surface exchange.

## Automatic Algebraic Turbulence Model

The usual option for program calculated viscosity where the solution domain is highly cluttered with objects. See “[Automatic Algebraic Model](#)” on page 172.

---

## Auxiliary Variables

Additional derived variables which may be calculated after project solution. Provided that the appropriate solution data is available, you can calculate Flow Angle and Total Pressure.

See “[Available Auxiliary Variables](#)” on page 540.

## Base Grid

The basic system grid is known as the base grid. Localized grid spaces created over this basic grid become children of the grid. If localized grid spaces are created over existing localized grid spaces, then in turn, the localized grid spaces themselves have children and the basic grid becomes a grandparent. See “[Grid Localization](#)” on page 628.

## Backup File

The safety copy of the project data file. The project data file is called group and is stored in the project solution directory. When a project is saved, the backup file *group.bak* is also created in the project solution directory. If you need to restore your project for some reason, then copy *group.bak* over the file group.

## Block with Holes

A SmartPart created by punching holes through a primitive cuboid block.

## Bonded Heat Sink

A heat sink SmartPart with fins bonded into grooves cut into the base.

## Buoyancy Force

The force resulting from temperature differences within an enclosure, see “[Ambient Temperature When Modeling Buoyancy Force](#)” on page 182.

## CC

Abbreviation used for the Command Center application window.

## Cells

The domain of integration is sub-divided into a regular array of non-overlapping, contiguous, cuboidal volumes known individually as cells and collectively as the grid. This sub-division is done for the purpose of discretizing the flow-governing differential equations. Within each cell, the value of a field variable is taken as constant. The solution algorithm provides the values of the field variables in each cell of the domain. See also [Solver Grid](#).

## Characteristic Length

The typical length scale for the flow under investigation.

## Characteristic Velocity

The typical velocity scale for the flow under investigation.

## Collapsed Assembly

An assembly that does not have its constituent parts on view. A collapsed assembly is represented in two ways:

- a ‘+’ icon in the Project Manager data tree

- 
- a green boundary outline in the drawing board.

To expand the data tree to display the assembly components, click the ‘+’ icon that precedes the assembly name in the Project Manager data tree. All the immediate children of the assembly then appear in the data tree. This expansion is conveyed to the drawing board, where the green boundary outline is replaced by a view of the assembly components.

## **Collapsed Object**

The representation of a 3D solid as a 2D plate with an assigned thickness in the collapsing direction.

All collapsed objects can be selected and highlighted using the Find dialog box.

## **Constraint**

See [Grid Constraint](#).

## **Continuation**

A calculation which carries on from where the previous one left off with the initial fields of the new calculation taken as the final fields of the previous calculation.

## **Control Cells**

See [Cells](#).

## **Convergence**

The solution algorithm is iterative. A calculation is converging when the residuals show a trend of diminishing with an increasing outer-iteration (or solution pass) number. The calculation stops when the calculation is converged which occurs when the residual of each variable divided by the its termination residual, set in the Variable Solution Control section of the **Solver Control** tab, is less than unity.

## **Coupling Metric**

A measure of the amount the results change if a selected region is replaced by a Zoom-in project originally created from the region. See “[Determining the Coupling Metric](#)” on page 103.

## **Create**

The mode used to create a model. In Create mode, geometry can be added to the data tree and manipulated in the GDA. See also “[Analyze](#)” on page 623.

## **Cutout**

A user-defined area cutout from the solution domain and, therefore, excluded from the program calculations. It is mainly used for irregular shaped enclosures.

## **Data Tree**

The representation of the project data as a hierarchical tree structure, see “[Data Tree](#)” on page 47.

## **Datum Pressure**

The pressure against which all other pressures are relative, see “[Reference Temperature and Pressure](#)” on page 180.

---

## Decompose

Depending on the context:

- The conversion of a parametrically defined Simcenter Flotherm SmartPart into primitive shapes.
- The conversion of an MCAD Part or Body into Simcenter Flotherm objects.

## Defaults

The initial start-up values. The defaults can be recovered by choosing **Project > New**.

---

### Note

---



Recovering the defaults destroys any unsaved project changes.

---

## Dependent Variables

The fields which satisfy the flow-governing partial differential equations. The three velocity components, the temperature, and the pressure fields are all dependent variables in this sense.

## Derived Variables

The variables calculated from the dependent variables: mass and heat flow are examples.

## Device Velocity

The velocity of the fluid as it flows through a device.

## Dimensionality

Simcenter Flotherm is equipped to perform simulations of flow and heat transfer in two and three dimensions, that is, 2D and 3D analyses can be performed. The 2D option solves in the X-Y plane. The 3D option solves in X-Y-Z space. The dimensionality option is selected in the **Model Setup** tab.

## Diss. Turb.

The field of dissipation rate.

## Distribution Options

The selection options for the type and amount of information to be tabulated in the fields results table.

## Domain of Integration

The region of space over which Simcenter Flotherm provides an integration of the differential equations governing the fluid flow and heat transfer within the domain. The domain is a cuboid, the lengths of whose sides are defined in the System property sheet.

## Double-Precision Solver

The Mentor Graphics thermal solver is an iterative scheme that uses the imbalance in cells to predict new values. In certain cases, the imbalance will be of the order of the precision at which floating numbers are stored. In these cases convergence cannot be achieved because rounding errors are of the same order of magnitude as the imbalance. Switching the solver to double precision can solve this problem.

---

See “[Double-Precision Solver](#)” on page 418.

## **Embedded Conduction Solver**

Linear embedded conduction solver, enabling full conjugate heat transfer in 2D solid objects.

See the **Activate Plate Conduction** description in the “[Solver Control Tab](#)” on page 200.

## **End Field Data**

The field solution results.

## **Errors**

See [Residuals](#).

## **Estimated Free Convection Velocity (EFCV)**

A solver control setting, used when there is no forced flow (for example fans or fixed-flow SmartParts). A default value is set for typical cases but in some models this value may need to be modified, see “[Solver Control Tab](#)” on page 200.

## **Exchange Factor Generator (EFG)**

The radiation Exchange Factor Generator, which calculates the fraction of uniform diffuse radiant energy leaving one surface that is absorbed by a second surface. By default, the exchange factor values are not available, however, you can request Simcenter Flotherm to write out a log (this can be large) of all the radiation information, including the exchange factors, to an ASCII text file.

To request the radiation information log, you must first set the environment variable OUTPUTEFGLOG = 1 before starting the exchange factor calculation.

See [Generating a View Factor Log](#) in the *Simcenter Flotherm Background Theory Reference Guide*.

## **Expansivity**

The volumetric expansivity of the fluid. For an ideal gas, the expansivity is the reciprocal of absolute temperature.

## **External Ambient Temperature**

Constant temperature prevailing outside the enclosure.

## **External Radiant Temperature**

The temperature of a remote radiating source.

## **False Time Step**

Simcenter Flotherm provides an under-relaxation option for the solved variables which consists of setting a false time step size for selected variables. These are set in the Variable Solution Control section of the **Solver Control** tab, with provision for automatic and manual options. This is the preferred method for the control of the rate of change of field values from one outer-iteration to the next. See [Convergence](#).

## **Field**

A quantity whose value in every cell of the integration domain is stored.

---

## **Free Area Ratio**

A measure of any obstruction to the flow through a device, for example, a grille across the flow path. It sets the ratio of the actual free area to the total area across the device.

## **Gauge Pressure**

Pressure relative to the datum pressure.

## **GDA**

Graphics Display Area. This functions as both a drawing board when creating a model, and as a results viewer.

## **Geometry Grid**

The grid lines aligned with the borders of the geometry objects.

## **Geometry Objects**

The construction data defining the shape of the model. There are three types of geometry objects:

- Primitives — cuboids, prisms, resistances, and sources (the lowest level building blocks).
- SmartParts — generic models of complicated geometries defined parametrically (Heat Sink, PCB, Sloping Block, Hole, Enclosure, Cylinder, Fan, Re-circulation Device, Fixed Flow, Compact Component, Perforated Plate).
- Assemblies — groups of primitives, SmartParts, and/or other assemblies.

## **Global Units**

The default units Simcenter Flotherm uses to apply and interpret quantities and coordinates. These settings can be overridden for individual dialog boxes using the popup unit selection list buttons. For a dialog box containing only location information, the units may be set locally for all length measurements, or in the case of a multi-unit type dialog box, individual data field units may be changed.

## **Grid**

The *Solver Grid*. There are two other grids: the *Snap Grid* and the *Time Grid*.

## **Grid Constraint**

The minimum grid requirements for either the solution domain or individual geometry objects within the solution domain.

## **Grid Inflation**

The action of adding grid layers, that is grid lines surrounding an object, extending a set distance or percentage distance away from the object boundary.

## **Grid Localization**

The action of constraining object grid lines to within the object boundary. Grid lines passing through keypoints are not localized. A Localized Grid is a grid space containing grid lines not needed in the rest of the solution domain. The grid constraints attached to an object are either

---

confined within the boundaries of an object or allowed to extend a set distance from an object. All objects with localized grid can be selected and highlighted using the Find dialog box.

### Grid Patch

A grid refinement added between regions of geometry that would otherwise not have grid lines.

### Grid Smoothing

The removal of large variations in neighboring grid cells, and of large aspect ratios.

### Grid Space

A cuboidal volume of space which contains grid lines not needed in the surrounding region and only lets through lines that are needed.

### History Data

The field values in the cells marked by a monitor point for each solution pass.

### Ignored Geometry

Geometry is marked as ignored by activating **Ignore Geometry** in the **Location** tab of the Assembly property sheet. Ignored geometry has no thermal properties and is ignored by the Simcenter Flotherm grid and solution. It is also not visible in the GDA.

Ignored geometry can still be seen in the Project Manager tree if **Show Ignored Geometry** is checked on in the User Preferences dialog box.

### Initial Conditions

The solution of the equations for the field variables starts from a set of initial values of the fields. These are known as the initial conditions and they are set in the Variable Solution section of the **Solver Control** tab.

In steady-state operation, the initial conditions constitute guesses for the solution (and the better the guess the less work the solver will have to do).

In transient operation the initial conditions form a part of the problem specification.

### Inner Iteration

The solution algorithm is an iterative one. At each outer iteration the finite-difference equations (one for each cell) of each variable are solved. These equations are expressed in the form of a system of linear equations which are solved by iteration. These iterations are known as inner iterations to distinguish them from the outer iterations or equation-set passes.

See [Background of Computational Fluid Dynamics \(CFD\)](#) in the *Simcenter Flotherm Background Theory Reference Guide*.

### <install\_dir>

Used in pathnames to denote the Simcenter Flotherm installation folder/directory. The default is *C:\Program Files\ MentorMA*.

For Linux systems, the default is */opt/MentorMA*.

---

## Inverted Tet

A cuboid with one of its corners removed, that is, a heptahedron with three rectangular faces, three right-angled triangular faces, and one triangular face that is angled in two coordinate directions.

## Iteration

See [Inner Iteration](#) and [Outer Iteration](#).

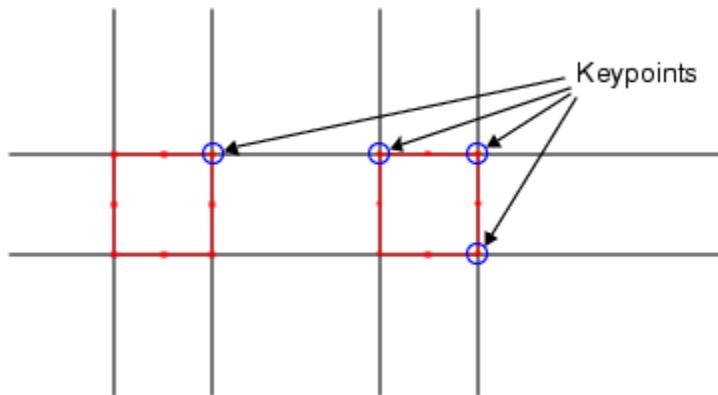
## KE Model

Calculates the turbulent viscosity as a function of two field variables, namely the kinetic energy of the turbulence ( $k$ ) and its rate of dissipation ( $\varepsilon$ ). These two field variables are determined by the solution of two additional differential equations which these variables satisfy.

The k-epsilon model is rarely needed for flows in electronic enclosures, being used for cases where turbulence is to be modeled over large empty volumes within an enclosure. See “[LEVEL K-Epsilon Model](#)” on page 172.

## Keypoint

An object boundary corner coincident with the grid lines. Initially, the program sets a default solution grid to align with the edges of the geometry you create, as shown for two cuboids in the figure below.



When geometry objects are deactivated, their keypoints are retained and the grid continues to match them: an object has to be deleted to remove its grid.

Transient keypoints locate the solution [Time Steps](#). By default, there is one time step, with a keypoint at the start and at the end of the solution period.

## Keypoint Tolerance

For spatial solution grids, the tolerance (distance) above which keypoints are treated as distinct.

For transient analyses, the keypoint tolerance determines the smallest [Time Step](#) that can be created. If you try to define time steps smaller than the keypoint tolerance they will not be created. This provides a way of suppressing very small unnecessary time steps that might otherwise be created.

---

## Laminar Viscosity

The molecular viscosity (or friction) between the fluid elements themselves and between the fluid and solid surfaces with which the fluid is in contact.

## Lib File

A lib file contains Simcenter Flotherm item or assembly information. It is used to build up a library of design specifications for later incorporation into Simcenter Flotherm models.

Lib files were normally created in V1.4 using Simcenter Flotherm GATE, but they can also be generated by external software systems to provide an interface with Simcenter Flotherm.

See “[Importing V1.4 Library Files](#)” on page 244, and the *FLOGATE V1.4 Instruction Manual* (hardcopy) for further details.

## Libraries

The Simcenter Flotherm libraries are storage and retrieval areas for commonly used items of geometry, properties, and attributes for both local or central use.

User libraries can be created to store customized designs, see “[Libraries](#)” on page 285.

## Linearized Equation

The flow-governing differential equations solved are non-linear. At each outer iteration, the discretized counterparts of these differential equations are expressed in linearized form (the non-linearities being temporarily hidden in the coefficients until they are re-calculated on the next iteration). The set of linearized equations are solved by one of methods selected by the user in the **Solver Control** tab.

## Local Coordinates

The definition of a location relative to the origin of the parent assembly.

## Localization

See [Grid Localization](#).

## LVEL

Length-velocity turbulence model, developed by Dereje Agonafer, Liao Gan-Li, and Brian Spalding. This turbulence model was developed for applications that involve many solids of differing shapes and sizes where only a small number of fluid grid cells between solid surfaces can be realized. The LVEL model is well suited to models that contain complex cluttered geometry where a sufficiently accurate grid (required for the more advanced turbulence models) goes beyond engineering constraints.

## LVEL K-Epsilon

The LVEL K-Epsilon model calculates the turbulent viscosity for the fluid cells not immediately adjacent to solid surfaces as a function of two field variables, namely the kinetic energy of the turbulence ( $k$ ) and its rate of dissipation ( $\epsilon$ ). These two field variables are determined by the solution of two additional differential equations which these variables satisfy.

This enables, more properly, for the turbulent variations in effective viscosity caused by velocity shears from free jets, and so on, in open spaces. See “[LVEL K-Epsilon Model](#)” on page 172.

---

## Mass Flux

Mass flow rate per unit area.

## MC

Abbreviation used for the MCAD Bridge application window.

## Monitor Cells

Solver grid cells in which the values of the variables are recorded at each outer-iteration or time-step.

The monitor point values can be plotted as a Profiles plot to display the convergence behavior of the field values in the monitor cells, that is, it provides a means of monitoring the course of convergence. A permanent record of this information can be exported to an ASCII profile definition file.

See “[Solution Monitoring and Profile Plots](#)” on page 377.

## Multi-fluids

Different fluid regions within the solution domain, created by defining Region SmartParts and attaching a different fluid property to each. Two distinct fluids must be separated by a solid.

## Multi Grid Solver

Uses multi-grid acceleration to solve the linear equations for temperature. For problems with conjugate heat transfer it can improve convergence and significantly reduce overall computation time. Use the **Solver Control** tab to activate the multi-grid solver.

## Multi Grid Damping

Controls any divergent behavior that may appear in some exceptional cases when the Multi Grid solver is used. Use the **Solver Control** tab to activate the damping.

Note, however, that the damping may cause some slow down in convergence in normal situations.

## Normalization

Division by a reference value.

## Open Boundaries

A free boundary of constant pressure through which air can flow. See “[System Property Sheet Boundaries Tab](#)” on page 186.

## Orthotropic

Values varying according to coordinate direction.

## Outer Iteration

The solution algorithm is an iterative one. In an outer iteration the equations for each variable in turn are solved, namely first temperature followed by the velocities u, v, and w, and the pressure. Outer iterations are also known as equation-set passes or passes. In general many outer-iterations are needed to achieve convergence. See also [Inner Iteration](#).

---

## Pack

The compression of the project and solution data to allow easier data transfer.

## Pass and Passes

The phrase ‘equation-set pass’ is synonymous with *Outer Iteration*.

## Patch

See *Grid Patch*.

## Physical Design Modeling Language (PDML)

A file format used to transfer data to and from the external file system for either the complete project or any selected assembly in the Project Manager.

Physical Design Modeling Language (PDML) files are used to provide library objects.

## \*.prb

The generic filename and its extension for Version 1.4 problem specification files.

## Primitive

The most basic modeling representations in Simcenter Flotherm. There are six types of primitive: cuboid, prism, tet, inverted tet, source region, and resistance region.

When converting ACIS models to Simcenter Flotherm objects, only cuboid and prism primitives are available.

## Project

A single Simcenter Flotherm case-study. All data relating to the study is stored in its own project directory structure located below the solution directory to which the project was saved.

## Realized

Converted into Simcenter Flotherm format.

## Reference Temperature

The global ambient temperature set in the Global System Settings section of the **Model Setup** tab.

## Residuals

The sum over all cells of the absolute value of the error remaining in the finite-difference equation for each cell. For continuity the error is the mass flow imbalance in a cell ( $\text{kg s}^{-1}$ ). For temperature the error is the heat imbalance in a cell (W). For momentum (that is, velocity) the error is the force imbalance in a cell (N). The quantity plotted in the display area when the equations are solved is the residual divided by the termination residual.

## Response Surface

A response surface shows how a Command Center output variable responds to a change in the input variables.

If a system has,  $< n >$  input variables, a  $< n > + 1$  dimensional response surface may be generated for any of its output variables.

---

The Command Center estimates the true response surface from a small finite number of experiments (for which the input and output are known). The accuracy of the estimated response surface is dependent on, the number of experiments, number of input parameters, and the complexity of the real-world response surface.

## Revised Algebraic Turbulence Model

For accurate modeling at air-solid surfaces, see “[Revised Algebraic Model](#)” on page 171.

## Root Assembly

The top-level assembly containing all the project geometry.

If the Root Assembly is not visible in the data tree, press F4 (View Reset).

## Roughness Height

Represents the surface roughness which enhances friction and heat transfer.

## Save Time

Only applicable to transient analyses. Save Times are time steps at which the 3D results fields will be saved to disk. After the solution is completed these results can be loaded and viewed in Analyze mode. Only the 3D results fields are saved, derived properties such as flow rates and heat fluxes are not available.

## S-F

Solid-to-Fluid

## SmartPart

The complex geometries modeling electronic component types such as heat sinks, PCBs, fans, perforated plates, re-circulation devices, cylinders, and so on, which can be quickly generated parametrically.

## Smoothing

See [Grid Smoothing](#).

## Snap Grid

A drawing board alignment tool. As you draw or drag an object closer to the lines of the snap grid, the object moves automatically to align with the grid.

## Solution Directory

Storage location for the set-up and solution results of Simcenter Flotherm projects. The solution directory may be changed when saving the project using the Save Project dialog box.

Also, see “[Project Overview](#)” on page 36 for a description of the project directory structure.

## Solution Domain

Defines the extent of the geometry model included in the Simcenter Flotherm calculations. Set in the [System Property Sheet](#).

---

## Solver Grid

A regular array of cuboidal cells in to which the domain of integration is sub-divided for the purpose of the numerical integration of the flow-governing differential equations. See also [Cells](#) and “[Spatial Solution Grid](#)” on page 303.

## S-S

Solid-to-Solid

## Stagger

The displacement from the grid cell centers of the locations of the values of the velocity components. It is a recurrent theme in CFD methodologies employing the finite-volume representation. See [View Stagger](#).

## Start Field Data

The [Initial Conditions](#) that the program uses to start the solution process.

## Stratification

Option for including the buoyancy generation terms in the calculation of turbulence. Use when temperature differences in the fluid are large.

At flow boundaries, turbulent kinetic energy and turbulent dissipation are convected in by the flow. The level of these values are set by the user in the Ambient Attribute property sheet, or in the appropriate flow dialog box. If left at the default of zero, an estimate is made based on the flow rate calculated from the average inflow speed. See “[LVEL K-Epsilon Model](#)” on page 172.

## Subdomain

A subdomain is a region of the solution domain over which initial values can be set, see “[Setting Initial Field Values Over Subdomains](#)” on page 146.

## Surface Exchange

Fixes heat transfer coefficient for cuboids, blocks with holes, heat sinks, and enclosures. You define a surface exchange using a Surface Exchange Attribute.

## Surface Finish

Geometry surface attributes. You define a surface finish type using a Surface Attribute.

## Symmetry Boundary

A frictionless, impermeable, and adiabatic planar surface, see “[Examples of Open and Symmetry Boundaries](#)” on page 141.

## TB

Abbreviation used for the Tables application window.

## Termination Residual

The calculation stops when the termination residuals of all the equations solved fall below their respective termination residuals. See [Convergence](#).

---

## Tet

A tetrahedral object that has three right-angled triangular faces that are aligned to the three coordinate directions, and one triangular face that is angled in two coordinate directions.

## Thick Wall

A wall modeled using a solid cuboid when you need heat conduction along the wall as well as through it.

For thick walls in enclosures, use the Enclosure SmartPart.

## Thin Wall

A wall modeled using a collapsed cuboid to save grid cells and therefore variable calculations. A thin wall can model heat through it, but not along it.

For thin walls in enclosures, use the Enclosure SmartPart.

## Time Grid

Only applicable to transient analyses. The Time Grid specifies the number, size, and distribution of the *Time Step*s used in transient calculations.

## Time Patch

Only applicable to transient analyses. Users have the option of adding Time Patches to the default *Time Grid*. Each patch is identified by a name and can be subdivided into a number of *Time Steps*.

## Time Step

Only applicable to transient analyses. Solutions are calculated at the end of each time step. In effect this defines the resolution of a transient analysis. One or more time steps make up a *Time Patch* and the time steps within a patch may be uniform (all steps are of equal duration), or may vary over the length of the patch, typically to give greater detail at the beginning and/or the end of a patch.

## Tolerance

The range limit within which gridlines are assumed to be coincident. If two gridlines are closer than the specified tolerance, in a particular direction, then they are treated as one. See the Minimum Size definition in the “[System Grid Property Sheet](#)” on page 328.

## Top

Chooses the data tree level from which to view the model. You can top any geometry object by selecting it, and then pressing F3, see “[Topping Assemblies or Parts](#)” on page 92. The display area will then display only the topped object, and in the case of an assembly, its immediate children.

The topped object becomes the head (or top) of the data tree.

## Transitional Flow

The transition between laminar and turbulent flow.

## Translation

The conversion of incoming geometric data to ACIS entities.

---

## Uniform Grid

The solution data can be interpolated to the uniform grid to aid visualization particularly in the cases of fine grid, for example, for displaying vectors clearly.

## Unit Class

The type of measurement used to represent a quantity.

In the profiles plots, a unit class is plotted along the variables axis, and variables measured using the same unit classes can be displayed on the same plot. For example, residuals for all variables can be shown on the same plot. Similarly, profiles for the X, Y, and Z-velocities can be shown on the same velocity plot.

## Variables

Variables are field quantities, that is, they have a distinct value for each cell.

- Dependent variables are variables that satisfy the flow-governing differential equations (for example, the temperature).
- Additional variables are variables for the storage of the fluid properties when they vary from cell to cell (for example, the density when the ideal gas law is used).
- Auxiliary variables are variables requested for calculation after solving.

## View Factor

A view factor is the fraction of uniform diffuse radiant energy leaving one surface that is incident upon a second surface. By default, the view factor values are not available for display. However, you can request Simcenter Flotherm to write out a log of all the radiation information (which includes the view factors) to an ASCII text file, which you can view using any normal text file. As the log file can be very large, make sure there is sufficient disk space.

To request the radiation information log, you must first set the environment variable OUTPUTEFGLOG = 1 before starting the exchange factor calculation by choosing **Solve > Exchange Factors**.

For information on the radiation log, see [Generating a View Factor Log](#) in the *Simcenter Flotherm Background Theory Reference Guide*.

## View Stagger

Displays the true calculated velocity components at the solution grid cell faces, not the coordinates of the cell center.

## Visualize SmartParts

Show all the composite primitive parts of SmartParts.

## Wall

[Thick Wall](#) or [Thin Wall](#).

## Work Plane

A 2D reference plane used when drawing and moving geometry objects.

---

## **Zoom-In Project**

The creation of a model from the contents of a volume region. It enables the geometrical and modeling complexity to be altered and re-solved without incurring the computational overhead of solving the whole, original, computational solution domain.

See “[Creating a Zoom-In Project](#)” on page 98.

# Index

---

## — A —

Absolute Coordinates, 215  
Acceleration Due to Gravity, 176  
Alignment  
    Align Selected dialog box, 279

Ambient  
    default temperature, 195

Angled  
    gravity, 193

Animations  
    animation controls, 473

Annotations  
    Profile Plots, 380

Application Windows  
    automatic opening, 128

AS CII PDML, 65

Aspect Ratio Levels (Solution Grid), 306, 329,  
    330

Automatic Algebraic Turbulence Model, 194

Automatic False Time Step, 205

Automatic Termination Residual, 205

Axes  
    Profile Plots, 380

## — B —

Backup Files, 37

Batch Mode  
    solving transients, 371

Bn, 155

brd Files, 241  
    board description file, 232

Buoyancy, 173, 182, 203

## — C —

Characteristic Length and Velocity, 171

Cloudiness, 189

Compact Components  
    table, 498

Components  
    Component Fluxes Table, 500

dimensions, 46  
IDF preference, 129  
Compute Capture Index, 192  
Conduction  
    conducting PCBs  
        IDF preference, 129  
    conduction only solutions, 150  
    conductive transport, 151

Continue Solution, 361

Contours  
    contour plots, 433

Control Solution  
    monitoring, 632  
    start/stop, 361  
    termination residual, 635

Convection  
    convective transport, 151

Convergence  
    control, 200  
    rules for assessment, 408

Cooler  
    table, 503

Coordinate System  
    setting, 128

Create  
    geometry, 212

Create a New Library, 287

Create Category dialog box, 108

Cuboids  
    intersection with angled plate, 223

Cutout/Overall Results Summary Table, 505

## — D —

Datum  
    datum pressure, 195

DCM.log, 166

Default Object Size, 51

Default Templates, 120

Die  
    table, 506

- 
- Diffusive Transport
    - diffusion in very slow flows, 412
    - diffusive heat flow, 151
  - Directory Structure, 37
  - Display
    - Options
      - Drawing Board, 130
  - DissTurb, 144
  - Domain (see Solution Domain)
    - Domain Boundary Faces, 186
  - Drawing Board, 254
  - E —
  - EFCV, 203, 411
  - Embedded Conduction Solver
    - definition, 627
    - mesh message details, 594
  - emm Files, 232, 241
  - emp Files, 232, 241
  - Environment Shell
    - Start menu option, 610
  - Estimated Free Convection Velocity, 203
  - Example of Application
    - templates, 120
  - Exchange Factors Calculation Progress dialog box, 375
  - External
    - ambient temperature, 195
    - boundary conditions, 186
    - radiant temperature, 195
  - F —
  - False Time Steps
    - setting, 205
  - Fan
    - Table, 512
  - FAQs
    - all, 545
  - Find dialog box, 109
  - Find objects within topped geometry, 109
  - Fixed Flows
    - fixed flows table, 513
  - FloSCRIPT
    - log files, 41
  - FloVolunteer
    - maximum number of jobs, 619
  - maximum number of threads, 619
  - queue path, 618
  - solution directory path, 617
  - Flow
    - flow angle auxiliary variable, 548
      - Flow Angle dialog box, 544
    - flow direction displayed, 130
    - flow models
      - conduction only, 150
      - flow and heat transfer, 150
      - flow only, 150
  - FoXML
    - examples, 230
  - Fluid Properties
    - speed, 158
  - Forced Convection, 205
  - Freeze Flow, 202
  - G —
  - Geometry
    - ignored, 128
  - Global Settings
    - external ambient temperature, 195
    - global system settings
      - Global System Settings, 195
      - Global Units dialog box, 115
  - GradT, 153
  - Gravity
    - Gravity setting, 193
    - vector display in Drawing Board, 131
  - Grid
    - aspect ratios, 306, 329, 330
    - Grid Summary dialog, 330
    - interpolation, 304
    - smallest cell, 306
    - total grid size, 306, 329
  - group Project Data File, 69
  - H —
  - Heat Pipes
    - table, 514
  - Heat Sink
    - libraries, 293
  - Heat Sinks
    - table, 515
  - Heat Transfer Coefficient

- 
- changing using table cells, 480
  - Holes
    - in cuboids and enclosure walls, 546
  - Hyper-Threading, 359
  - I**
    - Identification of Projects, 123
    - IDF Files, 232
      - power and thermal resistance, 232
      - preferences, 129
    - Ignore Geometry
      - show ignored geometry, 128
    - Initial Settings
      - field variables, 206
    - Inner Iterations, 205, 629
    - Installation Directory, 37
    - Interface Materials Library, 293
    - Interpolation of Grid, 304
    - Iterations, 630
      - inner, 629
      - outer, 629, 632
  - K**
    - KETurb, 144
    - Keypoint
      - transients
        - keypoint tolerance, 352
    - Keypoints, 630
      - tolerance, 636
  - L**
    - Laminar Flow, 193, 194, 195, 197, 199
    - Legend
      - profile plots, 380
    - lib Files
      - defined, 631
    - lib Files, 631
      - defined
        - see also Libraries
    - Libraries
      - lib Files, 232, 241, 631
      - Library Data property sheet, 300
    - License Timeout dialog box, 116
    - Load
      - project
        - Load Project dialog box, 117
  - Local Coordinates, 215
  - Locate
    - geometry, 215
  - Log files
    - xml, 41
  - Logarithmic Axis in Profile Plot, 401
  - Low Reynolds Number Flows, 412
  - LVEL
    - K-Epsilon Turbulence Model, 194
  - M**
    - MagGradT, 153
    - Manipulator
      - particle sources, 465
    - Mass Continuity
      - termination residual, 405
    - Maximum and Minimum Values, 455
    - Maximum aspect ratio
      - button, 55
    - Mean Flows
      - table, 522
    - Measure Distances
      - Measure dialog box, 280
    - Message Window, 119
    - Monitor Cells, 632
    - Monitor Points
      - convergence for temperature, 204
      - display in Drawing Board, 130
      - Monitor Points v Iteration Profile Plots, 390
      - Monitor Points v Time Profile Plots, 392
      - solution control settings, 204
    - Monitoring Convergence, 356
    - Move dialog box, 281
  - N**
    - Name
      - editing, 46
      - unique, 97
    - Natural Convection, 203
    - Net Radiant Flux, 195
    - Network Assembly Block Correction, 201
    - New
      - library, 287
      - project
        - New Project dialog box, 120

---

Normal Gravity, 193  
Notes, 122  
Number of Ticks in Legend  
    Visual Edit or, 456

— O —

Object Coordinates, 215  
Open Boundary, 186  
Outer Iterations, 629  
Outer Iterations, 632  
Overall Solution Domain, 185  
Overall/Cutouts Table, 504

— P —

Parallel Solver, 128  
Pass and Passes, 633  
Patterns  
    Pattern Creation dialog box, 251  
PDML Files  
    batch mode support, 363  
Perforated Plates  
    table, 518  
Possible Heat Flow Paths, 155  
Potential  
    termination residual, 406  
Power Density, 197  
Power Map  
    table, 519  
Power Maps  
    geometry, 214  
prb Files, 633  
Preferences, 127  
Profile Plots  
    annotations, 380  
    configuration, 380  
    highlighting, 380  
    two y-axes plots, 380  
    window, 379  
    zooming, 380  
Project  
    backup, 69  
    load, 117  
    notes, 123  
    solution directory, 37, 123  
    title, 123

— R —

Rack  
    table, 520  
Radiant Temperature, 195  
Radiation  
    external radiant flux (net), 195  
    radiative exchange factors  
        batch mode, 606  
        remote source, 195  
        switching on/off, 191  
Recirculation Device  
    Table, 521  
Reference Density, 176  
Reference Temperature, 176  
Regions  
    summary table, 523  
Remote Radiating Source  
    external radiant temperature, 195  
Residual Error, 404, 633  
    Residuals Iteration Curves v Time Profile  
        Plots, 388  
    Residuals v Time Profile Plots, 389  
Response Surface  
    definition, 633  
Results  
    range, 455  
    Results v Distance Profile Plots, 394  
Root Assembly  
    definition, 634  
Rotate View dialog box, 282

— S —

Save  
    Save Project dialog box, 123  
Sc, 155  
Selection  
    buttons, 634  
Slow Flows, 412  
Smallest Grid Cell, 306  
Snap  
    snap grid, 130  
Solar Radiation  
    Solar Radiation dialog box, 189  
        solar vector display, 131  
solexe, 362

---

Solid Conductors  
  fluxes table, 525  
  summary table, 528  
  temperatures table, 527

Solution  
  directory, 37  
  initialize  
    Solution Set dialog box, 208, 209  
  process, 403

Solution Directory, 37

Solution Domain  
  overall boundary, 185

Solver  
  advanced control  
    freeze flow, 202  
  control options, 200  
  outer iterations, 200  
  parallel solver, 128  
  segregated conjugate residual, 200  
  Solver Progress dialog box, 374

solver  
  multi grid, 200

Solver Control Tab, 200

Solver Options property sheet, 200

Solver Progress dialog box, 374

Sources  
  display direction in Drawing Board, 130

Stagger, 128

Standard Supplied Templates, 59

Stationary Flow, 150

Steady Solution  
  initial conditions, 629

Stratification, 173

Subdomains  
  edit location, 187  
  initial values, 188

Sun (effects of), 189

Surface Temperature  
  store, 197

Symmetry Boundary, 186

System  
  coordinates, 128  
  grid  
    System Grid property sheet, 328

## — T —

T3Ster  
  importing xCTM geometry files, 244

TEC  
  table, 531

Temperature  
  residual error, 405  
  solution control, 412  
  termination residual, 405

Templates, 120

Termination Residual, 205, 404, 635

Thermal Bottlenecks, 155

Three Dimensional Analysis, 191

Time

  time steps  
    defined, 636

Title

  setting, 123

Tolerance (keypoints), 636

Total Pressure, 548

Transient Grid

  Transient Solution dialog box, 352

Transient Solution

  in Command Center, 371  
  in stages, 371  
  initial conditions, 629  
  validity checking, 371

Turbulence

  Turbulence setting, 193, 194, 195, 197, 199

TurbVis, 144

Two Dimensional Analysis, 191

Two Y-Axes Plots, 380

Types of Solution, 150, 191

## — U —

Undo

  Flushing the Undo Stack, 84

Units, 115

Upwind Principle, 152

User Preferences dialog box, 127

uVelocity, 157

## — V —

Variables

  auxiliary variables, 637

Bn, 155

---

Density, [144](#)  
dependent, [637](#)  
DissTurb, [144](#)  
initializing, [206](#)  
KETurb, [144](#)  
MagGradT, [153](#)  
Potential, [144](#)  
Pressure, [143](#)  
Sc, [155](#)  
Temperature, [144](#)  
TurbVis, [144](#)  
variable solution control settings, [205](#)  
XGradT, [153](#)  
XVelocity, [143](#)  
YGradT, [153](#)  
YVelocity, [143](#)  
ZGradT, [153](#)  
ZVelocity, [144](#)  
Vector Plots, [433](#)  
Velocities  
    residual error, [405](#)  
    solution control, [411](#)  
View Stagger  
    effect on Results Table, [453](#)  
Viewpoints  
    properties, [475](#)  
Volume Regions  
    display in Drawing Board, [130](#)  
Volumetric Expansivity, [176](#)  
vVelocity, [157](#)

## — W —

w Velocity, [157](#)  
whats this button, [426](#)

## — Z —

Zoom  
    Profile Plots, [380](#)  
Zoom-In Creation dialog box, [136](#)

## **Third-Party Information**

Details on open source and third-party software that may be included with this product are available in the `<your_software_installation_location>/legal` directory.

