Simcenter™ Flotherm™ MCAD Bridge User Guide

Software Version 2021.1

SIEMENS

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Chapter 1
Getting Started

This document describes the main features of the MCAD Bridge application window of Simcenter™ Flotherm™ software, and details how to interface with the MCAD packages.

Introduction

MCAD Bridge interfaces between Simcenter Flotherm and MCAD software packages.

- Use MCAD Bridge to import MCAD objects in a variety of MCAD formats and convert them to Simcenter Flotherm objects, simplifying where necessary, before transferring them to a Simcenter Flotherm project.

  Geometry imported from MCAD files is initially displayed in the graphics display area as an ACIS model. MCAD files that are not in **.sat* file format are converted into ACIS models before being shown in the display area.

- Use MCAD Bridge to export Simcenter Flotherm project geometry for use in MCAD software packages.

Starting MCAD Bridge

You can start MCAD Bridge from Simcenter Flotherm or run MCAD Bridge as a stand-alone application.

MCAD Bridge License

MCAD Bridge is supplied as a separate Simcenter Flotherm module.

Ensure that you have the MCAD license for inputting MCAD data.

Additional licenses are required for the Pro/E, SolidWorks, and CATIA readers.

Starting MCAD Bridge

You can start MCAD Bridge from any of the Simcenter Flotherm application windows.

Procedure

Click the **Launch MCAD Bridge** icon  in a Simcenter Flotherm application.

A splash screen showing the program build version is displayed momentarily. To re-open the splash screen, choose **Help > Version**.

Starting MCAD Bridge in Stand-Alone Mode

MCAD Bridge can be run as a stand-alone program on Microsoft® Windows systems.

Procedure

You have a choice:

If you want to...	Do the following:
Run MCAD Bridge from the Start menu.	Choose: **MentorMA > Simcenter Flotherm <*version*> MCAD Bridge**

If you want to...	Do the following:
Run MCAD Bridge from any command window.	1. Open a command window and change the directory to: *<install_dir>\flosuite_v<version>\flotherm\WinXP\bin* where *<install_dir>* is the directory in which you have installed Simcenter Flotherm. 2. Type the start-up string: flomcad.bat The **flomcad** command has optional parameters to start up with a loaded model.

Related Topics

flomcad

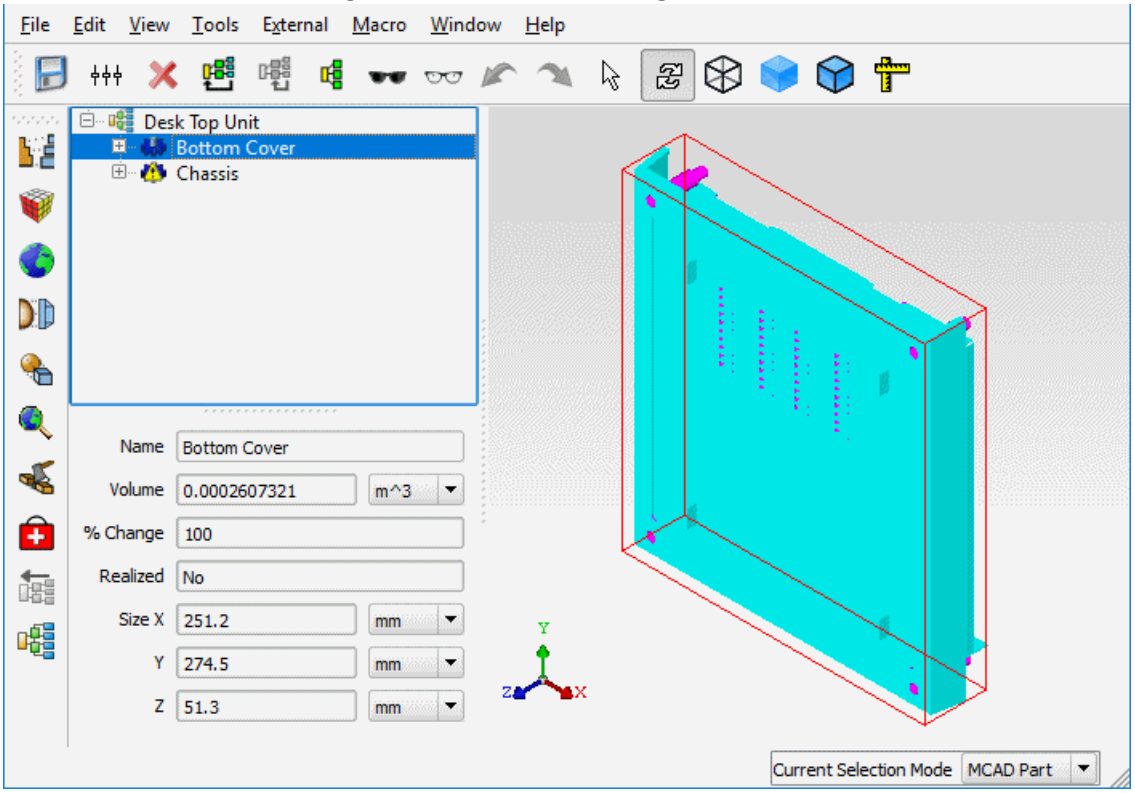
MCAD Bridge GUI Features and Concepts

MCAD Bridge operates as an independent application.

MCAD Bridge Window

MCAD Bridge opens in a separate window with its own menu and toolbars.

Figure 1-1. MCAD Bridge Window



The following are the main features of the window:

- The Graphics Display Area (GDA), for displaying and selecting geometry.
- The data tree, for selecting geometry and displaying the geometry hierarchy. Simcenter Flotherm geometry is identified using the same icons as in the Project Manager data tree. MCAD geometry is identified by the following icons:

  MCAD Part

MCAD Part with non-manifold edges, see “Geometry Rendering and Color Conventions” on page 15.

MCAD Body

Icons are dimmed when objects are hidden, and the root assembly is always shown as a Simcenter Flotherm assembly.

- Property sheets, below the data tree, display property values of items selected in the data tree, see “MCAD Bridge Property Sheet” on page 25.
- Context-Sensitive popup menus provide quick access to common commands.
- There are two toolbars, which can be moved to different sides of the window, or undocked from the window.

Geometry Rendering and Color Conventions

MCAD Bridge uses colors to distinguish between MCAD geometry and geometry converted to Simcenter Flotherm primitives.

Red	Bounding boxes around selected objects and assemblies.
Dark Blue	Simcenter Flotherm primitives.
Dark Green	Simcenter Flotherm SmartParts.
Cyan	MCAD planar surfaces.
Magenta	MCAD non-planar surfaces. This can be switched off using the Preferences dialog box.
Yellow	MCAD non-manifold edges, that is, face edges that could not be joined together, creating gaps in the surface and zero volume.

Geometry can be rendered as wireframe, solid, or solid with edges.

Figure 1-2 shows a partially converted design rendered as solid.

The assembly, named Chassis, has been converted and is displayed in dark blue. Note also that the Simcenter Flotherm assembly icon is shown against Chassis in the data tree.

The MCAD body, named Bottom Cover, has not been converted and is displayed in cyan. Note also that the MCAD part icon is shown against Bottom Cover in the data tree.

Figure 1-2. Partially Converted Design Rendered as Solid

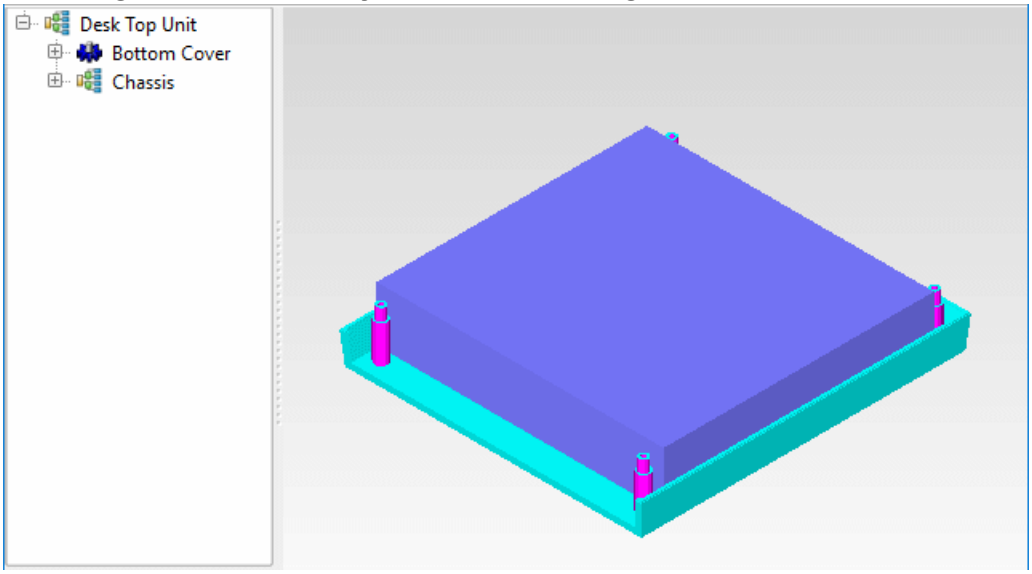


Figure 1-3 shows the design rendered as solid with edges.

Figure 1-3. Partially Converted Design Rendered as Solid With Edges

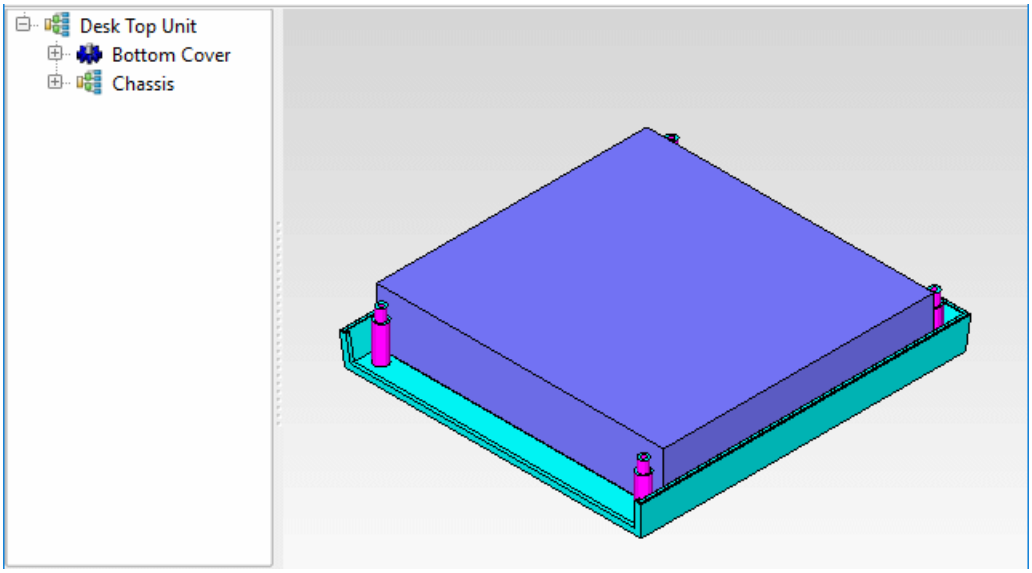
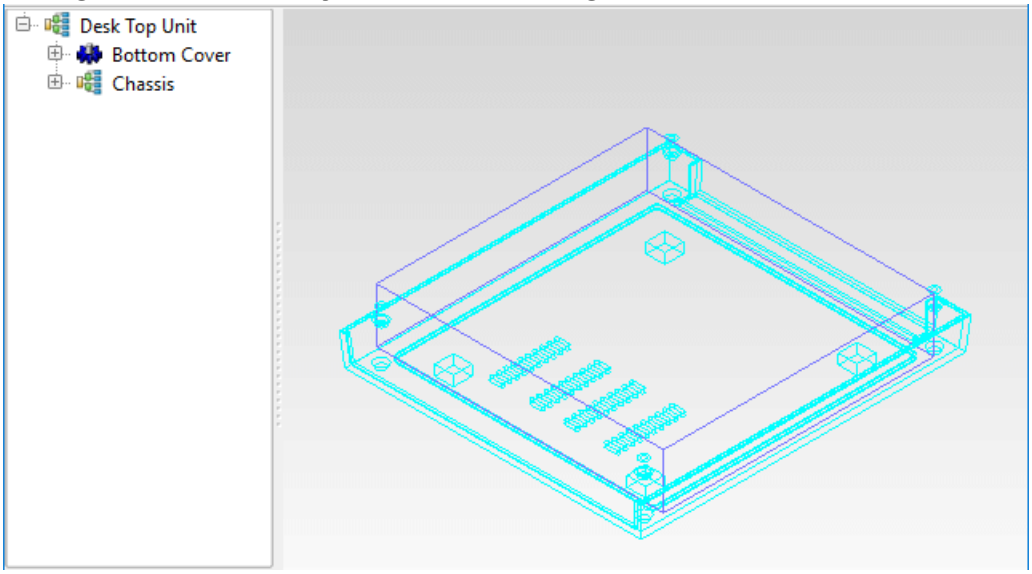


Figure 1-4 shows the design rendered as wireframe. The ability to see MCAD bodies and Simcenter Flotherm objects as wireframes enables you to gauge the complexity of the MCAD geometry and also determine the amount of MCAD geometry remaining to be simplified and dissected.

Figure 1-4. Partially Converted Design Rendered as Wireframe



Geometry Selection and View Manipulation

Depending on the current view mode, you can use the mouse to either select geometry and to change the view in the GDA.

The simplest way to swap between the two modes is to press the F9 key.

The mouse pointer changes, depending on what you are pointing to or the task you are performing.

- An arrow icon indicates Select mode. When you are in Select mode, there is a further control, current selection mode, that restricts what can be selected in the GDA.

  The current selection mode is set using the Current Selection Mode selector at the bottom-right of the window, or using a context-sensitive menu when right-clicking in the GDA. The following are the options:

  - MCAD Part
  - MCAD Body
  - Feature (a hole or hump)
  - Face (planar face)
  - Edge
  - Vertex

  It is important that you are the correct current selection mode when using MCAD Bridge.

- A double curved arrow icon indicates Manipulation mode. Use this mode to freely rotate, pan, or zoom the view.
  - Rotate

    Drag the mouse across the picture to rotate the object about its center. See also “Rotating the GDA View Incrementally” on page 22.
  - Pan

    Shift+drag, Ctrl+drag or Center-button+drag to pan the view in the direction of drag. The mouse icon changes to a pair of crossed double-headed arrows.
  - Zoom
    - Shift+Ctrl+drag-up to zoom out of the view.
    - Shift+Ctrl+drag-down to zoom in on the view. The mouse icon changes to a single vertical double-headed arrow.
    - Right-button click and drag to define a zoom-to bounding box.

      Tip

      Press R to refit the object within the GDA.

MCAD Bridge Shortcut Keys

Also known as hotkeys.

Table 1-1. MCAD Bridge Shortcut Keys

Function	Key
Top	F3
Reset tree	F4
Up to Parent	F5
Expand All	F6
Toggle between Select and Manipulate mode	F9
Show Preferences dialog box	F11
Show MCAD Bridge Monitor dialog box	Ctrl+M
Save As	Ctrl+S
Undo	Ctrl+Z
Redo	Ctrl+Y
Changing the view	

Table 1-1. MCAD Bridge Shortcut Keys (cont.)

Function	Key
View Selected	V
Hide selected items	F12
Reveal All	Shift+F12
Solid With Edges rendering	S
Wireframe rendering	W
Refit View(s)	R
View from plus X	X
View from minus X	Shift+X
View from plus Y	Y
View from minus Y	Shift+Y
View from plus Z	Z
View from minus Z	Shift+Z
Isometric view	Shift+I
Local Simplify	
Add bounding box	+
Subtract bounding box	-
Flatten face	F
Level face	L
Mark faces	M
Un-mark faces	U

MCAD Bridge Routine Procedures

Use the following procedures for simple routine operations.

Viewing an Assembly

The same view options, such as Topping, that are available in Simcenter Flotherm are available in MCAD Bridge.

Procedure

1. To redraw the display area to only contain the currently selected object, press F3, click the **Top** icon or choose **View > Top**.
2. To return to view the assembly containing this part, press F5, click the **Up To Parent** icon or choose **View > Up To Parent**.
3. The following are other View options:
   - To display the top level assembly for the MCAD model, press F4, click the **Reset** icon or choose **View > Reset**.
   - To expand all the bounding boxes of the geometry revealing the individual component parts, press F6 or choose **View > Expand All**.
   - To hide selected objects, press F12, click the **Hide** icon or choose **View > Hide**.
   - To re-display all hidden objects, press Shift+F12, click the **Reveal All** icon or choose **View > Reveal All**.

Un-Highlighting Non-Planar Faces

By default, non-planar faces are highlighted in magenta.

Procedure

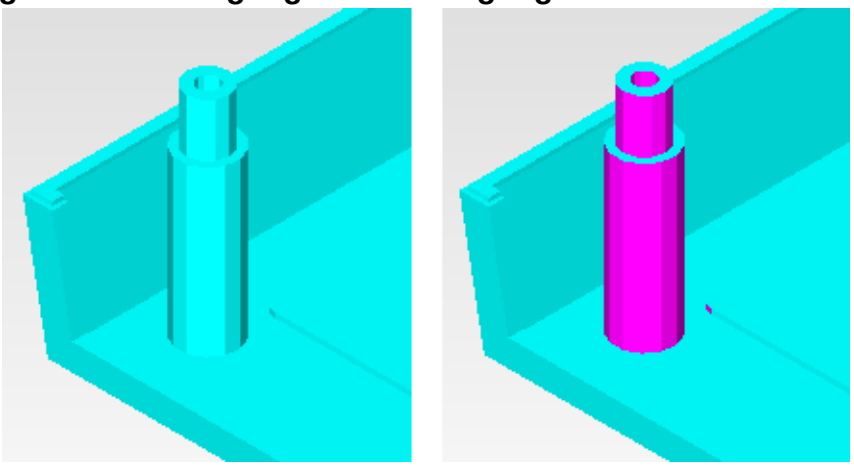
1. To un-highlight non-planar faces, click the **Preferences** icon or choose **Edit > Preferences** to open the Preferences dialog box.

2. Uncheck the Highlight Non Planar Faces check box and click **OK**.

Results

An example of un-highlighted and highlighted non-planar faces is shown in Figure 1-5.

Figure 1-5. Un-Highlighted and Highlighted Non-Planar Face



Measuring Distances Between Two Vertices

Distances between two vertices of the ACIS geometry can be shown in the GDA.

Procedure

1. Select the relevant MCAD Body and click the **Measure** icon  or choose **Tools > Measure** to open the Measure dialog box.

   The mouse pointer changes to cross-hairs.

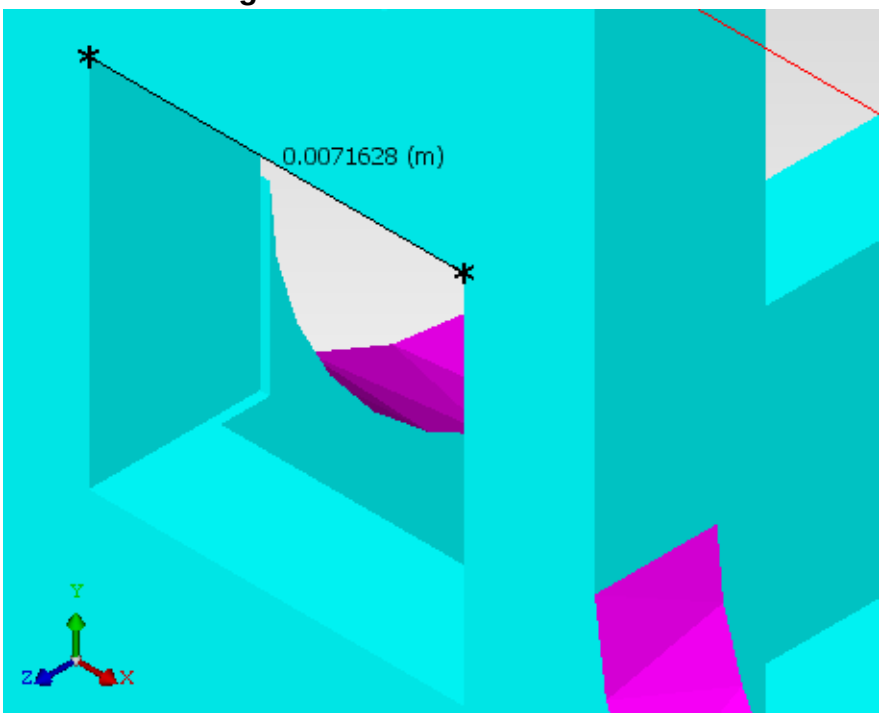
2. Select the first vertex by clicking near a vertex. A yellow cross-hair icon appears when you are close enough to select the vertex.

3. Select the second vertex.

Results

After the second selection, the distance is calculated and displayed in the window (see Figure 1-6) and the Measure dialog box.

Figure 1-6. Measured Distance



Related Topics

Measure Dialog Box

Rotating the GDA View Incrementally

You can rotate the graphics display area view with respect to the model or the viewer.

Procedure

1. Choose **View > Rotate View** to open the Rotate View dialog box.
2. Select the Rotate Mode (Model or View), depending around which set of axis you want the rotation to be made.
3. Select the Angle increment (in degrees or radians).
4. Use a rotate buttons to incrementally rotate the view.
5. 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

Rotating Parts and Assemblies to Align With Major Axes

A selected face can be aligned with one of the major axes.

Restrictions and Limitations

If the selected part belongs to a group of instanced parts created in the MCAD package, then the rotate operation breaks the relational links between the instanced parts. That is, each part becomes a separate entity.

Procedure

Select a face.

- To rotate the part containing the selected face to align with the nearest coordinate axis, choose **Tools > Rotate Align Part**.
- To rotate the assembly containing the selected face to align with the nearest coordinate axis, choose **Tools > Rotate Align Assembly**.

MCAD Bridge Property Sheets and Dialog Boxes

The following dialog boxes are called from MCAD Bridge.

MCAD Bridge Property Sheet

To access: Select an item in the data tree.

Use this property sheet to change the name of an item or to identify whether it has been fully converted (realized) into Simcenter Flotherm objects.

Objects

Field	Description
Name	Displays the name of the MCAD Part that was assigned in the MCAD package. If no name was assigned, Simcenter Flotherm uses the filename.
The following fields are only shown when an MCAD Part or a converted Simcenter Flotherm assembly is selected:	
Volume	Reports the volume of the part. The units are the current default for volume as set in the Simcenter Flotherm Global Units dialog box. A non-manifold body  has a volume of 0 (zero).
% Change	Reports the change of the part since loading as a percentage of its original volume. It is important to check for small changes of the MCAD Parts, particularly thermal calculations that could adversely affect results.
Realized	Indicates the conversion status of the part: • Yes — the part has been fully converted to Simcenter Flotherm format. • No — the part remains made up of ACIS components.
Size X, Y, Z	Indicates the current size of the object.

Related Topics

Global Units Dialog Box [Simcenter Flotherm User Guide]

Measure Dialog Box

To access: **Tools > Measure**

Use this dialog box to display the distance between two vertices selected in the MCAD Bridge GDA.

Objects

Field	Description
Absolute Distance	Read-only. The measured distance between the two selected vertices.
Distance X, Y, Z	Read-only. The measured distance between the two vertices 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 GDA.
Close	Closes the dialog box.

Usage Notes

If no MCAD Body is selected when the dialog box is opened, a message requests you to select one or more bodies.

Related Topics

Measuring Distances Between Two Vertices

Rotate View Dialog Box

To access: **View** > **Rotate View**

Use this dialog box to rotate the display area 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. • Model — Rotate about the major axes of the geometry: X, Y, and Z. • 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.
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.

Field	Description
 - 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 GDA View Incrementally

Chapter 2
Introduction to the Processes

MCAD Bridge can be used both to import MCAD assemblies for use in Simcenter Flotherm, and to export Simcenter Flotherm assemblies for use in MCAD applications software.

Overview of the Import Process

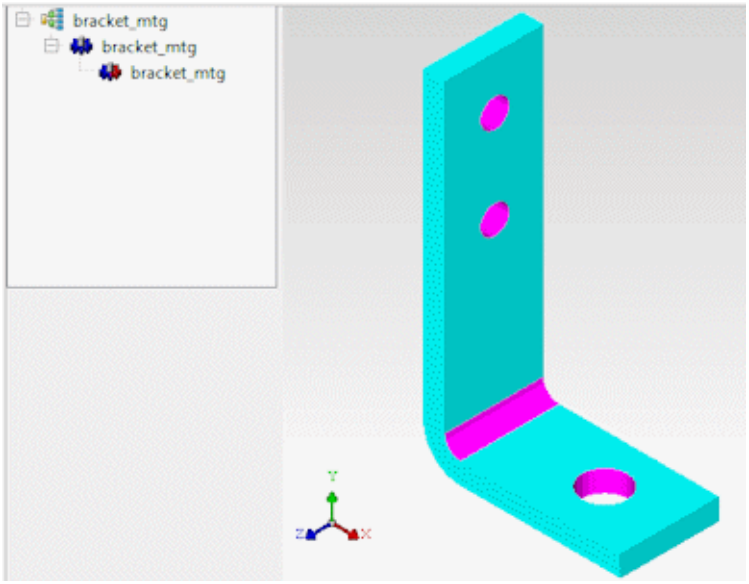
Before MCAD assemblies can be used in Simcenter Flotherm, they must be reformatted to represent Simcenter Flotherm objects. The whole process is a 3-stage operation.

Procedure

1. Start the import of the MCAD file by choosing one of the **External** > **Import** menu options.

   On import, the external MCAD Assemblies are translated by MCAD Bridge into ACIS® models, see Figure 2-1.

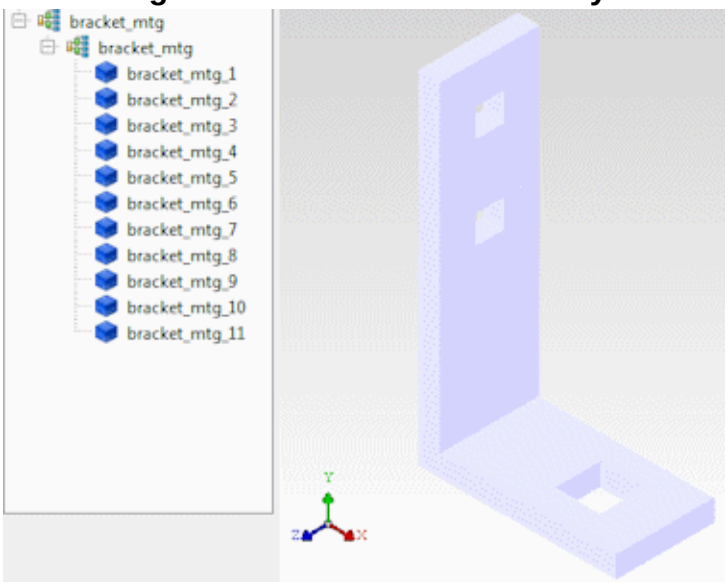
Figure 2-1. Imported MCAD Geometry



2. Simplify the MCAD geometry and then decompose it into Simcenter Flotherm geometry, see Figure 2-2.

   The simplest method is to let the program automatically simplify and convert the geometry.

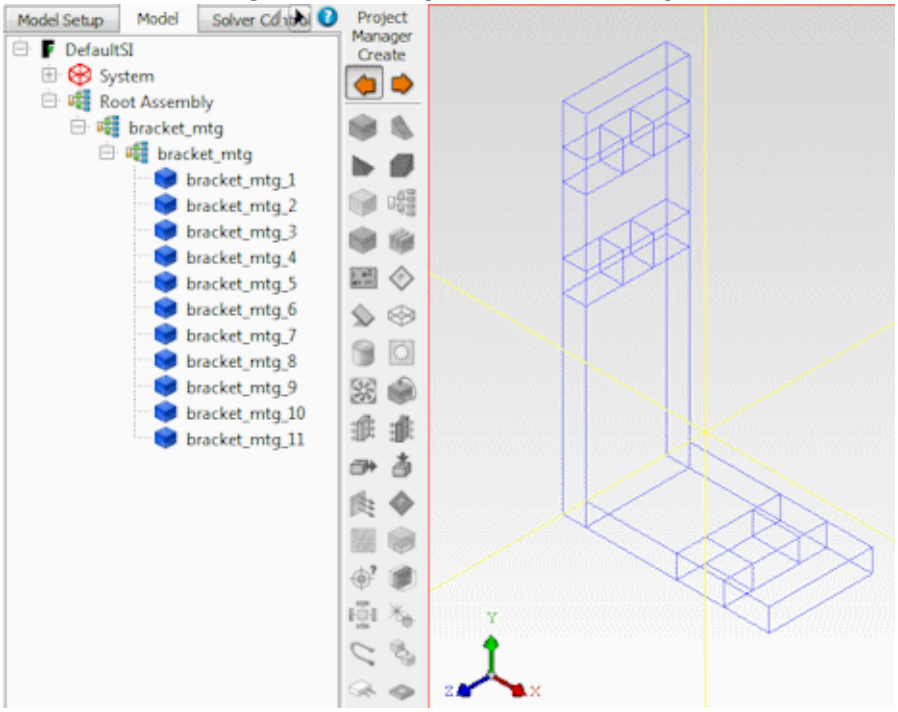
Figure 2-2. Converted Geometry



   However, for complicated models, it is more efficient to simplify the model by removing the geometry not required for thermal calculation. You can selectively simplify the model, for example, by removing holes and humps, leveling near-level features, straightening angled faces, planing non-planar faces, removing parallel features, flattening draft angles as well as splitting bodies, before replacing the MCAD objects with selected Simcenter Flotherm geometry.

3. Transfer the converted assembly to Simcenter Flotherm Project Manager by choosing **Tools > Transfer Assembly**.

Figure 2-3. Imported Geometry



Related Topics

Import

Overview of the Export Process

Simcenter Flotherm project geometry is loaded into the MCAD Bridge window from the Project Manager and then exported from MCAD Bridge in a selected file format.

Procedure

1. Import Simcenter Flotherm geometry into MCAD Bridge.
2. Export to the desired file format from MCAD Bridge.

   The process is described in detail under “Exporting Project Geometry” on page 71.

Identification of MCAD Assemblies and Parts

MCAD Bridge will typically contain a mixture of MCAD and realized Simcenter Flotherm objects.

Before the model imported into MCAD Bridge can be transferred to the Simcenter Flotherm project, it must be converted (realized) to Simcenter Flotherm geometry.

To do this, the assembly must be in a hierarchical structure:

- **MCAD Assembly —** the ACIS model.
- **MCAD Part —** an ACIS single solid part of the assembly.
- **MCAD Body —** a subset of the MCAD Part.
- **Simcenter Flotherm Objects —** Simcenter Flotherm primitives or SmartParts.

The MCAD Assembly is made up at least one MCAD Part. Each MCAD Part is made up of at least one MCAD Body. Each MCAD Body is made up of zero (not yet realized) or more Simcenter Flotherm objects (realized).

Naming Convention

Realized parts are named according to a convention.

- MCAD Parts

  If there is only one MCAD Part, this will be named:

  <*mcadpartname*>

  where <*mcadpartname*> is the name of the original MCAD Part.

  If there are multiple parts, then each part will be named:

  <*mcadpartname*>_#

  where # is an import sequence number.

- Simcenter Flotherm Geometry

  The Simcenter Flotherm assemblies are named:

  <*mcadpartname*>

  where <*mcadpartname*> is the name of the original MCAD Part.

  If there are multiple assemblies created, then each assembly will be named <*mcadpartname*>_# where # is a sequence number.

  When transferred to the Project Manager, if an assembly of the same name exists, the names will be appended by: #.

  Objects contained in the realized assemblies will be named:

  <*parentassemblyname*>_#.

FloSCRIPT

Advanced users can run a FloSCRIPT file from MCAD Bridge.

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

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

Each file has a unique name, using the naming convention *MCADLogFile<number>.xml*.

Note

Only the last five log files are retained when a new MCAD Bridge 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) “MCADLogFile” string.

A FloSCRIPT file can be run from the GUI by choosing **Macro > Play FloSCRIPT** and then selecting the file, or from the command line, by specifying the file in a **flomcad** command.

Related Topics

FloSCRIPT [Simcenter Flotherm User Guide]

flomcad

Chapter 3
Import and Export

How to use MCAD Bridge to import MCAD files and export Simcenter Flotherm files.

Advice on how to prepare third-party MCAD files for import is given in “Creating MCAD Files Using External Software” on page 93.

Import

Use the following procedures when importing MCAD files.

Overview of Importing MCAD Designs

The following is a high-level sequence to use when importing MCAD designs for thermal modeling.

1. Simplify the assembly by removing as much as possible of the geometry not required for the thermal calculation before creating the MCAD model file, see “Creating MCAD Files Using External Software” on page 93.
2. Import the MCAD file into MCAD Bridge. If the model fails to heal on import, try setting a different healing level, see “Using the Healing Tool to Join Entities to Create a Solid Object” on page 48, and either re-import the file, or choose **Tools > Heal** for the translated geometry to create a solid model.
3. Realize the ACIS model into Simcenter Flotherm objects using one or more of the following procedures.
   - Simplify an entire MCAD Assembly prior to decomposition into Simcenter Flotherm objects.
   - Simplify individual faces or features prior to decomposition into Simcenter Flotherm objects.
   - Automatically simplify a selected body or parts into Simcenter Flotherm objects.
   - Replace a body with a single cuboid, prism, resistance, fan, PCB, or enclosure.
   - Dissect a part or body into objects.
   - Voxelize a part or body into cuboids.
   - Split a body into more bodies. Each body can then be broken down into Simcenter Flotherm objects.

   The procedures are described under “Simplifying MCAD Parts or Bodies” on page 45 and “Converting MCAD Geometry” on page 59.

Supported Import File Formats

MCAD Bridge can read common MCAD file formats.

Table 3-1 lists the file formats that can be imported by MCAD Bridge

Table 3-1. Import File Formats

File Format	File Extensions	Versions Supported
CATIA V4	**.model* **.exp* **.session*	4.1.9 to 4.2.4
CATIA V5	**.CATPart* **.CATProduct* **.CGR*	V5R8 to V5-6R2020
IGES	**.igs* **.iges*	Up to 5.3
ACIS	**.sat* **.sab*	ACIS R1 to 2020 1.0
Pro/E	**.prt* **.prt.** **.asm* **.asm.**	Pro/E 16 to Creo 7.0
SolidWorks	**.sldprt* **.sldasm*	98 to 2020
STEP	**.stp* **.step*	AP203, AP214, AP242 (Geometry only)

Supported IGES Entities

The Connect IGES reader supports IGES entities.

The IGES entities supported are listed in Table 3-2. If the Connect reader fails, the code will use the in-house legacy reader, which supports the IGES entities in Table 3-3.

Table 3-2. Supported IGES Entities

IGES Entity	Form	Description	ACIS Interpretation
100	0	Circular Arc	EDGE with ellipse curve
102	0	Composite Curve	LOOP

Table 3-2. Supported IGES Entities (cont.)

IGES Entity	Form	Description	ACIS Interpretation
104	0,2,3	Conic Arc - General	Intcurve
104	1	Conic Arc - Ellipse	EDGE with ellipse curve
106	11,12,63	Linear Path	Curve List
108	1	Plane	FACE with plane surface
110	0	Line	EDGE with straight curve
112	0	Parametric Spline Curve	Intcurve
114	0	Parametric Spline Surface	Spline surface
116	0	Point	Point
118	0 or 1	Ruled Surface	Spline surface
120	0	Surface of Revolution	Spline surface
122	0	Tabulated Cylinder	Spline surface
123	0	Direction	Vector
124	0	Transformation Matrix	TRANSFORM
126	0	Rational B-Spline Curve	Intcurve
128	0	Rational B-Spline Surface	Spline surface
130	0	Offset Curve	Curve
140	0	Offset Surface	Surface
141	0	Boundary	LOOP
142	0	Curve on Parametric Surface	LOOP
143	0	Bounded Surface	FACE
144	0	Trimmed (Parametric) Surface	FACE
186	0	MSBO	Body
190	0	Plane Surface	Plane surface

Table 3-2. Supported IGES Entities (cont.)

IGES Entity	Form	Description	ACIS Interpretation
192	0	Rt. Circular Cylindrical Surface	Cone surface
194	0	Rt. Circular Conical Surface	Cone surface
196	0	Spherical Surface	Sphere surface
198	0	Torroidal Surface	Torus surface
502	1	Vertex List	Vertex

Table 3-3. In-House IGES Supported Entities

IGES Entity	Form	Description	ACIS Interpretation
100	0	Circular Arc	EDGE with ellipse curve
102	0	Composite Curve	LOOP
104	1	Conic Arc	EDGE with ellipse curve
106	11,12	Linear Path	LOOP
108	0	Plane	plane surface
110	0	Line	EDGE with straight curve
118	0 or 1	Ruled Surface	sweep spline surface or cone surface or plane surface.
120	0	Surface of Revolution	rotated spline surface or cone surface or sphere surface
122	0	Tabulated Cylinder	sweep spline surface or cone surface
124	0	Transformation Matrix	TRANSFORM
126	0,3 - 5	Rational B-Spline Curve	EDGE with intcurve curve
	1	Rational B-Spline Curve	EDGE with straight curve

Table 3-3. In-House IGES Supported Entities (cont.)

IGES Entity	Form	Description	ACIS Interpretation
	2	Rational B-Spline Curve	EDGE with ellipse curve
128	0,5 - 9	Rational B-Spline Surface	spline surface or plane surface or cone surface or sphere surface
	1	Rational B-Spline Surface	plane surface
	2,3	Rational B-Spline Surface	cone surface
	4	Rational B-Spline Surface	sphere surface
141	0	Boundary	LOOP
142	0	Curve on Parametric Surface	LOOP
143	0	Bounded Surface	FACE
144	0	Trimmed (Parametric) Surface	FACE
308	0	Subfigure Definition	BODY or sub-Assembly
402	1	Group	FACE & its LOOPs
406	12	External Reference File	BODY or sub-Assembly
408	0	Singular Subfigure Instance	Instance of BODY or sub-Assembly

Importing MCAD Files Into MCAD Bridge

Although the import process is simple, for best results, the MCAD file should be suitably prepared.

Restrictions and Limitations

- Not all versions of file formats are supported, see “Supported Import File Formats” on page 38.
- By default, any MCAD geometry color data imported into MCAD Bridge is lost when you transfer the model to the Project Manager.

To retain the MCAD colors, then set the FLOMCAD_SET_COLOR_ON_TRANSFER environment variable to any value, and re-start MCAD Bridge.

When MCAD geometry color is retained, it is done so by using an attached Surface attribute. The default emissivity for such Surface attributes is always 1.0. If this is not representative of the true emissivity, then the value must be changed in Simcenter Flotherm.

- If geometry is less than 10^{-6} times the size of the Length unit set in the Simcenter Flotherm Global Units dialog box, it will not be transferred successfully between MCAD Bridge and Simcenter Flotherm.

  This can happen when transferring micron-scale geometry when the Length unit is meters (m). In such cases, set the Length in the Simcenter Flotherm Global Units dialog box to a small unit, say millimeters (mm).

Prerequisites

- A model may fail to heal if the accuracy of the imported file is too coarse. In such cases, surfaces that are meant to join up have a gap between them and so a solid manifold object cannot be derived. This may happen with **.igs* or **.stl* data that contains just 1D and 2D geometric entities. It is because of this that it is critical that the geometry is regenerated to the required accuracy of 10^{-6} in the MCAD tool prior to creating the **.igs* or **.stl* file.

Procedure

1. Before importing and MCAD file, you are advised to simplify using the MCAD tool first. See “Creating MCAD Files Using External Software” on page 93.
2. If you are importing complicated geometry with many non-planar faces, you may want to restrict the number of facets to improve the import performance. To restrict the number of facets, choose **Edit > Preferences** to open the Preferences dialog box and set Graphics Faceting Level to Low.
3. Check that the unit of length setting in the Simcenter Flotherm Global Units dialog box is that required for the output model file. Choose a value that will not be too large and cause tolerance/accuracy problems (for example, in/cm).
4. If you want to view messages generated during importing, activate the Message Window by checking the Report ACIS Model Errors to Message Window check box in the Preferences dialog box.
5. Select one of the menu options from the **External** menu:
   - To import an IGES file, choose **Import IGES** to open the Import IGES data file selection dialog box, and then select and load the **.igs* file.

MCAD Bridge has two IGES readers: the robust ACIS Connect reader and an in-house IGES reader. If MCAD Bridge fails to load the IGES file using the Connect reader, it automatically tries again using the in-house IGES reader.

- To import an STL file, choose **Import STL** to open the Import STL data file selection dialog box, and then select and load the **.stl* file. As **.stl* files do not support unit definition, after confirming the file selection, you are prompted to choose the units Simcenter Flotherm is to use to model the MCAD Assembly.
- To import a SAT file, choose **Import SAT** to open the Import SAT data file selection dialog box, and then select and load the ACIS **.sat* file. For older **.sat* files with no unit definition, you are prompted to choose the units Simcenter Flotherm is to use to model the MCAD Assembly.
- To import a STEP file, choose **Import STEP** to open the Import Step data file selection dialog box, and then select and load the **.stp* file.
- To import a Pro/E file, choose **Import Pro/E** to open the Import Pro/E data file selection dialog box, and then select and load the **.prt** or **.asm** file.
- To import a CATIA V4 file, choose **Import Catia V4** to open the Import Catia V4 data file selection dialog box, and then select and load the **.model* or **.exp* file.
- To import a CATIA V5 file, choose **Import Catia V5** to open the Import Catia V5 data file selection dialog box, and then select and load the **.CATPart* or **.CATProduct* file.
- To import a DXF file, see "Importing DXF Files Into MCAD Bridge" on page 44.
- To import a SolidWorks file, choose **Import SolidWorks** to open the Import SolidWorks data file selection dialog box, and then select and load the **.sldprt* or **.sldasm* part or assembly file.

Results

- When an MCAD file is loaded into the application, it is translated into an ACIS assembly. The MCAD Bridge Monitor dialog box displays the load progress.
- To interrupt the loading of MCAD files, click **Interrupt MCAD** in the MCAD Bridge Monitor dialog box.
- Any MCAD bodies unrecognized during transfer are ignored and a warning message is issued.

Related Topics

Preferences Dialog Box

MCAD Bridge Monitor Dialog Box

Global Units Dialog Box [Simcenter Flotherm User Guide]

Message Window

Importing DXF Files Into MCAD Bridge

DXF data supports 0D (vertices), 1D (edges) or 2D (faces). For MCAD Bridge to successfully transfer DXF models to the Project Manager, all the data must be in at least 2D format.

There are two approaches to converting 1D and 2D DXF data to Simcenter Flotherm objects:

- Convert 1D edges to 2D faces, then convert the 2D faces directly into collapsed cuboids.
- Convert 1D edges to 2D faces, then extrude the 2D faces into 3D bodies which are processed in the normal way.

Procedure

1. Choose **External > Import DXF**, select a DXF file and click **Open**.

   DXF files created prior to AutoCAD 2000 do not support a unit definition. After confirming the file selection, you will prompted to choose the units Simcenter Flotherm is to use to model the MCAD Assembly.

   AutoCAD 2000 or later DXF files have the unit value embedded and this will be read and used by MCAD Bridge.

2. Delete or hide unwanted, non-geometry, layers. DXF data is often constructed from a number of layers (sometimes referred to as 2.5D data). It is common that many of these layers do not represent geometry but other data such as mark-up, annotation, drawing boarders, and so on. Layers are represented as MCAD Parts in MCAD Bridge and can be deleted.

3. Press W to show geometry in wire frame. By default, MCAD geometry is rendered as solid. Any DXF 2D faces will be visible, but 1D edges will not.

   Any 1D or 2D faces will be colored yellow as they form part of an unhealed body. Subsequent extrusion or direct conversion to collapsed cuboids will resolve this as follows:

   If required, extrude the edges to convert them to faces (see Extruding MCAD Geometry Faces or Edges), then do one of the following:

   - Convert the 2D face into collapsed cuboids as described in Voxelizing an MCAD Part. Voxelization will create a stepped representation of curved faces.
   - Convert into collapsed cuboids or angled thin cuboids as described in Dissecting a Body Based on Tolerances. Dissection will totally square off a curved face.
   - Use the extrude process to convert 2D faces into 3D bodies and treat in the usual way. The main advantage of taking this approach for DXF data is that prisms and cylinders will be created when dissecting the body and an exact curvature representation will be obtained.

Simplifying MCAD Parts or Bodies

The MCAD structure can be simplified to ease the conversion to Simcenter Flotherm. For best conversion results, use a combination of global and local simplification.

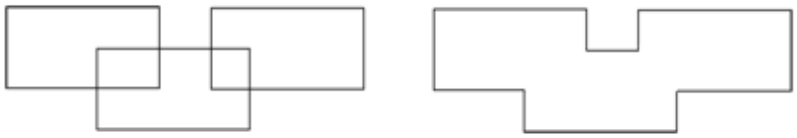
Uniting, Intersecting, and Subtracting Multiple Bodies

Options are available for combining multiple objects.

Procedure

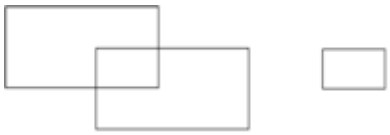
1. To unite all selected intersecting bodies into one body, select the bodies then choose **Tools > Unite Bodies**. Figure 3-1 shows a typical result.

Figure 3-1. Three Bodies United



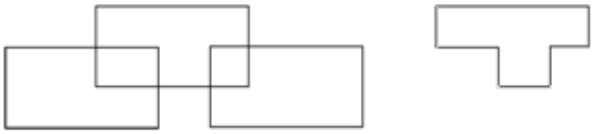
2. To deletes all but the intersecting regions of a number of bodies, select the bodies then choose **Tools > Intersect Bodies**. Figure 3-2 shows a typical result.

Figure 3-2. Two Bodies Intersected



3. To subtracts all selected intersecting bodies from the first selected body, select the bodies then choose **Tools > Subtract Bodies**. Figure 3-3 shows a result where three bodies have been replaced by a single body. The middle body was the first to be selected, subsequent selected bodies have been subtracted from that body.

Figure 3-3. Subtracted Bodies



Using Global Simplify to Automatically Simplify Features and Faces

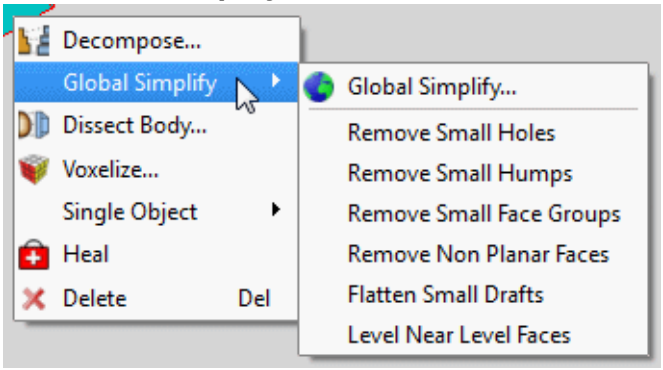
Use global simplification to remove small features and flatten faces from all sections of a selected MCAD part/body. Global simplification is a ‘batch’ run of a number of local simplification processes.

Procedure

1. Select the part and click the **Global Simplify** icon or choose **Tools > Global Simplify** to open the Global Simplify dialog box.
2. Perform one global simplify operation at a time. For example, remove small holes, then remove non-planar faces.

   If failures arise, that is, if “F”s appear in the MCAD Bridge Monitor dialog box, then try some local simplification or healing, then retry.
3. Repeat until you simplify to the extent where the geometry could have been created normally in Simcenter Flotherm.
4. Alternatively, select a part or body and right-click to display the context-sensitive menu, see Figure 3-4.

Figure 3-4. Global Simplify Context-Sensitive Menu Options



5. The **Global Simplify > Global Simplify** option opens the Global Simplify dialog box. The other options act on the selected object.

Results

Figure 3-6 to Figure 3-10 show the results of using the global simplify options.

Figure 3-5. Removal of Small Face Groups

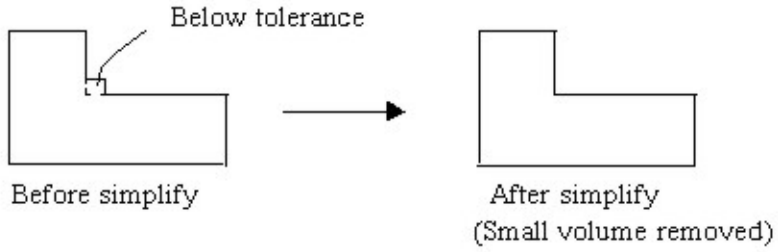


Figure 3-6. Replacement of Non Planar Faces

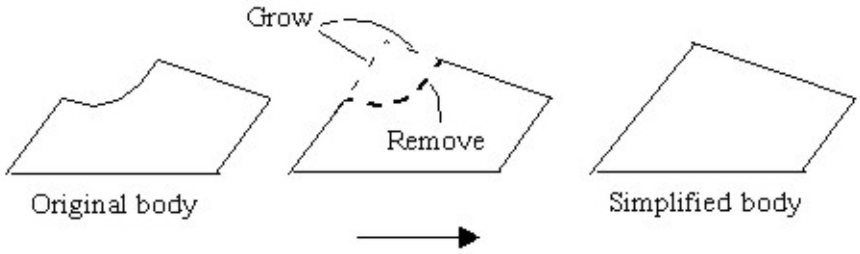


Figure 3-7. Replacement of Complex Non Planar Faces

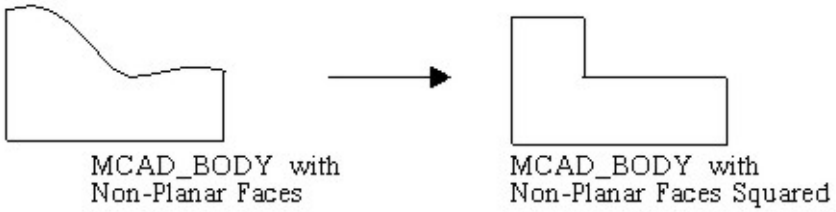


Figure 3-8. Flattening Small Drafts

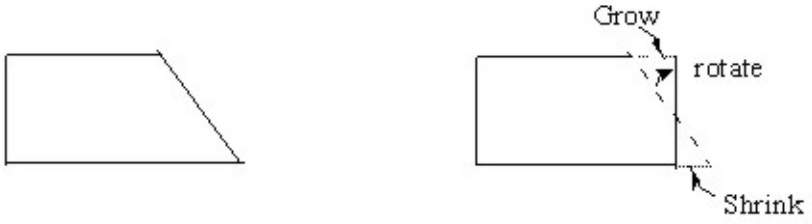


Figure 3-9. Flattening of a Complex Small Draft

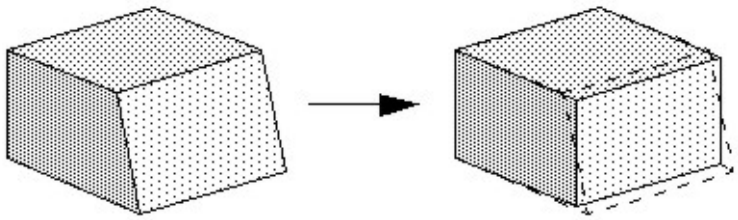
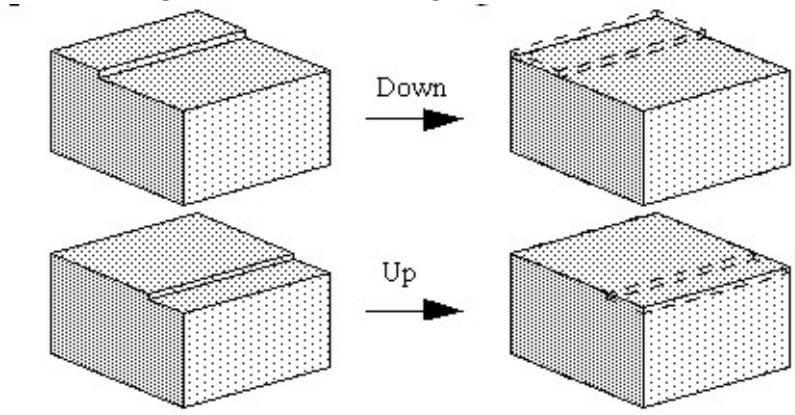


Figure 3-10. Leveling 'Near Level' Faces



Related Topics

Global Simplify Dialog Box

Using Local Simplify to Simplify Selected Features and Faces

Using the Healing Tool to Join Entities to Create a Solid Object

If a global simplify fails, try "healing" to create solid ACIS objects before global simplifying again.

It is important to understand the reason why healing is necessary. Nearly all simplification and dissection routines rely on the calculation of a body volume. For example, a body volume may be used to determine whether it is too small and should be removed. A healed model will aid the robustness of simplification and dissection.

Healing can be done:

- After input. If the original part or body is partially healed on input, this option can be activated at any time to attempt a complete heal.
- During simplification. If an ACIS model error occurs during simplification, healing can help to allow continuation of the simplification process.

Procedure

1. Choose **Edit > Preferences** to open the MCAD Preferences dialog box.
2. Check the healing level. By default, this is set to Full but it can be set to Stitch Only.
3. Click the **Heal** icon or choose **Tools > Heal**, especially just before a major simplification. This is sometimes extremely important as a model may become after much simplification has been performed.

Results

The selected MCAD part or body is “healed” by joining any separate ACIS entities that were formed during the translation to create a solid object. If no part or body is selected, then the complete model is healed.

The healing process can take a number of minutes depending on the complexity of the model.

After healing, you can continue simplification.

Related Topics

Preferences Dialog Box

Using Local Simplify to Simplify Selected Features and Faces

Use these methods to simplify selected features of an assembly, for example, to even out the surface of an object by removing holes and humps, or leveling sloping sides.

Adding a Bounding Box

Use the addition of a bounding box to fill in faces or features.

Prerequisites

- You are in Select mode and the Current Selection Mode is set to Feature or Face, see "Geometry Selection and View Manipulation" on page 17.

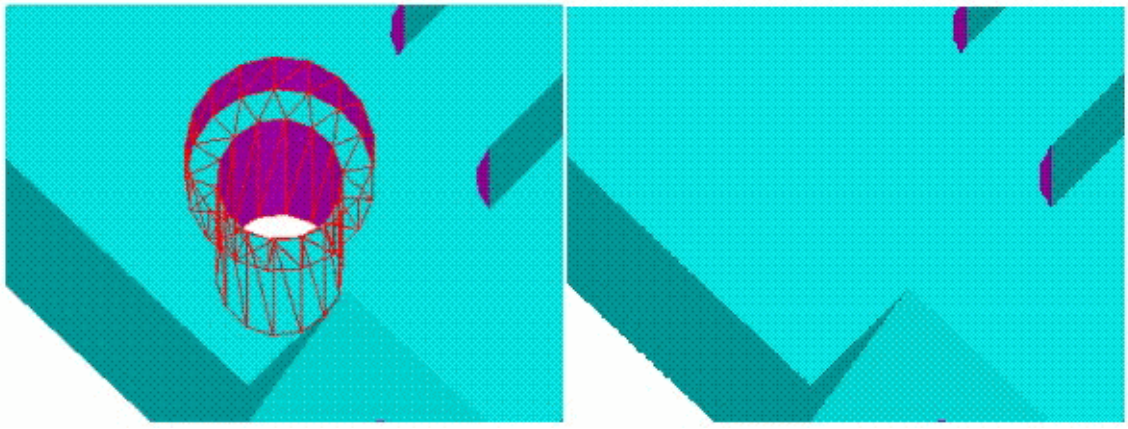
Procedure

1. Select the feature or face.
2. Click the **Local Simplify** icon  or choose **Tools > Local Simplify** to open the Local Simplify dialog box.
3. Click the **Add Bounding Box** button or press the + key.

Results

Figure 3-11 shows an example where a bounding box has been added to close a complex hole.

Figure 3-11. Addition of a Bounding Box



Subtracting a Bounding Box

Use the subtraction of a bounding box to remove the bounding box of either the selected faces or of each of the selected features from the owning MCAD Body.

Prerequisites

- You are in Select mode and the Current Selection Mode is set to Feature or Face, see “Geometry Selection and View Manipulation” on page 17.

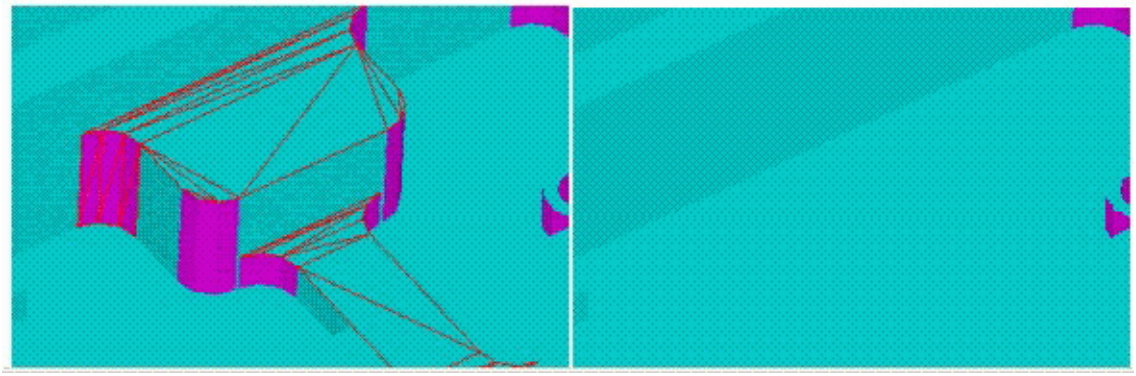
Procedure

1. Select the feature or face.
2. Click the **Local Simplify** icon or choose **Tools > Local Simplify** to open the Local Simplify dialog box.
3. Click the **Subtract Bounding Box** button or press the - key.

Results

Figure 3-12 shows where selected planar and non-planar faces has been removed by subtraction of a bounding box.

Figure 3-12. Subtraction of a Bounding Box



Removing Selected Features

You can remove selectable features or faces to simplify the model.

Prerequisites

- You are in Select mode and the Current Selection Mode is set to Feature or Face, see "Geometry Selection and View Manipulation" on page 17.

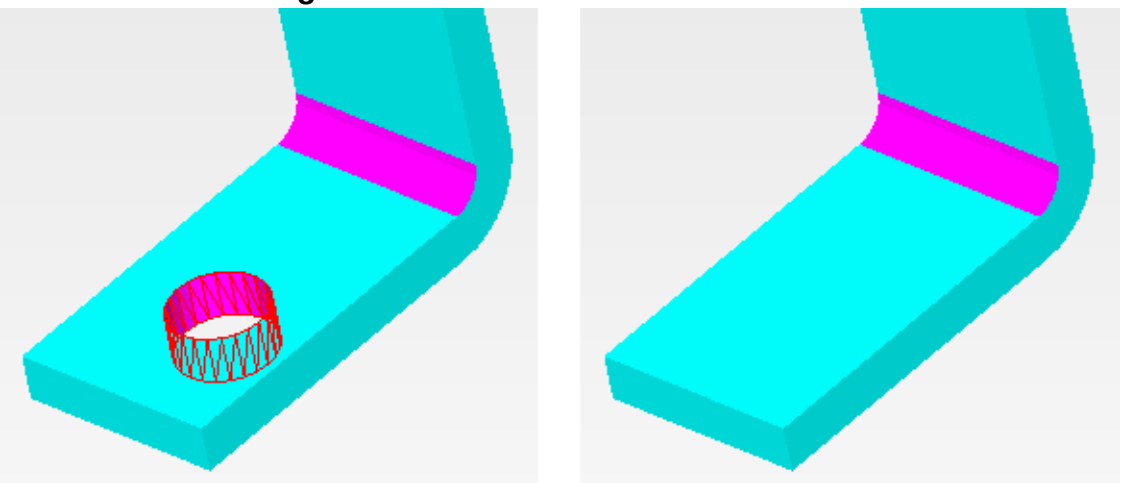
Procedure

1. Select the feature or face.
2. Click the **Local Simplify** icon or choose **Tools > Local Simplify** to open the Local Simplify dialog box.
3. Click the **Remove** button or press the Delete key.

Results

Figure 3-13 shows an example of a removed face.

Figure 3-13. Removal of Selected Face



Flattening Selected Faces

To simplify the model, you can flatten selectable faces.

Prerequisites

- You are in Select mode and the Current Selection Mode is set to Face, see "Geometry Selection and View Manipulation" on page 17.

Procedure

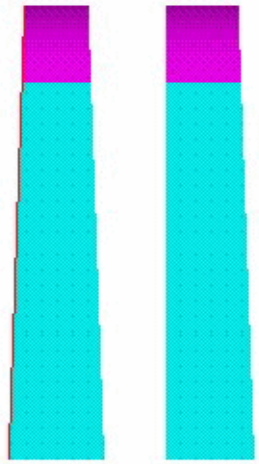
1. Select the face.

2. Click the **Local Simplify** icon or choose **Tools > Local Simplify** to open the Local Simplify dialog box.
3. Click the **Flatten** button or press F.

Results

Figure 3-13 shows an example of flattening one (the left-hand side) face.

Figure 3-14. Flattened Selected Face



Leveling Selected Faces

To simplify the model, you can level selectable faces.

Prerequisites

- You must be in Select mode and the Current Selection Mode must be Face, see "Geometry Selection and View Manipulation" on page 17.

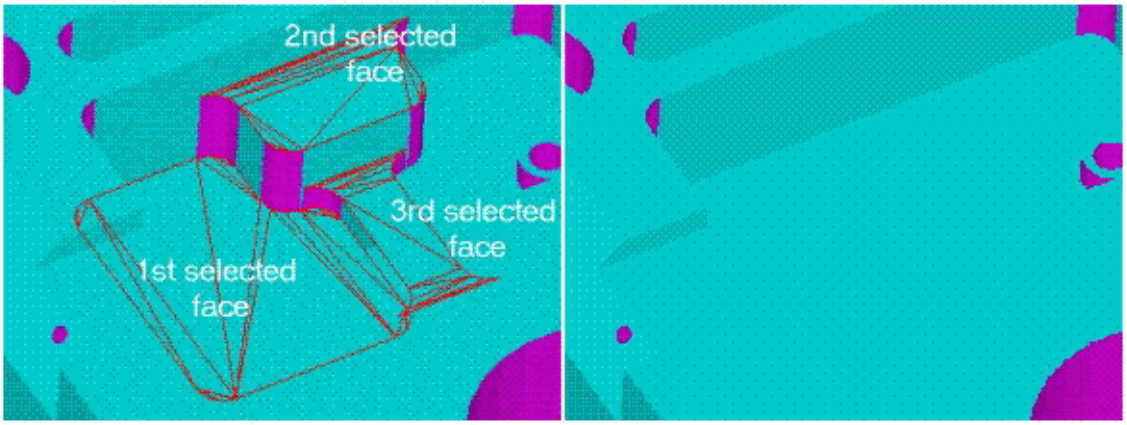
Procedure

1. Select one or more faces.
2. Click the **Local Simplify** icon or choose **Tools > Local Simplify** to open the Local Simplify dialog box.
3. Click the **Level** button or press L.

Results

Figure 3-15 shows an example of leveling three selected faces.

Figure 3-15. Leveled Selected Faces



Related Topics

Local Simplify Dialog Box

Using Global Simplify to Automatically Simplify Features and Faces

Suppressing Removal of Non-Planar Faces When Using Global Simplify

Non-planar faces can be excluded from global simplification, a technique best used to retain large holes that may have a dominant effect on the airflow.

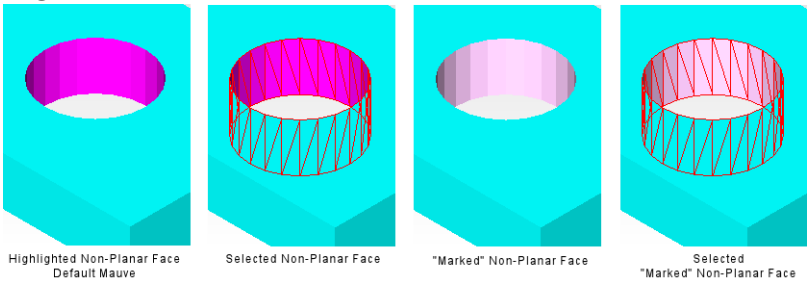
Prerequisites

- You are in Select mode and the Current Selection Mode is set to Face, see "Geometry Selection and View Manipulation" on page 17.

Procedure

1. Select the non-planar face or faces. Non-planar faces are highlighted in magenta by default, but this can be overridden using the Preferences dialog box.
2. Click the **Local Simplify** icon or choose **Tools > Local Simplify** to open the Local Simplify dialog box.
3. Click the **Mark** button or press M to mark the selected non-planar faces. Marking non-planar faces changes the color of a highlighted non-planar face from a deep to a light magenta as shown in Figure 3-16. Click the **Un-Mark** button or press U to un-mark selected marked non-planar faces.

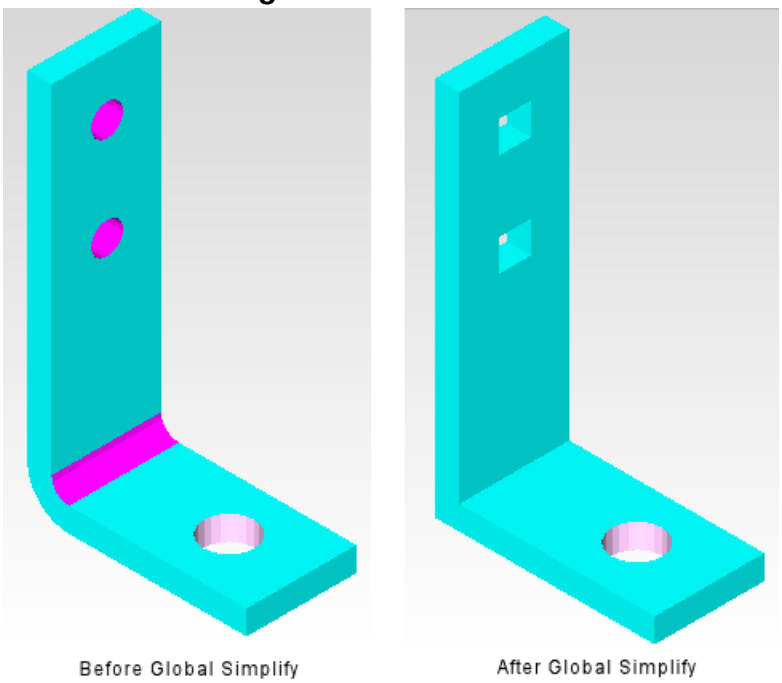
Figure 3-16. Un-Marked and Marked Non-Planar Face



Results

Figure 3-17 shows an example where one hole (the bottom hole) in a bracket has been marked before a global simplification. The marked hole retains its non-planar face.

Figure 3-17. Effect of Marking a Non-Global Face When Global Simplifying



Related Topics

Local Simplify Dialog Box

Using Global Simplify to Automatically Simplify Features and Faces

Splitting an MCAD Body

Use this method to split an MCAD Body into two or more MCAD bodies.

Procedure

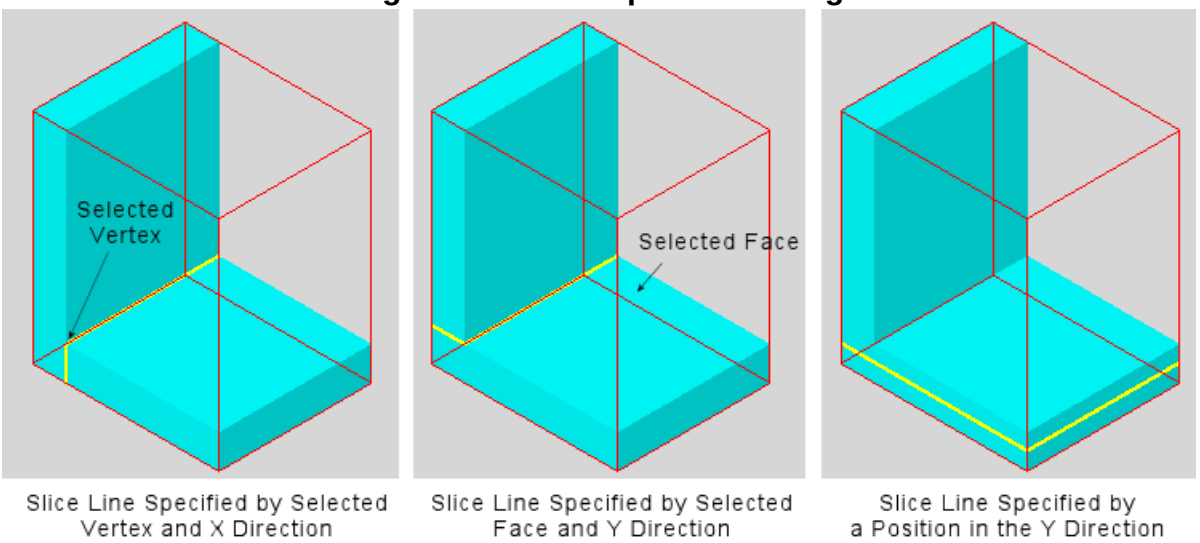
1. Select the MCAD Body and click the **Split Body** icon or choose **Tools > Split Body** to open the Split Body dialog box.
2. Use one of the following methods to mark the location(s) of split(s).

If you want to...	Do the following:
Mark the location from a vertex.	1. Click the **Select Vertex** button. The mouse pointer changes to a cross. 2. Use the mouse to select the MCAD vertex. 3. Select the Direction (plane) of the cut.
Mark the location from a face.	1. Click the **Select Face** button. The mouse pointer changes to a cross. 2. Use the mouse to select the MCAD face. 3. Select the Direction (plane) of the cut.
Mark the location by specifying an orthogonal plane.	1. Select the Direction (plane) of the cut. 2. Either use the Position slider bar or enter an axial coordinate to specify the position. The slider bar limits correspond to the minimum and maximum coordinates of the bounding box of the selected body

3. Click **Slice** to mark where the split is to occur.

   Figure 3-18 shows examples of slice marks (in yellow) that have been created from a vertex, from a face and by specifying a position.

Figure 3-18. Examples of Slicing

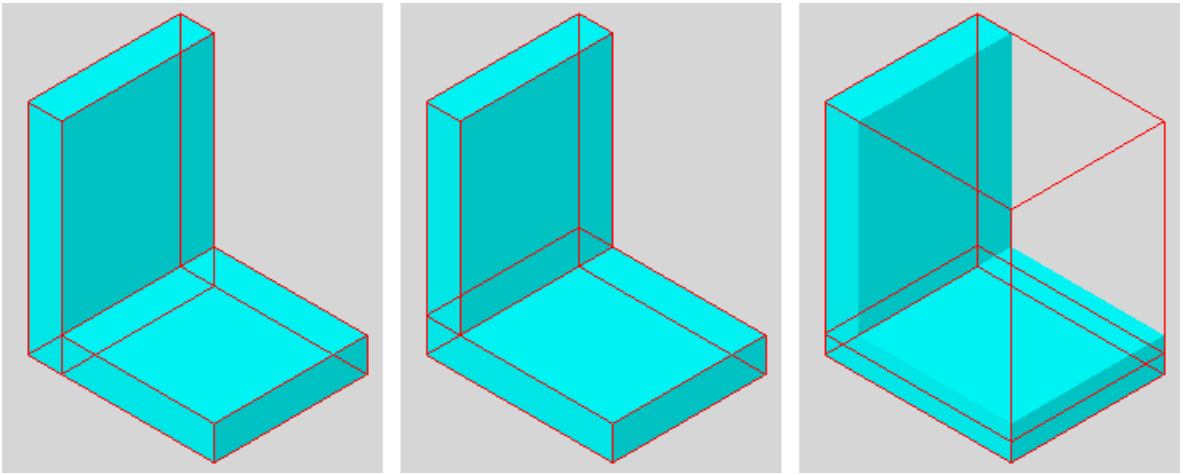


4. Click **Split** to split the MCAD body into smaller bodies.

Results

Separate MCAD bodies are created by splitting along slice lines. Figure 3-19 shows the examples of Figure 3-18 after splitting.

Figure 3-19. Examples of Splitting



Related Topics

Simplifying MCAD Parts or Bodies

Split Body Dialog Box

Extruding MCAD Geometry Faces or Edges

Extruding expands the ACIS model normal to a selected face, or in a specified direction from a selected edge.

Procedure

1. Ensure the mouse is in pick mode and the Current Selection Mode is set to Face or Edge as appropriate.
2. Select the MCAD face or edge to be extruded.
3. Choose **Tools > Extrude** to open the Extrude dialog box.
4. Enter the distance that the face or edge is to be moved in the Depth data entry field.
5. If an edge is selected, also choose the Direction of the extrude.

   Edge extrusion is most commonly used when representing walls or other building fabrics. Often DXF data contains edge descriptions for both the inner and outer wall faces. Selection of only those edges that represent the desired wall profile is required. This can be done by careful selection of those edges, or deletion of the bodies containing

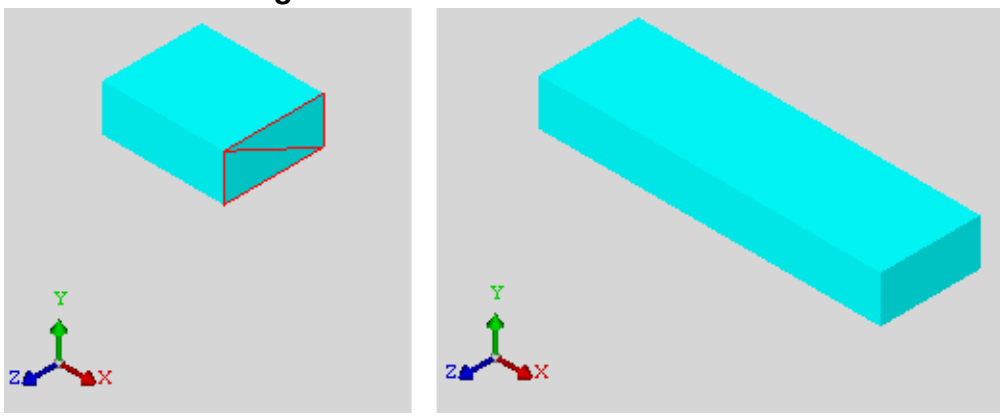
the unwanted edges followed by a rubber band select of the remaining desired edges. This latter method is often preferable.

6. Click **Apply**.

Results

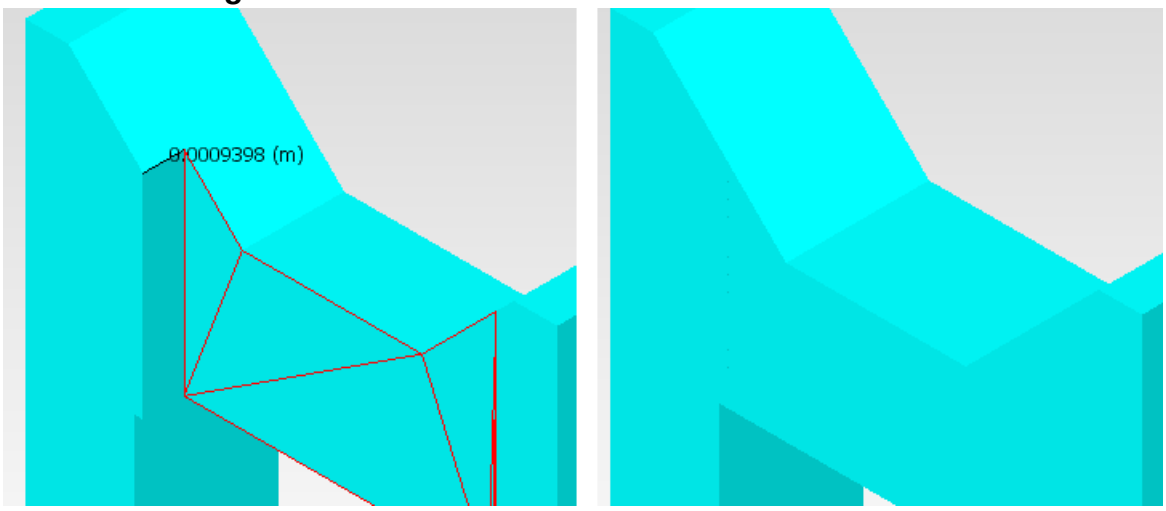
An example of extruding a selected face is shown in Figure 3-20. The direction is normal to the face.

Figure 3-20. Extruded MCAD Face



In Figure 3-21, the Measure tool has been used to determine the exact extrude distance of a face to align with another face.

Figure 3-21. Extruded Face to a Measured Distance



Related Topics

Extrude Dialog Box

Converting MCAD Geometry

Methods for converting MCAD parts and bodies into Simcenter Flotherm geometry.

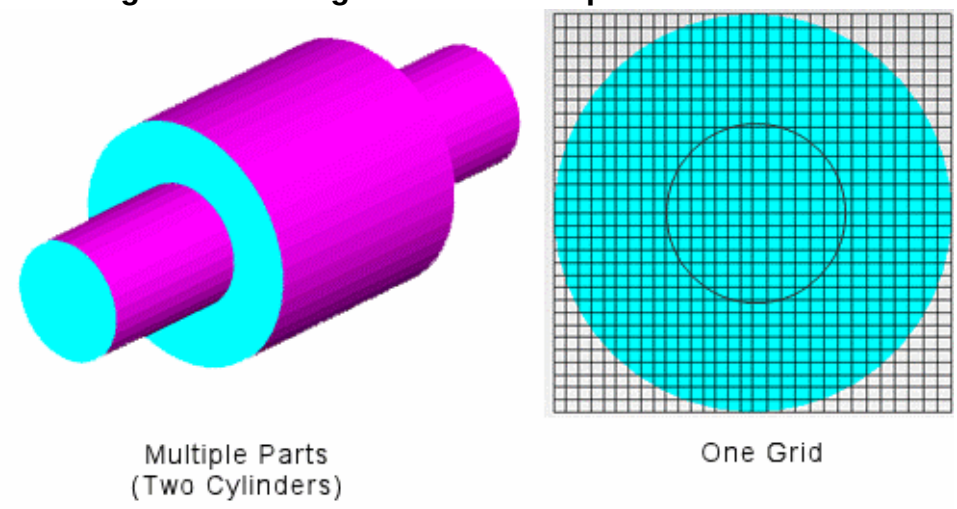
Voxelization

The voxelization method uses a grid (mesh) to convert MCAD geometry to Simcenter Flotherm geometry in the form of grid-sized cuboids, known as voxels.

Voxelization conversion is best suited to predominantly cartesian-type geometry that can be effectively represented by cuboids alone. The advantage of voxelization has is that it is a more robust method of geometry conversion than the Dissect Body conversion. The disadvantage of voxelization is that it creates a stepped representation of any non-orthogonal surface.

The voxelization grid is based on the geometry, similar to the Simcenter Flotherm spatial solution grid, that is, it attempts to align with the faces of the selected geometry. All selected geometry is voxelized on the same grid, ensuring that no breaks in the geometry occur at coincident faces of different parts, see Figure 3-22.

Figure 3-22. Single Grid for Multiple Selected Parts

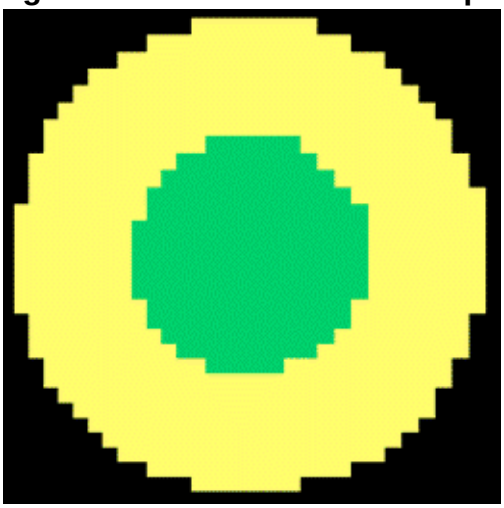


The grid is modified by whether you specify a Minimum Number of Cells or a Maximum Cell Size. You can also specify the Minimum Cell Size. It is possible to specify a grid that does not align with geometry faces and, therefore, produces Simcenter Flotherm geometry that is not

dimensionally accurate to the original MCAD model. A visual check can be made before confirming the conversion.

Voxelization creates cuboids that fill voxel cells provided that the centroid of the cell is found to be in solid. If the centroid of the cell is not in solid then no cuboid is created, see Figure 3-23.

Figure 3-23. Voxelization Example



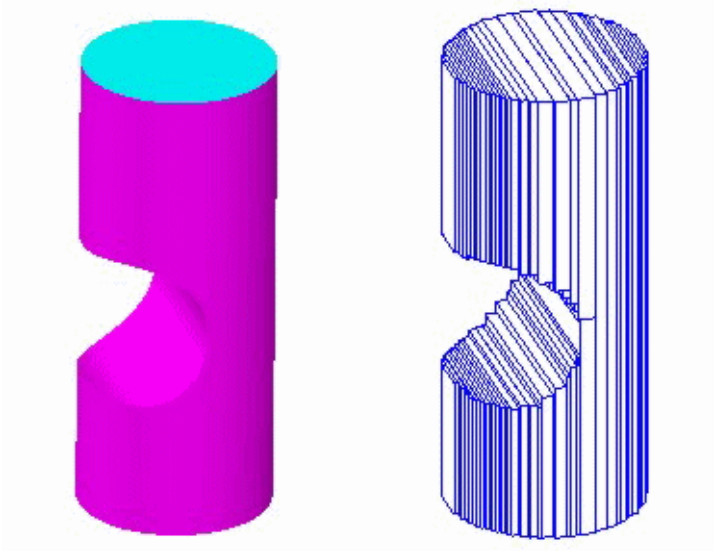
When the option to create collapsed cuboids is turned on, then collapsed cuboids are created for cells that do not have their centroids located in solid, but do have a solid face located in the cell.

Note

Although creating collapsed cuboids can increase translation accuracy in some cases, it adds significant processing time to voxelize, so it is turned off by default.

Neighboring cubes that share a common face are merged together to minimize the total number of cuboids needed to represent the geometry, see Figure 3-24.

Figure 3-24. Voxelization Merging of Cuboids



Related Topics

Voxelizing an MCAD Part

Voxelize Dialog Box

Simple Simplifying and Converting

This is a high-level one-step approach for decomposing the geometry into Simcenter Flotherm objects.

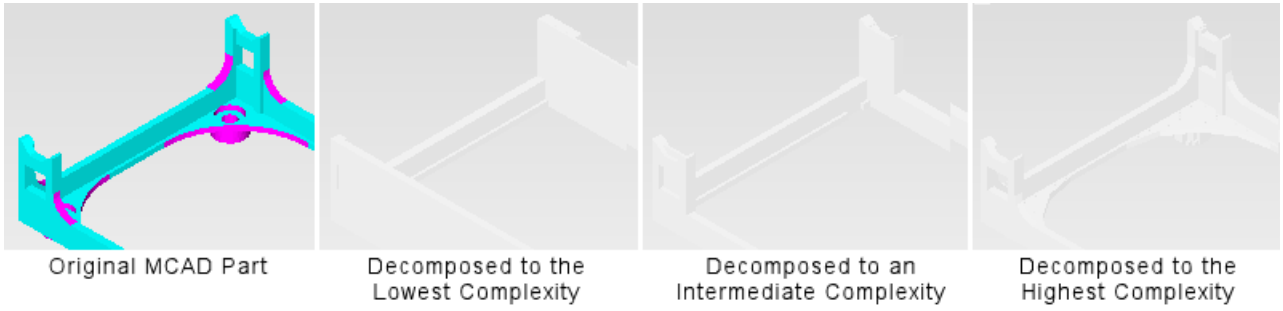
Procedure

1. Optionally, select a Part then click the **Decompose** icon or choose **Tools > Decompose** to open the Decompose dialog box.
2. Use the dialog box to simplify and dissect the selected MCAD Body or Part into Simcenter Flotherm objects according to the tolerances you define. If nothing is selected, then the entire MCAD Assembly is decomposed.

Results

Figure 3-25 to illustrates the effect of applying different levels of decomposition. In this example, the number of primitives created by the decompose process was 26, 87 and 247 respectively.

Figure 3-25. Varying the Level of Decompose Complexity



If this method does not give the desired result, then try splitting the body.

Related Topics

Decompose Dialog Box

Splitting an MCAD Body

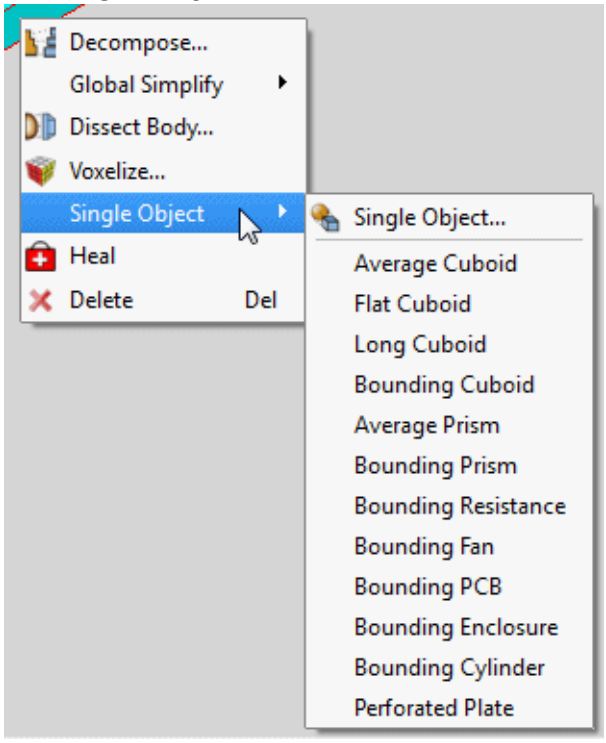
Replacing With a Single Object

Use this method to replace a selected MCAD Body with a single Simcenter Flotherm primitive or SmartPart.

Procedure

1. Select an MCAD Part or Body and click the **Single Object** icon  or choose **Tools > Single Object** to open the Single Object dialog box.
2. Select an object type for the replacement object and, if required, set a Collapse Below limit. If the Collapse Below value is left at the default 0 (zero) then no faces will be collapsed.
3. Click **OK** to replace the selected object.
4. Alternatively, select a part or body and right-click to display the context-sensitive menu, see Figure 3-26.

Figure 3-26. Single Object Context-Sensitive Menu Options



5. The **Single Object** > **Single Object** option opens the Single Object dialog box. The other options replace the selected object with the respective primitive or SmartPart.

Results

The data tree will display the relevant Simcenter Flotherm icon for the replacement. Primitives are displayed in dark blue and SmartParts are displayed in dark green in the GDA.

Figure 3-27, Figure 3-28 and Figure 3-29 show some examples of simplifying an MCAD Part to a single Simcenter Flotherm object.

Figure 3-27. Replacement Cuboids

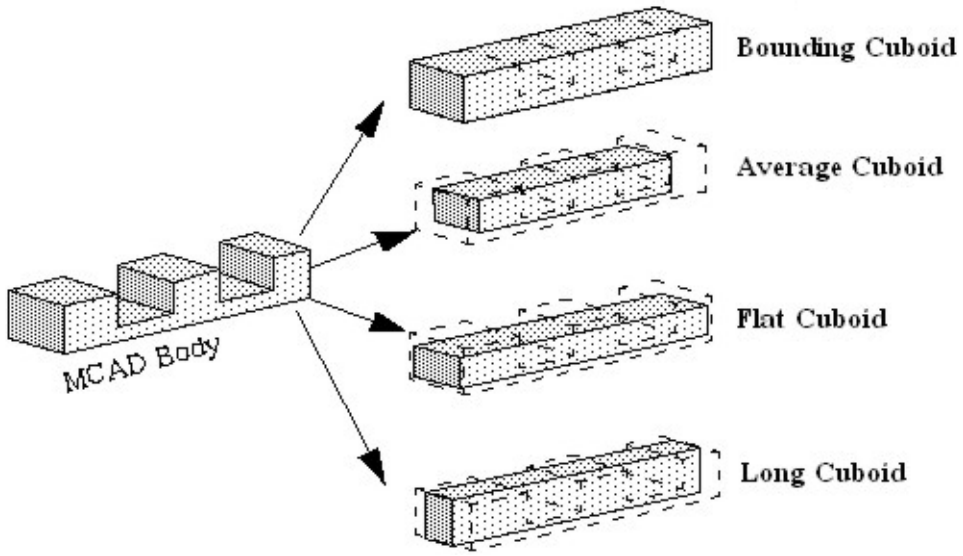


Figure 3-28. Replacement Prisms

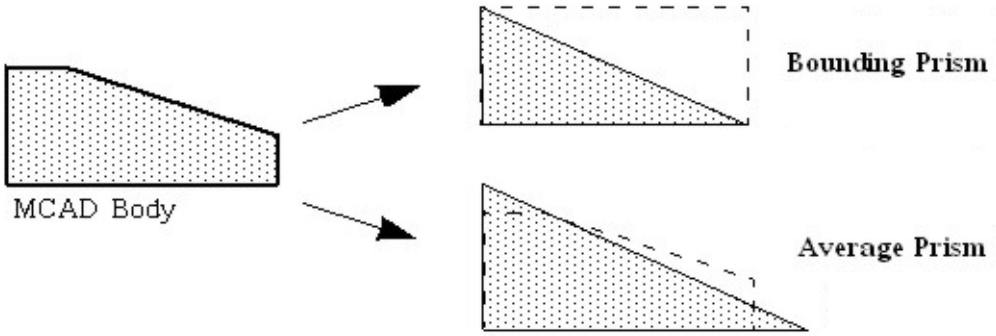
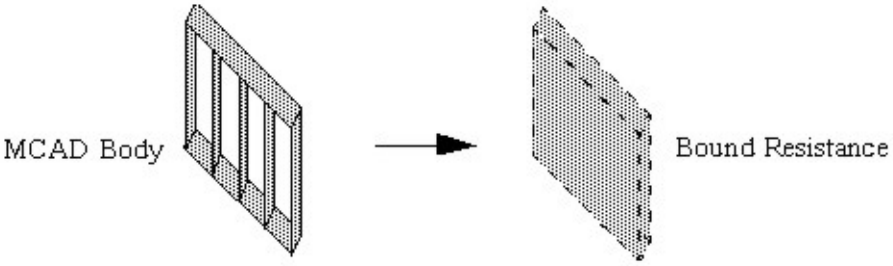


Figure 3-29. Replacement Resistance



Related Topics

Single Object Dialog Box

Dissecting a Body Based on Tolerances

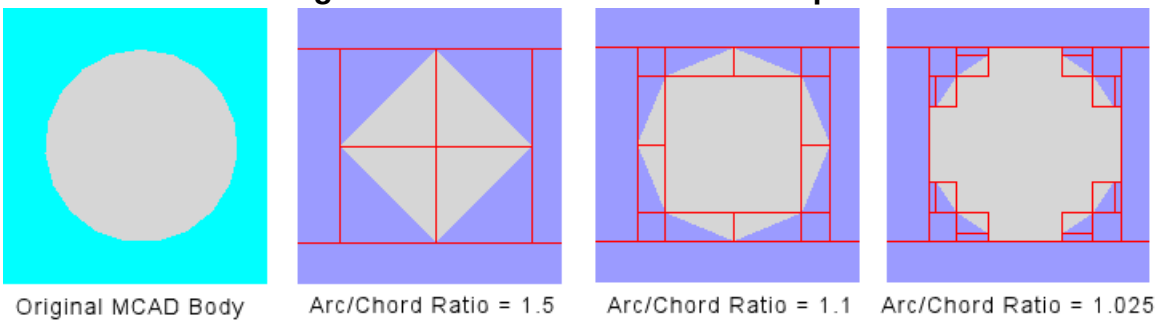
Dissect splits the MCAD Body of an MCAD Part into several smaller MCAD Bodies based on specified tolerances.

Procedure

1. Select an MCAD Part or Body and click the **Dissect Body** icon or choose **Tools > Dissect Body** to open the Dissect Body dialog box.
2. Specify tolerances to decide whether to approximate these new MCAD Bodies with MCAD primitives or to further split them into more MCAD Bodies as follows:
   - Set Minimum Volume Tolerance % to delete MCAD bodies that are less than this value.
   - Set Empty Body Tolerance % to replace MCAD bodies with less than this value with a cuboid.
   - Set Full Body Tolerance % so that MCAD bodies greater than this tolerance are approximated to MCAD primitives

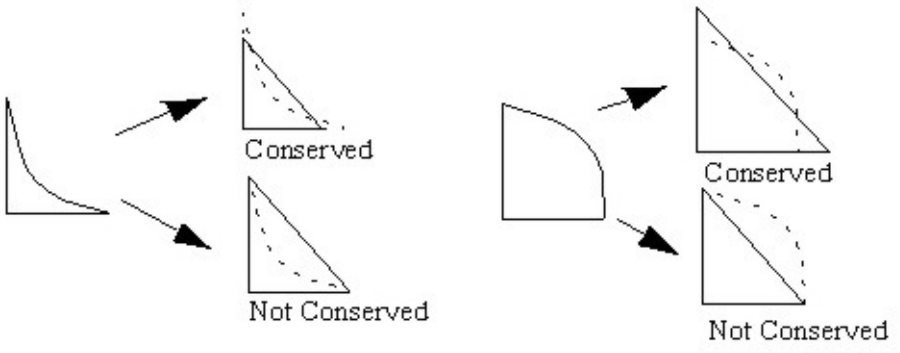
   MCAD Bodies greater than Empty Body Tolerance % and less than Full Body Tolerance % are approximated to MCAD prism primitives.
3. Set an Arc/Chord Ratio to approximate curves using cuboids and prisms. Non-planar MCAD Bodies with an arc/chord ratio greater than that specified are sliced at their mid-points, see Figure 3-30. The most accurate representations are achieved using an Arc/Cord Ratio of between 1.01 and 1.001.

Figure 3-30. Arc/Chord Ratio Examples



4. When dissecting 2D faces into collapsed cuboids, ensure the resulting thickness in the collapse direction is set using **Collapse Below**. This is important for the preservation of through and in-plane thermal resistance.
5. Check the Conserve Volume check box to conserve volume, see Figure 3-31.

Figure 3-31. Conserved and Non-Conserved Volume Replacement



Results

Figure 3-32 to Figure 3-35 shows some examples of the Dissect Body tool.

Figure 3-32. Dissected Body

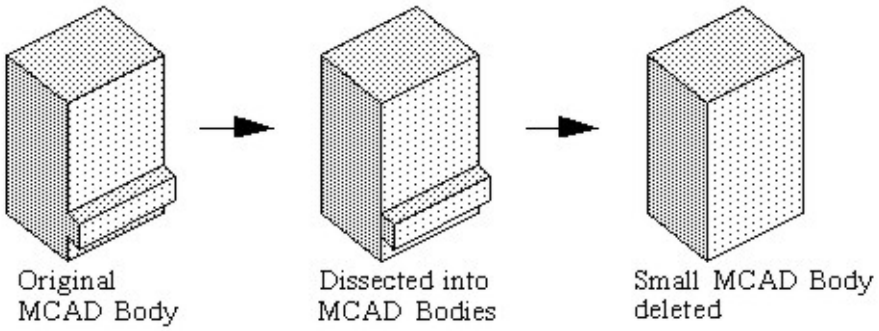


Figure 3-33. Empty Body Replacement

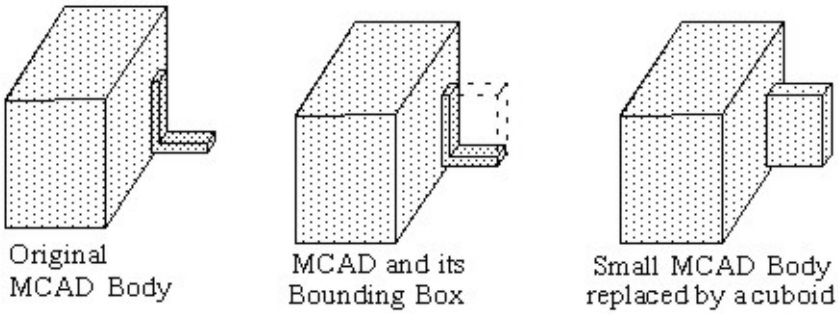


Figure 3-34. Full Body Replacement

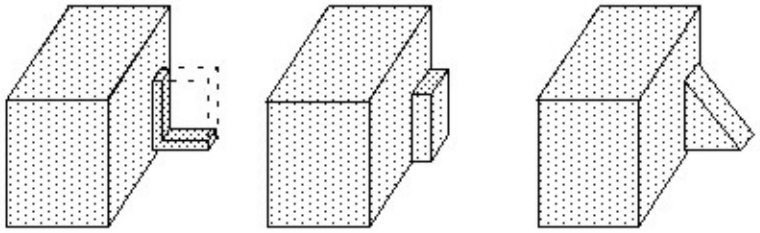


Figure 3-35. Sliced Curve

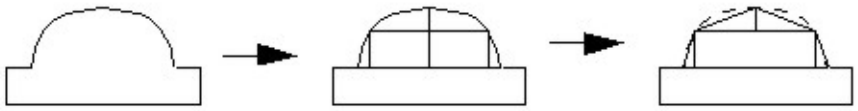
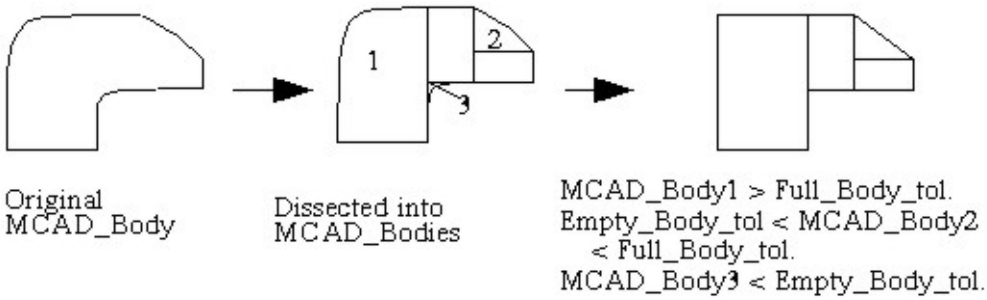


Figure 3-36 shows a dissected MCAD Body where one MCAD Body less than Empty Body Tolerance is deleted, and one MCAD Body greater than Full Body Tolerance is replaced with a cuboid.

Figure 3-36. Example of Dissected Body



Related Topics

Simplifying MCAD Parts or Bodies

Dissect Body Dialog Box

Voxelizing an MCAD Part

Voxelizing, or dissecting into cuboids based on a grid, is best suited to predominantly cartesian geometry.

Procedure

1. Select one or more MCAD parts and click the **Voxelize** icon or choose **Tools > Voxelize** to open the Voxelize dialog box.

   If more than one part is selected then only one grid (mesh) is created, ensuring that no breaks in the geometry will occur for coincident faces on different parts.

2. Set the Minimum Cell Size value to be equal to the smallest geometric feature that requires representation.

3. Define the mesh by either a Minimum Number of cells or a Maximum Size of cell.

4. It is recommended that you check the Show Preview Mesh check box in the Voxelize dialog box so that you can see the mesh and the effect that changes these values has on the mesh.

The mesh is displayed normal to one of the axes. You can change the view of the mesh either by:

- using the **Preview Normal** selector in the Voxelize dialog box, or
- using the keyboard View shortcuts X, Y and Z. This method changes both the view of the geometry and the view of the mesh, such that both views are along the same axis.

5. To create collapsed cuboids, check the Create Collapsed Cuboids check box. Although this may increase translation accuracy in some cases, it adds significant processing time to voxelize.

   When voxelizing into collapsed cuboids, make sure that the resulting thickness in the collapse direction is set using the 2D Face Collapse Thickness data entry field. This is important for the preservation of through and in-plane thermal resistance.

6. When you are satisfied with the mesh, click **OK** to voxelize the selected part.

   You can undo the process.

Results

Note

After transfer to Simcenter Flotherm, it is sometimes advisable to localize and inflate the assembly grid to minimize the total number of grid cells in the model.

Related Topics

Voxelization

Voxelize Dialog Box

Changing Names and Checking for Full Conversion

The MCAD Bridge data tree enables part selection, similar to the Simcenter Flotherm Project Manager.

Procedure

1. Select the assembly.

   The property sheet the name associated with the assembly in the display area and this is editable.

2. In addition, the data tree lists all the assembly component parts and can be used to check for full realization of the assembly into Simcenter Flotherm objects. To select an item, click on an item in the data tree.

   The item is selected in the GDA and its property sheet is opened. For MCAD parts, the property sheet provides realization and volume information for the selected part.

Results

A fully realized assembly is indicated by Simcenter Flotherm assembly icons as shown in Figure 3-37.

Figure 3-37. MCAD Parts Before and After Conversion

Related Topics

MCAD Bridge Property Sheet

Transferring a Converted MCAD Assembly

Fully-realized objects can be transferred at any time, building the Simcenter Flotherm project incrementally.

Caution

A “Transfer Assembly” operation transfers the remaining realized objects to join the previous transferred assembly. Ensure you do not delete the destination assembly structure before the transfers of all required parts are complete.

Procedure

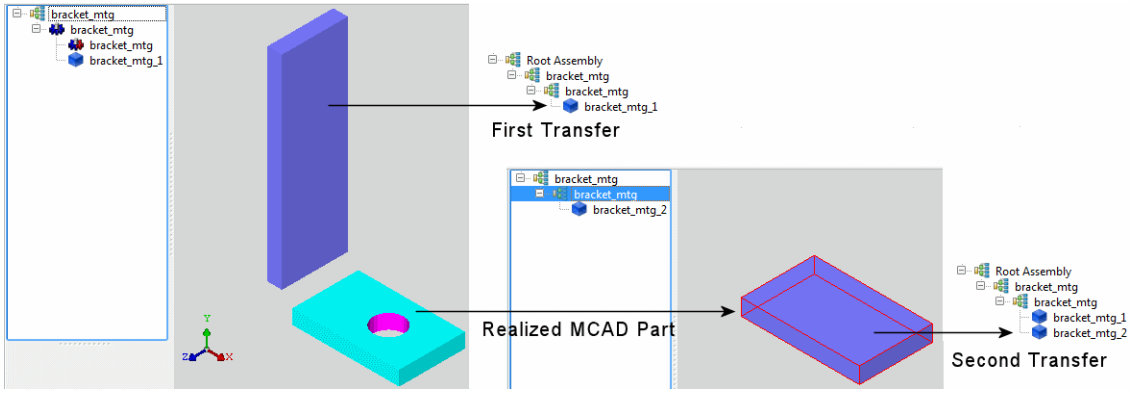
1. Select the assembly to be transferred.
2. Click the Transfer Assembly icon or choose **Tools > Transfer Assembly**.

Results

Any fully-realized Simcenter Flotherm objects are transferred into the current Simcenter Flotherm project, leaving only unrealized MCAD parts on display in the MCAD Bridge window. The realized parts are transferred to a Simcenter Flotherm assembly, named after the original MCAD assembly.

If any MCAD parts remain, then they can be converted to Simcenter Flotherm format and then transferred to join the previously transferred assembly, maintaining the original MCAD assembly structure, see Figure 3-38.

Figure 3-38. Transferring an Assembly - One Part at a Time



Export

Use the following procedures when exporting MCAD files.

Supported Export File Formats

MCAD Bridge can write common MCAD file formats.

Table 3-4 lists the file formats that can be exported by MCAD Bridge.

Table 3-4. Export File Formats

File Format	File Extension	Versions Supported
CATIA V4	**.model*	4.1.9
IGES	**.igs*	5.3
ACIS	**.sat*	R27, R28, R29 or 2020 1.0
STEP	**.stp* or **.step*	AP214 (Geometry only)

Exporting Project Geometry

Using MCAD Bridge to transfer Simcenter Flotherm project geometry to an external file system.

Restrictions and Limitations

- See "Supported Export File Formats" on page 71.
- Simcenter Flotherm assembly hierarchy is flattened, that is, all geometry from Simcenter Flotherm become children of the MCAD assembly.

Prerequisites

- The project geometry is in Simcenter Flotherm Project Manager.
- Optional. To view detected errors while exporting MCAD files, then check the Report ACIS Model Errors to Message Window check box in the Preferences dialog box.
- Optional. To create a log file when writing STEP, CATIA, or IGES files, set the FLOMCADKEEPLOGFILES environment variable to TRUE and restart Simcenter Flotherm.

  Log files are written to:

  C:\Users\<username>\AppData\Local\Temp

Log file names include the exported file type and date stem-name (*mm-dd-hh-mm-ss*), for example:

STEP_WRITE-05-04-16-35-39.log

Procedure

1. In MCAD Bridge, choose **External > Import Project Geometry**. If there is an existing model in MCAD Bridge, then you will be asked to overwrite that model.

   Project geometry in Simcenter Flotherm Project Manager is read into MCAD Bridge and converted to ACIS geometry.

2. You have a choice:

If you want to...	Do the following:
Export the MCAD assembly in SAT file format.	1. Choose **External > Export SAT > ACIS <*version*>**. 2. Use the Export ACIS dialog box is opened to navigate to a folder and save to a **.sat* file. The exported file does not include the assembly hierarchy, object names or units.
Export the MCAD assembly in IGS file format.	1. Choose **External > Export IGES**. 2. Use the Export IGES dialog box to navigate to a folder and save to an **.igs* file. The exported file does not include the assembly hierarchy, object, or names.
Export the MCAD assembly in STEP file format.	1. Choose **External > Export STEP**. 2. Use the Export STEP dialog box to navigate to a folder and save to an **.stp* file. The exported file does not include the assembly hierarchy, object names, or units.
Export the MCAD assembly in CATIA V4 file format.	1. Choose **External > Export Catia V4**. 2. Use the Export Catia V4 dialog box to navigate to a folder and save to a **.model* file. The exported file does not include the assembly hierarchy, object names or units.
Export the MCAD assembly in STL file format.	1. Choose **External > Export STL**. 2. Use the Export STL dialog box to navigate to a folder and save to an **.stl* file.

3. If you have set FLOMCADKEEPLOGFILES to TRUE for logging, then reset this environment variable to FALSE when you are finished to prevent creating unwanted temporary files when exporting STEP, CATIA, or IGES file.

MCAD Bridge Import and Convert Dialog Boxes

MCAD Bridge uses the following dialog boxes when realizing MCAD objects.

MCAD Bridge Monitor Dialog Box

To access: Automatically displayed at the start of the import operation. If closed, the dialog box can be re-instated by choosing **Window > Monitor Dialog**, or by pressing Ctrl+M.

This dialog box reports the progress of the importing and conversion operations.

Objects

- Each operation is named and the MCAD Body to which it applies is indicated inside angled brackets. The following message conventions are used:
    - Dots — operation in progress.
    - C, P, or Y — cuboid, prism, or cylinder creation respectively.
    - F — ACIS operation failed.
    - S — small MCAD Body discarded from model.
    - U — unclassified MCAD Body left after an interrupt now added to part as a new Body.

Usage Notes

For machines supporting multiple threads, the dialog box includes a progress bar which fills proportionately as the operation approaches completion.

You can cancel the file import by clicking **Interrupt**.

Alternatively, you can skip the current operation by clicking **Skip**.

Related Topics

Import

Preferences Dialog Box

To access: **Edit > Preferences**

Use this dialog box to customize the basic MCAD Bridge setup.

Objects

Field	Description
Import Healing Level	Sets the level to which Simcenter Flotherm attempts to build a solid (manifold) body. This happens automatically when importing an MCAD model into MCAD Bridge, and by users choosing **Tools > Heal**. • Full — moves faces, edges, and vertices to close up all gaps to perform a full heal on the model. Robust, but slow[1]. The default. • Stitch Only — uses tolerant modeling techniques, that is, edges and vertices are shuffled around until close enough together. Faster than Full, but not as robust. • None — no healing is done.
Graphics Faceting Level	Sets the faceting level for the graphics to LOW, MEDIUM, or HIGH. The higher the accuracy (number of facets) the lower the performance. If you are importing complicated geometry, it may be necessary to restrict the number of facets by using the LOW option.
Report ACIS Model Errors to Message Window	Provides two options for handling the ACIS model errors. When activated, it causes the error messages to be output to the Message Window; otherwise they will be ignored and, therefore, not displayed.
Highlight Non Planar Faces	Color non-planar faces in magenta.
Gradient Background	Sets the background of the GDA to be merged from the top background color to the bottom background color. If unchecked, the GDA background is a uniform color.
Background Color	The background color of the GDA.
Top Background Color	(Gradient Background) The background color at the top of the GDA.
Bottom Background Color	(Gradient Background) The background color at the bottom of the GDA.

1. On balance, the quality of the resulting manifold object as a result of Full healing is much higher, for the time taken, than Stitch Only healing.

Usage Notes

The settings made in the MCAD Preferences dialog box are saved to a configuration file named *username-useridMCAD.cfg* held in the directory:

<install dir>/flosuite_v<version>/flotherm/config

Related Topics

Importing MCAD Files Into MCAD Bridge

Using the Healing Tool to Join Entities to Create a Solid Object

Global Simplify Dialog Box

To access: **Tools > Global Simplify** or click the **Global Simplify** icon

Use this dialog box to simplify the complete selected MCAD Body or part in preparation for its conversion to Simcenter Flotherm objects.

Description

A number of different simplification techniques that can be applied. These techniques are selected by check box and operate on features that are smaller than the defined tolerances.

Objects

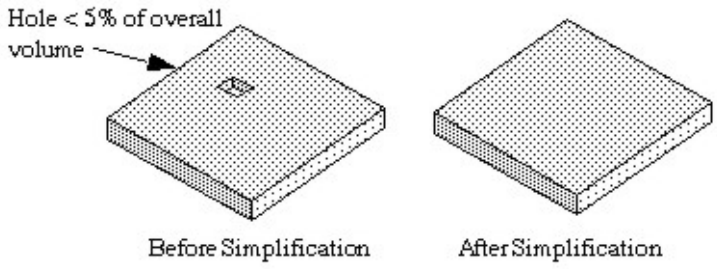
Field	Description
Tolerance Fields	
Small Hole Tolerance %	Small hole tolerance as the percentage of the volume of the bounding box of the hole to the original MCAD Body volume.
Small Hump Tolerance %	Small hump tolerance as the percentage of the volume of the bounding box of the hump to the original MCAD Body volume.
Small Faces Tolerance %	Small face tolerance as the percentage of the bounding box volume of either a single or a pair of neighboring faces to the volume of the MCAD Body.
Small Draft Angle	Small draft angle of slope tolerance.
Near Level Tolerance	Near level tolerance as the percentage of the diagonal length of the MCAD Body's bounding box.
Check Boxes	
Remove Small Holes	Remove holes, from the surface of the body, that are smaller than the Small Hole Tolerance %.
Remove Small Humps	Remove humps, from the surface of the body, that are smaller than the Small Hump Tolerance %.
Remove Small Face Groups	Removes a face or pair of neighboring faces that are below the Small Faces Tolerance %.

Field	Description
Remove Non Planar Faces	Remove non-planar faces. This function first attempts to remove non-planar faces and ‘grow’ the remaining faces to remake the solid as shown in Figure 3-6. If this method cannot be used, then a block is created from the bounding box and is either united with or subtracted from the MCAD Body. The decision as to whether to unite or subtract is based on the curvature of the surface. Figure 3-7 shows a non-planar face being squared. **Note:** You can “mark” non-planar faces, using the Local Simplify Dialog Box, so that they will not be simplified when using Remove Non Planar Faces.
Flatten Small Drafts	Flatten planar faces not aligned with an axis having slopes less than the small draft angle. This function first attempts to rotate the face and make it flat by ‘growing’ and ‘shrinking’ the surrounding faces to remake the solid as shown in Figure 3-8. If the above method cannot be employed, then a draft face is flattened into a box half of the height of its bounding box. The direction of the height is taken to be the average of the surface normals for all vertices of the face, snapped to the nearest coordinate axis. Figure 3-9 shows a small draft face being flattened.
Level Near Level Faces	Level faces that are within the near level tolerance. Two faces are considered ‘Near Level’ if they: • are within a small distance of each other in their normal directions, • overlap in another coordinate direction, • touch in the last coordinate direction. The small distance is defined by the near level tolerance, which is a percentage of the length of the diagonal of the bounding box of the MCAD Body. This percentage can be set by the user. The global simplification of the near level faces involves finding each of the pairs of near level faces. The face with the smallest area is then moved up or down to the level of the larger face, see Figure 3-10.

Usage Notes

The size percentage values entered in this dialog box relate to the overall MCAD Body volume. Figure 3-39 shows the removal of a small hole during simplification when the small hole tolerance is set to 5%.

Figure 3-39. Small Hole Global Simplification



Related Topics

Using Global Simplify to Automatically Simplify Features and Faces

Simplifying MCAD Parts or Bodies

Local Simplify Dialog Box

To access: **Tools > Local Simplify** or click the **Local Simplify** tool icon

Use this dialog box to simplify individually selected faces or features.

Objects

Field	Description
Add Bounding Box	Active when faces or features are selected. Adds a bounding box of either the selected faces or each of the selected features to the owning MCAD Body. The shortcut key is '+'
Subtract Bounding Box	Active when faces or features are selected. Removes the bounding box of either the selected faces or of each of the selected features from the owning MCAD Body. The shortcut key is '-'
Remove	Active when faces or features are selected. Removes the selected face or feature. If a non-planar face is selected and removed, the neighboring faces are 'grown' to replace it. This operation works most effectively with selected features. The shortcut is the Delete key.
Flatten	Active when faces are selected. Flattens each of the selected faces. The shortcut key is F.
Level	Active when two or more faces are selected. Levels all selected faces to that of the first selected. To create a manifold object, it may have to 'grow' all the surrounding faces. To select multiple faces, press Ctrl and click each face. The shortcut key is L.
Non-Planar Faces	
Mark	Active when one or more faces are selected. Marks any selected non-planar face. Marked faces are excluded from
Un-Mark	Active when a face is selected. Un-marks any selected and marked non-planar face.

Related Topics

Using Local Simplify to Simplify Selected Features and Faces

Split Body Dialog Box

To access: Select an MCAD Body then choose **Tools > Split Body** or click the **Split Body** icon 

Use this dialog box to split a MCAD Body, for example, when you want to work on a small section independently.

Objects

Field	Description
Select Face	Enables you to select a face of the body. The cursor changes to cross-wires as the mouse moves across the display area.
Select Vertex	Enables you to select a vertex of the body. The cursor changes to cross-wires as the mouse moves across the display area.
Direction	X, Y or Z
Position X, Y or Z	• Automatically set if Select Face or Select Vertex are used. • Manually use the slider bar or entry field to specify the position of the plane where the split is to be made.

Usage Notes

Click **Slice** to mark where a cut is to be made.

Click **Split** to split the body along the slice lines.

Related Topics

Splitting an MCAD Body

Extrude Dialog Box

To access: **Tools > Extrude**

Use this dialog box to extend a selected MCAD Face or Edge.

Description

If an MCAD Face is selected, the direction of extension is normal to the face. If an MCAD Edge is selected, the direction of extension is selectable.

Objects

Field	Description
Depth	The distance that the selected face or edge will be extruded.
Direction	(MCAD Edge) The axis direction (+ or -, X, Y or Z) in which the edge will be extruded.

Related Topics

Extruding MCAD Geometry Faces or Edges

Decompose Dialog Box

To access: **Tools** > **Decompose** or click the **Decompose** icon 

Use this dialog box to decompose (convert) the selected MCAD Part or Body into Simcenter Flotherm objects. If nothing is selected, then the entire MCAD Assembly is decomposed.

Note

Decompose both simplifies AND dissects in a single operation.

Objects

Field	Description
Complexity Level	Set the tolerances for the decomposition. The position of the slider bar determines the complexity of the conversion from the MCAD Assembly to Simcenter Flotherm objects. The further the slider moves to the right, the greater the number of Simcenter Flotherm objects are generated. The program will automatically set up tolerances according to the slider bar setting, from 0 (least complex) to 20 (most complex).
Collapse Below	The thickness below which Cuboids and Sloping Blocks are collapsed.
Prisms	Use prism primitives to represent MCAD geometry. Prisms are best suited to represent spherical and hemispherical geometries.
Cylinders	Convert cylindrical shapes into Simcenter Flotherm Cylinder SmartParts with a default number of facets. Use when cylindrical shapes are important. You can change the faceting level of the Cylinder SmartPart within Simcenter Flotherm without re-importing data from the MCAD package.
Sloping Blocks	Convert angled plates or walls into Simcenter Flotherm Sloping Block SmartParts.

Related Topics

Converting MCAD Geometry

Simple Simplifying and Converting

Single Object Dialog Box

To access: Select an MCAD object then choose **Tools > Single Object** or click the **Replace with Single Object** tool icon 

Use this dialog box to replace the selected MCAD Body, Part, Bounding box of any selected Face(s) or Feature(s) with a single cuboid, prism, resistance, fan, PCB, enclosure, cylinder, or perforated plate.

Description

For a selected Face or Feature, the bounding box volume of the Face or Feature will be removed before inserting the selected single object.

Objects

Field	Description
Type The replacement object type. Select from the following:	
Average Cuboid	Replace with a single cuboidal primitive that has the same volume and length ratio as the bounding box of the original object, see Figure 3-27.
Flat Cuboid	Replace with a single cuboidal primitive that has the same volume as the original object. The cuboid's two longest side lengths are the same as those of the bounding box of the original object, see Figure 3-27.
Long Cuboid	Replace with a single cuboidal primitive that has the same volume as the original object. The cuboid's longest side length is the same as that of the bounding box of the original object, see Figure 3-27.
Bounding Cuboid	Replace with a single cuboidal primitive that has the same dimensions of the bounding box of the original object, see Figure 3-27.
Average Prism	Replace with a single prism primitive that has the same volume as the original object, see Figure 3-28.
Bounding Prism	Replace with a single prism primitive that is half the bounding box of the original object, see Figure 3-28.
Bounding Resistance	Replace with a resistance occupying the same space as the object's bounding box, see Figure 3-29.

Field	Description
Bounding Fan	Replace with a 3D 12 facets, simple 2D, or rectangular fan, depending on the two longest lengths of the bounding box of the MCAD Body or Part and the Collapse Below value. • 3D 12 Facets Fan — requires that the two longest lengths of the bounding box of the body/part differ by less than 10% and the shortest length is greater than the Collapse Below value. The geometric center of the fan will be placed coincident with the geometric center of the body/part it is replacing. The Zo depth of the fan will be the same as the body/part shortest length direction. • Simple 2D Fan — requires that the two longest lengths of the bounding box of the body/part differ by less than 10% and the shortest length is less than the Collapse Below value. The geometric center of the fan will be placed coincident with the geometric center of the body/part it is replacing. The Zo depth of the fan will be the same as the body/part shortest length direction. • Rectangular Fan — requires that the two longest lengths of the bounding box of the body/part differ by more than 10%. The Zo location of the fan will be the middle of the shortest length of the body/part.
Bounding PCB	Replace with a single conducting board of the same bounding box size and location. Fr4 and Cu at 10% conductor by volume will be set. The serials of FR4 and Cu will be the same as that in the supplied Simcenter Flotherm materials library.
Bounding Enclosure	Replace with a six-sided enclosure of the same bounding box size and location. The enclosure will be Thick, with a thickness of 2.0 mm.
Bounding Cylinder	Replace with a cylinder. The cylinder's local z-coordinate will be in the direction of the longest length of the body's bounding box. The cylinder's radius will be half the length of the middle sized side of the bounding box. The cylinder will be conducting, and have 12 facets.
Perforated Plate	Replace with a perforated plate SmartPart. The local z-coordinate of the perforated plate will be in the direction of the shortest length of the bounding box.
Collapse Below	

Field	Description
Collapse Below	(Average, Bounding, Flat and Long Cuboids, and Bounding Resistance) Sets the limit below which the length of a side of the replacement object is collapsed. (Bounding Fan) Sets the limit below which the length of a side of the replacement object is collapsed to create a 2D Fan, see Bounding Fan above.

Related Topics

Replacing With a Single Object

Dissect Body Dialog Box

To access: **Tools > Dissect Body** or click the **Dissect Body** tool icon.

Use this dialog box to define how selected MCAD Bodies are dissected into Simcenter Flotherm primitives, cylinder SmartParts or sloping block SmartParts, working within given tolerances.

Note

The FLO_OVERCOMPLEX_CELL_LIMIT environment variable sets a limit to the dissection of geometry.

Objects

Field	Description
Minimum Volume Tolerance %	The minimum volume tolerance for MCAD dissection, specified as the percentage volume of the MCAD body being dissected. Figure 3-32 shows an MCAD part being dissected with a small MCAD body falling below this value.
Empty Body Tolerance %	The percentage of the volume of an MCAD Body to the volume of its bounding box. MCAD bodies less than Empty Body Tolerance % are replaced with a cuboid, see Figure 3-33.
Full Body Tolerance %	The percentage of the volume of an MCAD body to the volume of its bounding box. MCAD bodies between Empty Body Tolerance % and Full Body Tolerance % are approximated with prism MCAD primitives. MCAD bodies larger than Full Body Tolerance % are approximated with cuboidal MCAD primitives.
Arc/Chord Ratio	Determines whether to slice non-planar faces at the mid point of the curve. This value represents the ratio of the arc length of a curve on the face to the chord length of that curve. For faces with values above this ratio, mid-point slices are performed as shown in Figure 3-35.
Collapse Below	Any cuboids or sloping blocks that are thinner than this value are collapsed.
Conserve Volume	Volume is conserved for an MCAD body approximated to a prism.
Prisms/Tets	Translate prisms and tets. When unchecked, prisms or tets are not generated (see Usage Notes).
Sloping Blocks	Replace angled plates or walls with Simcenter Flotherm default sloping block SmartParts.

Field	Description
Cylinders	Replace cylindrical parts with Simcenter Flotherm default cylinder SmartParts, that is, cylinders with 16 facets. When dissecting without cylinders, any cylinders that would have been created are dissected into cuboids and prisms according to the Arc/Chord Ratio.

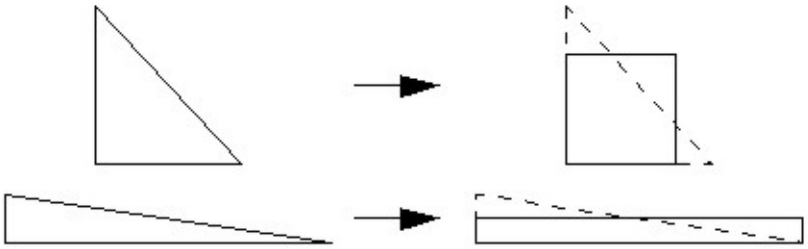
Usage Notes

Representation of Prisms by Cuboids

When the Prisms/Tets option in the Dissect Body dialog box is unchecked, then any would-be prisms are represented by one of these two methods:

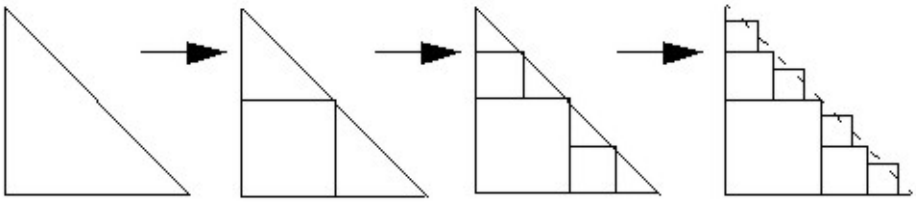
1. If a prism is smaller than the Empty Body Tolerance % of the original MCAD Body, it is replaced with a cuboid of the same volume.

Figure 3-40. Prism Smaller Than Empty Body Tolerance



2. If a prism is larger than the Empty Body Tolerance % of the original MCAD Body, then it is dissected into two prisms and a cuboid recursively, until the prisms are smaller than the Empty Body Tolerance %, then these are replaced by single cuboids as above.

Figure 3-41. Prism Larger Than Empty Body Tolerance



Related Topics

Dissecting a Body Based on Tolerances

Voxelize Dialog Box

To access: **Tools** > **Voxelize** or click the **Voxelize** icon.

Use this dialog box to define a dissection mesh by specifying limits to mesh cell numbers or size, and to dissect an MCAD part in accordance with the mesh.

Objects

Field	Description
Define Mesh By	The method of defining the X, Y and Z grid over the geometry: Minimum Number or Maximum Size. The grid defined by these settings is in addition to the default grid based on every vertex in the model.
Minimum Number of Cells in X, Y, Z	(Minimum Number) The minimum number of cells in each coordinate direction. The default is 1.
Maximum Size of Cells in X, Y, Z	(Maximum Size) The maximum size of cells in each coordinate direction. The default is 10 mm.
Minimum Cell Size	A minimum cell size can be used to ensure no extremely small cuboids are created by the Voxelization method. This in itself is a type of simplification in that no geometric feature smaller than this minimum cell size will be represented. The default is 1 mm.
Create Collapsed Cuboids	When checked, the voxelize process detects cells that have MCAD faces but are not completely solid, and builds collapsed cuboids as needed. Un-checked by default, to reduce the processing time.
2D Face Collapse Thickness	(Create Collapsed Cuboids) The thickness of the collapsed side for any collapsed cuboids created by the voxelize process. This is equally relevant when voxelizing geometry combining 2D faces. The default is 2 mm.
Show Preview Mesh	Show the voxelize mesh in the GDA. This enables you to preview the amount of voxelization.
Preview Normal	(Show Preview Mesh) The preview mesh is shown normal to this direction (X, Y or Z). **Note:** The preview mesh is synchronized with the GDA view direction when switching between X, Y and Z direction views so that it remains visible.

Related Topics

Voxelizing an MCAD Part

Voxelization

Input File has no Units Definition Dialog Box

To access: Displayed when importing an MCAD file that does not have units defined. For example, when importing STL files or older SAT and DXF files.

Use this dialog box to specify length units for the imported MCAD file.

Objects

- Length

  Select the appropriate unit of length for the imported model.

Related Topics

Importing MCAD Files Into MCAD Bridge

Importing DXF Files Into MCAD Bridge

Message Window

To access: Check the Report ACIS Model Errors to Message Window check box in the Preferences dialog box.

This window reports MCAD Bridge error, warning, and information messages.

Objects

Field	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.
Clear 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 GDA, double-click the title bar.

Related Topics

Importing MCAD Files Into MCAD Bridge

MCAD Messages (12000 Series) [Simcenter Flotherm User Guide]

Chapter 4
Creating MCAD Files Using External Software

Basic advice for creating External MCAD files ready to import into MCAD Bridge.

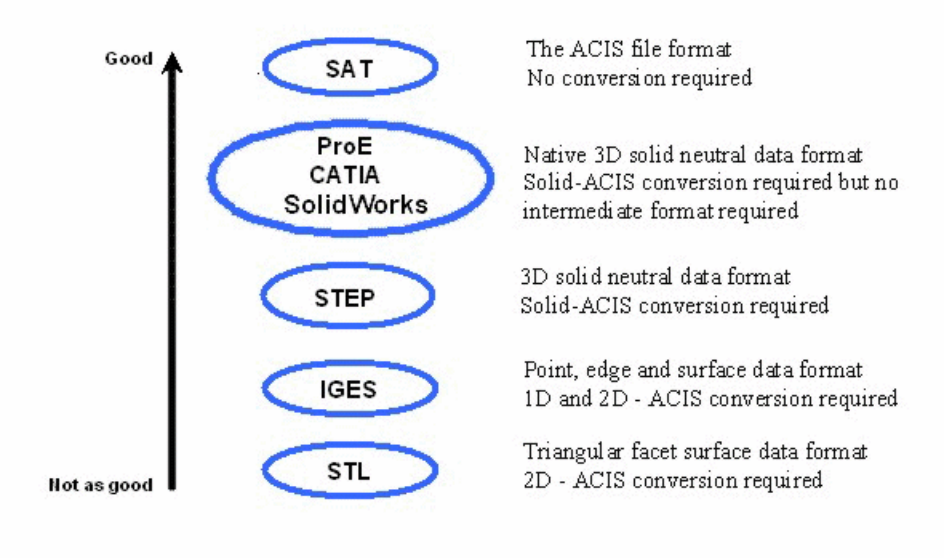
Recommended Neutral File Format

The preferred neutral file formats for robust and error free import and manipulation within MCAD Bridge.

Preferred file formats are based upon a combination of translation, healing, and subsequent simplification and dissection concerns.

Figure 4-1 shows, on balance, the preferred file types.

Figure 4-1. Preferred File Types



If neither SAT or STEP formats are available, then the instructions for the generation of IGES data must be followed.

If Pro/ENGINEER **.prt* or **.asm* files up to release Creo 2.0 are available, then these should be used in preference to the STEP, IGES, or STL files. The same is true for CATIA V4 files. By definition these files need no manual export from their MCAD tool.

Reduce the Geometric Complexity of the Model

MCAD Bridge has some methods for reducing the geometric complexity of models but *it is recommended that as much unnecessary detail as possible be removed from the model before exporting it from the generating MCAD package*.

Simplification involves removing small Features and Parts that are unimportant to the Simcenter Flotherm calculation. This will result in smaller files with faster translation and dissection in the MCAD Interface.

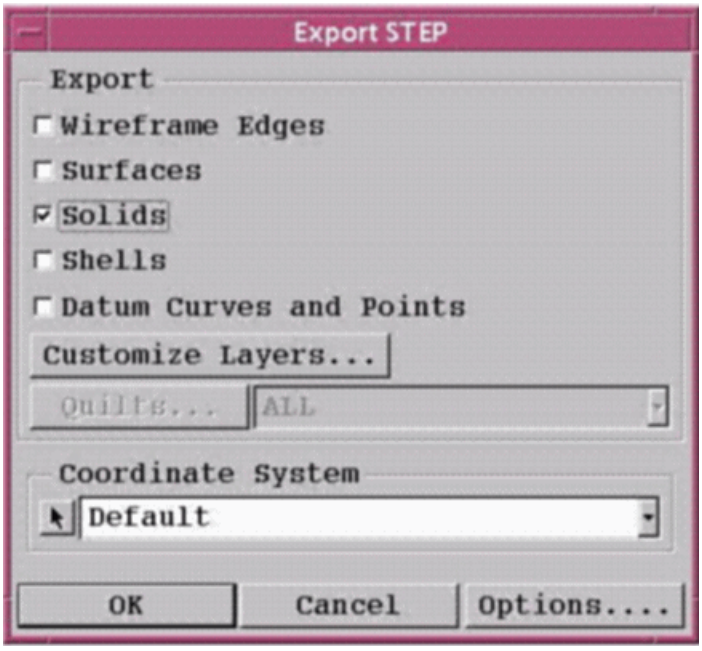
Exporting STEP Files From Pro/ENGINEER

Use the following procedure to export STEP files from Pro/ENGINEER for import into MCAD Bridge.

Procedure

1. Choose **File > Export > STEP** to display the Export STEP dialog box (Figure 4-2).

Figure 4-2. Export STEP Dialog Box



2. Accept the default filename
3. Ensure that the Export STEP dialog box has Solids selected and that Shells is not selected (it is by default).

Preparing and Exporting IGES Files From Pro/ENGINEER

Use these procedures when exporting IGES files for use in MCAD Bridge.

Changing the Accuracy of a Pro/ENGINEER Part

Before importing models into MCAD Bridge, regenerate each Part of the model in Pro/ENGINEER to an accuracy of 1e-6.

Procedure

1. Add the following line to the Pro/ENGINEER configuration file:

   ACCURACY_LOWER_BOUND 0.000001

2. Start Pro/ENGINEER
3. Select Set Up from the PART menu
4. Select Accuracy from the PART_SETUP menu
5. Enter the value 0.000001.

Results

This will cause the part to be regenerated. For more information see ‘CHANGING PART ACCURACY’ in the *Pro/ENGINEER Part Modeling User’s Guide*.

Reducing the Geometric Complexity of a Pro/ENGINEER Model

Reduce the complexity of models in Pro/ENGINEER to speed up the export of models to Simcenter Flotherm.

Procedure

Remove small Features and Parts that are unimportant to the Simcenter Flotherm calculation.

> A method of managing simplified models is provided in Pro/ENGINEER. For more information see ‘SIMPLIFIED REPRESENTATIONS’ in the *Pro/ENGINEER Assembly Modeling User’s Guide*.

Exporting a Part From Pro/ENGINEER

Use this procedure to export a Part from Pro/ENGINEER.

Prerequisites

- Change the accuracy as described in "Changing the Accuracy of a Pro/ENGINEER Part" on page 96.
- Reduce the complexity as described in "Reducing the Geometric Complexity of a Pro/ENGINEER Model" on page 96.

Procedure

1. Set up the accuracy to 0.000001 as described above.
2. Select Export from the File menu.
3. Select Model from the Export menu.
4. Select IGES from the Export menu.
5. Check the Surfaces option in the Export IGES dialog box.
6. Ensure the Dtm Curves and Wireframe options are unchecked in the Export IGES dialog box
7. Ensure the File Structure option is set to All Levels (Assemblies only).
8. Select the Options button in the Export IGES dialog box. This will enable the IGES configuration file to be edited using Pro/TABLE. The following options should be entered in this file:
   - Iges_out_all_srfs_as DEFAULT
   - Iges_out_spl_crvs_as_126 YES
   - Iges_out_spl_srfs_as_128 YES
   - Iges_out_trim_xyz YES
   - Iges_out_trm_srfs_as_143 NO
   - Iges_out_trim_curve_deviation 0.000001
   - Intf_out_blanked_entities YES
   - Intf3d_out_extend_surface YES
9. Close Pro/TABLE and select **OK** from the Export IGES dialog box, The IGES file should now be created, and can be imported into MCAD Bridge.
10. For more information on generating IGES files from Pro/ENGINEER see 'IGES FILES' in the *Pro/ENGINEER Interface Guide*.

Exporting an Assembly From Pro/ENGINEER

Exporting an Assembly from Pro/ENGINEER to Simcenter Flotherm is similar to the process of exporting a Part.

Prerequisites

- Change the accuracy as described in "Changing the Accuracy of a Pro/ENGINEER Part" on page 96.
- Reduce the complexity as described in "Reducing the Geometric Complexity of a Pro/ENGINEER Model" on page 96.

Procedure

1. Follow the procedure "Exporting a Part From Pro/ENGINEER" on page 97, with the following exceptions:
   - Set up the accuracy of each of the Parts of the Assembly as described in Accuracy.
   - The Export IGES dialog box has an extra option, select All Levels from this dialog box option.
   - A number of IGES files should now be created, and can be loaded into MCAD Bridge by loading in the top level assembly IGES file.

     This will automatically load in the Part IGES files assuming they are all present in the same directory and their names have not been changed.
2. For more information on generating IGES files from Pro/ENGINEER, see IGES FILES in the *Pro/ENGINEER Interface Guide*.

Preparing and Exporting IGES Files From IDEAS 7 and 5

Use these procedures when exporting IGES files for use in MCAD Bridge.

Changing the Accuracy of an IDEAS Model

A model accuracy of 1e-6 is required.

Procedure

You have a choice:

If you want to...	Do the following:
Set the default accuracy before creating the model.	1. Choose **Modify > Special Techs > Tolerance > Default Tolerance**. 2. Change the value to 1e-6.
Change the model's accuracy after creation.	1. Choose **Modify > Special Techs > Tolerance > Modify Tolerance**. 2. Select the part(s) to be modified. 3. Set the tolerance to 1e-6.

Reducing the Geometric Complexity of an IDEAS Model

Reduce the complexity of the models in IDEAS to speed up the process of exporting models to Simcenter Flotherm.

Procedure

Remove small Features and Parts that are unimportant to the Simcenter Flotherm calculation.

> A method of managing simplified models is provided in IDEAS. For more information, see SIMPLIFIED REPRESENTATIONS in the *IDEAS Assembly Modeling User's Guide*.

Configuring the Flavor

Follow this procedure to add the required Simcenter Flotherm flavor data.

Procedure

1. Copy the file *<IGES install dir>/gif/USERFLAVOR.CON* to the directory you start IDEAS from.
2. Read the setup directions in this file and add the following flavor.

```
FLOTHERM
0
1
4
7 1 2 9 9 9 9 9
9 3 7 6 6 6 6 5 5 9
3
```

Note

If there were already extra flavors defined in the file, increment the first integer by one, otherwise add a 1 to the file before the FLOTHERM word, as the instructions in USERFLAVOR.CON describe.

Exporting Parts From IDEAS 7

Use this procedure to export Parts from IDEAS 7.

Prerequisites

- Change the accuracy as described in "Changing the Accuracy of an IDEAS Model" on page 99.
- Reduce the complexity as described in "Reducing the Geometric Complexity of an IDEAS Model" on page 99.
- Configure the flavor as described in "Configuring the Flavor" on page 100.

Procedure

1. Choose **External > Export IGES** then select the part(s) to export. In the Export IGES dialog box select **Specify IGES Header**.
2. Set the following values in the IGES Header dialog box:
   - Model Resolution Tolerance = 1e-6
   - Model Space Scale = 1
3. Select **Analytic** from the system Apply Flavor dropdown menu in the Export IGES dialog box.

4. The first time you export a part you need to set up a flavor. To do this perform the following steps. On successive IGES exports go straight to Step 8.
5. Then select **Flavors** from the Export IGES dialog box. This opens the Edit IGES Export Flavor dialog box.
6. Enter a name for the flavor (that is, MentorMA). Select the Assemblies - “Map I_DEAS Assemblies to IGES subfigures” option in the Edit IGES Flavor dialog box.
7. Select Choose Geometry Types to export option. Click on the **Types** to export button. This opens the types to export to IGES dialog box; in this dialog box set the following options:
   - Points = Off
   - Groups = On
   - Curves = On
   - Convert Curves to type = Analytic
   - Convert arcs to conics = On
   - Surfaces = On.
   - Convert to trimmed surface type = 144:0 Trimmed with 3D model and 2D.
   - Convert base surface type = Analytic

   If applicable prefer this:
   - Type = 120 surface of revolution
   - Trim curves = Convert to Analytic
   - Solids = On
   - Convert to Representation type = Surfaces
8. Once a local flavor has been set up I-DEAS should pick it up. Select that flavor from the local apply flavor option.
9. Finally enter a filename for the IGES file in the Export IGES dialog box and select **Export**. This generates an IGES file that can be read.

Exporting Parts From IDEAS 5

Use this procedure to generate an IGES file that can be read into Simcenter Flotherm.

Prerequisites

- Change the accuracy as described in “Changing the Accuracy of an IDEAS Model” on page 99.

- Reduce the complexity as described in “Reducing the Geometric Complexity of an IDEAS Model” on page 99.
- Configure the flavor as described in “Configuring the Flavor” on page 100.

Procedure

1. Choose **External** > **Export IGES** then select the part(s) to export.
2. In the Export IGES dialog box select **IGES Options**.
3. Set the following values in the IGES Options dialog box:
   - Parameter Delimiter [,]
   - Record Delimiter [;]
   - Model Space Scale [1]
   - Product Identification [name for Simcenter Flotherm Assembly]
   - Resolutions: Angular [1e-10]
   - Resolutions: Perpendicular [1e-6]
   - Resolutions: Parallel [1e-6]
   - Resolutions: Point Coincident [1e-6]
4. Select **Flavors** from the Export IGES dialog box.
5. Select **Use Standard Flavor**, and select the standard flavor of Simcenter Flotherm.
6. Enter a filename for the IGES file in the Export IGES dialog box and select **Export**.

Related Topics

Supported IGES Entities

Chapter 5
Frequently Asked Questions

This chapter provides answers for frequently asked questions (FAQs) about the Simcenter Flotherm MCAD Bridge.

Color Attributes

Frequently asked questions about color attributes.

1. How can I retain color attributes after transferring the MCAD assembly to the Project Manager?

   If you need to retain the MCAD part colors, set the environment variable FLOMCAD_SET_COLOR_ON_TRANSFER to any value before starting MCAD Bridge.

Appendix A
Command Reference

This appendix contains reference information for command line commands.

flomcad

Runs on Windows systems

Opens the MCAD Bridge application window. In stand-alone mode, you can specify the optional arguments to open the window with a loaded model, or run a FloSCRIPT file.

Usage

flomcad [-f *floscript*] [-*filetype*] [*filename*]

The -f option takes priority over -*filetype*, which takes priority over *filename*.

Arguments

- *floscript*

  Run file *floscript* at startup, where *floscript* is a FloSCRIPT xml file. If there are any spaces in the filename, then, following standard Microsoft® Windows conventions, the project name must be enclosed in single or double quotation marks. For example, \"my script file.xml\".

- *filetype*

  The type of CAD file to be loaded: CATIAV4, CATIAV5, DXF, IGES, PROE, SAT, SOLIDWORKS, STEP, or STL.

- *filename*

  The name of the file, including the extension, to be loaded.

Table A-1. File Extensions Supported

Filetype	Extension
CATIAV4	*.model* and *.exp*
CATIAV5	*.CATPart* and *.CATProduct*
DXF	*.dxf*
IGES	*.igs* and *.iges*
PROE (PRO/E)	*.prt* and *.asm*
SAT	*.sat* and *.sab*
SOLIDWORKS	*.sldprt* and *.sldasm*
STEP	*.stp* and *.step*
STL	*.stl*

Examples

- To start MCAD Bridge with file, *example.sat*, loaded:

  flomcad -SAT example.sat

or

flomcad example.sat

- To run MCAD Bridge from FloSCRIPT file, *My_MCADLogFile_saved.xml*:

  flomcad -f \"<*install_dir*>\flosuite_v<*version*>\flotherm\WinXP\bin\LogFiles\My_MCADLogFile_saved.xml\"

Related Topics

Starting MCAD Bridge in Stand-Alone Mode

FloSCRIPT [Simcenter Flotherm User Guide]

Glossary

Decomposition
The simplification and dissection of MCAD Parts and Bodies into a collection of Simcenter Flotherm objects.

Dissection
The conversion of an MCAD Part or Body into a collection of Simcenter Flotherm objects.

Feature
MCAD Bridge features including holes, humps, and small groups of faces.

Global Simplification
The simplification of a selected MCAD Part or Body.

Healing
The process of creating a solid object out of a collection of ACIS geometric entities.

Local Simplification
The simplification of MCAD Faces or Features selected in MCAD Bridge.

MCAD Assembly
A collection of MCAD Parts. In MCAD Bridge, the ACIS model before conversion into Simcenter Flotherm modeling objects.

MCAD Body
A part of an MCAD Part. In MCAD Bridge, an ACIS Body which has not been converted into Simcenter Flotherm modeling objects.

MCAD Face
One surface section of an MCAD Body or Part.

MCAD Feature
A geometric entity made up of a number neighboring faces.

MCAD Part
A collection of one or more MCAD Bodies. In MCAD Bridge, an ACIS solid part.

Realized Assembly
An imported MCAD assembly with all of its component parts converted into Simcenter Flotherm primitives.

Simplification

The modification of MCAD geometry by removing thermally insignificant features such as small holes, humps, curved faces, and so on, which are unimportant to the Simcenter Flotherm calculations.

STL

Stereolithography Interface, developed by 3D Systems Inc. for imaging 3D solid models using a collection of triangular facets. A common CAD/CAM interfacing format in most Rapid Prototyping systems.

There are two STL formats, BINARY and ASCII. MCAD Bridge only supports the line-structured ASCII version.

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