Simcenter™ Flotherm™ SmartParts Reference Guide

Software Version 2021.1

Support Center: support.sw.siemens.com
Send Feedback on Documentation: support.sw.siemens.com/doc_feedback_form

Table of Contents

List of Figures

List of Tables

Chapter 1
SmartParts and Primitives

The general properties of SmartParts and primitive geometry shapes of Simcenter™ Flotherm™ software.

EDA SmartParts are particular types of SmartParts that are created when a MotherBoard is transferred from EDA Bridge, see "EDA SmartParts" on page 323.

What are SmartParts and Primitives?

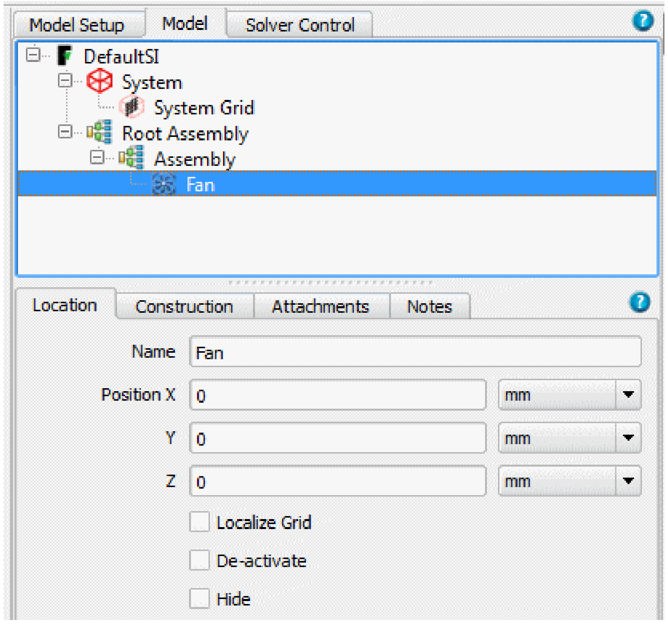
To save time when building complicated structures, pre-built model macros (or templates) of shapes and equipment, called SmartParts, are supplied which are easily adapted to suit your model.

In addition, a set of simple 3D shapes (cuboids, prisms, tetrahedra (tets), and inverted tetrahedra), called primitives, that can be used to build more complex shapes, are provided.

SmartParts and primitives are added to the model from the New Object Palette.

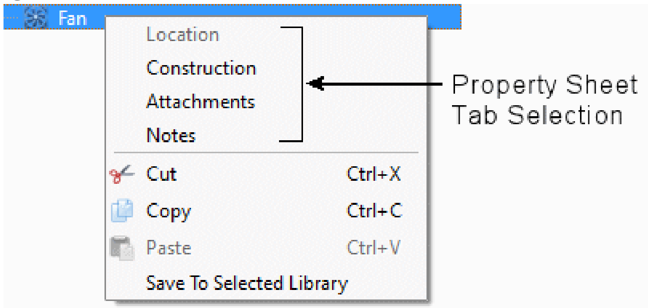
SmartPart and primitive dimensions, locations, and thermal properties are defined in their property sheets, which are displayed below the data tree when a SmartPart or primitive is selected. For example, see Figure 1-1.

Figure 1-1. Fan Property Sheet Displayed When a Fan is Selected



By default, the first tab of the property sheet is opened, however, you can open a tab directly by selection from the context-sensitive (right-click) menu, see Figure 1-2.

Figure 1-2. SmartPart Context-Sensitive Menu



Related Topics

Coordinate Systems

General Procedures Applicable to SmartParts and Primitives

Generic Property Sheets and Dialogs

Coordinate Systems

SmartParts and Primitives are created as members of a parent assembly, and have, by default, their origins located at the origin of the parent assembly.

Origins of SmartParts

Each SmartPart has its own coordinate system, and, therefore, its own origin, so that its structure can be defined independent of the overall model. This enables SmartParts to be held in libraries and transferred between projects.

The SmartPart axes are denoted by Xo, Yo, and Zo to differentiate between the X, Y, and Z system axes of the overall model.

Positions of SmartParts

The location of a SmartPart is changed either by using the **Location** tab of the SmartPart's property sheet, or by dragging in the drawing board.

Plane of Location

SmartParts are geometry representing well defined features which can be defined parametrically with numerical and text input via property sheets. SmartParts defined in this way will always appear as co-planar in the X-Y plane (that is Xo and Yo axes of the SmartPart align with the system X and Y axes). All other orientations require the SmartPart to be rotated and translated into position.

In the drawing board, the SmartPart is drawn by dragging the bounding box across any of the available 2D views. In each case, the SmartPart is created as co-planar with the selected view.

When drawing a part in a 2D view, the part is drawn on the work plane. If you are working on a multi-floor model, it is often useful to set the work plane co-incident with the floor you are working on so that objects are drawn on the floor. Objects will be drawn in a positive direction normal to the work plane with a size the average of the two dimensions drawn in the work plane. If an object produces flow, the flow will by default be in the positive out of work plane direction.

Related Topics

Generic Property Sheet Location Tab

General Procedures Applicable to SmartParts and Primitives

Generic Property Sheets and Dialogs

Thick (3D) and Thin (2D) Walls, Holes, and Sloping Blocks

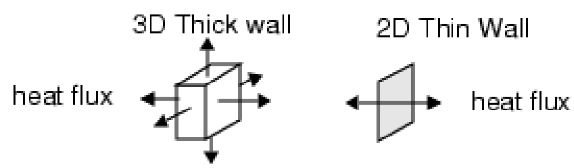
Enclosure Walls, Holes, and Sloping Blocks can be modeled as Thick or Thin.

- "Thin" walls are displayed as 2D in the drawing board and only allow conduction through the plane. Because they are geometrically 2D, the grid is simplified. The solver considers the thickness from a heat transfer perspective. Heat passing through Thin walls experiences the same thermal resistance as they would were they to be modeled 3D.

By default, heat spreading in the plane of a Thin wall is ignored. If this is required to be calculated then **Active Plate Conduction** must be set in the **Solver Control** tab.

- "Thick" walls are displayed as 3D in the drawing board and allow conduction through and in the plane of the block, see Figure 1-3.

Figure 1-3. Heat Flux Through and Along Thick and Thin Walls



Normally, modeling the walls as collapsed objects is sufficient to capture the thermal and barrier to flow characteristics, but in some cases it can be useful to model the actual thicknesses.

Related Topics

Solver Control Tab [Simcenter Flotherm User Guide]

General Procedures Applicable to SmartParts and Primitives

The following procedures are general procedures applicable to SmartParts and primitives.

Collapsing 3D Cuboids to 2D Planes

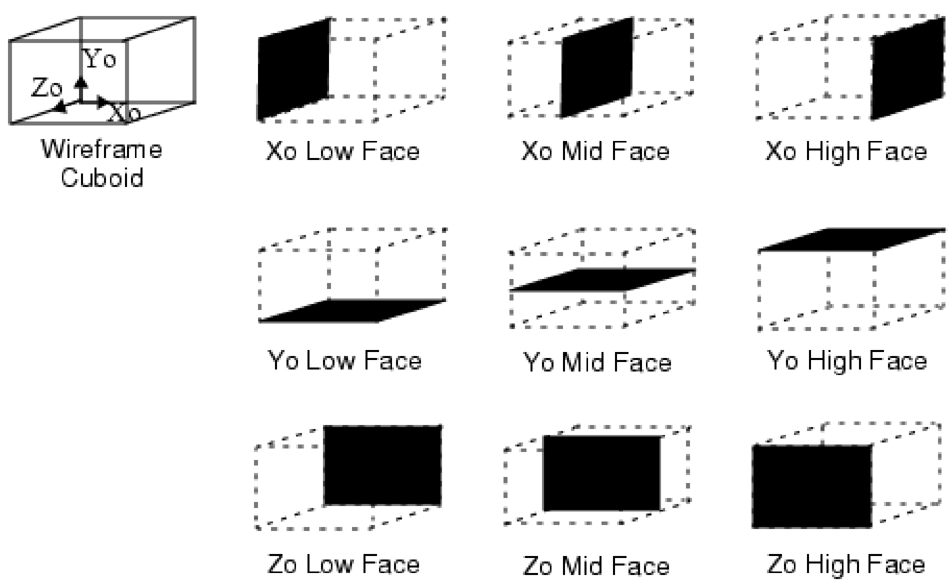
Cuboids and Network Cuboids can be collapsed from their 3D default structures to 2D planes. Collapsing to 2D increases the efficiency of modeling.

Procedure

1. Select the **Location** tab of the property sheet of the cuboid you want to collapse.
2. Select one of the Collapse options to specify the position of the plane.

   Figure 1-4 shows the possible collapsed plane positions.

Figure 1-4. Collapsed Planes



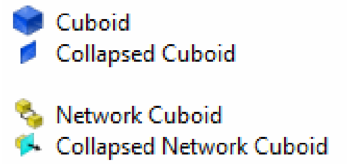
Results

The 3D cuboid becomes a 2D plane in the drawing board.

Collapsed cuboids do not have resize handles.

A collapsed Cuboid or Network Cuboid has a different icon in the data tree, see Figure 1-5.

Figure 1-5. Collapsed Cuboid Icons



The collapse direction is relative to the local coordinate system of the primitive, that is, if you rotate the primitive, the collapse direction remains the same, see Figure 1-6.

Figure 1-6. Collapsed Plane Before and After Rotation

Decomposing SmartParts

Some SmartParts can be decomposed into their constituent independent primitives.

Restrictions and Limitations

- You can only decompose the following SmartParts:
  - Block with Holes
  - Compact Components
  - Conducting PCB
  - Cylinder
  - Die
  - Enclosure
  - Heat Sink
  - Thick Sloping Block
- Multi-select decomposing is not allowed.

Procedure

1. Right click the object in the data tree.

   If the object can be decomposed then the **Decompose** option will be listed in the context menu.

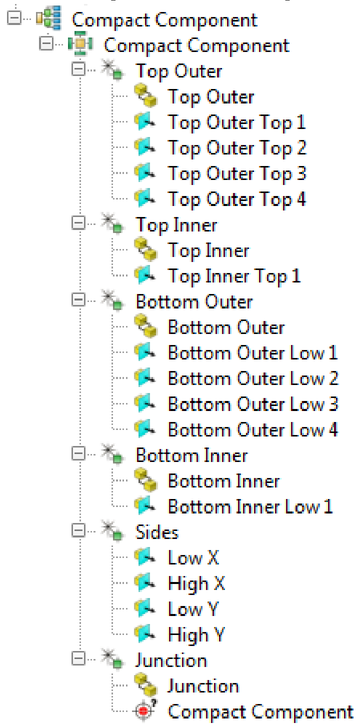
2. Select **Decompose**.

Results

A new assembly is created at the same level of the SmartPart, the SmartPart is deleted. To reverse the process, use Undo.

A Compact Component is decomposed into a new assembly that has a child Network Assembly.

Figure 1-7. Decomposed Compact Component



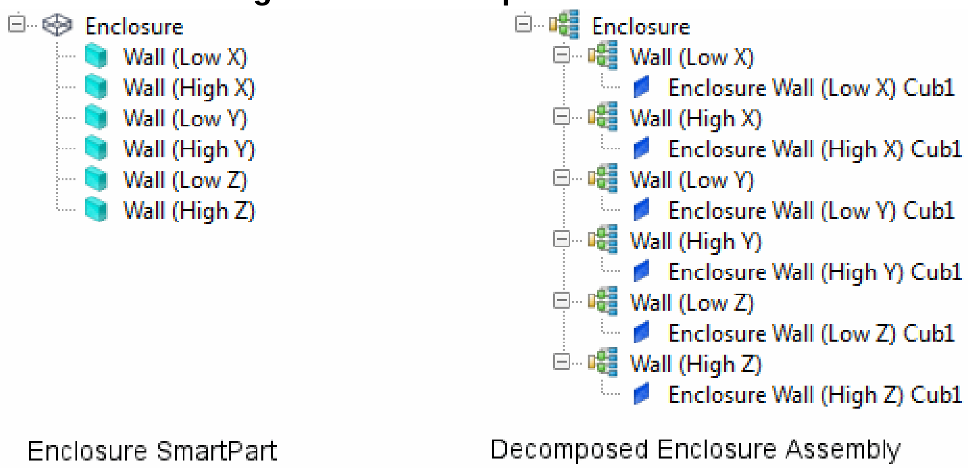
Note

A Network Assembly interfaces to the model in a slightly differently way than a Compact Component, so slight result differences are to be expected if Compact Components are decomposed. A Network Assembly holds solid cells adjacent to a 2D Network Cuboid isothermal, but a Compact Component does not.

Examples

Figure 1-8 shows an enclosure that has been decomposed into six primitive parts which can now be individually moved and modified.

Figure 1-8. Decomposed Enclosure



Another example is a plate fin Heat Sink that can be decomposed into an assembly of parts so that individual fins can be modified.

For an Enclosure, Sloping Block or Block with Holes, a thermal attribute may be applied which defines the object as conducting, or non-conducting at a fixed temperature. If a library attribute is attached which has a power output set, the power setting is ignored. However, on decomposing the object, the resulting cuboids each retain the thermal attachment and the power setting now becomes valid and is applied.

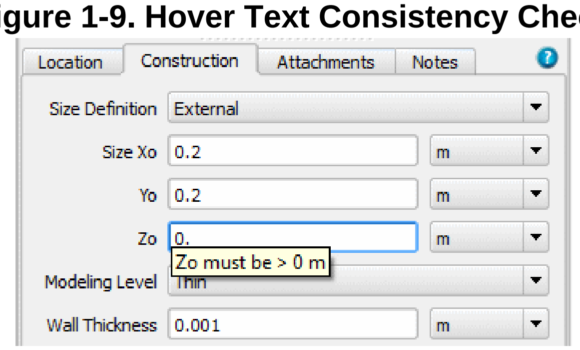
Consistency Checking

Consistency checks are performed for SmartParts during property sheet data input.

Any inconsistencies detected are displayed as hover text.

For example, Figure 1-9 shows hover text when attempting to enter a Size value for an Enclosure SmartPart.

Figure 1-9. Hover Text Consistency Check



Generic Property Sheets and Dialogs

Property sheet tabs that are common to more than one SmartPart.

Generic Property Sheet Location Tab

To access: Select an object and then select the **Location** tab of the property sheet.

Use this tab to relocate the object by entering its new position, to collapse the object to 2D, to localize the grid on the object, to deactivate or hide the object, or to remove children from the data tree (lightweight tree view).

Objects

Field	**Description**
Name	Identifies the object. You cannot edit the name in the data tree.
Position X, Y and Z	Locates the origin of the object.
Size X, Y and Z	Defines the length of each side of the shape.
Collapse	Collapses a 3D object down to a 2D planar surface.
Localize Grid	Confine any attached grid constraints to just the SmartPart grid space.
De-activate[1]	Removes the object from the simulation. The object remains in the project, but is removed from the drawing board. The deactivation of objects does not change the solver grid.
Hide[1]	Removes the object from the drawing board. The hide operation is conveyed to other application windows. The simulation remains unaffected as the object does not change its Active/De-Active state.
Transparency	Use this slider bar to change the transparency of the selected object, and all child objects. When the slider is at the far-left position (default), the object is opaque.
Ignore Geometry[2]	All geometry is ignored by the grid and solution. Ignored geometry is not visible in the data tree or GDA, but can be made visible, in red, in the data tree only, by setting the Show Ignored Geometry option in the User Preferences dialog box.
Lightweight[3]	Improves the readability of the data tree by removing objects. • For assemblies, checking this check box removes the children of the assembly from the data tree. • For EDA SmartParts, this check box is checked by default to simplify the data tree of a MotherBoard transferred from EDA Bridge. When unchecked, EDA SmartParts are extended, see "Extending EDA SmartParts" on page 324.

1. Not applicable to the Root Assembly.
2. Only applicable to assemblies, but not the Root Assembly.

3. Only applicable to assemblies, and EDA SmartParts except EDA Layer Assemblies and EDA Electrical Vias Assemblies.

Usage Notes

- The available fields are dependent on the selected object.
- The position of the origin can be the Absolute Coordinates origin or the Local Coordinates origin, depending on the “Display Positions in” setting in the User Preferences dialog box.
- This tab is used in the System property sheet to define the position and size of the solution domain. The third direction is always needed, even for 2D models.

Related Topics

Coordinate Systems [Simcenter Flotherm User Guide]

Collapsing 3D Cuboids to 2D Planes

Grid Localization [Simcenter Flotherm User Guide]

Changing Geometry Transparency [Simcenter Flotherm User Guide]

Data Tree [Simcenter Flotherm User Guide]

Generic Property Sheet Attachments Tab

To access: Select an object and then select the **Attachments** tab of the property sheet

Use this tab to attach or detach attributes to or from the selected object.

Objects

The attributes that can be attached depend on the selected object.

The attribute dropdown selection includes:

- No Attachment

  No attribute is attached.

- Create New

  You can create a new attribute.

- Load From Library

  You can load an attribute from a library.

- A list of project attributes.

  You can select an existing project attribute.

Usage Notes

Use the expansion buttons when you want to define different attributes in different directions or planes.

Related Topics

Simcenter Flotherm Project Attributes Reference Guide

Generic Property Sheet Groups Tab

To access:

- For Coolers and Racks, select the Groups tab of the property sheet.
- For Recirculation Devices, Fixed Flows or Perforated Plates, first check the Calculate check box in the Capture Index section of the **Model Setup** tab, then select the **Groups** tab of the property sheet.

Use this tab to assign group and sub-group names to objects for CI (Capture Index) analysis, and to assign a Cooler Group name to a Rack or Cooler when using remote rack temperature to control cooler airflow.

Note

Names are deleted automatically when they are not attached to any objects.

Objects

Field	Description
Hot Aisle Group[1]	(Coolers, Racks, Recirculation Devices, Fixed Flows.) If used, the name of Hot Aisle Group to which the object belongs.
Cold Aisle Group[1]	(Coolers, Racks, Recirculation Devices, Fixed Flows, Perforated Plates.) If used, the name of Cold Aisle Group to which the object belongs.
Hot Aisle Sub-Group[2]	(Coolers, Racks, Recirculation Devices, Fixed Flows.) If used, the name of Hot Aisle Sub-Group to which the object belongs.
Cold Aisle Sub-Group[2]	(Coolers, Racks, Recirculation Devices, Fixed Flows, Perforated Plates.) If used, the name of Cold Aisle Sub-Group to which the object belongs.
Cooler Group	(Coolers, Racks.) If used, the name of Cooler Group to which the object belongs.

1. Only shown if the Calculate check box is checked in the Capture Index section of the **Model Setup** tab.
2. Only shown if both the Calculate and Sub-Groups check boxes are checked in the Capture Index section of the **Model Setup** tab.

Usage Notes

Select an existing name from the dropdown list or **Create New** to open the Create Group dialog box.

Names are deleted when they are not attached to any objects.

Related Topics

Capture Index [Simcenter Flotherm User Guide]

Cooler Groups

Create Group Dialog Box

Create Group Dialog Box

To access: Select **Create New** from the dropdown options in the **Groups** tab of a property sheet.

Use this dialog to create and assign a new Aisle Group name to a Cooler, Rack, Recirculation Device, Fixed Flow or Perforated Plate, or a new Cooler Group name to a Cooler or Rack.

Objects

Field	Description
Group Name	A user-defined name.

Related Topics

Generic Property Sheet Groups Tab

Generic Property Sheet Notes Tab

To access: Select an object in the data tree, or an attribute from the Project attributes, and then select the **Notes** tab of the property sheet

Use the **Notes** tab of property sheets to add any relevant information about an object or attribute.

Objects

Field	Description
Notepad area	Enter or paste text as required. Right-click anywhere in the notepad area to display a context-sensitive menu.

Chapter 2
Assemblies

Use Assembly SmartParts to group together components so that they can be moved as a single entity.

Modeling Assemblies

All modeling objects belong to a parent assembly. By default, the “Root Assembly” is the top-level assembly.

Assemblies and Coordinate Systems

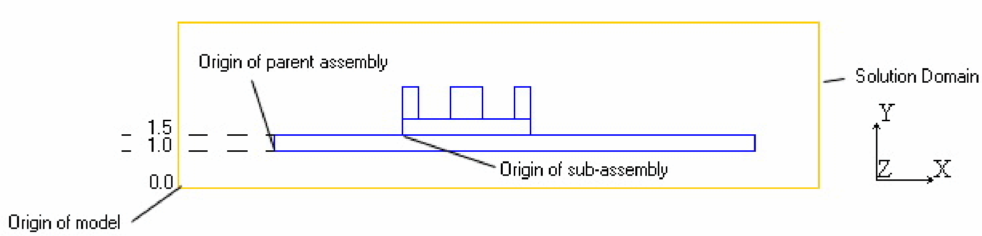
By default, a new assembly origin is always located at the origin of the coordinate system being used.

When using absolute coordinates, the position of the assembly are relative to the model origin.

When using local coordinates, the position of the assembly depends on the display method used.

- In the project tree, local coordinates are relative to the parent assembly. For example, consider the assembly and sub-assembly shown in Figure 2-1.

Figure 2-1. Locating an Assembly



  The sub-assembly is located at Y=1.5 (absolute), or Y=0.5 (local). The difference is because the parent assembly origin is at Y=1.0.

- In the drawing board, local coordinates are relative to the origin of the assembly on view. Local coordinates are the same as the absolute coordinates if the model is viewed from the root assembly (providing the root assembly is still at the default origin). Note, however, that the local coordinates displayed in any property sheets are always be relative to the parent assembly.

Assembly Procedures

Organize your model by adding assemblies to the Root assembly and then adding objects/ geometry to those assemblies.

Adding Assemblies

Use assemblies to organize the objects in your model.

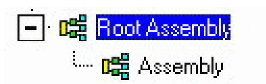
Procedure

To add a new (empty) assembly, select the parent assembly to hold the new assembly and click the **Assembly** icon in the New Object Palette.

Results

The new assembly appears in the tree as a child of the selected parent assembly (Figure 2-2).

Figure 2-2. A Parent-Child Assembly Hierarchy



Initially, a new assembly (containing no objects) exists only as a reference origin and does not appear in the drawing board. When objects are added to the assembly, then its bounding box, that is, the bounding box of all the objects contained within the assembly, and its local coordinate system are displayed in the drawing board when the assembly is selected.

Moving the assembly, moves all of the objects that are children of the assembly.

Adding Child Geometry to Assemblies

Geometry can be added as new default specification SmartParts or as configured library items. In addition, you can import assemblies from other systems.

Procedure

Select the assembly to which the geometry is to be associated with then use one of the following methods:

- To add a new object, click the object icon on the New Object Palette.
- To add a library object, either click the object icon in the library or drag and drop the icon.
- You can also move or copy geometry within the tree using Cut, Copy, and Paste commands which are available via menus or shortcut keys.

Assembly Property Sheet

To access: Select an Assembly in the model.

Use this property sheet to configure an assembly.

Objects

Object	Description
Location tab.	See “Generic Property Sheet Location Tab” on page 28.
Attachments tab.	See “Generic Property Sheet Attachments Tab” on page 30.

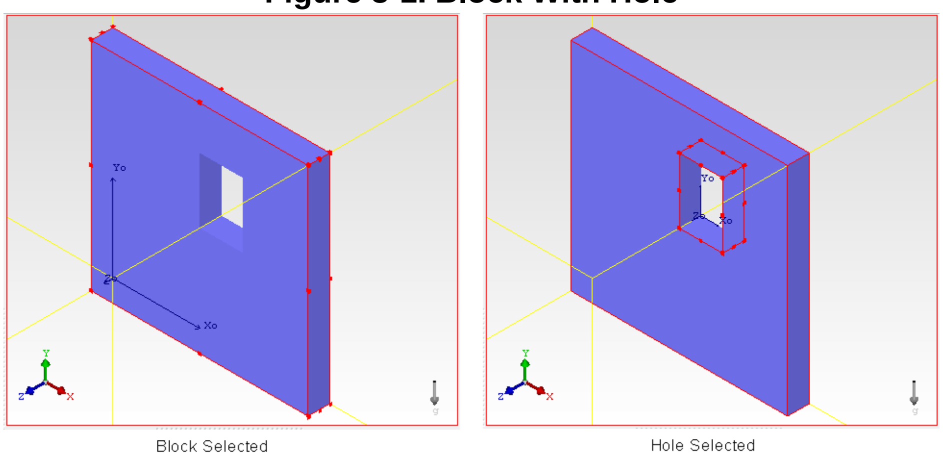
Chapter 3
Blocks With Holes

Use Block with Holes SmartParts when modeling apertures in materials, or composite materials such as windows, in which case the hole is filled with an alternative material to the block.

Block With Hole Properties

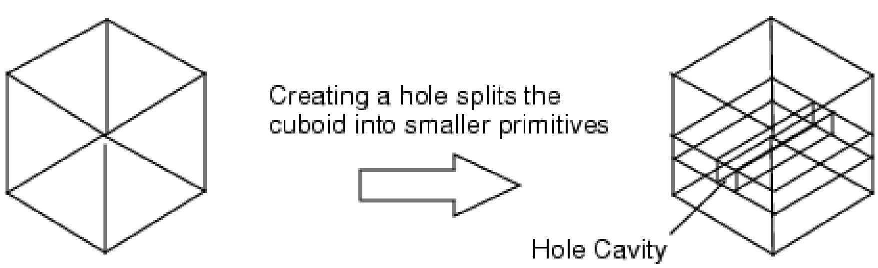
Blocks with Holes are cuboids with holes, like cuboids, they can be collapsed but they can also be decomposed.

Figure 3-1. Block With Hole



Holes run through the full length of the cuboid, see Figure 3-2.

Figure 3-2. Hole Cavity Through Whole Cuboid



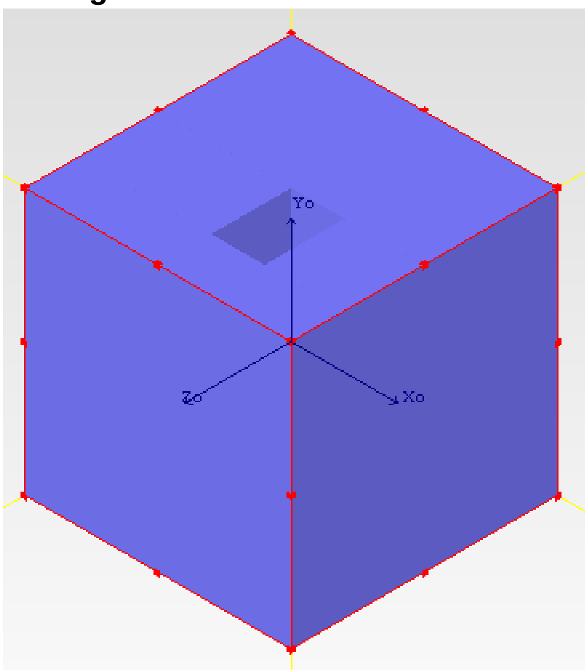
Holes are SmartParts and their properties (size, open, or filled, and so on) are defined separately, refer to “Holes” on page 193.

Hole Direction

The hole direction can only be changed if the cuboid is not collapsed. If the cuboid is collapsed, the hole is created in the collapse direction.

The possible hole directions are the local axis directions: Xo, Yo, or Zo. Figure 3-3 shows the hole in the Yo direction.

Figure 3-3. Hole in Yo Direction



Collapsed Blocks With Holes

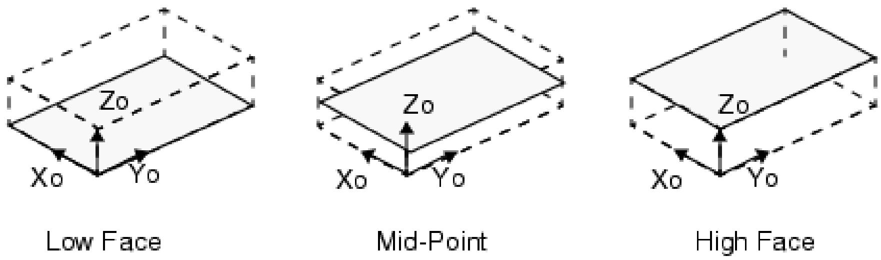
When modeling a planar surface with holes you can collapse a Block with Holes.

The collapse direction is in the direction of the hole(s). All holes are in the same direction.

A Block with Holes can be collapsed to the low face, the mid-point or the high-face in the hole (collapse) direction.

Figure 3-4 shows the possible options when a block is collapsed in the Zo direction.

Figure 3-4. Collapse Options



Thermal Attachments After Decomposition

A Block with Holes can be decomposed into its primitive parts by choosing **Geometry** > **Decompose**. The resulting cuboids each retain any thermal or surface exchange attributes previously attached to the original block with holes.

Creating a Block With Holes SmartPart

Block With Holes SmartParts are not added from the Objects Palette like other SmartParts, they are created from existing Cuboids in the model.

Restrictions and Limitations

- Holes that are 0.01% or less than the smallest side of the block face are ignored by the grid, the solver, and the decompose command, and are only displayed in wireframe rendering mode.

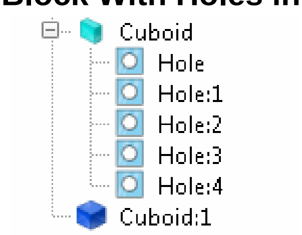
Procedure

1. Select a cuboid and then add one or more holes.
2. Define the properties using the Block With Holes property sheet and the Hole SmartPart property sheet(s).

Results

A Block With Holes has a different icon from a cuboid, see Figure 3-5.

Figure 3-5. A Block With Holes in the Data Tree



Related Topics

Block With Holes Property Sheet

Hole Property Sheet

Block With Holes Property Sheet

To access: Select a Block with Holes SmartPart in the model.

Use this property sheet to configure a block with holes.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Construction tab.	See "Block With Holes Property Sheet Construction Tab" on page 44.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.

Block With Holes Property Sheet Construction Tab

To access: Select a Block with Holes SmartPart in the model and then select the **Construction** tab.

Use this tab to change the overall size of a Block with Holes SmartPart and/or alter the direction of any of its holes.

Objects

Field	Description
Size Xo, Size Yo, Size Zo	Defines the overall lengths of each side of the block. The dimensions are relative to the local origin of the original cuboid.
Hole Direction	The direction perpendicular to the hole aperture, that is, the direction of the depth of the hole. See “Hole Direction” on page 40.
Collapse To	Collapses the block to the high face, low face or mid point of the cuboid in the direction of the hole(s). See “Collapsed Blocks With Holes” on page 40.

Usage Notes

If you change the Hole Direction, the Length and Width field name in the Holes property sheet change in line with this. For example, changing Hole Direction from the default Zo to Xo, changes the Length and Width from Length (Xo), Width (Yo) to Length (Yo), Width (Zo).

Related Topics

Creating a Block With Holes SmartPart

Block With Hole Properties

Hole Property Sheet

Chapter 4
Compact Components

Use Compact Component SmartParts to represent array and peripheral lead packages.

Modeling Compact Components

Compact Component SmartParts are resistor network representations of packages and can be used when the resistance values are known for the component.

The resistance values may be available from the component supplier, or can be generated by FloTHERM PACK. FloTHERM PACK can also be used to generate detailed component models if the highest accuracy is needed, but these require additional mesh.

Compact Component SmartParts provide an intermediate level of accuracy when predicting the junction temperature of components. The Compact Component SmartPart can represent one of two thermal network models; the 2-Resistor model and the General model.

There are two main structural types of Compact Components covering classical area of array and peripheral lead packages (mono, dual, and quad) in both square and rectangular shapes.

TO-style, SIMMS, and stacked packages are not modeled.

Model Types

There are two computational network models provided for simulating the resistor networks of these packages: the 2-Resistor Model and the General Model.

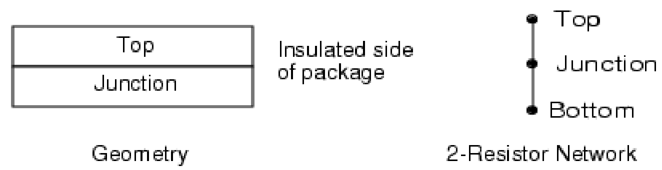
2-Resistor Model

The 2-Resistor Model is the simplest representation generic to all package types, consisting of two cuboid layers.

The internal resistances to the board are included in the 2 resistances:

- from the Junction to the Top (case) and
- from the Junction to the Bottom (board on which it is mounted), see Figure 4-1.

Figure 4-1. 2-Resistor Network Model



General Model

Use the General Model when you want to model compact components in detail.

The main body is divided into:

- An inner core, which may be either raised above, or level with, the package top, and
- Outer regions.

Solder balls, pins, and leads can all be represented, with the option of underfilling the package.

The General network model permits completely flexible connectivity between the computational nodes of the compact model. A maximum of 20 junction nodes to model stacked die packages, plus two internal nodes may be included in this model type. However, these nodes have no physical representation in the package and are only included, where required, to achieve a more accurate representation of the thermal behavior of the package.

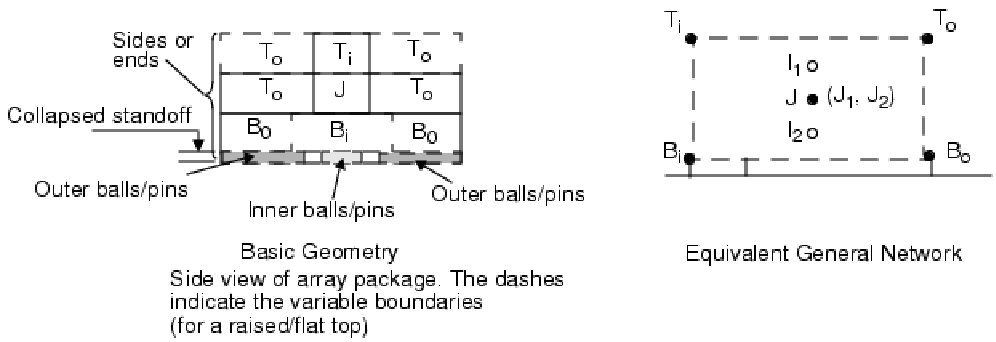
General Models comprise two package types: Area Array packages and Peripheral packages.

Area Array Packages

Area array packages have ball or pin connectors on the bottom of the package.

Figure 4-2 shows the basic geometry for an area array package and its representation by the General network model.

Figure 4-2. Basic Geometry for an Area Array Package



J indicates the junction node, chosen from the number of junctions specified for the model, to have its temperature displayed in Analyze mode.

The dashed lines indicate the extent of the package.

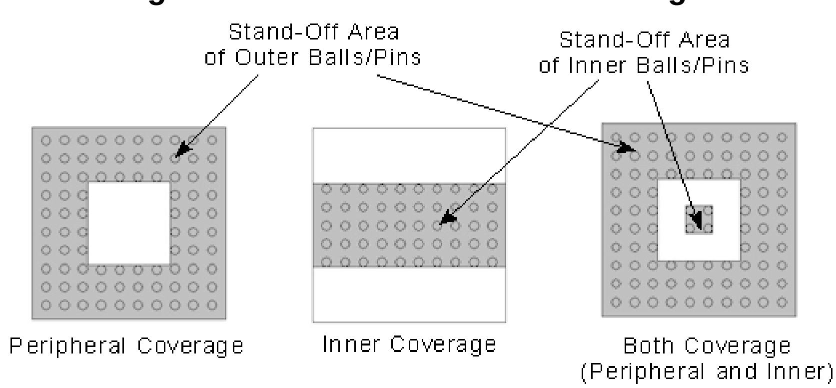
Nodes representing the external surfaces of the package are shown on the surface of the package, and nodes internal to the model are shown inside the dashed region. Figure 4-2 does not show how the nodes are connected.

Solder Balls/Pins Arrays

For an area array package, the connection between the bottom and the PCB is achieved by coverages of solder balls/pins.

The different coverages available for models are Full, Peripheral, Inner, and Both. The stand-off areas are shown in gray in Figure 4-3.

Figure 4-3. Solder Balls/Pins Coverages



- The inner region is an inner array of solder balls, and is set over the area corresponding to the smaller of Top Inner and Bottom Inner. The outer periphery of the inner region is considered to coincide with the edge of the inner array of solder balls. The inner solder ball/pins resistance is applied over the inner region having the smallest area (either Bottom Inner or Top Inner).
- The outer region is a peripheral array of solder balls, and is set over the area corresponding to the smaller of Top Outer and Bottom Outer. The peripheral solder balls/pins resistance is applied over the outer region having the smallest area (ether Bottom Outer or Top Outer).

When the Both Coverage is used, the solder ball/pin resistance for each region is the total resistance of the package solder/balls associated with that region.

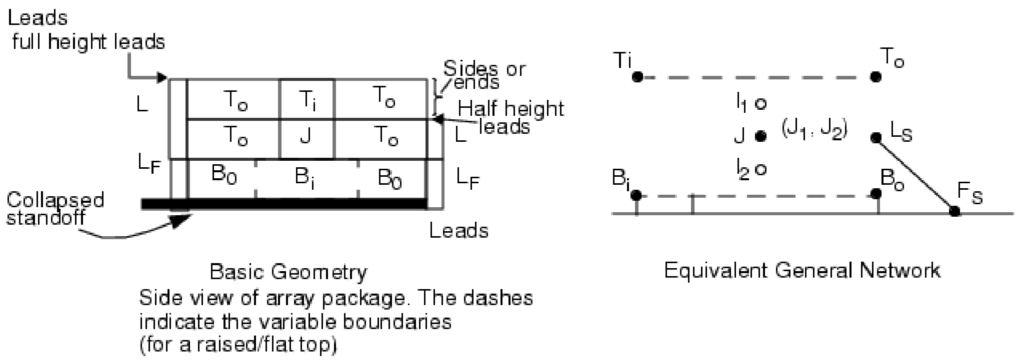
When the Full, Peripheral, or Inner Coverage is used, the solder ball/pin resistance is the total resistance of the package solder balls/pins (that is, the resistances of the individual solder balls/ pins summed in parallel).

Peripheral Packages

Peripheral packages have lead connectors at the periphery of the package.

Figure 4-4 shows the basic geometry for a peripheral package and its representation by the General network model.

Figure 4-4. Basic Geometry for a Peripheral Package



J indicates the junction node, chosen from the number of junctions specified for the model, to have its temperature displayed in the GDA when in Analyze mode.

The dashed lines indicate just the extent of the package body and not resistances.

The nodes connected by dashed lines may also be connected by resistances, but this is not a requirement.

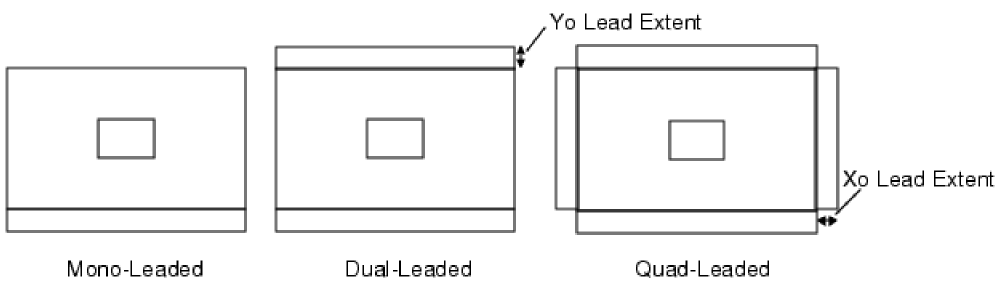
The solid lines represent the resistances connecting the package and the board. These resistances are computed from the supplied data, that is, resistances of leads, standoff height and the properties of the underfill material.

In the case of a single lead node, the leads resistance is the total resistance of the package leads (that is, the resistances of the individual leads summed in parallel). Where two lead nodes are used, the leads resistance for each node is the total resistance of the package leads associated with that node.

The Compact Component SmartPart supports Mono, Dual, and Quad-leaded package styles.

- For Mono-leaded packages, only a single leads node is allowed.
- For Dual-leaded packages the leads can be treated as a single node, or the leads on each side of the package can be represented as a separate node.
- For Quad-leaded packages, the leads on all four sides can be represented as a single node. Alternatively, pairs of leads on opposite sides can be treated as a single node, or the leads on each side can be represented as a separate node.

Figure 4-5. Mono-, Dual-, and Quad-Leaded Packages



The leads are assumed to extend along the entire length of the side/end of the package to which they are attached.

Geometry Representation of Peripheral Packages

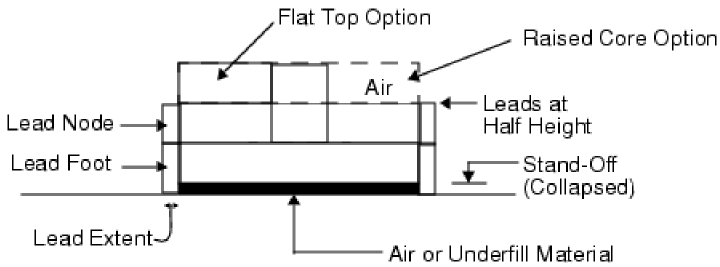
The leads interconnections are represented by two layers of solid cuboids. The bottom layer models the leads foot, the top layer models the leads where they emerge from the package body.

There are no balls/pins at the bottom of a peripheral package so the standoff region is represented by a single collapsed cuboid spanning the plan dimensions of the main body of the package. The leads interconnections may be either the Half Height or Full Height of the main body.

- Leads at Half Height

  Use Mono, Dual, or Quad Leads at Half Height to represent plastic encapsulated parts, for example, PQFP, SOP, TSOP. The geometric structure is shown in Figure 4-6.

Figure 4-6. Leads at Half Height



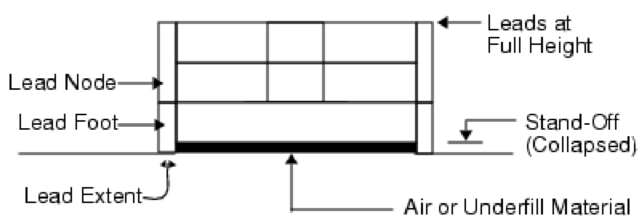
The geometric model supports the option of a raised top core as the outer region on the top layer can either be the Top Outer node or absent (air).

The physical leads are represented as two layers of solid cuboids. The lower layer represents the Lead Foot while the upper layer represents Lead node(s) in the network. With the Half Height option, the upper layer (Lead node) coincides with the middle layer in the package body. As the top of the lead frame normally coincides with the mid-height of the package body, the relative thickness of the bottom plus middle layers to the top layer (that is, $Z_B + Z_M : Z_T$) will be approximately 1:1.

- Leads at Full Height

  Use Mono, Dual, or Quad Leads at Full Height to represent ceramic parts, for example, CQFP. The geometric structure is shown in Figure 4-7.

Figure 4-7. Leads at Full Height



The physical leads are represented as two layers of solid cuboids. The lower layer represents the Lead Foot while the upper layer represents the Lead node(s) in the network. With the Full height option, the lower layer (Leads Foot) extends over the lower and middle layers in the package body.

Note

Mono, Dual, or Quad Leads at Full Height do not allow a raised top core region.

Examples of Compact Components

Included are some examples of different Compact Component geometry.

By default, an area array component is created, one-tenth the size of the solution domain in the X and Y directions.

Figure 4-8 and Figure 4-9 and Figure 4-10 show geometric examples of models. A Monitor Point is included with the SmartPart.

Figure 4-8. 2-Resistor Model in the Drawing Board

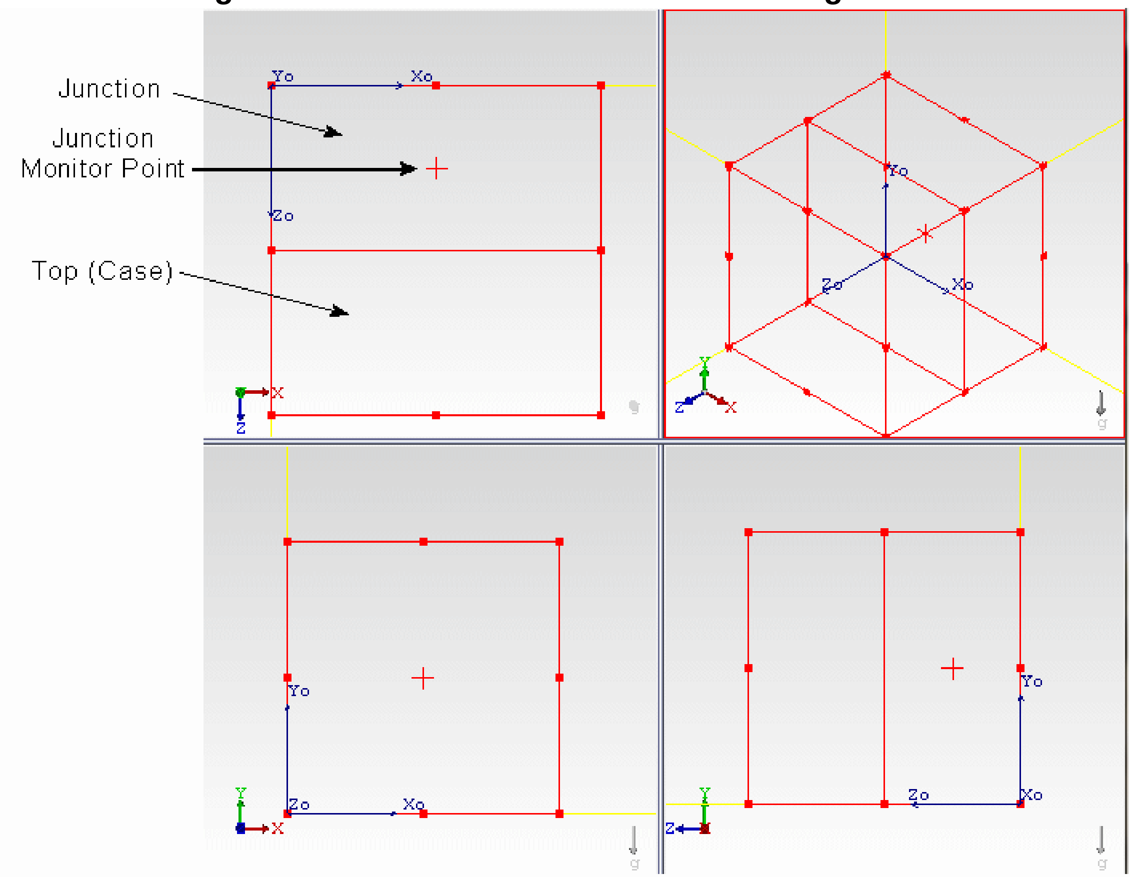


Figure 4-9. General Model Area Array in the Drawing Board

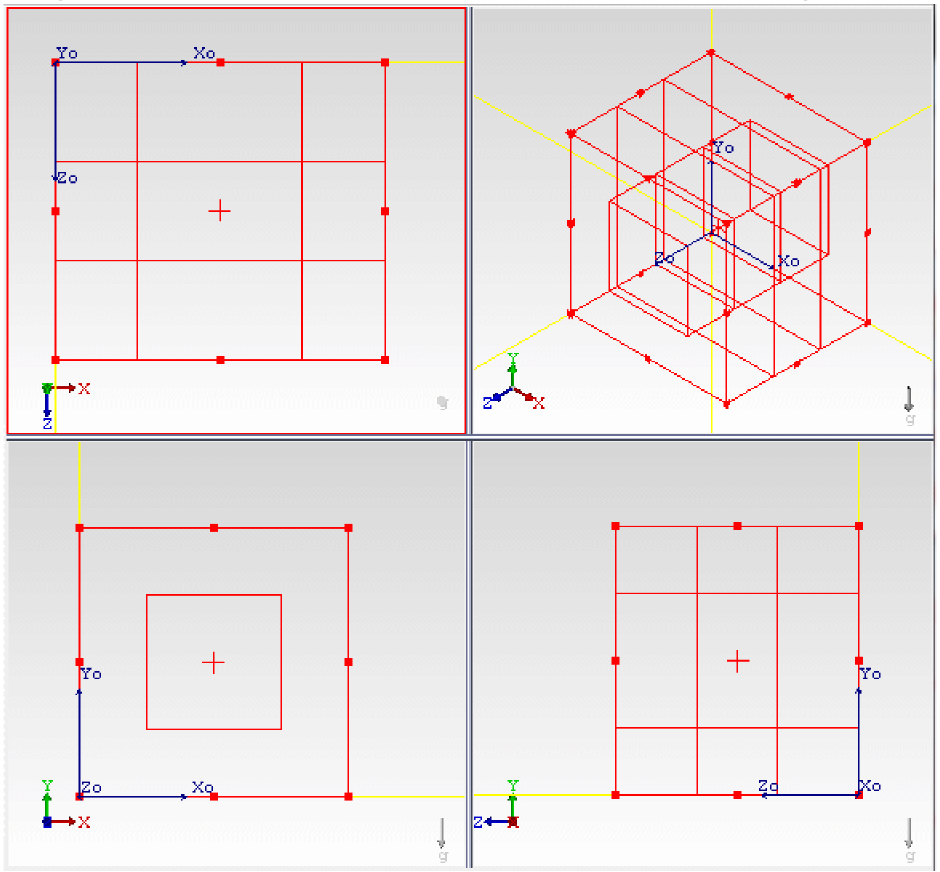
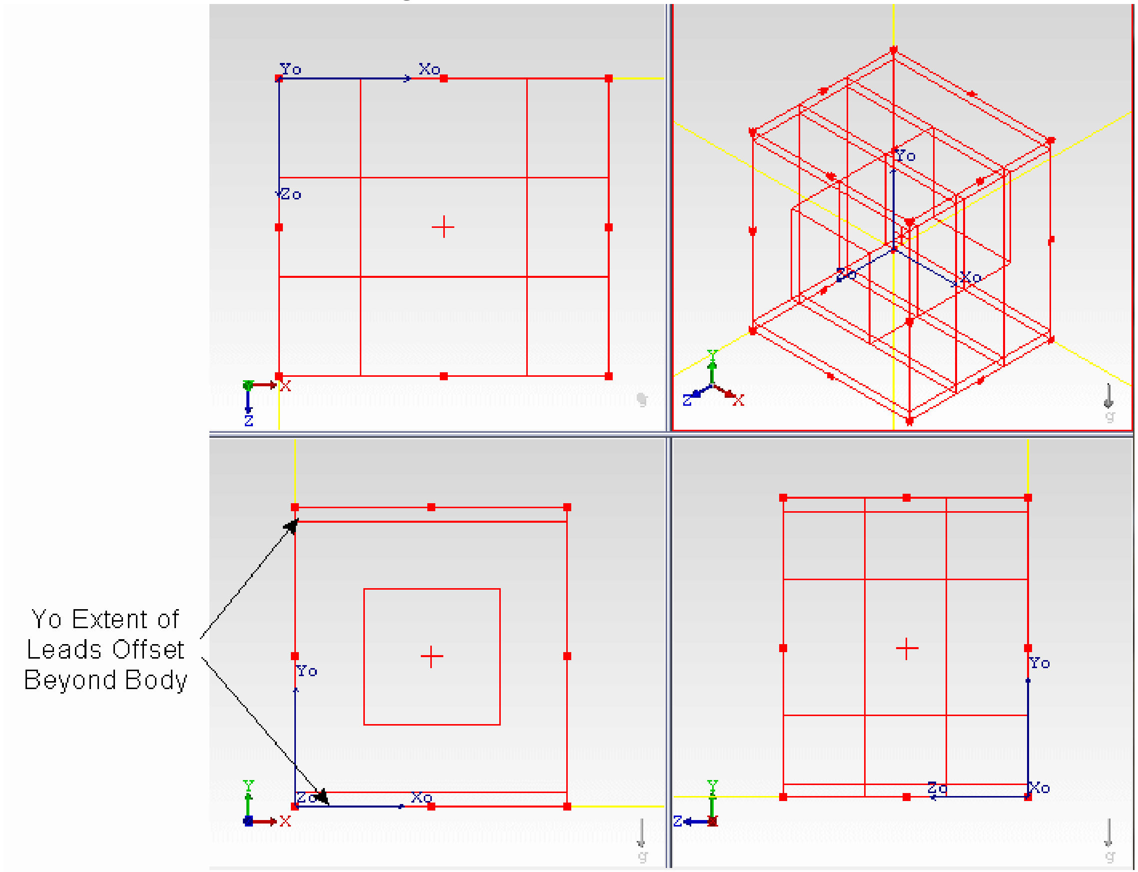


Figure 4-10. General Model Peripheral Package Type in the Drawing Board Showing Leads Offset in Yo Direction



Related Topics

Modeling Compact Components

Compact Component Property Sheet

Compact Component Geometry

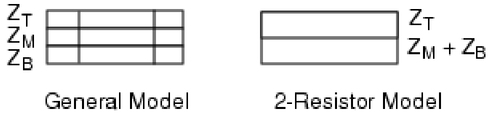
2-resistance models comprise two equal layers. General models comprise three layers where the top and bottom core areas can be separately defined.

Layer Thicknesses

The General and 2-Resistance model options are stored independently ensuring equal height when switching between the two model representations.

The General Model is three cells thick and the 2-Resistor model is two cells thick as shown in Figure 4-11.

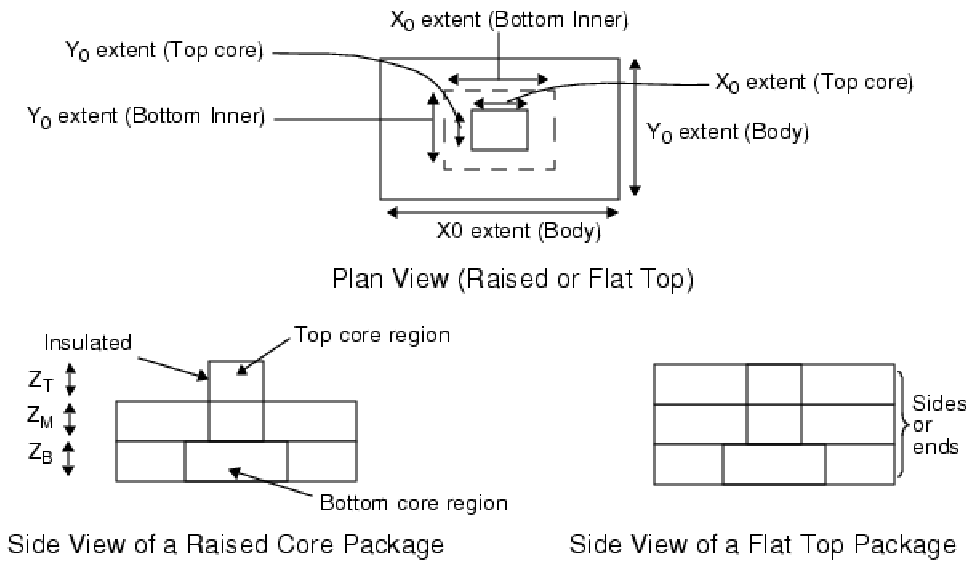
Figure 4-11. Layers of Compact Component Models



General Model Geometry and Cores

A General model package is represented using three layers of cuboids, allowing the possibility of a raised top. This is shown in Figure 4-12.

Figure 4-12. General Model Geometry



The outer region of the top layer is solid (Top outer node) for flat topped packages, and is occupied by the circulating fluid (normally air) for packages with a raised core.

The middle layer is the junction, surrounded by Top outer.

The bottom layer is Bottom inner, surrounded by Bottom outer.

Network Connectivity

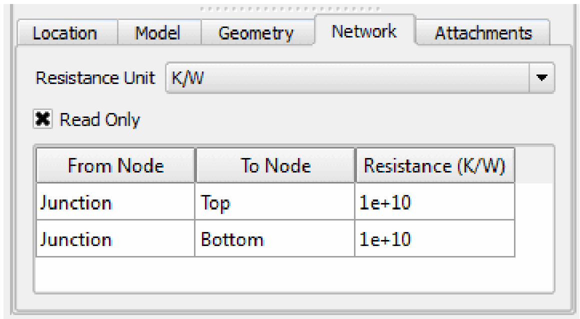
Network connectivity is fixed for 2-Resistor models but unfixed for General models.

Network Connectivity of 2-Resistor Models

Since only two nodes connect to a central junction node, irrespective of the package type, the network connectivity is fixed.

The resistances are defined using the **Network** tab of the Compact Component property sheet as shown in Figure 4-13.

Figure 4-13. Default Resistances for 2-Resistor Model

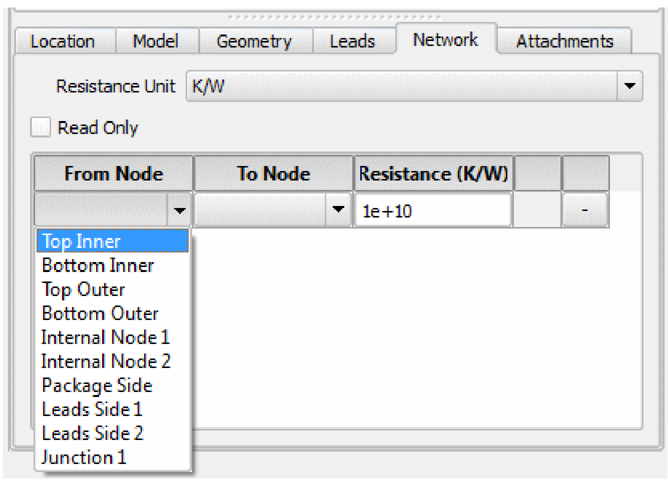


Network Connectivity of General Models

The network connectivity is not fixed, and the table is not populated.

The nodes in the network vary with the options chosen, the number of side nodes and, for peripheral leaded packages, the number of leads nodes. The resistances are defined using the **Network** tab of the Compact Component property sheet as shown in Figure 4-14.

Figure 4-14. Possible Nodes of a Dual-Leaded Package With Two Leads and One Side Nodes



The maximum permissible number of flow paths or links in a general model is calculated by the following:

$$\text{Number of Combinations} = \frac{n!}{(n-2)! \times 2}$$

where n is the number of nodes (excluding the Lead Foot node(s) if present).

For example, a seven-node network can have a maximum of 21 resistances for the main body of the package.

Displaying Compact Component Results

After solution, the temperatures for the package case, PCB, leads (for leaded packages), the displayed junction temperature and the maximum junction temperature (where the model contains more than one junction) can be displayed in tables. Fluxes are shown for the case top, case sides, PCB, and leads (for leaded packages).

Prerequisites

- The Project Manager is in Analyze mode.

Procedure

1. Select a Compact Component.
2. Make sure that the Tables pane is displayed (**Window** > **Launch Tables**).

Results

Two tables of data are provided:

- The Compact Component table shows the junction, top, and bottom temperatures, providing a quick summary of the temperature distribution of the component models included in the simulation.
- The Component Fluxes table gives details of heat transfer mechanism from each face of the package. The sum of the heat output from each package should balance the junction power input.

Related Topics

Reporting Project Data and Results in Tables [Simcenter Flotherm User Guide]

Compact Component Property Sheet

To access: Select a Compact Component SmartPart in the model.

Use this property sheet to define the properties of the Compact Component (compact resistor package).

Description

The **Balls/Pins** tab is not available if Peripheral package type has been selected in the **Model** tab.

The **Leads** tab is not available if Area Array package type has been selected in the **Model** tab.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Model tab.	See "Compact Component Property Sheet Model Tab" on page 58.
Geometry tab.	See "Compact Component Property Sheet Geometry Tab" on page 60.
Balls/Pins tab.	See "Compact Component Property Sheet Balls/Pins Tab" on page 61.
Leads tab.	See "Compact Component Property Sheet Leads Tab" on page 63.
Network tab.	See "Compact Component Property Sheet Network Tab" on page 65.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.

Compact Component Property Sheet Model Tab

To access: Select a Compact Component SmartPart, then select the **Model** tab.

Use this tab to select the package type, configuration (raised/flat top), junction power input and whether to simulate the effects of radiation.

Objects

Field	Description
Model Type	Sets the complexity of the thermal network. There are two options: • 2-Resistor Model — 3 node model requiring the resistances from the junction to the top of the package and to the bottom. • General Model — for highest level of flexibility. See "Modeling Compact Components" on page 45.
Package Type	(General Model) • Area Array — for packages connected to PCB by pins/solder balls. Activates the **Balls/Pins** tab, see Compact Component Property Sheet Balls/Pins Tab. • Peripheral — for packages connected to the PCB by leads. Activates the **Leads** tab, see Compact Component Property Sheet Leads Tab.
Top Style	(General Model) • Flat Topped — for packages with flat tops, that is a top core surrounded by solid. • Raised Core — for components with a raised central region in the top of the package, that is, a top core surrounded by fluid. Raised core packages can only have leads at half-height.
Underfill Material	(General Model) The underfill material filling the standoff. Select an existing Material attribute or create a new one.
Standoff Thickness	(Underfill Material selected) The standoff thickness is the height of the gap between the package body and the board. By default, there is no gap.
Radiation On	Switches the modeling of radiation effects on or off.
Power	(General Model) Select a unit of power.
Power Table	(General Model) Use to set power values for selected junctions up to 20 junctions.
Junction Power	(2-Resistor Model) Sets the junction input power.
Displayed Junction	(General Model) The junction number Monitor Point values that will be displayed in the Profiles and GDA (Analyze mode) windows. Values at only one junction can be displayed.

Related Topics

Modeling Compact Components

Compact Component Property Sheet Geometry Tab

To access: Select a Compact Component SmartPart, then select the **Geometry** tab.

Use this tab to specify the size of the structure. The geometry representation depends on the Model Type chosen in the **Model** tab.

Objects

Field	Description
Body Length Xo and Body Width Yo	The package length and width.
Top Layer Thickness Zo Middle Layer Thickness Zo Bottom Layer Thickness Zo	• 2-Resistor Models — only the Top and Bottom layer thicknesses can be defined. • General Models — all three layer thicknesses can be defined.
Total Body Thickness Zo	The sum total of the layer thicknesses (Read Only).
Top Core Length Xo and Top Core Width Yo	(General model) Sets the dimensions of the central core in the top surface of the package.
Bottom Core Length Xo and Bottom Core Width Yo	(General model) Sets the dimensions of the central core in the bottom surface of the package.

Related Topics

Compact Component Geometry

Compact Component Property Sheet Balls/Pins Tab

To access: Select a Compact Component SmartPart, then select the **Balls/Pins** tab. This tab is displayed when Model Type is set to General Model, and Package Type is set to Area Array, in the **Model** tab.

Use this tab to define the balls/pins connections for an area array package when using a star or general network model.

Description

You are required to make a selection regarding the distribution of the solder balls/pins on the bottom of the package and to provide information of the area(s) enclosing the balls/pins. The thermal resistance of the balls/pins in each region may also be edited.

Objects

Field	Description
Balls/Pins Coverage	Full Coverage — an even distribution over the entire bottom area of the package. Peripheral Coverage — a distribution around the periphery of the package. Inner Coverage — a distribution over one of the inner regions of the package. Both Coverage (that is Peripheral and Inner) — a distribution around the periphery and in the center of the package. Refer to "Solder Balls/Pins Arrays" on page 47.
Number of Side Nodes	The number of nodes used to represent the side surfaces of the package. 1 — the side and end surfaces of the package are modeled as a single network node. 2 — the side and end surfaces of the package are modeled as separate network nodes.
Stand-Off Area of Balls/Pins (Peripheral, Inner, or Both Coverage) Refer to "Solder Balls/Pins Arrays" on page 47.	
Inner Balls/Pins	(Inner or Both Coverage) The area enclosing the inner solder balls. The area is compared to the areas calculated by the SmartPart for Top Inner and Bottom Inner and is applied over the region that most closely represents the area.
Outer Balls/Pins	(Peripheral or Both Coverage) The area enclosed between the inner and outer rows of balls in the periphery region.
Interconnect Resistance to PCB	

Field	Description
Total Balls/Pins	(Full Coverage) Thermal Resistance of the entire array of solder balls.
Inner Balls/Pins	(Inner or Both Coverage) Thermal Resistance of the inner array of solder balls.
Outer Balls/Pins	(Peripheral or Both Coverage) Thermal Resistance of the outer array of solder balls.
Case Temperature	
Calculation Factor	A factor, between 0 and 1, that provides an estimate of the Case temperature from Junction 1 and Top Inner temperatures by: $Case\ Temperature = factor * Junction\ 1\ Temperature + (1 - factor) * Top\ Inner\ Temperature$

Related Topics

Modeling Compact Components

Compact Component Property Sheet Leads Tab

To access: Select a Compact Component SmartPart, then select the **Leads** tab. This tab is displayed when Model Type is set to General Model and Package Type is set to Peripheral in the **Model** tab.

Use this tab to define the lead distribution for general-model, peripheral-package-type compact components.

Objects

Field	Description

Leads Interconnect	The lead arrangement.
	• For Flat Top packages, the choice is:
	  • Quad Leads at Half Height
	  • Quad Leads at Full Height
	  • Dual Leads at Half Height
	  • Dual Leads at Full Height
	  • Mono Leads at Half Height
	  • Mono Leads at Full Height
	• For Raised Core packages, the choice is:
	  • Quad Leads at Half Height
	  • Dual Leads at Half Height
	  • Mono Leads at Half Height
	See “Peripheral Packages” on page 48.
	Note: Flat Top or Raised Core is selected in the **Model** tab.

Number of Leads Nodes	The number of lead nodes in the model: 1 (Mono), 2 (Dual), or 4 (Quad).
Number of Side Nodes	(not Quad Leads at Full Height) The number of nodes used to represent the side surfaces of the package. 1 — represent the side and end surfaces of the package by a single network node. 2 — treat the side and end surfaces of the package as separate network nodes.
Leads Offset Beyond Body	
Xo Extent	(Quad) Sets the Xo lead extent. Quad-leaded packages have leads along the entire length and width of the package and so require both the Xo and Yo extents specified. Normally these will be the same value.

Field	Description
Yo Extent	Sets the Yo lead extent. Mono-leaded packages have leads along the entire length (Xo-direction) of the low Yo side of the package, and Dual-leaded packages have leads along the entire length (Xo-direction) of the body, and so, for these packages, only the Yo extent can be specified. See Figure 4-10 on page 53.
Interconnect Resistance to PCB	
Leads Resistance	(Mono, Dual, or Quad with 1 Lead Node) The parallel thermal resistance sum of all the package leads.
Leads Along Xo	(Quad with 2 or 4 Lead Nodes) The parallel thermal resistance sum of all the package leads along the two Xo-direction sides of the package body.
Leads Along Yo	(Quad with 2 or 4 Lead Nodes) The parallel thermal resistance sum of all the package leads along the two Yo-direction sides of the package body.
Case Temperature	
Calculation Factor	A factor, between 0 and 1, which provides an estimate of the Case temperature from Junction 1 and Top Inner temperatures by: *Case Temperature* = *factor* * *Junction 1 Temperature* + (1 − *factor*) * *Top Inner Temperature*

Related Topics

Compact Component Property Sheet

Compact Component Property Sheet Network Tab

To access: Select a Compact Component SmartPart, then select the **Network** tab.

Use this tab to specify the network resistances between the nodes of the component.

Objects

Field	Description
Resistance	Select units of resistance.
Resistance Table	Tabulate and edit the network resistances for the compact component, see "Network Connectivity" on page 54. The Node options depend on the Model Type set in the Compact Component Property Sheet Model Tab.

Related Topics

Compact Component Property Sheet

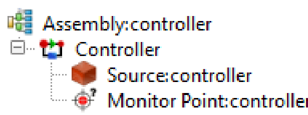
Chapter 5
Controllers

Use the Controller SmartPart to create a frequency-based power controller.

Modeling Controllers

Use Controller SmartParts to change power when set high and low temperatures are reached.

Figure 5-1. Controller in Data Tree Hierarchy



The only objects that can be children of a Controller SmartPart are a source and one or more monitor points.

When there is more than one monitor point child of a controller, you have a choice between using any monitor point to switch between the frequencies, or using all monitor points to switch between the frequencies. See the "THigh Frequency Switching Criteria" description in "Controller Property Sheet Construction Tab" on page 69.

The size of the heat source is defined by a single Source SmartPart, that is, a cuboid shape.

The heat output is defined by a Control attribute: the heat output set by a Source attribute attached to the Source SmartPart is ignored. A Control attribute defines a set of control (Power vs Temperature) curves with high and low temperature thresholds.

After solving for a transient analysis, Controllers results tables show Controller Frequency and Controller Power values.

Related Topics

Frequency-Based Power Control [Simcenter Flotherm Project Attributes Reference Guide]

Controller Property Sheet

To access: Select a Controller SmartPart in the model.

Use this property sheet to configure a controller.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Construction tab.	See "Controller Property Sheet Construction Tab" on page 69.

Controller Property Sheet Construction Tab

To access: Select a Controller in the model then the **Construction** tab.

Use this property sheet tab to attach a Control attribute to the controller and to select the initial power control curve. Also, use this property sheet to find out which power curve is in use if a solve is interrupted or stopped.

Objects

Object	Description
Control	Only active if a Source SmartPart has been attached to the controller. A dropdown list enables you to select a Control attribute, create a new Control attribute or load an attribute from a library. Click **Edit** to open the property sheet of the attribute.
Starting Frequency	The power curve to be used at the start of a solve. You can select any power curve that is defined in the Control attribute.
THigh Frequency Switching Criteria	This selection comes into effect at THigh when there is more than one monitor point child of a controller. • Any Monitor Point − Provided the temperatures of the monitor points are below THigh on the current frequency curve, then when THigh is exceeded by *any* monitor point, the frequency is reduced. In effect, a Boolean OR. • All Monitor Points − Provided the temperatures of the monitor points are below THigh on the current frequency curve, then when THigh is exceeded by *all* monitor points, the frequency is reduced. In effect, a Boolean AND. For more details, refer to Frequency-Based Power Control in the *Simcenter Flotherm Project Attributes Reference Guide*. After switching up or down, the monitor point with the highest temperature is used to extract the power to be applied for the next time step, whichever option is selected.
Current Frequency	If a solve is interrupted, this shows the current frequency in use.

Related Topics

Control Attributes [Simcenter Flotherm Project Attributes Reference Guide]

Chapter 6
Coolers

Use Cooler SmartParts to model Data Center cooling units.

Modeling Coolers

The Cooler SmartPart can represent any row-based or room-based Data Center cooling unit.

There are three main components when modeling a Cooler SmartPart in addition to the positions of the air supplies and extracts:

- Airflow

  You have the options to specify a fixed airflow, a fan curve, a flow rate so as to achieve a fixed temperature difference (across the cooler), or a remote rack temperature control (that is, the cooler's airflow is dependent on the temperature at the inlet of one or more racks).

- Temperature Set Point

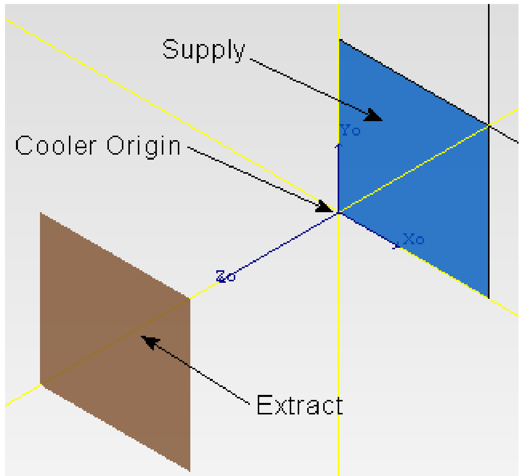
  You can specify whether the set point is a supply or return temperature.

- Capacity Limit

  You can define the maximum power that can be extracted from the airflow at a given temperature

Each supply and extract of a cooler is represented by a planar surface. For identification, a supply is shown on the drawing board in light blue, and an extract in brown, see Figure 6-1.

Figure 6-1. Cooler SmartPart in the Drawing Board

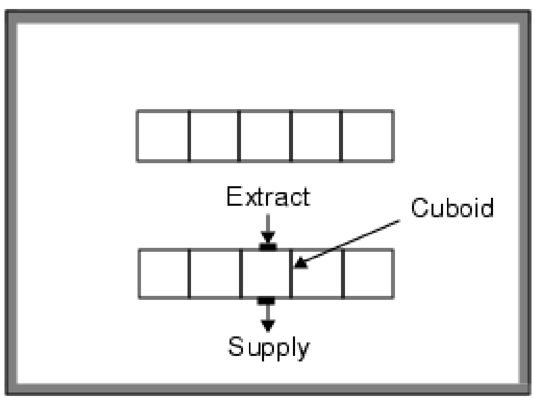


To ensure that the cooler is included in the simulation, the supplies and extracts must sit on the edges of a cuboid, see Figure 6-2.

Note

When using a cooler on the edge of the solution domain or a cutout, ensure that this is not coincident with an open boundary. Either make the boundary a symmetry plane or add a collapsed cuboid on the edge, on which to place the cooler extract or supply.

Figure 6-2. Arrangement of an In-Row Cooler in a Computer Room



It is important that the air cannot move directly from Extract to Supply in the model, therefore, you must introduce a primitive cuboid block to represent the portion of the calculation domain occupied by the cooler.

Using a Temperature Difference Set Point to Control Cooler Airflow

In this case, a Cooler's airflow is controlled by a temperature difference from the set point.

Where greater airflow is required to achieve the set point, the airflow increases, within the maximum and minimum flow rate limits, to lower the temperature difference.

If the difference set point cannot be reached because of a capacity limitation, the cooler operates at maximum airflow, but will fail to achieve the desired set-point temperature. In this case, the power extracted from the airflow is determined from the specified capacity limit.

In cases where less airflow is required to achieve the set point, the airflow decreases to increase the temperature difference. If the unit cannot achieve the specified set point, the airflow is fixed at the minimum airflow limit and the set point temperature may or may not be achieved based on the specified capacity limit.

Using Remote Rack Temperature Control to Control Cooler Airflow

In this case, a Cooler's airflow is controlled by the inlet temperature of one or more Racks that belong to the same Cooler Group.

The cooler and *at least one* Rack SmartPart must be assigned to the same Cooler Group.

This control can be based on the average rack inlet temperature, or the temperature at the 2/3-height of the rack.

The cooler airflow is reduced to the minimum value, subject to the Maximum and Minimum Flow Rate constraints, at which the rack or racks remain at or below the Target Temperature.

If *more than one rack* belong to the same Cooler Group, then the cooler airflow is determined by *the hottest rack* in the group.

If more than one cooler belong to the same Cooler Group, then the airflow of all coolers increase or decrease linearly together from their Minimum Flow Rate to their Maximum Flow Rate. In this case, the Target Temperature must be set to the same value for all coolers in the Cooler Group.

Note

When either of the fan curve options is selected, the change in pressure (ΔP) used to compute the volume flow rate is calculated by averaging over all the extract and supply instances.

Related Topics

Cooler Groups

Creating New Coolers

When Coolers are created, they have Extract and Supply SmartParts added as children.

Procedure

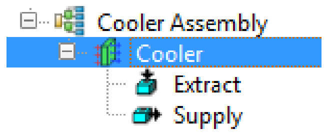
1. Select an assembly in the data tree.

2. In the New Object Palette, choose Project Manager Create or Drawing Board Create and double-click the **Cooler** icon.
3. By default, coolers have no structure, but a cuboid can be colocated with a cooler's bounding box by selecting the cooler and then adding a cuboid.

Results

A Cooler is created with an Extract and a Supply (see Figure 6-3), which are separate objects with their own property sheets. More Supplies and Extracts can be added to the Cooler as children, but there must always be at least one supply and extract for each cooler.

Figure 6-3. Cooler SmartPart in the Data Tree



The bounding box of a cooler can be selected and moved, but not resized, because the overall size is determined by the sizes of the extract and supply entities that the bounding box contains.

The display of a cooler in the drawing board is dependent on the data tree. For example, collapsing the cooler in the data tree displays the bounding box in the drawing board, whilst expanding the cooler in the data tree displays the supply and extract in the drawing board.

When the cooler is expanded, the supply and extract become visible and can be separately edited.

As the supply and extract components of a cooler are always attached to a solid, they are not transferred to MCAD Bridge and hence not included in outputs to external MCAD format files.

Related Topics

Extracts

Supplies

Creating New Recirculation Devices

Capacity Curve

Cooler Property Sheet

Cooler Groups

Cooler groups are used when a cooler's airflow is set to be controlled by remote rack temperature.

Attaching Cooler Group Names to Coolers and Racks

When using Rack temperature to control Cooler airflow, at least one Cooler and one Rack must be assigned to the same Cooler Group.

Procedure

1. Select the Cooler or Rack and open the **Groups** tab of the property sheet.
2. Open the **Cooler Group** dropdown menu and either select an existing name or choose **Create New** to open the Create Group dialog box and enter a new name.

Related Topics

Generic Property Sheet Groups Tab

Create Group Dialog Box

Modeling Coolers

Checking Cooler Groups in Data Tree Summary Information

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

Prerequisites

- The Cooler Group check box in the **Summary** tab of the User Preferences dialog box is checked.

Procedure

1. Select part of the data tree and either click the **Show Summary** icon or press the I key.
2. Optionally, to improve readability by filtering the data tree, do the following:
    a. Press Ctrl+F to open the Find dialog box. Stay in the **Quick Criteria** tab.
    b. For Type, select Cooler or Rack.

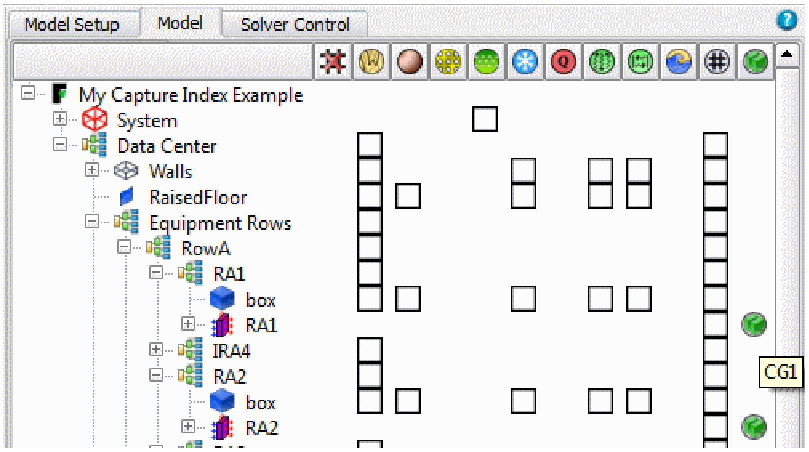
c. Check the Filter check box.

d. Click **Find**.

Results

Hover text displays group names, for example, see Figure 6-4.

Figure 6-4. Display of Cooler Groups in the Data Tree Summary



Capacity Curve

A capacity curve provides the capability to manually define, or import data from a CSV file, a curve definition defining Cooler Capacity limit (power) versus Return Temperature.

Running at Less Than Maximum Flow Rate

The Cooler Capacity vs. Return-Temperature curve for a real cooler is almost always measured at maximum airflow rate.

However, there are times when a cooler runs at less than maximum airflow rate. In these cases, the cooler capacity will be reduced relative to the maximum-airflow performance. An adequate approximation for this reduction is:

$$C_2 = C_1 \times (Q_2 \div Q_1)^k$$

where:

C_1 is the capacity at the maximum airflow rate and C_2 is the capacity at the reduced airflow rate.

Q_1 is the maximum airflow rate and Q_2 is the reduced airflow rate.

k is a constant between 0 and 1:

- When $k = 0$, the capacity is independent of airflow rate.
- When $k = 1$, the capacity scales linearly with airflow rate.
- The default value is 0.8.

This additional scaling is added to the data model.

Related Topics

Defining a Cooler Capacity Curve

Creating New Coolers

Defining a Cooler Capacity Curve

A cooler capacity curve can be defined either by importing a CSV file or adding points interactively. Such data is available from manufacturers.

Prerequisites

- A Cooler SmartPart is being edited and the Capacity Limit is set to Variable in the **Construction** tab of the property sheet:

Procedure

1. Click the Capacity Curve **Click to Edit** button.

   The Capacity Curve dialog box is opened.

2. Select the desired units for Temperature Unit and Power Unit.

3. Depending on the how you want to create the capacity curve, choose one of the following methods.

If you want to...	Do the following:
Enter data manually.	1. Enter a pair of values for Return Air Temperature and Sensible Cooling Capacity. 2. Click the + button to add another row in the table and enter another pair of values. 3. Continue adding data in this way. The chart is updated as you enter data. You can delete rows by clicking the - button.
Enter data by CSV file import.	1. Click **Import CSV File**. 2. Navigate to, and select, a valid CSV file and click **Open**.

4. Click **OK** to save the data and close the Capacity Curve dialog box.

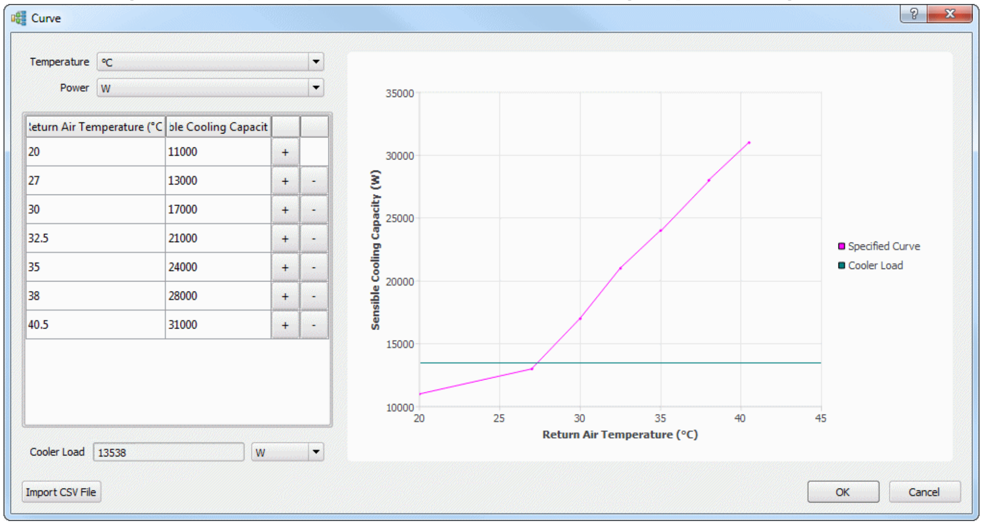
Results

After importing from a CSV file, the capacity curve is plotted using the values from the file.

If the cooler capacity is insufficient to achieve the specified set point, the relevant (supply or return) temperature “floats” as required. For example, if the cooler has insufficient capacity to meet a specified supply temperature, the actual supply temperature will be warmer such that the rate of heat extraction from the air will be equal to the maximum cooler capacity at the current return temperature.

On completion of the solution, the Capacity Curve dialog box is updated to show the operating point for the cooler unit as the Cooler Load value and, if possible, as a line on the curve, see Figure 6-5.

Figure 6-5. Capacity Curve Showing Operating Point



Related Topics

Capacity Curve Dialog Box

Capacity Curve CSV File Format

Capacity Curve CSV File Format

A comma-separated point-pair curve text data file.

Use this file to create a Cooler SmartPart capacity curve by file import as an alternative to manually adding data directly into the Capacity Curve dialog box.

Format

A Capacity Curve CSV File must conform to the following formatting and syntax rules:

- The units are those currently set by the dialog box.
- The list can be any length.
- The values are in floating point format.
- The file format is two columns of data separated by a comma.

```
<return_air_temperature>,<cooling_capacity>
<return_air_temperature>,<cooling_capacity>
...
```

Parameters

- return_air_temperature

  The return air temperature.

- cooling_capacity

  The cooling capacity at the return air temperature.

Examples

The following is an example where the return air temperature is in degC and the cooling capacity is in Watts.

```
20,11000
27,13000
30,17000
32.5,21000
35,24000
38,28000
40.5,31000
```

Related Topics

Defining a Cooler Capacity Curve

Capacity Curve Dialog Box

Capacity Curve Dialog Box

To access: In the **Construction** tab of a Cooler property sheet, click the Capacity Curve **Click to Edit** button. This button is only visible when Capacity Limit is set to Variable.

Use this dialog box to define a cooler capacity curve from a sequence of points.

Objects

Field	Description
Temperature Unit	The units of Return Air Temperature used.
Power Unit	The units of Sensible Cooling Capacity used.
Table	Pairs of values of Return Air Temperature and Sensible Cooling Capacity.
Import CSV File	Click to import a CSV file.

Related Topics

Defining a Cooler Capacity Curve

Capacity Curve CSV File Format

Cooler Property Sheet

Cooler Property Sheet

To access: Select a Cooler SmartPart in the model.

Use this property sheet to configure a cooler.

Objects

Object	Description
Location tab.	See “Generic Property Sheet Location Tab” on page 28.
Construction tab.	See “Cooler Property Sheet Construction Tab” on page 83.
Attachments tab.	See “Generic Property Sheet Attachments Tab” on page 30.
Groups tab.	See “Generic Property Sheet Groups Tab” on page 31.

Related Topics

Extract Property Sheet

Supply Property Sheet

Cooler Property Sheet Construction Tab

To access: Select a Cooler SmartPart, then select the **Construction** tab.

Use this property sheet tab to specify the flow, set point and capacity of a Cooler SmartPart.

Objects

Object	Description
Flow Type	The flow rate of air passing through the cooler is defined by one of the following: • Fixed — The default setting. A fixed volume flow rate. The total volume flow rate is divided proportionally, by area, between all the supply entities. The total volume flow rate is similarly divided between all the extract entities. • Curve — A variable flow rate, defined by a characteristic curve (flow rate dependent on pressure). • Temperature Difference Set Point — A variable flow rate, controlled by a temperature difference across the cooler, relative to the specified Supply or Return temperature. See "Using a Temperature Difference Set Point to Control Cooler Airflow" on page 72. • Remote Rack Temperature Control — A variable flow rate, controlled by the inlet temperature of one or more Racks that belong to the same Cooler Group. See "Using Remote Rack Temperature Control to Control Cooler Airflow" on page 73.
Volume Flow Rate	(Curve) The volume flow rate at 0 (zero) pressure. Defines one of the end points of the characteristic curve.
Pressure	(Curve) The pressure at 0 (zero) volume flow rate. Defines one of the end points of the characteristic curve.
Fan Curve Chart	(Curve) Click **Click to Edit** to open the Fan Curve Chart Dialog Box.
Temperature Difference	(Temperature Difference Set Point) The temperature difference across the Cooler that the Cooler tries to achieve by adjusting the flow rate (within the limits specified by Maximum and Minimum Flow Rate). For example, if 20 degC is specified at the Supply (Temperature Set Point), and the Temperature Difference is specified as 11 degC, then the Cooler will try to achieve a Return temperature of 31 degC. The default is 11.111 degC (20 degF).
Rack Inlet Temperature	(Remote Rack Temperature Control) The location of the remote rack target temperature: • Average — use the average inlet temperature for the rack. • 2/3 Rack Height — use the inlet temperature at 2/3 the height of the rack, that is, 1/3 of the height from the top of the rack.

Object	Description
Maximum and Minimum Flow Rate	(Temperature Difference Set Point and Remote Rack Temperature Control) When the Cooler flow rate varies, it does so within the limits specified here.
Target Temperature	(Remote Rack Temperature Control) The temperature at the remote rack inlet that the Cooler tries to achieve by adjusting its flow rate (within the limits specified by Maximum and Minimum Flow Rate).
Temperature Set Point	
Location	The location of the set point temperature: • Supply — The temperature will be that of the air issued from the Cooler. • Return — (Fixed or Curve) The temperature will be that at the return (Cooler Extract) that the Cooler tries to achieve, by adjusting the Supply temperature.
Temperature	The cooler unit attempts, within capacity and minimum and maximum airflow constraints, to maintain this temperature at the Location specified in the previous field. The default set point is 20 degC (68 degF).
Capacity	
Capacity Limit	The capacity limit is the maximum power that can be extracted from the airflow at a given return temperature, and can be specified in one of three ways: • None — The cooler has infinite capacity, and always achieves the desired supply or return set point. • Fixed • Variable — The cooler capacity depends on the return air temperature relationship, defined by a capacity curve.
Power	(Fixed) The cooler capacity is fixed to this value.
Airflow	(Variable) Defines the value of Q_2 used in the formula: $C_2 = C_1 \times (Q_2 \div Q_1)^k$
Airflow Reference Exponent	(Variable) Defines the value of k in the formula: $C_2 = C_1 \times (Q_2 \div Q_1)^k$
Capacity Curve	(Variable) Click **Click to Edit** to open the Capacity Curve Dialog Box.

Related Topics

Running at Less Than Maximum Flow Rate

Chapter 7
Cuboids, Prisms, Tets, and Inverted Tets

Use cuboids, prisms, tets (tetrahedra), and inverted tets, collectively referred to as primitives, to model structures.

Primitive Usage

Primitives are used to model structures whose shapes are considered hydraulically equivalent by setting dimensions to achieve the same surface area.

They can also be used in combination, to build up detailed thermally non-uniform structures for which there is not an equivalent SmartPart.

Cuboids can be collapsed to two dimensions to represent planar surfaces, see "Collapsing 3D Cuboids to 2D Planes" on page 23.

If cuboids have Holes added as child objects, they become Blocks with Holes SmartParts, see "Blocks With Holes" on page 39.

The default dimensions of primitives are one-tenth the size of the solution domain in the three coordinate directions.

Note

Primitives can be used in a negative way, for example, to block out regions of an enclosure with which there is no concern for a particular analysis. Within a computer, it may not be required to obtain solution in the case containing the disk drives.

Primitive Attributes

By default, primitive primitives are created as smooth, non-conducting solids.

Attach physical definition attributes to complete the definition of the block. Some attributes can be attached to one or all sides of the primitive.

Heat Transfer

Full conjugate heat transfer, that is, conduction in all directions, as well as convection and radiation from all sides is available by default for 3D cuboids.

Heat Transfer for Collapsed Cuboids

To model full conjugate heat transfer for collapsed cuboids aligned with the grid, select Activate Plate Conduction in the Solver Options property sheet before solving. The heat conduction along and across the collapsed cuboid will then be computed. When Activate Plate Conduction is not activated, only heat transfer in the collapse direction is modeled.

Note

For 2D cuboids not aligned with the grid, Activate Plate Conduction has no effect. Only heat transfer normal to the surface (that is, through the cuboid) is modeled for 2D cuboids.

See Construction Precedence Rules in the *Simcenter Flotherm User Guide* for information on the thermal effects at the boundaries of objects.

Primitives and Solar Radiation

Depending on the transparency definition in the materials attached to cuboids, solar radiation will be either fully blocked, partially blocked/absorbed or fully transparent to solar radiation.

Prisms, unlike cuboids, are either fully transparent or fully opaque to solar radiation, depending on whether the attached material is marked as transparent or not. There are no intermediate states (that is, no absorption of solar radiation).

Primitive Property Sheet

To access: Select a primitive (Cuboid, Prism, Tet, or Inverted Tet) in the model.

Use this property sheet to configure a cuboid, prism, tet, or inverted tet.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.

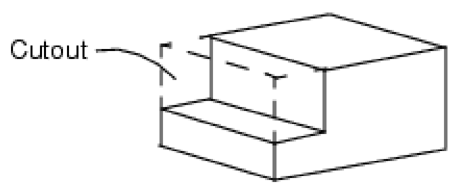
Chapter 8
Cutouts

If the required solution domain is not of a regular cuboid shape, then use Cutout SmartParts to remove sections of the solution domain.

Cutout Usage

A cutout defines the size and location of regions of the domain regarded as external to the model and, therefore, excluded from the program solution.

Figure 8-1. Cutouts in the Solution Domain



The default Cutout cuts out a cube from the solution domain one-tenth the size of the domain in each direction and located at the origin of the selected assembly.

Cutout Boundaries

The faces of a cutout are open by default, but symmetry boundaries can be set using the **Boundaries** tab of the property sheet.

Related Topics

Cutout Property Sheet

Cutout Property Sheet

To access: Select a Cutout in the model.

Use this property sheet to configure a cutout.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Boundaries tab.	See "Cutout Property Sheet Boundaries Tab" on page 91.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.

Cutout Property Sheet Boundaries Tab

To access: Select a Cutout in the model then the **Boundaries** tab.

Use this property sheet to define all or individual (six) faces of the Cutout and their ambient attribute(s).

Objects

Field	Description
Faces	• Open — an open face of constant pressure through which air can flow. • Symmetry — a frictionless, impermeable, and adiabatic planar surface through which neither air nor heat can flow.
Ambient	If required, attach an ambient attribute to all or individual faces.

Usage Notes

By default the Faces and Ambient settings apply to all faces. To define individual faces and ambients, expand the fields to include definitions for each of the six faces: Xo High, Xo Low, Yo High, Yo Low, Zo High, and Zo Low.

Related Topics

Examples of Open and Symmetry Boundaries [Simcenter Flotherm User Guide]

Chapter 9
Cylinders

Use Cylinder SmartParts to model curved volumes.

Cylinder Properties

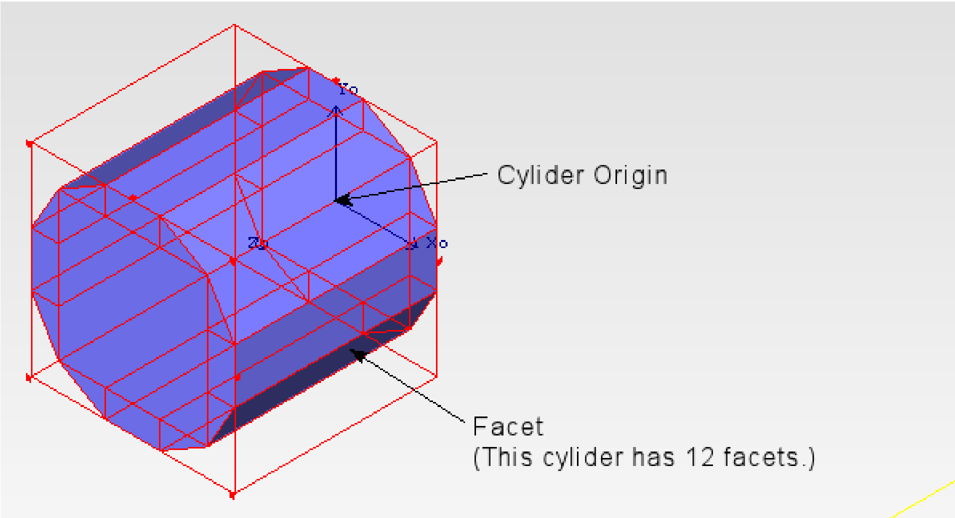
The Cylinder SmartPart uses a combination of primitives (cuboids and prisms) to approximate a cylindrical shape. Cylinder SmartParts are added to the data tree as children of assemblies.

Cylinder Size and Shape

The size of a cylinder is determined by the radius and length in the coordinate directions indicated, relative to its own origin.

The origin of a cylinder is at the low co-ordinate diameter center, see Figure 9-1.

Figure 9-1. Cylinder SmartPart



By default, a new cylinder has a length and diameter one-tenth the dimension of the solution domain in the Z-direction and Y-directions respectively.

The modeling level is the number of facets, which defines the precision of the cylindrical shape.

Cylinder Modeling Precision Considerations

If a cylinder is relatively large compared to the solution domain, (for example, a large electrolytic capacitor in a power supply unit), then a high precision can often be used without significantly increasing the number of grid cells.

If the spacing between the cylinder and other objects has an important effect on the flow through the system, then a precise representation of the cylinder may be needed to give the correct inter-object spacing.

If, however, the cylinder is small compared to the size of the solution domain and does not form an important part of the thermal management strategy of the system, then it can often be represented as a single cuboid.

Exchanging Heat With Surroundings

By default, cylinders do not exchange heat with their surroundings, however, they can do so by attaching a material attribute with conductivity.

To fix a surface temperature, set the material to have a high conductivity and define a source in the body of the cylinder with a fixed temperature. A fixed dissipation can be achieved in a similar manner by setting the dissipation from the source.

Related Topics

Cylinder Property Sheet

Cylinder Property Sheet

To access: Select a Cylinder SmartPart in the model.

Use this property sheet to configure a cylinder.

Objects

Object	Description
Location tab.	See “Generic Property Sheet Location Tab” on page 28.
Construction tab.	See “Cylinder Property Sheet Construction Tab” on page 96.
Attachments tab.	See “Generic Property Sheet Attachments Tab” on page 30.

Cylinder Property Sheet Construction Tab

To access: Select a Cylinder in the model then the **Construction** tab.

Use this property sheet tab to modify the representation of cylindrical components, such as capacitors.

Objects

Field	Description
Radius	The radius of the cylinder.
Length	The length of the cylinder.
Modeling Level	The number of facets (flat surfaces) of the cylinder (default is 12) which model a cylinder’s curved surface.

Related Topics

Cylinder Properties

Chapter 10
Dies

Use Die SmartParts to model dies, the rectangular sections of silicon that are the heart of a package.

Applications of Die SmartParts

Die SmartParts can be used to investigate changes in die power distribution and packing.

Investigating Changes in Die Power Distribution

The Die SmartPart can be used to investigate the effect of variations in the distribution of power dissipation on the surface of a die. The results may be used to determine the best location for functional groups of transistors or check whether the resulting thermal performance is acceptable.

The descriptions of where the powered functional groups are placed can be defined directly, or in other EDA design software and transferred to Simcenter Flotherm using CSV files.

Investigating Changes in Packaging

The Die SmartPart can be used to investigate the effect of placing a die in different packaging environments and provide insights into the dependent maximum power dissipation.

Related Topics

Creating Dies

Die Properties

Modeling Dies

Die Property Sheet

Creating Dies

Dies can be modeled as uniform or non-uniform sources of heat.

Procedure

1. Select an assembly in the data tree.
2. In the New Object Palette, choose Project Manager Create or Drawing Board Create and double-click the **Die** icon.

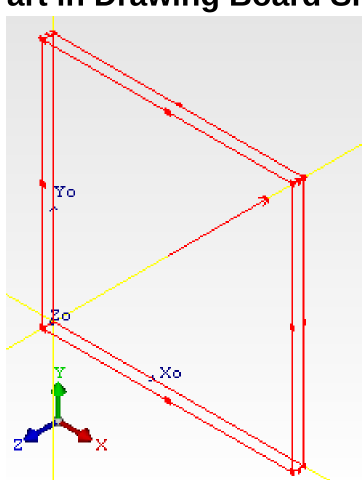
   By default, a die is modeled with a single uniform source spread across its surface.
3. To define a non-uniform distribution, open the **Construction** tab of the property sheet, change Power Dissipation Type from Uniform Dissipation to Non-Uniform Dissipation and either define the power distribution based on total coverage or by defining discrete sources.

Results

By default, the Die SmartPart is located at the origin of the parent assembly with a physical size 4.9 mm × 4.9 mm × 0.22 mm, with a defined material type of Silicon (Pure).

A die is displayed in the drawing board showing the source direction, see Figure 10-1.

Figure 10-1. Die SmartPart in Drawing Board Showing Source Direction



Dies can be decomposed. Decomposed dies comprise a Cuboid and one or more Sources, depending on the source distribution over the surface of the die.

Related Topics

Decomposing SmartParts

Applications of Die SmartParts

Die Properties

Modeling Dies

Die Property Sheet

Die Properties

By default, the Die SmartPart has a Silicon (Pure) Material attribute attached via the **Construction** tab of the property sheet, rather than the **Attachments** tab.

The Silicon (Pure) Material attribute has a Polished Plate Aluminum Surface attribute, which is defined in the **Attachments** tab.

Related Topics

Applications of Die SmartParts

Creating Dies

Modeling Dies

Die Property Sheet

Modeling Dies

The Die SmartPart is modeled as a cuboid with one or more 2D source objects on the Zo high face of the cuboid representing discrete or total coverage sources.

Modeling Non-Uniform Dissipation Using Discrete Sources

Non-uniform power dissipation is modeled by defining the power output at specified rectangular locations, termed discrete sources.

Procedure

1. In the Die Construction property sheet, select Discrete Sources Source Type.
2. Click the Source Definition **Click to Edit** button.

   The Non-Uniform Discrete Source Definition dialog box is opened.
3. Select the desired units for Power Unit and Size Length Unit.
4. Depending on the how you want to enter data, choose one of the following methods.

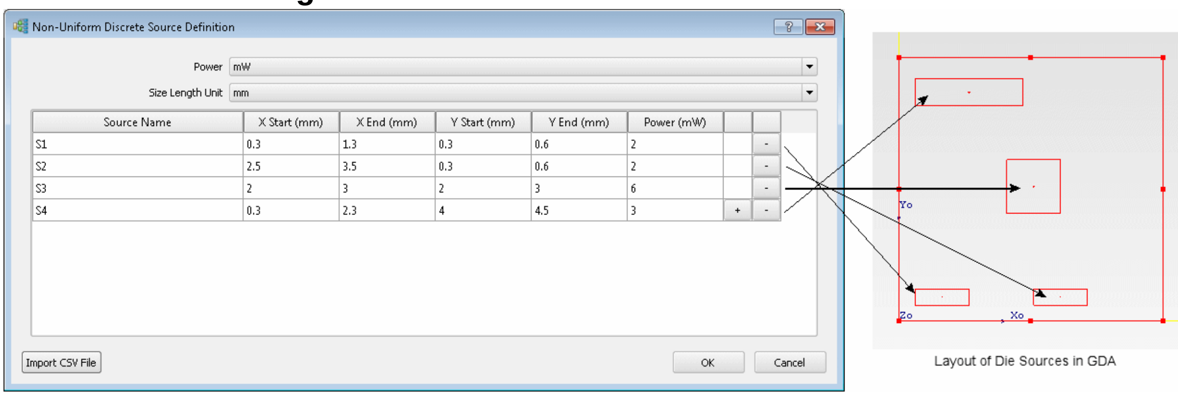
If you want to...	Do the following:
Enter data manually.	**Note:** By default, a single source, DiscreteSource1 of dimensions 1 mm × 1 mm and no specified power is created. 1. Edit the default DiscreteSource1 row. 2. If another source is to be defined, click the + button to add another row in the table. 3. Enter another set of values. 4. Continue adding data in this way. The chart is updated as you enter data. You can delete rows by clicking the - button.
Enter data by CSV file import.	1. Click **Import CSV File**. 2. Navigate to, and select, a valid CSV file and click **Open**.

5. Click **OK** to save the data and close the Non-Uniform Discrete Source Definition dialog box.

Results

- The Total Power value in the **Construction** tab of the Die property sheet is updated to show the total calculated power.
- The drawing board shows the layout of the sources on the Die surface, for example, see Figure 10-2.

Figure 10-2. Placement of Discrete Sources



Related Topics

Discrete Source Definition CSV File Format

Non-Uniform Discrete Source Definition Dialog Box

Modeling Non-Uniform Dissipation Using Total Coverage Sources

Non-uniform power dissipation is modeled by defining the power output at selected rectangular locations across the face of the die.

Procedure

1. In the Die Construction property sheet, select Total Coverage Sources Source Type.
2. Click the Source Definition **Click to Edit** button.

   The Non-Uniform Total Coverage Source Definition dialog box is opened.
3. Select the desired units for Power Unit.

4. Depending on the how you want to enter data, choose one of the following methods.

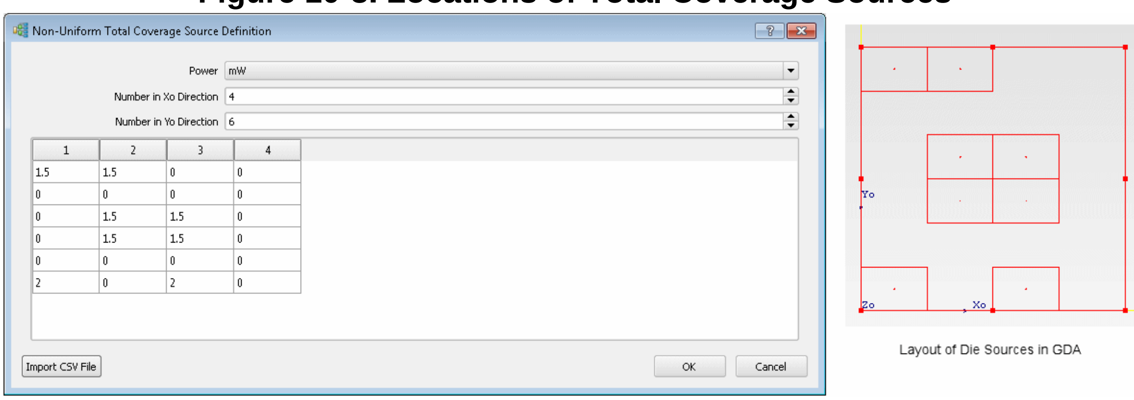
If you want to...	Do the following:
Enter data manually.	1. Enter values for Number in Xo Direction and Number in Yo Direction. The number of table rows and columns changes. 2. Enter power values in the table as required.
Enter data by CSV file import.	1. Click **Import CSV File**. 2. Navigate to, and select, a valid CSV file and click **Open**.

5. Click **OK** to save the data and close the Non-Uniform Total Coverage Source Definition dialog box.

Results

- The Total Power value in the **Construction** tab of the Die property sheet is updated to show the total calculated power.
- The drawing board shows the layout of the sources on the Die surface, for example, see Figure 10-3.

Figure 10-3. Locations of Total Coverage Sources



Related Topics

Total Coverage Source Definition CSV File Format

Non-Uniform Total Coverage Source Definition Dialog Box

Dies in Application Windows

The following subsections give information about Die SmartParts represented in results tables, the Command Center, and MCAD Bridge.

Die Results in Tables

Power Dissipation and Maximum Die Temperature data are available after solution in the Dies table when in Analyze mode. If the die has non-uniform sources, then results data is shown on separate rows for each source.

Input/Output Settings for the Command Center

The following variables are available as input variables to the Command Center:

- Absolute Location
- Activated
- Localized Grid
- Grid Constraints
- Parametric Data/Power Dissipation(s)

The following variables are available as output variables for the Command Center:

- Power Dissipation
- Maximum Temperature

Representation in MCAD Bridge

When a Die SmartPart is transferred to MCAD Bridge, it is represented as a single cuboid.

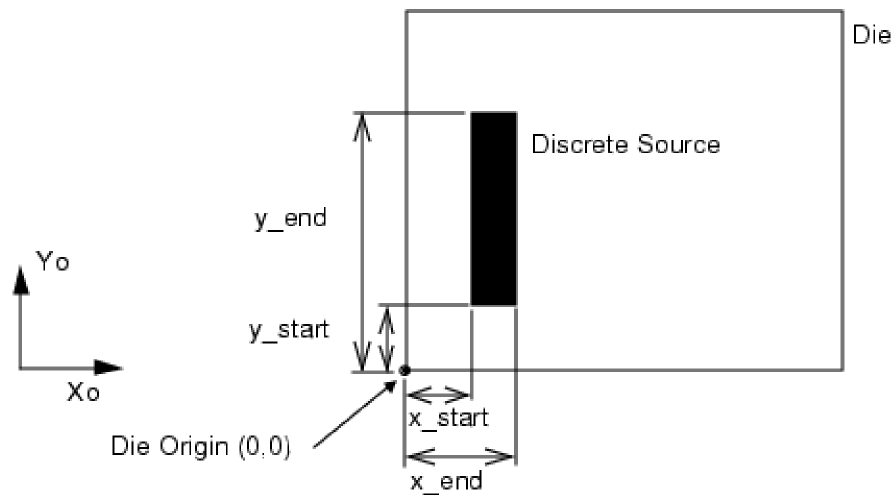
Discrete Source Definition CSV File Format

A comma-separated text data file.

Use this file to create a Die power distribution map by file import as an alternative to entering data into the table of the Non-Uniform Discrete Source Definition dialog box.

The position of a discrete source is defined relative to the Die origin as shown in Figure 10-4.

Figure 10-4. Discrete Source Position Definition



Format

A Discrete Source Definition CSV File must conform to the following formatting and syntax rules:

- The units are those currently set by the Non-Uniform Discrete Source Definition dialog box.
- A discrete source is created for each line present in the imported file.
- The list can be any length.
- The values are in floating point format.

```
<source_name>,<x_start>,<x_end>,<y_start>,<y_end>,<power>
<source_name>,<x_start>,<x_end>,<y_start>,<y_end>,<power>
...
```

Parameters

- source_name

  A name for the source.

- x_start

  The X coordinate of the bottom left corner of the source.

- x_end

  The X coordinate of the top right corner of the source.
- y_start

  The Y coordinate of the bottom left corner of the source.
- y_end

  The Y coordinate of the top right corner of the source.
- power

  The power of the source.

Examples

The following is an example of four sources where the length units are in mm and the power is in mW.

```
S1,0.3,1.3,0.3,0.6,2
S2,2.5,3.5,0.3,0.6,2
S3,2,3,2,3,6
S4,0.3,2.3,4,4.5,3
```

Related Topics

Non-Uniform Discrete Source Definition Dialog Box

Modeling Non-Uniform Dissipation Using Discrete Sources

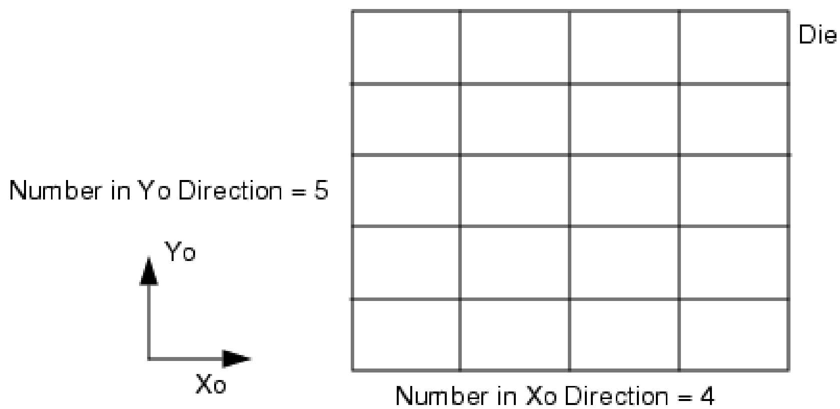
Total Coverage Source Definition CSV File Format

A comma-separated text data file.

Use this file to create a Die power distribution map by file import as an alternative to manually adding data directly into the Non-Uniform Total Coverage Source Definition dialog box.

The position of a source is defined as a “grid cell” on the Die as shown in Figure 10-4.

Figure 10-5. Total Coverage Source Position Definition



Format

A Total Coverage Source Definition CSV File must conform to the following formatting and syntax rules:

- The number of columns of data in the file is the number of sources in the X-direction.
- The number of rows of data in the file is the number of sources in the Y-direction.
- The units of power are those currently set by the Non-Uniform Total Coverage Source Definition dialog box.
- The values are in floating point format.

```
<power>,<power>, ...
<power>,<power>, ...
...
```

Parameters

- power

  The value of power in a grid cell where the grid is defined by the number of values in each row by the number of rows.

Examples

The following is an example is for a 4 by 6 grid where the power is in mW.

```
1.5,1.5,0,0
0,0,0,0
0,1.5,1.5,0
0,1.5,1.5,0
0,0,0,0
2,0,2,0
```

Related Topics

Non-Uniform Total Coverage Source Definition Dialog Box

Modeling Non-Uniform Dissipation Using Total Coverage Sources

Die Property Sheet

To access: Select a Die SmartPart in the model.

Use this property sheet to configure a die.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Construction tab.	See "Die Property Sheet Construction Tab" on page 109.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.

Die Property Sheet Construction Tab

To access: Select a Die SmartPart, then select the **Construction** tab.

Use this property sheet tab to define the thermal characteristics of the selected Die SmartPart.

Objects

Field	Description
Size Xo, Yo, Zo	Sets the dimensions of the bounding cuboid of the Die SmartPart in the three coordinate directions.
Die Material	By default the material type is set to Silicon (Pure). To change the material type, either select an existing project material attribute or create a new one.
Power Dissipation Type	• Uniform Dissipation — A single source, with the same Xo and Yo sizes as the cuboid, is located on the Zo-High face of the die cuboid. • Non-Uniform Dissipation — A number of sources located on the Zo-High face.
Total Power	(Uniform Dissipation) The total die dissipation value.
Non-Uniform Source	
Source Type	• Discrete Sources — defined by positions of corners of a rectangles. • Total Coverage Sources — defined by positions in a grid.
Source Definition	Click **Click To Edit** to open a dialog box. • For Discrete Sources, the Non-Uniform Discrete Source Definition Dialog Box is opened. • For Total Coverage Sources, the Non-Uniform Total Coverage Source Definition Dialog Box is opened.
Total Power	(Non-Uniform Dissipation) The total power calculated from the sum of individual sources. Read-only.

Related Topics

Modeling Dies

Applications of Die SmartParts

Non-Uniform Discrete Source Definition Dialog Box

To access: From the Die property sheet **Construction** tab, select Non-Uniform Dissipation as the Power Dissipation Type, then Discrete Sources as the Source Type and Click the Source Definition **Click to Edit** button.

Use this dialog box to define the locations and power values of rectangular sources over the Zo-High face of the die.

Objects

Field	Description
Power Unit	The units of power values entered into the table.
Size Length Unit	The units of start and end dimensions entered into the table.
Import CSV File	Click to open a file browser to select a valid CSV file.
Table containing a row for each power source	
Source Name	The name of the source.
X Start, X End, Y Start, Y End.	The location and size of the source defined by the four corners of a rectangle.
Power	The power of the source.

Usage Notes

- After initial editing, the table is opened in read-only mode.
- Undo and Redo operate on the total changes since opening the dialog box.
- When closing the dialog box, a message is output if validation fails and rows containing invalid values (overlapping, zero area, outside of physical die dimensions) are highlighted in red.

Related Topics

Modeling Non-Uniform Dissipation Using Discrete Sources

Discrete Source Definition CSV File Format

Non-Uniform Total Coverage Source Definition Dialog Box

To access: From the Die property sheet **Construction** tab, select Non-Uniform Dissipation as the Power Dissipation Type, then Total Coverage Sources as the Source Type and Click the Source Definition **Click to Edit** button.

Use this dialog box to define the power output at rectangular cells across the face of the die.

Objects

Field	Description
Power Unit	The units of power values entered into the table.
Number in Xo Direction, Number in Yo Direction	The number of power "cells" in the Xo direction and Yo direction. This produces a matrix of, for example, 4 by 6 equally-sized cells over the entire surface of the die.
Table	A table, representing the surface matrix, whose entries are the power values for each cell.
Import CSV File	Click to open a file browser to select a valid CSV file.

Related Topics

Modeling Non-Uniform Dissipation Using Total Coverage Sources

Total Coverage Source Definition CSV File Format

Chapter 11
Enclosures

Use Enclosure SmartParts to model containers. The size of a new Enclosure is that of the solution domain.

Note

Enclosure SmartParts are useful for modeling internal objects, but avoid using them for objects where the internal flow is immaterial as this will cause unnecessary calculation.

Enclosure Construction

An Enclosure SmartPart is made up of up to six 3D or 2D walls.

Each wall can be:

- Removed (non-existent).
- modeled as 3D (Thick) or 2D (Thin).
- Provided with heat transfer to the ambient surroundings by convection and radiation if the (Thick or Thin) enclosure wall lies either on the domain boundary or a cutout.

Enclosure walls can have holes, which can be:

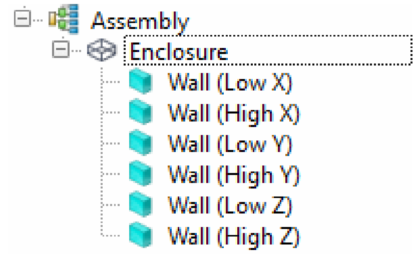
- Used to represent ventilation openings.
- Refined by adding resistance, for example, to represent grilles or louvres.
- Filled with another material of the same or different thickness, to represent changes in the wall.

New Enclosures

Enclosures are created as children of assemblies.

Expand the Enclosure node in the data tree to show the side walls, see Figure 11-1.

Figure 11-1. Expansion of an Enclosure SmartPart in the Data Tree



A new Enclosure has solid non-conducting thin walls coincident with the solution domain.

Enclosure Walls

Each wall of an enclosure can be individually defined.

The default names of the walls are Low X, High X, Low Y, High Y, Low Z and High Z where, for example, Low Y is the wall in the X-Z plane with it's local origin at Y = 0, and High Y is the wall in the X-Z plane with its local origin at Y = <y-height of enclosure>. Select each wall in the data tree to view the wall's local origin.

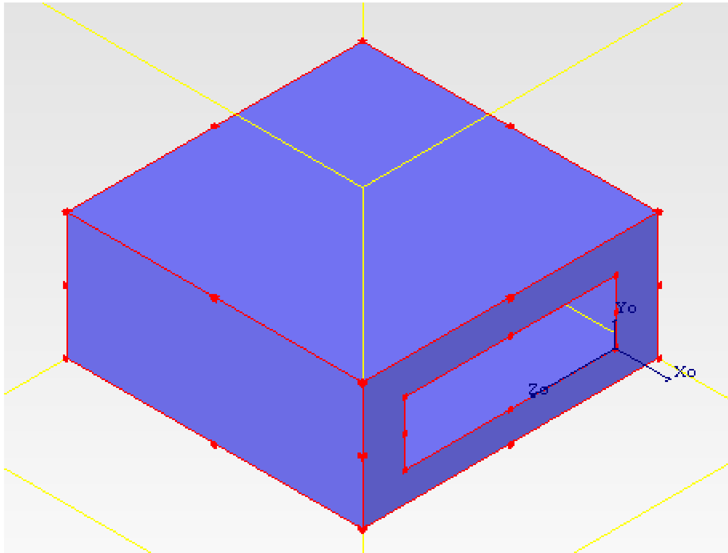
Enclosure Walls With Holes

Holes can be added to enclosure walls to allow the specification of airflow in and out.

Add holes in the enclosure walls for locating vents and fans.

Holes that are 0.01% or less than the smallest side of the enclosure wall are ignored by the grid, the solver, and the decompose command, and are only displayed in wireframe rendering mode.

Figure 11-2. Enclosure With Hole on Thin High X Wall



For more details, see "Creating Holes" on page 193.

Related Topics

Enclosure Modeling

Enclosure Property Sheet

Enclosure Wall Property Sheet

Enclosure Modeling

The following sub sections provide advice when modeling enclosures.

Walls of Multiple Materials

More than one material can be defined for a wall by adding holes and then filling them with different materials, see "Creating Holes" on page 193.

Radiation Exchange Between Enclosure Walls and Other Objects

Radiation exchange between an enclosure and other objects can be applied if the walls are Thick.

If the thick-walled enclosure is co-located with the overall boundary or a cutout, then you can set a Radiant Temperature in an Ambient Attribute property sheet which then enables radiative heat transfer between the enclosure walls and a fixed temperature.

Radiation exchange can be applied on Thin walls if they are located either on the domain boundary or on the edge of a cutout.

To define radiation exchange, attach a Radiation attribute to the enclosure.

Convective Exchange of Enclosure Walls on the Domain Boundary

If the enclosure walls lie on the domain boundary, then convective exchange with the surroundings can be applied for both thick and thin walls.

Thermal Attachments After Enclosure Decomposition

An enclosure can be decomposed into its primitive parts.

Prior to decomposition, any attached thermal attribute defines the enclosure as conducting or non-conducting at a fixed temperature. If a library attribute is attached which has a power output setting, then the power setting is ignored.

Related Topics

Enclosure Property Sheet

Enclosure Wall Property Sheet

Holes

Enclosure Construction

Enclosure Property Sheet

To access: Select an Enclosure SmartPart in the model.

Use this property sheet to configure an enclosure.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Construction tab.	See "Enclosure Property Sheet Construction Tab" on page 118.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.

Enclosure Property Sheet Construction Tab

To access: Select an Enclosure SmartPart, then select the **Construction** tab.

Use this tab to change the definition of the boundary walls of a cuboid space within the calculation domain. The enclosure boundaries can be solid, punctured with holes (which can be filled with a specified material, resistance, or set completely free). They are used to represent the box containing the electronics in the system being modeled, for example, PSU enclosures, system cabinet, sub-racking.

Objects

Field	Description
Size Definition	Sets the size values to be either the internal or external dimensions of the enclosure. If the internal dimensions are given, then the walls are built outside of this volume, allowing the external wall to be resized without affecting the inside of the enclosure.
Size Xo, Yo, Zo	The external or internal (depending on the above setting) enclosure dimensions in each local axis direction.
Modeling Level	There are two modeling options for the basic definition of each enclosure wall: • Thick — models the walls as 3D cuboids • Thin — models the walls as collapsed 2D cuboids The 2D model is used for economy in both grid and variable calculations, saving at least a grid cell in the direction of the collapse.
Wall Thickness	The default thickness for all the walls of the enclosure. This value can be overwritten for individual walls using the Enclosure Wall Property Sheet.

Related Topics

Thick (3D) and Thin (2D) Walls, Holes, and Sloping Blocks

Enclosure Wall Property Sheet

To access: Select an Enclosure Wall in the model.

Use this property sheet to configure an enclosure wall.

Objects

Object	Description
Location tab.	See "Enclosure Wall Property Sheet Location Tab" on page 120.
Construction tab.	See "Enclosure Wall Property Sheet Construction Tab" on page 121.

Enclosure Wall Property Sheet Location Tab

To access: Select an Enclosure Wall in the model and then select the **Location** tab.

Use this tab to change the wall name.

Objects

Object	Description
Name	Use to identify the wall.

Related Topics

Enclosure Wall Property Sheet

Enclosure Property Sheet

Enclosure Wall Property Sheet Construction Tab

To access: Select an Enclosure Wall in the model and then select the **Construction** tab.

Use this tab when individual walls of an enclosure require distinct modeling.

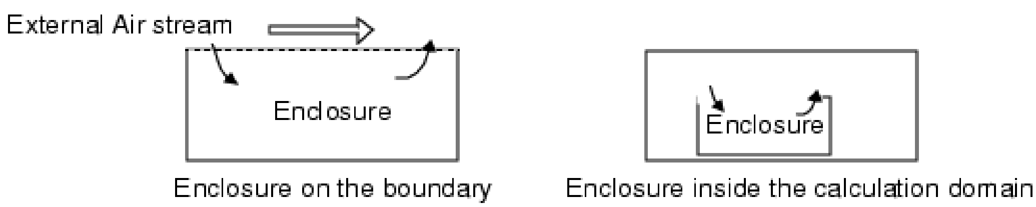
Objects

Object	Description
Side Exists	Defines a solid boundary wall. If unchecked, the enclosure side is removed, allowing air to flow through.
Modeling Level	(Side Exists) There are three modeling levels: • Enclosure Set — use the Modeling Level definition in the parent Enclosure Property Sheet Construction Tab. • Thick — model the wall as a 3D cuboid. • Thin — model the wall as a 2D collapsed cuboid (plane).
Wall Thickness	(Thick or Thin) The thickness of the wall.

Usage Notes

Uncheck the Side Exists check box when you want to represent the portion of the enclosure wholly exposed to an external air stream which can either flow parallel to the exposed surface (Figure 11-3) or remain stationary representing an open boundary.

Figure 11-3. Open-Sided Enclosure



Use the Ambient attribute to set up the external flow conditions. For example, this should normally be used for the edges of the solution domain where the flow is generally parallel to that face.

Related Topics

Enclosure Construction

Enclosure Wall Property Sheet

Enclosure Property Sheet

Thick (3D) and Thin (2D) Walls, Holes, and Sloping Blocks

Chapter 12
Extracts

Extracts are children of Recirculation Device, Rack, and Cooler SmartParts.

Modeling Extracts

Rotating and overlapping extracts.

Note

Extract locations may be on the solution domain boundary for an external device. For internal devices, they must be positioned on the surfaces of a region blanked out by a primitive cuboid as mentioned in the description of the "Recirculation Device Property Sheet" on page 286.

Rotating Extracts

The plane of a selected extract can be rotated independent of its parent. Rotating a parent device rotates all its child supplies and extracts.

Overlapping Extracts

If two or more extracts from *different* parent devices overlap, their flow rates are additive.

If two or more extracts from *the same* parent device overlap, they override, but in such a way that the total mass flow required by that device is maintained.

Grid Constraints on Extracts

Grid constraints are available to extracts but localized grid is only available to the parent device.

Related Topics

Rotating Objects [Simcenter Flotherm User Guide]

Grid Constraints [Simcenter Flotherm User Guide]

Extract Property Sheet

Recirculation Devices

Supplies

Extract Property Sheet

To access: Select an Extract SmartPart in the model.

Use this property sheet to configure an extract.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Construction tab.	See "Extract Property Sheet Construction Tab" on page 126.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.

Extract Property Sheet Construction Tab

To access: Select an Extract SmartPart, then select the **Construction** tab.

Use this property sheet tab to locate the extraction zone (for example, the intake of a heater) of a recirculation device.

Objects

Field	Description
Size Xo, Yo	Sets the dimensions of the rectangular area representing the extract, that is, the extent of the extract in the coordinate direction indicated.

Usage Notes

The dimensions should be set so as to give an area equivalent to that of the actual extract region, for example, if the regions are round ducts, the length of the side of an equivalent square is given by: $(\sqrt{\pi}/2) \times$ *diameter_of_duct*.

Related Topics

Modeling Extracts

Chapter 13
Fans

Use the Fan SmartPart to model an inlet, outlet, or internal axial or rectangular fan.

Applications of Fan SmartParts

The Fan SmartPart can be used to model axial or rectangular fans which can be configured to simulate a variety of designs.

Axial Fans

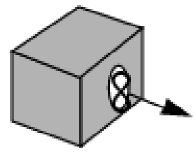
An axial fan is represented by building any required base, cowling, and hub out of cuboids and prisms, according to the level of detail required.

The fan blades are not individually modeled, only the expected flow passing through the region they occupy is required. Any expected swirl may also be included.

Tubeaxial Fans

A tubeaxial fan can be modeled by locating a fan in a hole in a cuboid (Block with Hole SmartPart), see Figure 13-1.

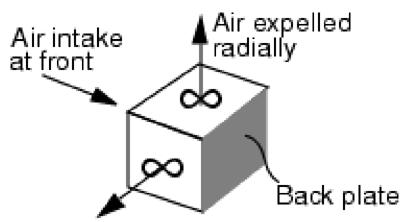
Figure 13-1. Tubeaxial Fan



Centrifugal Fans

You can create a centrifugal fan by combining a minimum of four axial fans and a collapsed cuboid backplate, see Figure 13-2.

Figure 13-2. Centrifugal Fan



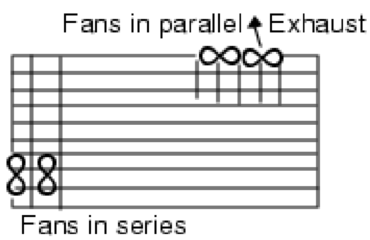
You can also create a centrifugal fan using the Blower Macro, available as a utility from the Simcenter Flotherm Related Downloads page on Support Center.

Combinations of Fans

More complex arrangements can be built up by the introduction of several fans, for example, fans in parallel and fans in series.

Fans in series must be separated by at least one grid cell, see Figure 13-3.

Figure 13-3. Fans in Parallel and in Series

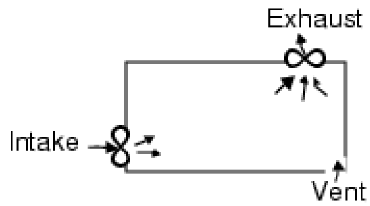


Fan Locations

Fans can be located:

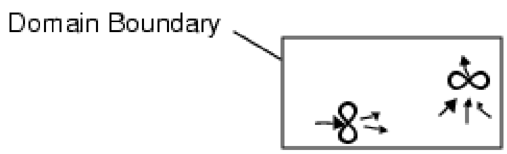
- On the domain boundary, blowing air into, or extracting air from, the domain.

Figure 13-4. Fans on the Domain Boundary



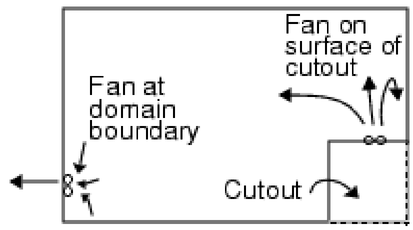
- Totally within the domain, stimulating fluid motion by imparting momentum to the fluid.

Figure 13-5. Fans Within the Domain



- Fans can be put inside the domain when they are connected on one side with a cutout, as illustrated in Figure 13-6.

Figure 13-6. Fan on Cutout



Transient Attributes Attached to Fans

It is possible to attach a Transient attribute to a fan. In such cases the Transient attribute multiplier is applied to the derating factor of the fan.

Modeling the Effects of Finger Guards or Grilles

When an axial fan is located on the domain boundary, by default the program assumes a zero loss resistance. To model the effects of finger guards or grilles, attach a resistance to the fan. This option only works if the fan has a 3D modeling level.

Related Topics

Transient Attributes [Simcenter Flotherm Project Attributes Reference Guide]

Fan Properties

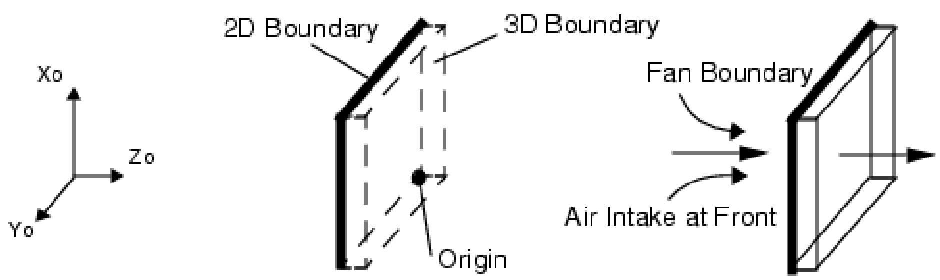
The properties of Fan SmartParts are described in the following subsections and in the property sheet description.

Fan Flow

For 3D models, the intake side of the fan assembly is always located on the boundary side that would still exist after collapsing the fan to 2D.

The flow direction is shown in Figure 13-7, an arrow indicates fan flow direction in the drawing board.

Figure 13-7. Intake Side of Fans



Flow can be directed normal or angled to the inlet. Axial fans with hubs can also have swirl applied.

The flow rate through the fan may be specified as a fixed volume, as a linear variation dependent upon pressure, or according to a fan characteristic curve.

Inlet Fan Flow

For inlet fans, the fluid is drawn in from the external environment. By default, the fluid brought in this way is at the ambient temperature as defined in the ambient attribute, and, by default, the program calculates the speed of the fluid as *Volume flow rate per unit area*.

This is the velocity normal to the plane of the fan. It is possible to allow for the effect of an inclined flow by setting the supply direction angle.

Exhaust Fan Flow

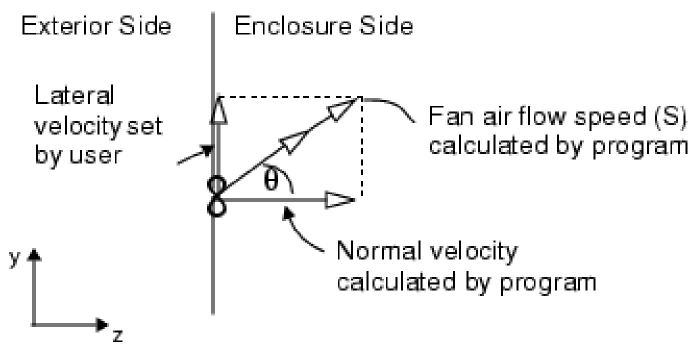
For exhaust fans, that is, outlets, the situation is different from that for inlet fans. The fluid leaving the enclosure leaves it with the conditions and properties that the fluid has on the enclosure side of the fan. Therefore, the temperature of the fluid leaving the enclosure through an exhaust fan is the temperature calculated by the program adjacent to the exit. The fluid leaves the enclosure with whatever angle the program computes for the flow towards the fan.

Angled Flow

Angled flow from intake fans, fixed flows and supplies is specified by three values, namely the local x-coordinate, Xo, the local y-coordinate, Yo, and the local z-coordinate, Zo.

- The axes used to define angled flow are those local to the object, which may not be the same as the solution domain axes, for example, if the object has been rotated.
- The program normalizes the components to create the resultant flow direction. For example, the settings XoN=0, YoN=1, ZoN=1 are transformed into 0, $1/\sqrt{2}$, $1/\sqrt{2}$ (that is, 0, Cos 45°, Sin 45°).
- The program checks the positive/negative sense of the direction and always sets it to conform with the inward normal.
- The direction cosines of the flow coming into an enclosure as illustrated in Figure 13-8.

Figure 13-8. Fan Angled Flow in One Plane



    - The z-direction cosine is $Cos(\theta)$.
    - The y-direction cosine is $Cos((\pi/2) - \theta)$, that is, $Sin(\theta)$.
- The ZoN value, which specifies the direction normal to the plane of supply, cannot be set negative or equal to 0.

Note

The user specification of a supply direction applies solely to intake fans. For exhaust fans (that is, outlets), the fluid leaving the enclosure leaves it with the conditions and properties that the fluid has on the enclosure side of the fan. Therefore, the temperature of the fluid leaving the enclosure through an exhaust fan is the temperature calculated by the program adjacent to the exit. The fluid leaves the enclosure with whatever angle the program computes for the flow towards the fan.

Related Topics

Defining a 2D Angled Flow

Defining a 3D Angled Flow

Dissipated Fan Power

The dissipated fan power is the power dissipated as heat (heat energy), as opposed to the power used to turn the fan (mechanical energy).

The dissipated fan power is calculated from the following:

$$\textit{Dissipated fan power} = \textit{Derated fan power} \times (1 - \textit{Fan efficiency})$$

The derated fan power is calculated by the program, see the Derated Power description under “Fan Property Sheet” on page 151. The fan efficiency is determined from the operating point of the fan.

Fan efficiency and dissipated fan power are displayed in tables, see Reporting Project Data and Results in Tables in the *Simcenter Flotherm User Guide*.

The dissipated fan power is distributed as follows:

- When the fan is at the 3D modeling level and a non-zero fan power is specified:
  - 80% in the fan hub
  - 20% in the air passing through the fan
- When the fan is at the 2D modeling level:
  - 100% in the air passing through the fan
- When the fan is at the 3D modeling level but does not have a material attached (V8.2 and earlier)
  - 100% in the air passing through the fan

Axial Fan Models

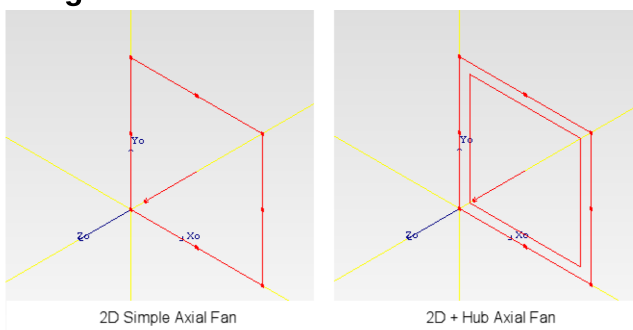
Axial fans can be 2D or 3D of varying complexity.

2D Axial Fan Models

There are two sub-types:

- 2D Simple — specified by the Outer Diameter.
- 2D + Hub — specified by the Outer Diameter and Hub Diameter.

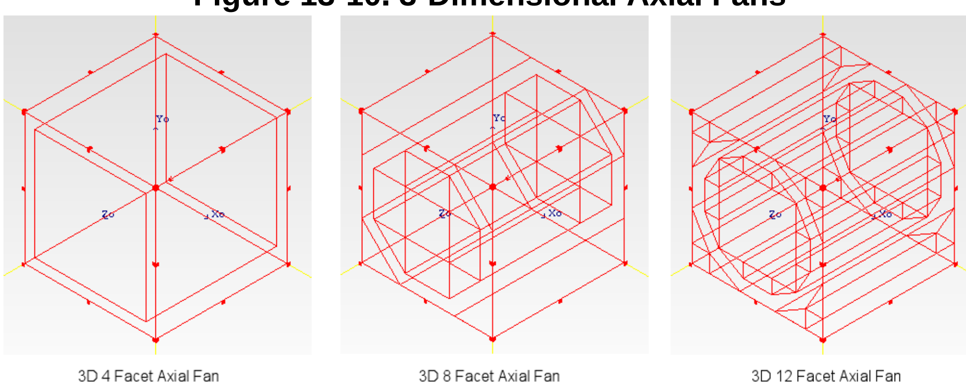
Figure 13-9. 2-Dimensional Axial Fans



3D Multi-Facet Axial Fan Models

These models are specified by the Outer Diameter, Hub Diameter and Fan Depth, see Figure 13-10.

Figure 13-10. 3-Dimensional Axial Fans



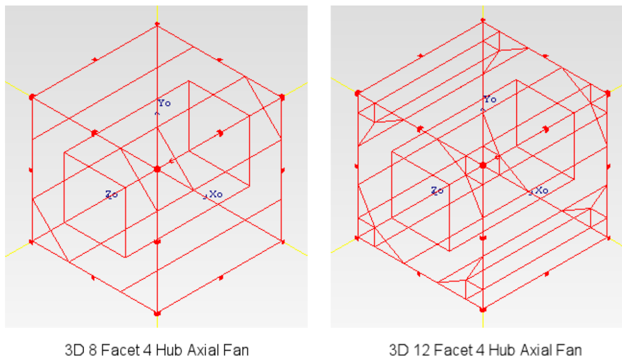
For these models, an additional grid is required to resolve the shape of the hub. If no additional grid is added, the triangular blocks that are contained within a single grid cell are not as accurately represented. The flow characteristics and thermal representation of the fan are not altered by additional grid cells.

3D Multi-Facet Axial Fan Models With 4-Facet Hubs

These models are specified by the Outer Diameter, Hub Diameter and Fan Depth, see Figure 13-11.

Where the hub is small compared with the fan and a 3D modeling option is chosen, it is recommended that a 4-facet hub option is chosen to minimize grid requirements.

Figure 13-11. 3-Dimensional Axial Fans With 4-Facet Hubs



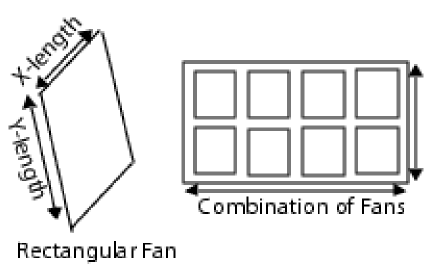
3D models with 4-facet hubs can only be defined when there is sufficient space between the Outer Diameter and the Hub Diameter. Hover Text advice is provided when inputting these dimensions.

Rectangular Fan Models

Rectangular fan models are always 2D objects, whose dimensions are defined by the side lengths, Xo and Yo.

In addition to the 90 degree-step rotation that can be applied to geometry (for example, **Geometry** > **Rotate Clockwise**), rectangular fans can be rotated more accurately using the Fan property sheet.

Figure 13-12. Rectangular Fans



Related Topics

Changing the Fan Angle of a Rectangular Fan

Fan Flow Types

The possible fan flow types are Normal, Angled, and Swirl.

All flow types (Normal, Angled, and Swirl) are available for axial fans. Normal and Angled are available for rectangular fans unless they are rotated, in which case only Normal is available.

By default, zero swirl is assumed. For electronics cooling, axial fans without guide vanes are normally used. The ratio of the tangential to axial velocity of the flow passing through the fan varies with position on the fan characteristic. This can be as low as 0.1 at the maximum throughput, increasing to 0.7 at the normal operating point, and increasing above 1.0 at lower flow rates. The presence of any form of grille or finger guard will reduce the swirl produced.

If the Constant Speed option is selected, the swirl speed determines the tangential speed of the flow at the fan.

If the Flow Dependent Speed option is selected, the swirl speed is computed from the fan flow rate and the user set swirl speed, which is set to the rotational speed of the blades/motor. It is used to reduce the swirl at high rotational speed as a consequence of slip.

Flow Specification

Flow can be specified as fixed volume, or by linear or non-linear relationships of flow rate with pressure.

Fixed Volume Flow

For an intake fan on the domain boundary, the fluid is drawn in from the external environment. By default, the fluid brought in this way is at the ambient temperature applied to the sides of the overall system domain using the Ambient attributes.

By default, the program calculates the axial momentum (that is, velocity) of the incoming fluid from *Fan volume flow rate per unit area*.

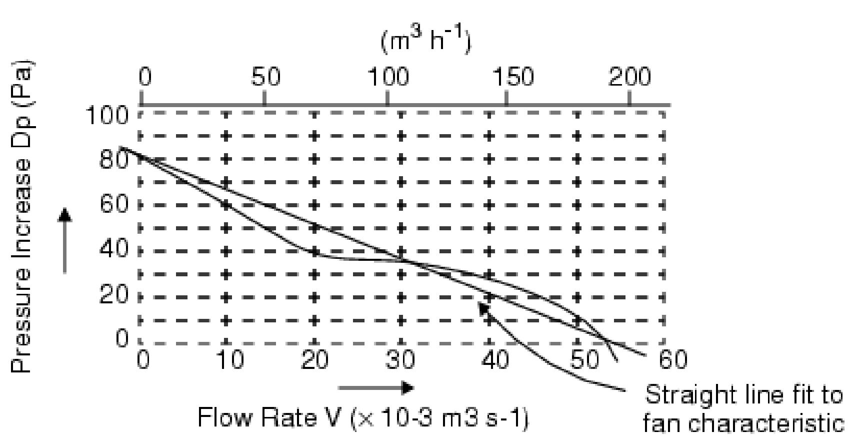
This is the velocity normal to the plane of the fan. However, it is possible to allow for the effect of an inclined flow by setting the Supply direction angle.

For exhaust fans (that is, outlets), the situation is quite different, the fluid leaving the enclosure leaves it with the conditions and properties that the fluid has on the enclosure side of the fan. Therefore, the temperature of the fluid leaving the enclosure through an exhaust fan is the temperature calculated by the program adjacent to the exit. The fluid leaves the enclosure with whatever angle the program computes for the flow towards the fan.

Linear Fan Flow

A linear fan flow specification is an attempt to model the fan according to a linear representation as close as possible to the manufacturer's fan characteristic.

Figure 13-13. Linear Approximation to a Fan Characteristic



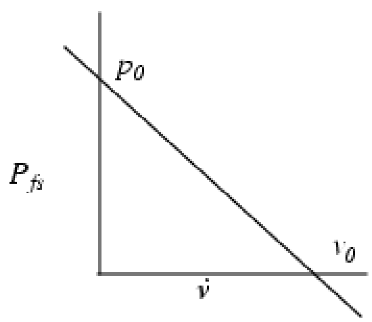
When defining a linear fan characteristic, set the Open Volume Flow Rate and Pressure at Stagnation so that the resultant linear representation best describes the fan curve at the point where it is expected to operate, these values may be different from those supplied by the manufacturer.

After solution, check the flow characteristics of the fan to make sure that the fan is operating as anticipated.

Linear Relationship Between Volume Flow Rate and Fan Static Pressure

A Linear fan relates the volume flow linearly to the fan static pressure as shown in Figure 13-14.

Figure 13-14. Volume Flow Rate vs Fan Static Pressure



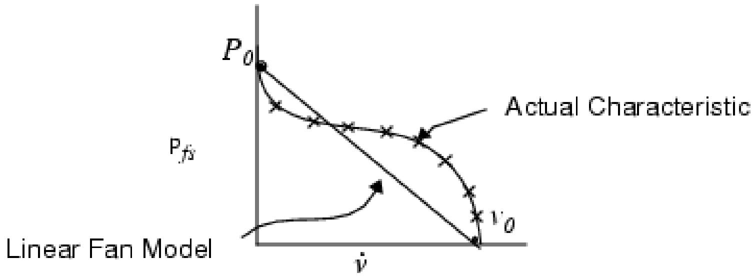
The intercepts of this straight line with the axis are denoted P_0 and v_0, which respectively signify the fan static pressure that causes zero flow through the fan, and the volume flow rate through the fan when the fan static pressure is zero (which pertains for a totally free standing fan, that is, not enclosed in any way).

Making a Best Fit

There are two ways of making a best fit of a linear characteristic to the actual characteristic.

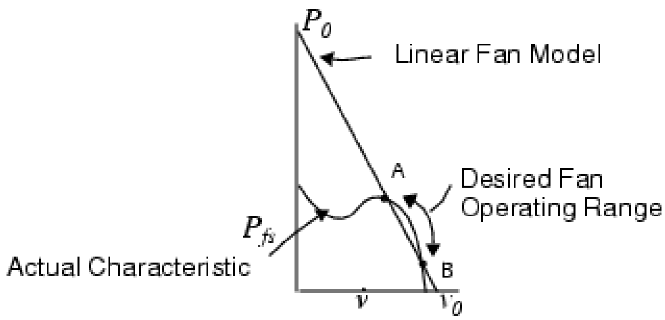
- The first method is to set P_0 and v_0 to those of the actual characteristic which gives a fit of the type shown in Figure 13-15.

Figure 13-15. Best Linear Fit Method 1



- The second method is to fit the line to cover the range over which you want the fan to operate as shown in Figure 13-16.

Figure 13-16. Best Linear Fit Method 2



If using the second method, the linear fan representation gives a good fit to the fan characteristic in the normal operating range but a poor fit elsewhere.

Equation of Linear Fan Characteristic

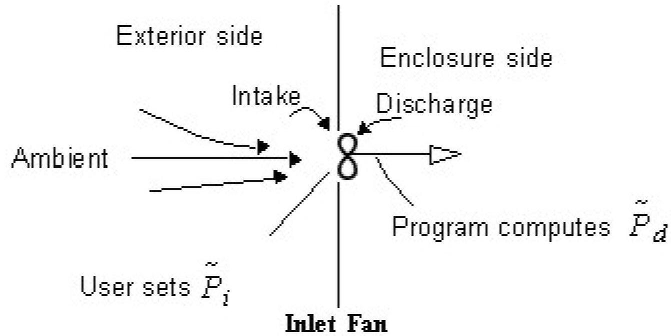
The equation of the linear fan characteristic is given by the following formula:

$$\dot{v} = v_o \times \left(1 - \frac{P_{fs}}{P_0}\right)$$

You set v_0 and P_0 as outlined in "Making a Best Fit" on page 138. The fan static pressure (P_{fs}) is defined as the (area averaged) static pressure $(\tilde{P}_d)$ on the discharge side of the fan minus the (area averaged) stagnation pressure $(\tilde{P}_i)$ on the intake side of the fan, that is:

$$P_{fs} = \tilde{P}_d - \tilde{P}_i$$

If the fan is on the solution domain or a cutout (whether blowing into or out of the domain) then the program calculates the pressure on one side of the fan (for example, for a fan blowing into the solution domain, this would be the stagnation pressure at discharge) while the other pressure will be derived from the relevant ambient setting (for example, fan blowing in solution domain — pressure on intake side).



Computed Operating Point

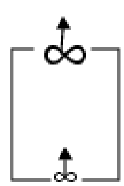
The program determines the operating point on the fan characteristic line specified. The accuracy of the computed fan flow rate is determined by the accuracy with which the program calculates the pressure head losses in the system. Therefore, for good results an accurate representation of the losses has to be present which will be achieved by a combination of judicious refinements of the grid and by correct settings of any loss factors for grilles, baffles, cabling, and so on. The computed operating point for each fan is given in the Tables application window, that is, it gives $\dot{v}$ and P_{fs} for each fan and also flags out-of-range operation.

Out-of-Range Operation

Out-of-Range operation occurs when the program finds that *either* the fan static pressure is less than zero in which case the flow rate through the fan exceeds $\dot{v}_0$ *or* the fan static pressure is greater than P_0 in which case flow is forced back through the fan in the wrong direction.

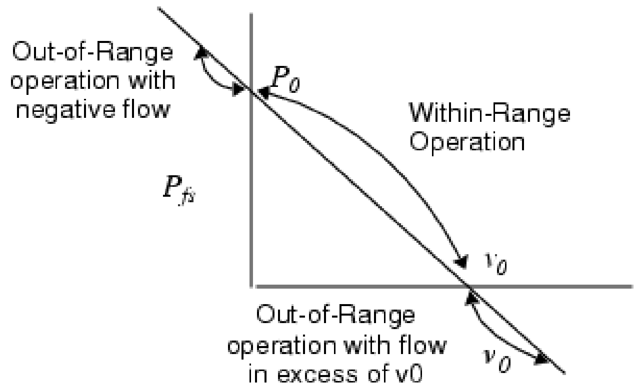
Figure 13-17 is an example of where the mass flow rate can exceed $\dot{v}_0$.

Figure 13-17. Mixing Large and Small Fans



In this enclosure, a large fan is used at the top of the enclosure and a small fan (for safety in case the large one fails) is used at the bottom. The small fan is driven well beyond its maximum flow rate v_0. Of course, fan manufacturers do not provide the characteristic beyond P_0 and v_0, but Simcenter Flotherm assumes that the linear line continues indefinitely in each direction as shown in Figure 13-18.

Figure 13-18. Assumed Continuation of Fan Characteristic



Affect of Fan RPM Derating Factor on Flow Types

Fans are often set to run below their maximum capacity, to reduce noise and increase their lifetime, but still achieve thermal cooling requirements.

Fans are run at less than maximum capacity by reducing the RPM that they operate at. This has the effect of derating (reducing) the fan curve and is simulated using the Derating Factor. The Derating Factor is applied to the different flow types as follows:

- Fixed Volume Flow

  $$\textit{Derated Flow Rate} = \textit{Derating Factor} \times \textit{Original Flow Rate}$$

- Linear Fan

  The volume flow rate and pressure drop values are derived from the original fan curve points as follows:

  $$\textit{Derated Open Volume Flow Rate} = \textit{Derating Factor} \times \textit{Defined Flow Rate}$$

  $$\textit{Derated Pressure at Stagnation} = (\textit{Derating Factor})^2 \times \textit{Defined Pressure at Stagnation}$$

- Non-Linear Fan

  The derated volume flow rate and pressure drop points are derived from the original fan curve points as follows:

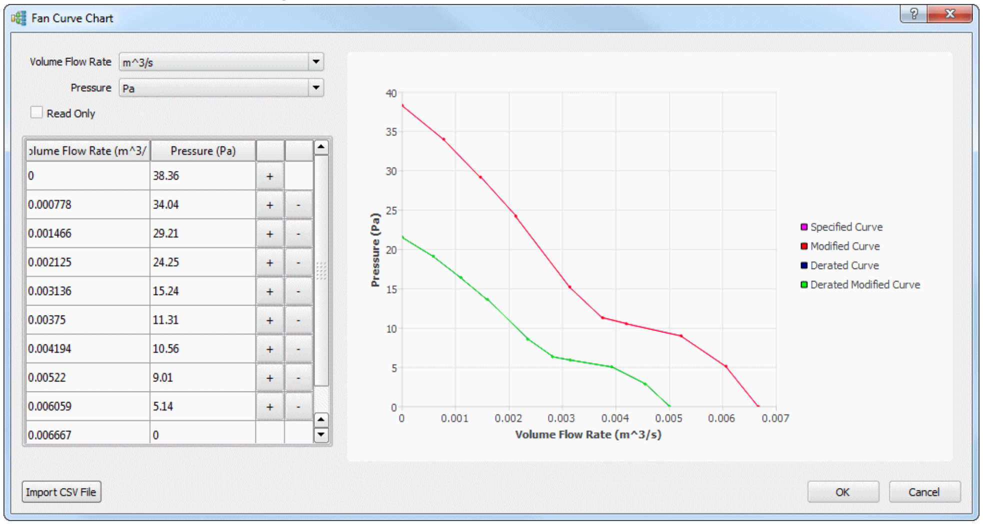
  $$\textit{Derated Flow Rate} = \textit{Derating Factor} \times \textit{Original Flow Rate}$$

  $$\textit{Derated Pressure Drop} = (\textit{Derating Factor})^2 \times \textit{Original Pressure Drop}$$

  When the Derating Factor is less than 1, a Derated Curve is displayed in the Fan Curve Chart dialog box, For example, Figure 13-19 shows a Derated Fan Curve when the

Derating Factor is 0.75. Changes made to the original fan curve are reflected automatically in the Derated Fan Curve.

Figure 13-19. Derated Fan Curve



- Swirl

  If a Constant Speed swirl model is selected, then the swirl speed is derived from the specified value as follows:

  $$Derated\ Swirl\ Speed = Derating\ Factor \times Original\ Swirl\ Speed$$

Fan Attributes

You can attach Material, Radiation, Resistance, and Transient attributes to fans.

- Material

  If you do not assign a material to a fan, the fan will be assumed to be adiabatic and, therefore, appear to be at ambient temperature in the solution.

  A Typical Fan Material attribute is supplied in the Library, under **Materials** > **Others**.

- Radiation

  Although you can attach a radiative attribute to a fan, fans are unlikely to require a radiation attribute if the (reasonable) assumption is made that the air will be moving quite fast over their surfaces. Ignoring radiation has the added benefit of computational efficiency.

- Resistance

  See “Modeling the Effects of Finger Guards or Grilles” on page 129.

- Transients

See "Transient Attributes Attached to Fans" on page 129.

Fan Procedures

This section describes how to define a characteristic curve and angled flow.

Defining a Fan Characteristic Curve

A fan characteristic curve of volume flow rates at different pressures can be defined either by importing a CSV file or adding points interactively.

Prerequisites

- A Fan, Recirculation Device, Rack, or Cooler SmartPart is being edited and the following is set in the **Construction** tab of the property sheet:
  - Fans: Flow Specification is Non-Linear Fan.
  - Recirculation Devices: Flow Type is Non Linear Fan Curve.
  - Racks and Coolers: Flow Type is Curve.

Procedure

1. Click the Fan Curve Chart **Click to Edit** button.

   The Fan Curve Chart dialog box is opened.

2. Select the desired units for Volume Flow Rate and Pressure Unit.

3. Uncheck Read Only if it is checked.

4. Depending on the how you want to create the chart, choose one of the following methods.

If you want to...	Do the following:
Enter data manually.	**Note:** The end values of the chart are defined from values of Open Volume Flow Rate (or Volume Flow Rate) and Pressure at Stagnation (or Pressure) in the property sheet and all manual data entry into the chart table is restricted by these values. 1. Click the + button to add another row in the table and enter a pair of values for Volume Flow Rate and Pressure. 2. Continue adding data in this way. The chart is updated as you enter data. You can delete rows by clicking the - button.
Enter data by CSV file import.	1. Click **Import CSV File**. 2. Navigate to, and select, a valid CSV file and click **Open**.

5. Click **OK** to save the data and close the Fan Curve Chart dialog box.

Results

After importing from a CSV file:

- The Pressure value corresponding to a Volume Flow Rate of zero (the pressure at stagnation), is entered into the following field in the property sheet:
    - For Fans: the Pressure at Stagnation field.
    - For Recirculation Devices, Racks, or Coolers: the Pressure field.
- The Volume value corresponding to a Pressure of zero (the open volume flow rate), is entered into the following field in the property sheet:
    - For Fans: the Open Volume Flow Rate field.
    - For Recirculation Devices, Racks, or Coolers: the Volume Flow Rate field.
- The Fan Curve Chart is set to Read Only to prevent accidental changes.

Related Topics

Fan Curve Chart Dialog Box

Fan Characteristic Curve CSV File Format

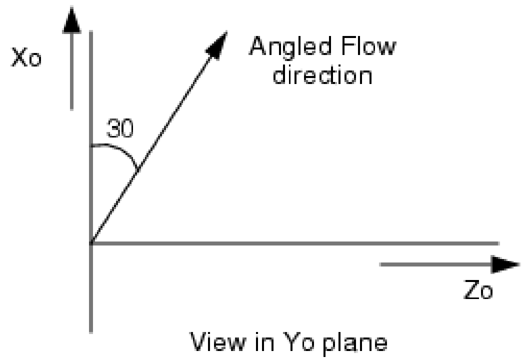
Defining a 2D Angled Flow

A 2D flow angle is specified by one angle from the Normal. In such cases, either the Xo-direction or Yo-direction component is zero.

Procedure

1. Calculate the components in the two directions, for example, consider an angled flow at 30° from the Xo axis, see Figure 13-20.

Figure 13-20. Example of 2D Angled Flow



The Xo-direction component is Cos 30°.

The Yo-direction component is 0.

The Zo-direction component is Sin 30°.

2. Therefore, enter the following values in the property sheet:

   XoN = 0.866, YoN = 0, ZoN = 0.5.

Results

After solution, tracks and velocity vectors showing the direction of flow can displayed in Analyze mode.

Related Topics

Angled Flow

Defining a 3D Angled Flow

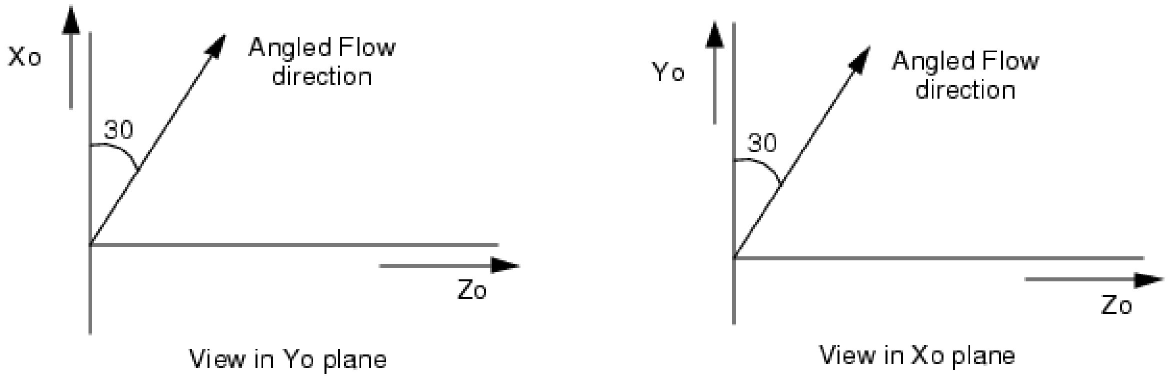
Defining a 3D Angled Flow

A 3D flow angle is specified by two angles from the Normal. In such cases, the Xo- and Yo-direction components are non-zero.

Procedure

1. Calculate the components in the three directions, for example, consider an angled flow at 30° from the x-axis and at 30° from the y-axis, see Figure 13-21.

Figure 13-21. Example of 3D Angled Flow



The X-direction component is Cos 30°.

The Y-direction component is also Cos 30°.

The Z-direction component is Sin 30°.

2. Therefore, enter the following values in the property sheet:

XoN = 0.866, YoN = 0.866, ZoN = 0.5.

Results

After solution, tracks and velocity vectors showing the direction of flow can displayed in Analyze mode.

Related Topics

Angled Flow

Defining a 2D Angled Flow

Changing the Fan Angle of a Rectangular Fan

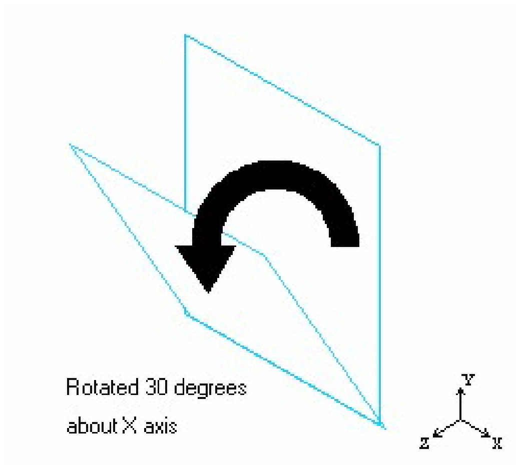
Angled fans can be created by rotating rectangular fans about a chosen axis.

Prerequisites

- Flow Type must be Normal
- Fan Power must not be activated.

Procedure

1. Choose the direction to be rotated: None (default), X Axis or Y Axis. The fan local coordinates are referenced, therefore Z is always normal.
2. Enter the angle of rotation between −90 and +90 degrees.



As this is a rectangular fan, no swirl can be impressed on the flow.

Fan Characteristic Curve CSV File Format

A comma-separated point-pair curve text data file.

Used this file to create a Fan SmartPart characteristic curve by file import as an alternative to manually adding data directly into the Fan Curve Chart dialog box.

Format

A Fan Characteristic Curve CSV File must conform to the following formatting and syntax rules:

- The units are those currently set by the chart.
- The first value for *volume_flow* must be 0 (zero).
- The last value for *pressure* must be 0 (zero).
- The list can be any length.
- The values are in floating point format.

```
0,<pressure_at_stagnation>
<volume_flow>,<pressure>
<volume_flow>,<pressure>
...
<open_volume_flow_rate>,0
```

Parameters

- pressure_at_stagnation

  The pressure at zero volume flow.
- volume_flow

  The volume flow rate at the corresponding pressure.
- pressure

  The pressure.
- open_volume_flow_rate

  The volume flow at zero pressure.

Examples

The following is an example where the volume flow is in m^3/s and the pressure is in Pa.

```
0,38.36
0.000778,34.04
0.001466,29.21
0.002125,24.25
0.003136,15.24
0.00375,11.31
0.004194,10.56
0.00522,9.01
0.006059,5.14
0.006667,0
```

Related Topics

Defining a Fan Characteristic Curve

Fan Curve Chart Dialog Box

Fan Property Sheet

To access: Select a Fan SmartPart in the model.

Use this property sheet to configure a fan.

Description

The basic model is an axial fan, but it can be adapted to represent centrifugal fans. There is also the capability of modeling banks of axial fans.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Construction tab.	See "Fan Property Sheet Construction Tab" on page 152.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.

Fan Property Sheet Construction Tab

To access: Select a Fan SmartPart, then select the **Construction** tab.

Use this tab to define the dimensions, flow rate and flow direction of a Fan SmartPart.

Objects

Field	Description
Fan Type	Axial or Rectangular.
Fan Model	(Axial) 2D and 3D geometry model templates are provided, see "Axial Fan Models" on page 134.
Hub Diameter	(Axial) Hub (inner aperture) diameter of an axial fan.
Outer Diameter	(Axial) Outer diameter of aperture of an axial fan.
Fan Depth (Zo)	(Axial) Applies to 3D models only.
Length Xo and Yo	(Rectangular) Side lengths of a rectangular fan.
Rotated About	(Rectangular) Use to change the angle of a rectangular fan: None, about the X Axis, or about the Y Axis.
Angle	(Rectangular and X Axis or Y Axis) The angle of rotation.
Derating Factor	Use to control the derating of the fan RPM, that is the ratio of *Operating RPM/Defined RPM*. Values must be greater than 0. • Modifies a Fixed Volume flow linearly. • Modifies a Linear or Non-Linear Fan based on the Fan Laws, which describe how the physical performance of a fan changes when it is derated by reducing the RPM.
Flow Specification	• Fixed Volume — use when modeling 'what if' studies. • Linear Fan — the simplest method of applying a flow rate dependent upon pressure drop in the system. • Non-Linear Fan — use when simple modeling is not sufficient.
Open Volume Flow Rate	The total flow rate for the expected conditions. For Linear and Non-Linear Fans, the flow at zero pressure.
Pressure at Stagnation	(Linear and Non-Linear Fan) The pressure when flow is zero.
Fan Curve Chart	(Non-Linear Fan) Opens the Fan Curve Chart Dialog Box.
Flow Type	Sets the inclination of the emerging flow or a swirl flow: • Normal — normal to the plane of the supply. • Angled — inclined to the normal. Not available for rotated rectangular fans, see "Angled Flow" on page 132. • Swirl — sets up swirl modeling, for axial fans only.

Field	Description
Flow Direction XoN, YoN, and ZoN	(Angled) The local coordinates that specify the direction of angled flow.
Swirl Model	(Swirl) • Constant Speed • Flow Dependent Speed
Swirl Direction	(Swirl) The swirl direction when facing the outflow of the fan: Clockwise or Counter-Clockwise.
Swirl Speed	(Swirl) The angular speed.
Power On	(Not available for rotated rectangular fans) Check to simulate fan power. When unchecked, no heat is imparted from the fan.
Power	(Power On) The amount of heat imparted to the air by the fan in operation.
Derated Power	(Power On) Read Only. The derated power calculated from: $\mathit{Derated\ Fan\ Power} = \mathit{Fan\ Power} \times (\mathit{Derating\ Factor})^3$ This is the reduced power that is added to the air. The exponent 3 can be overwritten using the FLO_FAN_POWER_EXP environment variable. The Command Center can be used to determine the lowest effective running conditions by solving different scenarios created for different values of the fan Derating Factor.
Noise	The fan noise in decibels.
Derated Noise	Read Only. The calculated derated noise is calculated from: $\mathrm{dB}_{\mathrm{derated}} = \mathrm{dB} + 50 \log_{10} \mathit{RPM\ Derating\ Factor}$ The dB noise value has no effect on the flow or heat transfer solution, it serves only to indicate the relationship between RPM derating and the resulting noise level. It is also available as an output variable in the Command Center. The coefficient 50 can be overwritten using the FLO_FAN_NOISE_COEFF environment variable.
Failed	Check to deactivate the fan flow functionality and simulate the effects of fan failure, keeping the geometry without the forced flow.

Related Topics

Fan Properties

Changing the Fan Angle of a Rectangular Fan

Angled Flow

Affect of Fan RPM Derating Factor on Flow Types

Fan Curve Chart Dialog Box

To access: In the **Construction** tab of a Fan, Recirculation Device, Rack or Cooler SmartPart property sheet, click the Fan Curve **Click to Edit** button. This button is only visible when specifying non-linear fans, that is:

- In the Fan property sheet, Flow Specification is set to Non-Linear Fan.
- In the Recirculation Devices property sheet, Flow Type is set to Non Linear Fan Curve.
- In Rack and Cooler property sheets, Flow Type is set to Curve.

Use this dialog box to define a fan characteristic curve from a sequence of points.

Objects

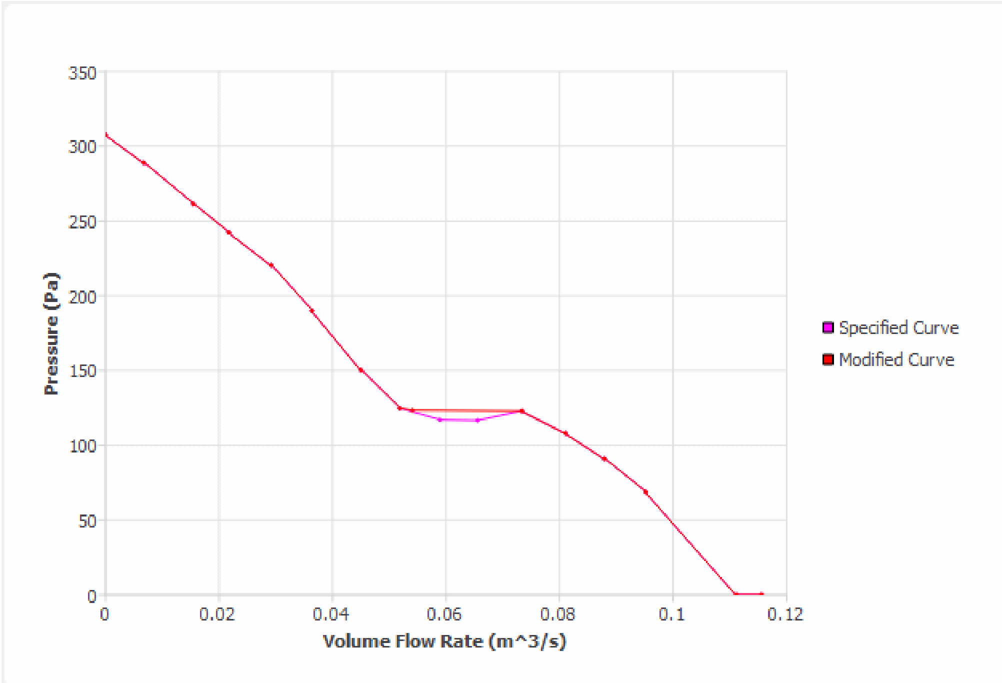
Object	Description
Volume Flow Rate	The units of Volume Flow Rate used.
Pressure Unit	The units of Pressure used.
Read Only	Check to protect data from changes.
Table	Pairs of values of Volume Flow Rate and Pressure.
Operating Flow Rate	(Post-Solve) The computed value of the operating flow rate.
Operating Pressure	(Post-Solve) The computed value of the operating pressure.
Import CSV File	Click to import a CSV file.

Usage Notes

- The Specified Curve is always displayed, this is the curve drawn using the table values. In addition, any of the following curves may be shown:
    - Derated Curve — shown when a Derating Factor which is less than 1 (one) has been specified, for example, see Figure 13-19.
    - Modified Curve and Derated Modified Curve

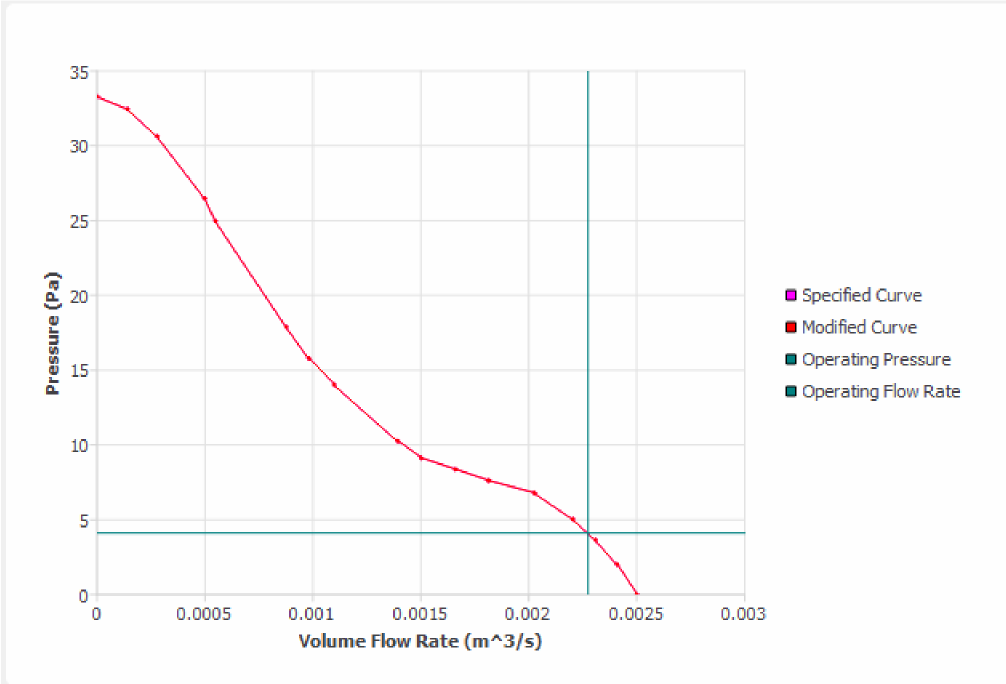
      If the fan curve has a local minima, then Simcenter Flotherm uses a modified fan curve which is monotonically decreasing and equal to the supplied curve except in the neighborhood of the local minima, see Figure 13-22, and a message to this effect is displayed in the message window.

Figure 13-22. Treatment of Local Minima



- After solving, the computed Operating Point is shown on the curve, as the intersection of the Operating Pressure and Operating Flow Rate lines.

Figure 13-23. Post-Solve Fan Operating Point



Related Topics

Defining a Fan Characteristic Curve

Fan Characteristic Curve CSV File Format

Linear Fan Flow

Chapter 14
Fixed Flows

Use the Fixed Flow SmartPart when you want to model a constant flow of fluid into or out of the solution domain.

Applications for Fixed Flow SmartParts

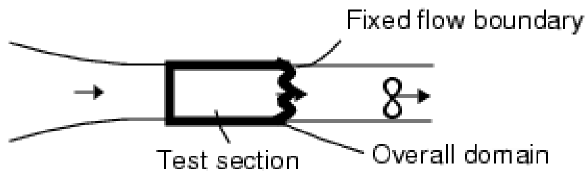
The Fixed Flow SmartPart specifies the flow conditions for a surface when the flow rate and the temperature entering the device is known.

A fixed flow either introduces, or extracts, fluid to, or from, the solution domain. As such, it needs to be defined on the surface of a solid object.

For systems which are internally located with the supply temperature part of the solution, the recirculation boundary should be used instead, see “Recirculation Devices” on page 271.

An example of using the fixed flow boundary is modeling the test section of a wind tunnel, see Figure 14-1.

Figure 14-1. Test Section of a Wind Tunnel



Fluid leaving the system through the outlet leaves it with the conditions and properties that the program has calculated on the solution domain side of the device.

Fixed Flow Properties

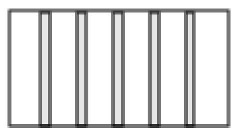
The properties of Fixed Flow SmartParts are described in the following subsections and in the property sheet description.

Free Area Ratio

Free Area Ratio is a measure of any obstruction to the flow through the device, for example, a grille across the flow path.

The Free Area Ratio is the ratio of the actual free area to the total area across the device. For example, an obstruction blocking 25% of the flow path is represented by a Free Area Ratio of 0.75, see Figure 14-2.

Figure 14-2. 25% Blockage



Where the Free Area Ratio is used to represent significant obstruction, care should be taken to define a sufficiently fine grid perpendicular to the air terminal to be able to capture the entrainment caused by the low pressure next to the obstructions.

- For Flow Type = Normal Velocity, the Free Area Ratio is used to modify the volume flow-rate for both inflow and outflow.
- For Flow Type = Volume-Flow Rate, the Free Area Ratio modifies the inflow velocity in the first cell only. This lowers the pressure locally and enables entrainment effects to be considered.

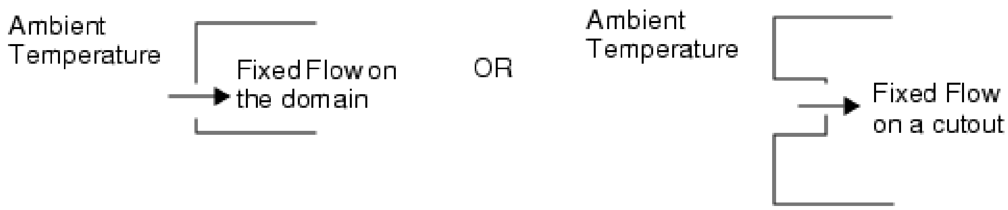
Flow Direction

Fixed Flows can be a flow direction into the solution domain (Inflow) or out of an enclosure (Outflow).

Inflow Flow Direction

When modeling inflows, an ambient flow must be set, otherwise the fixed flow device will not be included in the solution, and a message will be generated, informing you that the fixed flow has been deactivated.

Figure 14-3. Inflows



Additional fields for the turbulent kinetic energy and dissipation rate of the inflow are enabled in the Ambient attribute if the **LVEL K-Epsilon** turbulence model has been selected in the Turbulence section of the **Model Setup** tab.

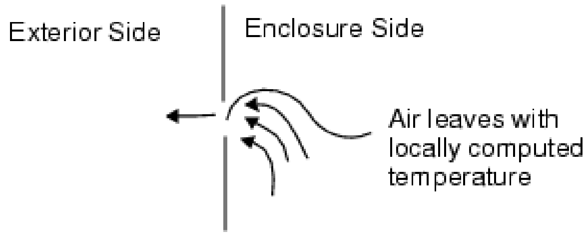
If left at the default value of zero, an estimate is made based on the flow rate calculated for the average fan speed.

Outflow Flow Direction

When modeling an outflow, the air leaves the enclosure with the conditions and properties that the program has calculated on the solution side of the device. For example, the temperature of the air leaving the enclosure through the outlet is the temperature calculated by the program adjacent to the outlet.

The fluid leaves the enclosure with whatever angle the program computes for the flow approaching the outlet.

Figure 14-4. OutFlow



Creating Fixed Flows

Fixed Flow SmartParts are 2D rectangles that represent apertures through which fluid flows.

Procedure

1. Select an assembly.
2. In the New Object Palette, choose Project Manager Create or Drawing Board Create and double-click the **Fixed Flow** icon.

Results

A fixed flow is represented in the drawing board as a 2D rectangle with a flow direction arrow. If the flow direction arrow is not visible, check that it has not been suppressed (User Preferences).

Positioning of Fixed Flows

Fixed flows must not be placed in free air, they must be placed on the solution boundary, the surface of a thick wall of an enclosure, or the face of an uncollapsed cuboid.

Overlapping Fixed Flows

If two or more fixed flows overlap, their volume flow rates are additive, and, by default, the cumulative flow is averaged across the cross-sectional area of the overlapping apertures.

The FLOFIXEDFLOWSOVERWRITE environment variable enforces hierarchical precedence such that a fixed flow with the highest precedence maintains its flow through its defined area, an overlapping fixed flow with a lower precedence maintains its flow through the remaining area, and so on. The total flow rate, as set, is still maintained, but its distribution is determined by the precedence of the fixed flows.

The use of FLOFIXEDFLOWSOVERWRITE only applies to fixed flows where the volume flow rate is set, it does not apply when the flow is set by velocity.

Fixed Flow Property Sheet

To access: Select a Fixed Flow SmartPart in the model.

Use this property sheet to configure a fixed flow.

Objects

Object	Description
Location tab.	See “Generic Property Sheet Location Tab” on page 28.
Construction tab.	See “Fixed Flow Property Sheet Construction Tab” on page 164.
Attachments tab.	See “Generic Property Sheet Attachments Tab” on page 30.
Groups tab.	See “Generic Property Sheet Groups Tab” on page 31.

Fixed Flow Property Sheet Construction Tab

To access: Select a Fixed Flow SmartPart, then select the **Construction** tab.

Use this property sheet tab to model a constant flow of air, either entering or leaving, through a device.

Objects

Field	Description
Flow Type	Select how the flow is to be defined: Volume Flow Rate or Normal Velocity.
Volume Flow Rate	(Volume Flow Rate) The volume flow rate through the device.
Normal Velocity	(Normal Velocity) The flow velocity normal to the device.
Length Xo and Yo	The size of the device, which is represented as a rectangular plane. The dimensions of this rectangular area should be set so as to give an area equivalent to the area enclosed by the perimeter of the vent.
Free Area Ratio	The ratio of the free area to the total area across the device.
Flow Direction	The flow direction can be set as: • Inflow — models fluid brought into the solution domain. • Outflow — models the flow as exiting from an enclosure.
Flow Angle	(Inflow) Sets the inclination of the emerging flow: • Normal — normal to the plane of the supply. • Angled — inclined to the normal.
XoN, YoN, and ZoN	(Inflow and Angled) The local coordinates that specify the direction of angled flow.
Ambient Conditions	(Inflow) Attach an Ambient attribute that defines the conditions outside the solution domain. Message I/9011 is output if there is no ambient defined.
Transparent to Solar Radiation	By default, fixed flows block solar radiation. Check this box to allow solar radiation through.

Related Topics

Free Area Ratio

Flow Direction

Angled Flow

Solar Radiation [Simcenter Flotherm User Guide]

Chapter 15
Heat Pipes

Use the Heat Pipe SmartPart to model a heat pipe by defining its thermal characteristics. Use Network Cuboid or Cuboid SmartParts to form the structure of the heat pipe.

Heat Pipe Applications

Advice when using heat pipe manufacturer's data.

Thermal Transport

The performance of a heat pipe changes with heat flux. Below the critical heat flux, the heat transfer occurs by boiling and condensation. Above the critical heat flux, heat transfer is inhibited by one of a number of heat transport limitations: sonic limit, entrainment limit, capillary limit and boiling limit.

Manufacturer’s performance data are measured under idealized conditions dissimilar to those used in the application. The performance of the heat pipe depends on a number of factors, including the environment in which it is used. Since their influence can only be determined experimentally, these are not included in the Heat Pipe SmartPart input data. Rather, a pragmatic approach is taken, whereby the effectiveness of the heat pipe and the critical heat flux for the application environment is required.

Simulation Method

The complex physics of the heat transfer within a pipe are not explicitly simulated, but are characterized by the Effective Thermal Resistance.

Manufacturers supply non-standardized, characterization data from which the Effective Thermal Resistance and Maximum Heat Flow can be obtained. The methodology required to derive the properties is highly dependent on the form of the supplied data.

The following qualitative statements can be taken as a general “rule of thumb” and their quantitative measures can be obtained from individual heat pipe manufacturers:

- Diameter — decreasing the diameter heat pipe will cause the effective thermal resistance to rise and the maximum heat flow to fall.
- Orientation — decreasing the angle of orientation will cause the effective thermal resistance to increase.
- Bends — each ninety degree blend will cause the effective thermal resistance to increase.

Manufacturers may explicitly provide an effective thermal resistance and maximum heat flow. However, if not, they should provide sufficient data/equations based on the dimensions, orientation, and number of bends from which an effective thermal resistance and maximum heat flow can be obtained.

Heat Pipe Selection Rule of Thumb

The following rule of thumb when selecting heat pipes is based primarily on an unflattened heat pipe (assuming manufacturing processes are good).

The dry-out heat, which is the maximum heat that a heat pipe can carry before all the water evaporates, can be related to the square of the heat pipe diameter. This can be written as:

$$Q\_dryout = Diameter^2$$

if *Diameter* is measured in millimeters and *Q_dryout* is in Watts.

This rule of thumb means that for a 6 mm heat pipe, *Q_dryout* is approximately 36 W and for a 8 mm heat pipe, *Q_dryout* is approximately 64 W. This maximum heat carried by the heat pipe drops once the heat pipe is bent and flattened.

The first step for a thermal engineer is to determine the total power that the heat sink needs to dissipate from a component then set the requirement to the vendors as the total heat pipes required for the design.

Related Topics

Creating a Heat Pipe

Heat Pipe Construction

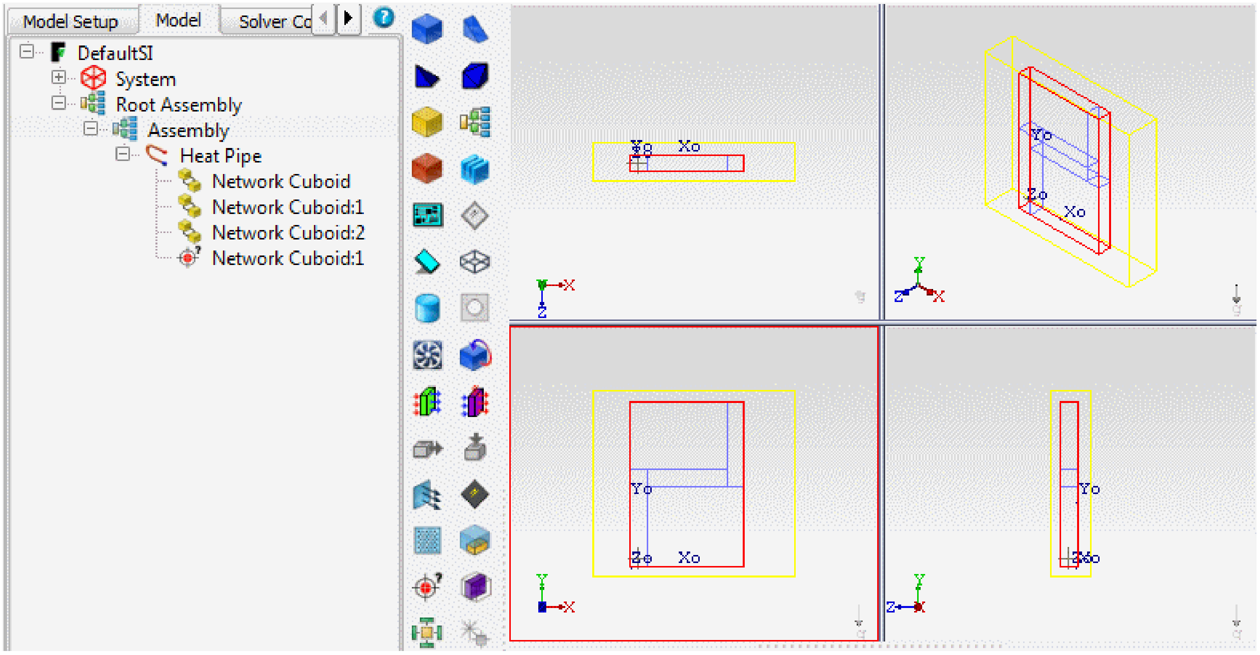
Heat Pipe Property Sheet

Heat Pipe Construction

A heat pipe modeled by Simcenter Flotherm is an assembly of network cuboids which have a very low thermal resistance as defined by heat pipe performance data.

Network Cuboid SmartParts are added to form the structure of the heat pipe. Monitor points are added to track the temperature in the heat pipe. An example structure is shown in Figure 15-1 where a monitor point has been added to track the central temperature of the heat pipe.

Figure 15-1. Heat Pipe Displayed in Data Tree and Drawing Board



Modeling Circular Heat Pipes

As the diameter of a heat pipe is small compared to its length and only a small percentage of the heat is lost in transit along the pipe, it suffices to model the physical representation of the circular heat pipe geometry as a series of network cuboids whose constant cross-sectional circumference is equal to that of the circular heat pipe such that:

$$x = \pi D/4$$

where:

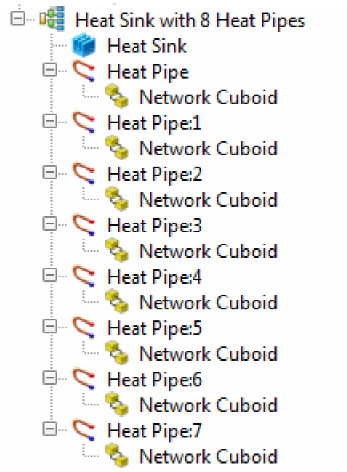
x is the network cuboid width.

D is the original heat pipe diameter.

Modeling Heat Sinks With Embedded Heat Pipes

Embedded heat pipes behave independently and are glued to a base with epoxy. Therefore, you will need to create a Heat Pipe SmartPart for each heat pipe and add a Network Cuboid to each of these, for example, see Figure 15-2.

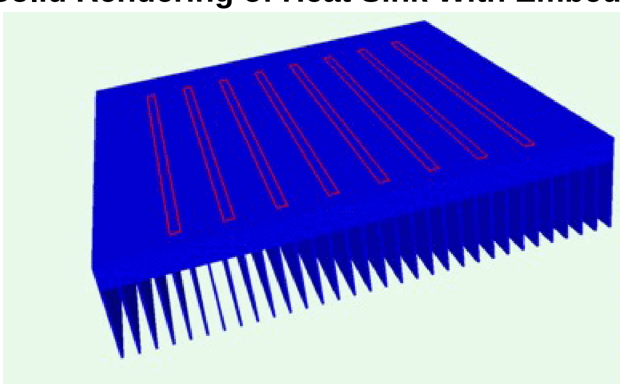
Figure 15-2. Example Heat Sink With Embedded Heat Pipes



Attach a Surface attribute to set a solid-to-solid surface resistance on the appropriate surface of the network cuboid.

The solid rendering for such a model is shown in Figure 15-3, with the surfaces of the heat pipes highlighted.

Figure 15-3. Solid Rendering of Heat Sink With Embedded Heat Pipes



Modeling Complex Geometry Using Cuboids

Cuboids can be copied and pasted into Heat Pipe SmartParts. This enables complex heat pipe geometry to be easily created when combined with the use of the Voxelization command in MCAD Bridge.

Restrictions When Using Heat Pipes

Direct abutment of compact components or network assemblies with heat pipes is not supported, and results in a denial of solve. Insert a highly conducting cuboid between the compact component/network assembly and the heat pipe to allow this configuration to be modeled.

Precedence Rules

The network cuboids exhibit the normal order of precedence rules within the heat pipe, that is, cuboids lower down the tree will overwrite those above.

As the heat pipe uses the Embedded Conduction Solver, its cuboids cannot be overwritten by other solid geometry lower down the data tree that is not solved by the Embedded Conduction Solver.

Related Topics

Creating a Heat Pipe

Heat Pipe Applications

Heat Pipe Property Sheet

Creating a Heat Pipe

A Heat Pipe SmartPart is a point in space, therefore, heat pipes are physically modeled by adding Network Cuboid or Cuboid SmartParts.

Procedure

1. Select or create an assembly in the data tree and add a Heat Pipe SmartPart.
2. Either select the Heat Pipe and add a Network Cuboid, or copy and paste a Cuboid to the Heat Pipe.

   Pasting cuboids enables more complex shapes of heat pipe to be defined, for example, from voxelized MCAD components.

   When pasting a Cuboid to a Heat Pipe, the Cuboid adopts the **Network Cuboid** icon, any Material, Thermal, or Surface Exchange attributes are detached, and any Surface, Grid Constraint, or Radiation attributes remain attached.
3. Repeat Step 2, adding more Network Cuboids or pasting more Cuboids to build the shape of the heat pipe.
4. Use the Heat Pipe property sheet to define the effective thermal resistance and maximum heat flow of the heat pipe.

5. Use the Network Cuboid property sheets to define the surface, radiation, and grid constraint attributes.
6. If required, add Monitor Points to Network Cuboids to track temperatures in grid cells.

Results

For a simple example of a heat pipe built using three network cuboids, see Figure 15-1.

Related Topics

Heat Pipe Construction

Heat Pipe Applications

Heat Pipe Property Sheet

Network Cuboid Property Sheet

Monitor Point Property Sheet

Heat Pipe Property Sheet

To access: Select a Heat Pipe SmartPart in the model.

Use this property sheet to configure a heat pipe.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Construction tab.	See "Heat Pipe Property Sheet Construction Tab" on page 172.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.

Heat Pipe Property Sheet Construction Tab

To access: Select a Heat Pipe SmartPart, then select the **Construction** tab.

Use this property sheet tab to define the effective thermal resistance of the selected Heat Pipe SmartPart.

Objects

Field	Description
Effective Thermal Resistance	Sets the effectiveness of the Heat Pipe Thermal Resistance.
Maximum Heat Flow	A reference, obtained or derived from manufacturer data, which will be used to calculate the Validity indicator of whether the heat pipe is moving more heat than it can handle, see Heat Pipes Results Table in the *Simcenter Flotherm User Guide*.

Related Topics

Heat Pipe Applications

Heat Pipe Construction

Creating a Heat Pipe

Chapter 16
Heat Sinks

Heat Sink SmartParts can have plate or pin fins.

Modeling Heat Sinks

This section gives modeling advice for heat sinks.

Design Variations

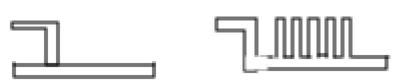
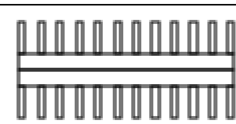
The design variations supported by Simcenter Flotherm are plate and pin fin heat sinks, which are formed by either extrusion, casting, or bonding.

Table 16-1 shows some of the variations that are possible when modeling heat sinks.

The design variations are limited to heat sinks with uniform base thickness and fin height (excluding end fins), with fins on one side of the base only. The fins style options are tapered or uniform. When modeling tapered fins, an equivalent uniform fin is used.

More complex designs may be created by decomposing the SmartPart into its constituent primitives using **Geometry** > **Decompose** and then editing individual parts.

Table 16-1. Heat Sink Variations

Variation	Appearance
Fins with a Center Gap.	
Offset End Fins.	
Non-Standard (Wider) End Fins.	
Mounting Brackets on Offset and Non-Offset End Fins.	
Heat sinks with fins on both sides of the base can be created by combining two heat sink designs in a single assembly.	

Compact Modeling

In some circumstances, the overhead in modeling the fins in detail in a system level analysis can be avoided by using the compact heat sink model.

For situations where an engineering approximation of heat sink performance is sufficient for assessing the overall cooling strategy in a system, this modeling option offers an efficient alternative to modeling each fin or pin explicitly.

Because of the algorithms used, the compact model is best suited to heat sinks in forced convection flow, where radiative losses are not a significant contribution to the overall heat transfer from the heat sink.

This treatment replaces the fins by volume and planar flow resistances to account for the friction and contraction/expansion losses as air passes through the fin or pin volume. In addition, the heat transfer is calculated in the fin/pin volume using a volume-based Surface Exchange attribute. The settings within the volume are calculated using standard Nusselt number and fanning friction factor correlations, appropriate for fully-developed duct flow; these settings use the local flow data in each grid cell along with the defined geometry of the heat sink.

The Nusselt number, *Nu*, is calculated from:

$$Nu = \max(7.54,\ 0.024\ Re^{0.786} Pr^{0.45})$$

The fanning friction factor, *f*, is calculated from:

$$f = \max(24/Re,\ 0.0791/Re^{0.25})$$

This is represented in Simcenter Flotherm using the existing advanced resistance formula:

$$f = k1/Re + k2/Re^{\alpha}$$

If $k1 = 19.0$, $k2 = 0.05$ and $\alpha = 0.3$, then, over the range $Re = 10$ to 5000, the maximum error is 20%

Thermal Resistance of Thermal Bonds

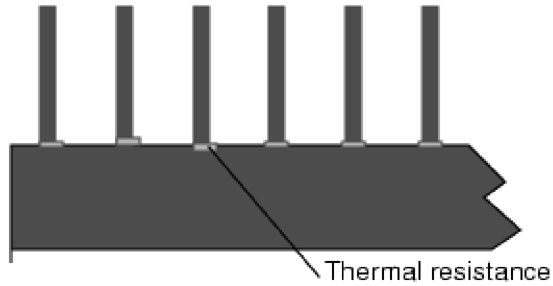
Bonded fin heat sinks are constructed by bonding fins into grooves cut in the base of the heat sink.

The thermal resistance resulting from the epoxy bond is difficult to quantify as it is affected by a number of factors, such as:

- Whether the fin is in the middle of its groove
- Air pockets in the epoxy
- Contact resistance between the bottom of the fin and the bottom of the groove
- Contact resistance between the fin and the epoxy

The thermal resistance of the epoxy is represented as a solid-to-solid surface thermal resistance at the root of the fin as shown in Figure 16-1

Figure 16-1. Heat Sink Fin-Root Thermal Resistance



In representing the thermal resistance it is assumed that:

- The fin is in the middle of the groove
- The epoxy is free of air pockets
- There is no contact resistance between the sides of the fin, epoxy, and the sides of the groove in the base of the heat sink
- There is an infinite contact resistance between the end of the fin and the base of the groove, so that all the heat enters through the sides of the fin

Although the fin may make contact with the base of the groove, some thermal resistance will be present due to the contact resistance.

Equating the conductances:

$$R_{bond} = \left(\frac{\delta}{kA}\right)^{\text{fin side}}$$

where:

δ is the thickness of the bonding material around the fin.

k is the thermal conductivity of the bonding material.

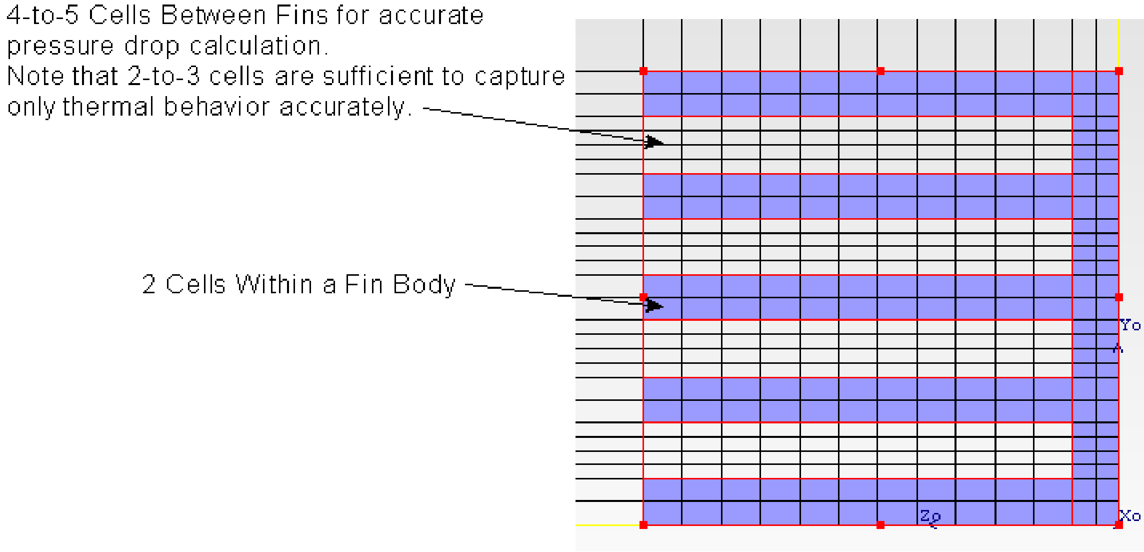
A is the surface area of the fin in contact with the epoxy (both sides of fin).

Heat Sinks and Gridding

One reason for using compact models is to reduce the grid. When modeling detailed heat sinks, a large number of grid cells are required.

To capture the pressure drop and heat transfer between the fins of a parallel fin heat sink, it is important to have an adequate grid, see Figure 16-2.

Figure 16-2. Adequate Level of Grid



The pressure loss through a heat sink is made up of two components: skin friction loss, and exit/entry loss.

The skin friction loss is induced by the fluid flowing over the surfaces of the channels between the fins. The boundary layer that develops between the channels results in a frictional pressure drop that varies roughly in inverse proportion to the velocity through the fins gaps (as the flow there is typically laminar for parallel fin heat sinks used in electronics cooling).

The entry/exit loss is caused because the flow streamlines contract/expand as the flow enters and leaves the heat sink, respectively. This loss of inertia in nature and varies roughly with the square of the velocity.

To capture the skin friction losses a reasonable grid density must be maintained in the span-wise direction, as shown in Figure 16-2.

Note

The grid configuration may not be as expected for staggered pin arrangements, because the number of cells between pins only takes into account the pins that lie along the 'in-line' axes, and cells within intermediate staggered pins are ignored.

For example, Figure 16-3 and Figure 16-4 show in-line and staggered arrangements. When the pins are staggered, as shown in Figure 16-4, the number of cells within the intermediate staggered pins (2 in this case) are counted as one cell.

Figure 16-3. Grid for In-Line Pins

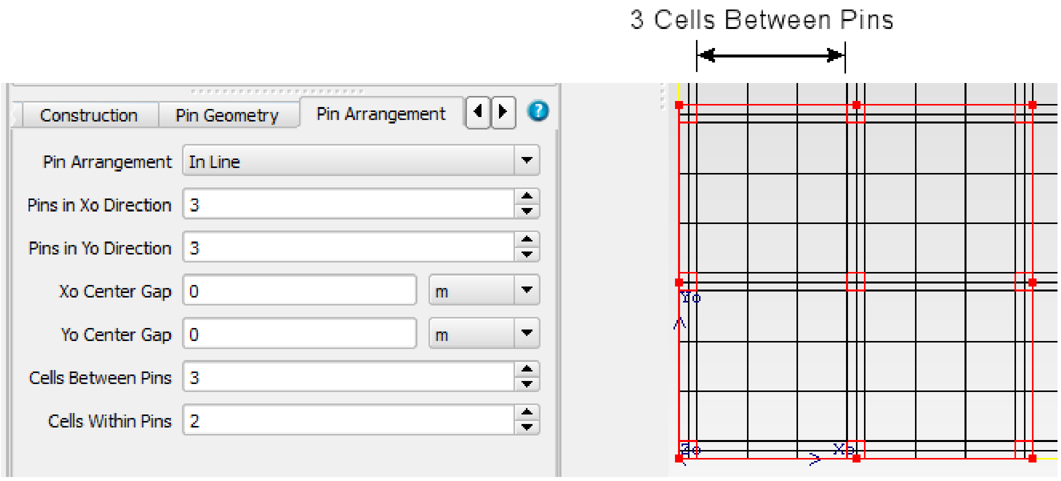
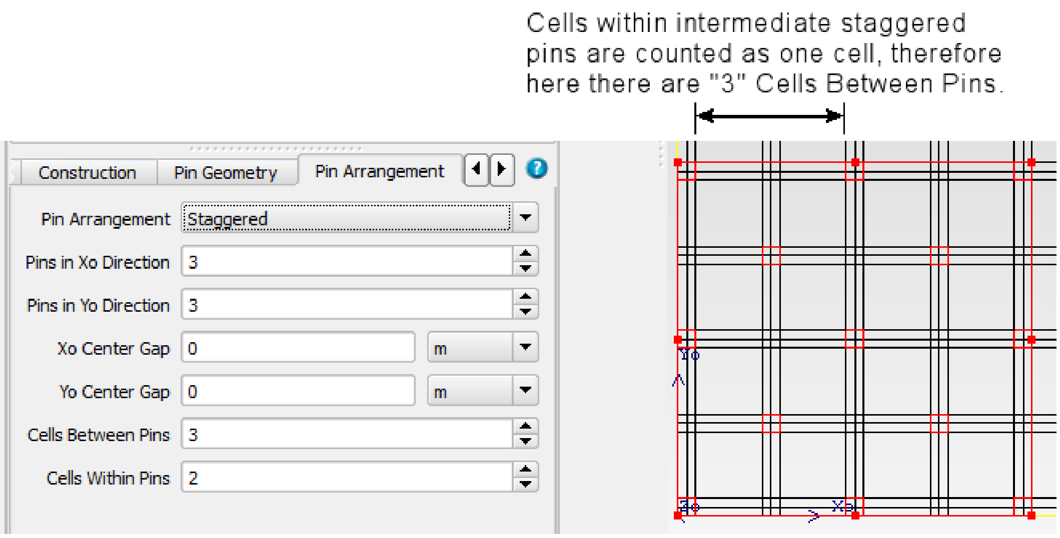


Figure 16-4. Grid for Staggered Pins



Creating Heat Sinks

There are two modeling methods: Detailed and Compact.

Procedure

1. Select an assembly in the data tree.
2. In the New Object Palette, choose Project Manager Create or Drawing Board Create and double-click the **Heat Sink** icon.
3. In the **Construction** tab, choose a Detailed or Compact modeling level.

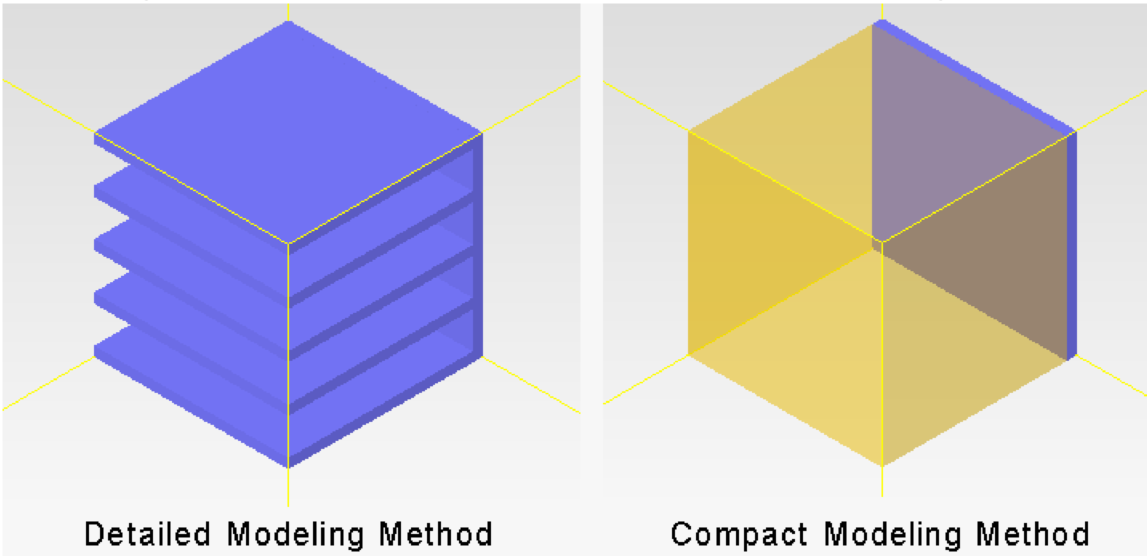
   A Detailed model provides the most accurate physical and thermal representation of the heat sink, generating a large number of grid cells.

A Compact model provides a thermally equivalent simplified representation. The heat sink base and fins are replaced by an equivalent overall shape, constructed from planar and volume resistances for the finned region and a cuboid for the base. This simplified shape reduces grid cell count and the solution time.

Results

The default is a Detailed model of an extruded/cast heat sink with five uniform fins, as shown on the left of Figure 16-5.

Figure 16-5. Default Heat Sink SmartPart in the Drawing Board



Heat Sink Properties

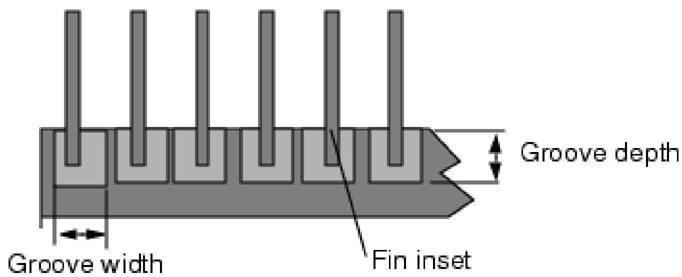
The properties of Heat Sink SmartParts are described in the following subsections and in the property sheet description.

Heat Sink Fabrication

There are two methods of fabrication: Extruded/Cast and Bonded

- Extruded/Cast — constructs the fins from a single homogenous material and manufactured using an extrusion or casting process.
- Bonded — constructs by bonding fins into grooves cut into the base of the heat sink. To enable the program to calculate the resistance effects, you specify the construction by setting the groove width and depth (see Figure 16-6), fin material and adhesive type.

Figure 16-6. Heat Sink Groove Dimensions



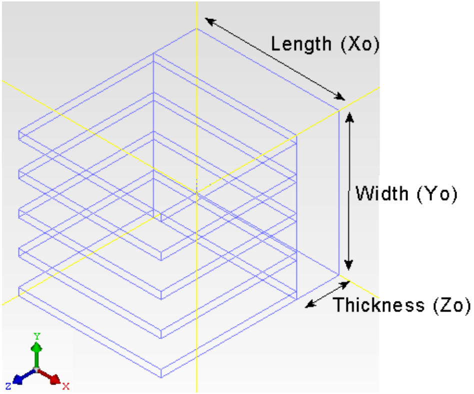
Related Topics

Thermal Resistance of Thermal Bonds

Heat Sink Base Properties

The size of the heat sink base is with reference to the heat sink origin.

Figure 16-7. Heat Sink Base Dimensions



The origin of the heat sink is located at the low coordinate corner of the base of the heat sink (Figure 16-7). The convention used is that the base of the heat sink is in the X-Y plane with the fins extruded in the Z-direction.

The base is directly represented as a block of conducting material. In a bonded fin heat sink, the base may be of a different material to the fins. The attributes of the base are that of the constituent material.

Heat Sink Fin Properties

Heat sink with either plate fins or pin fins can be modeled. The properties of those fins are described in the following subsections

Fin Properties Applicable to Plate and Pin Fins

Plate fins and pin fins share some common characteristics such as tapered fins and can be arranged with center gaps,

Fin Style

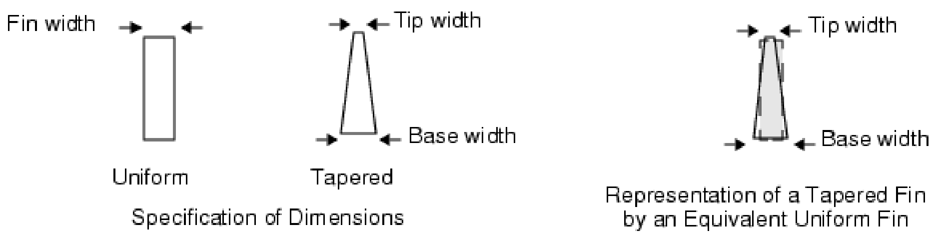
There are two options for selecting the fin style. The internal fins can be either uniform or tapered in cross-section.

Figure 16-8 shows how these are specified. For Uniform fins, the fin width is required. For Full Taper fins, the widths at the base and the top of the fin are required.

Tapered fins are represented in the drawing board by a block having the same thermal mass.

The effective width of a tapered fin is the mean value over the height, that is, the width of the representative block is calculated from $(W_B + W_T)/2$ where W_B and W_T are the base and tip widths respectively.

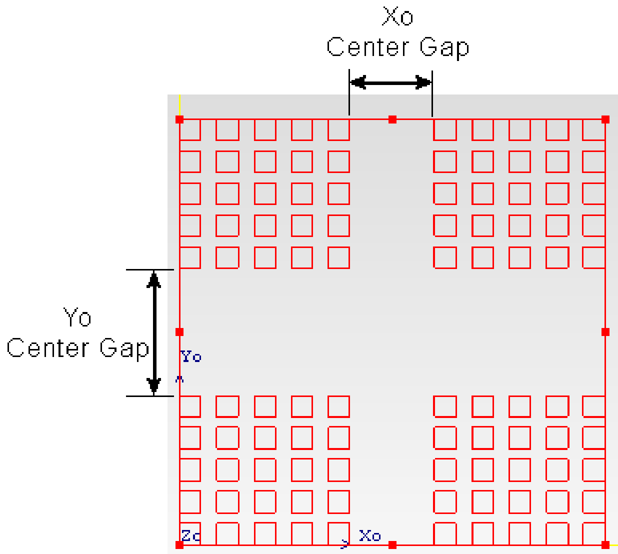
Figure 16-8. Heat Sink Uniform and Tapered Fins



Center Gaps

Figure 16-9 shows the effect of adding a center gap to a Pin fin heat sink. In this example there are 6 pins in the Xo direction and 4 pins in the Yo direction. Center gaps can also be specified for Plate fin heat sinks.

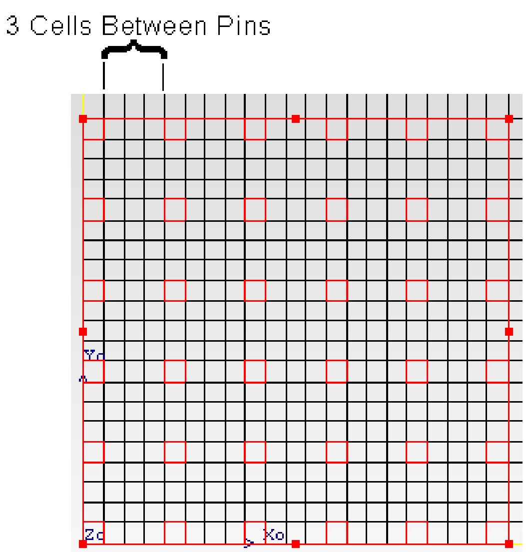
Figure 16-9. Center Gap



Cells Between Fins/Pins

Figure 16-10 shows the default grid arrangement, with three cells between pins. Three grid cells are recommended between a pair of pins to capture the thermal effects accurately. Four to five cells are needed if an accurate pressure drop value is required.

Figure 16-10. Cells Between Pins



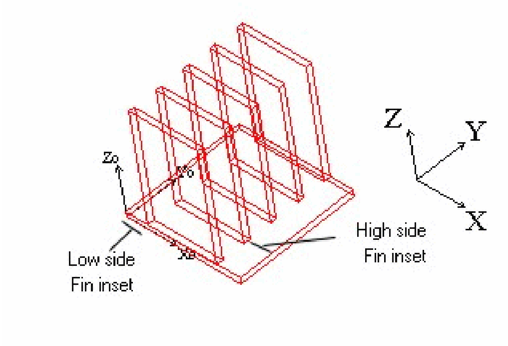
Fin Properties Applicable to Plate Fins

Plate fins can be inset from the heat sink base, and the end fins can be separately defined.

Plate Fin Insets

Plate fins can be inset from the base on the Low or High side in the X-direction, see Figure 16-11.

Figure 16-11. Heat Sink High and Low Side Fin Insets



End Fins

The end fins can be either the same as the internal fins, or a non-standard specification. If non-standard then you can define the end fin size and shape, and whether it has a bracket.

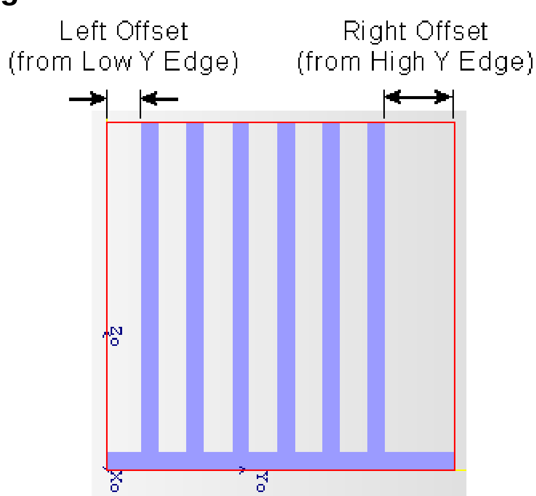
The height and width of the end fins can be more or less than that of the internal fins.

If the heat sink is bonded, then the end fins are the same material set as the fin material.

End Fin Offsets

Offsets are from either end of the base, see Figure 16-12.

Figure 16-12. Heat Sink End Fin Offsets



If the offset is zero, the block representing the fin is aligned with the edge of the base.

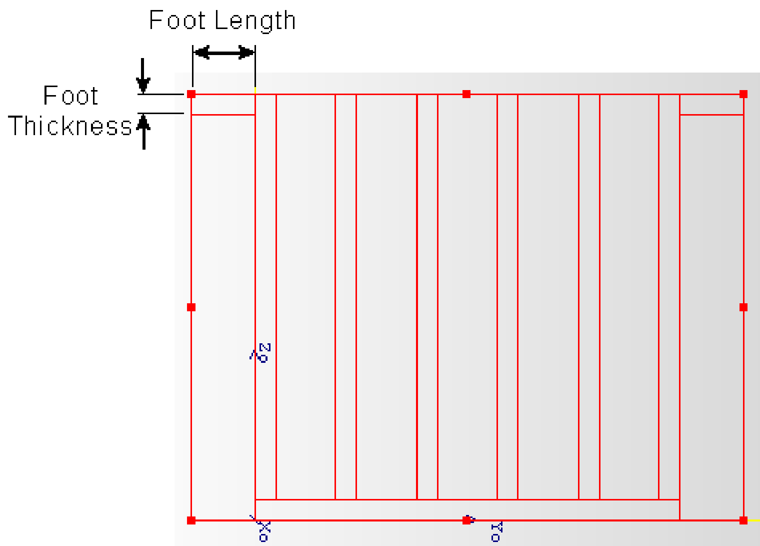
A Left Offset is the distance from the Low Y edge of the base to the start of the fin.

A Right Offset is the distance from the High Y edge of the base to the start of the fin.

End Fin Brackets

Option to attach a bracket to end fins. Selecting End Fin Brackets activates the bracket foot data entry fields, see Figure 16-13.

Figure 16-13. Heat Sink End Fin Brackets



Fin Properties Applicable to Pin Fins

Pin fins can be rectangular or circular in cross section, and can be arranged in line or staggered. The height of a pin fin is the distance from the base to the tip.

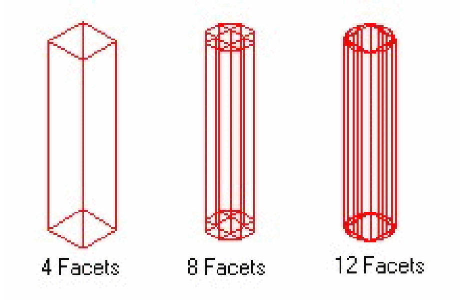
Rectangular Pins

There are two style options: Uniform or Full Taper. Each pin is always represented by a single block. If the pins are tapered, they are represented by a block having the same thermal mass, with the effective width being the mean value over the height, see “Fin Style” on page 182.

Circular Pins

For circular pins, you specify the Modeling Level as 4, 8 or 12 facets, see Figure 16-14.

Figure 16-14. Heat Sink Circular Pin Modeling Levels

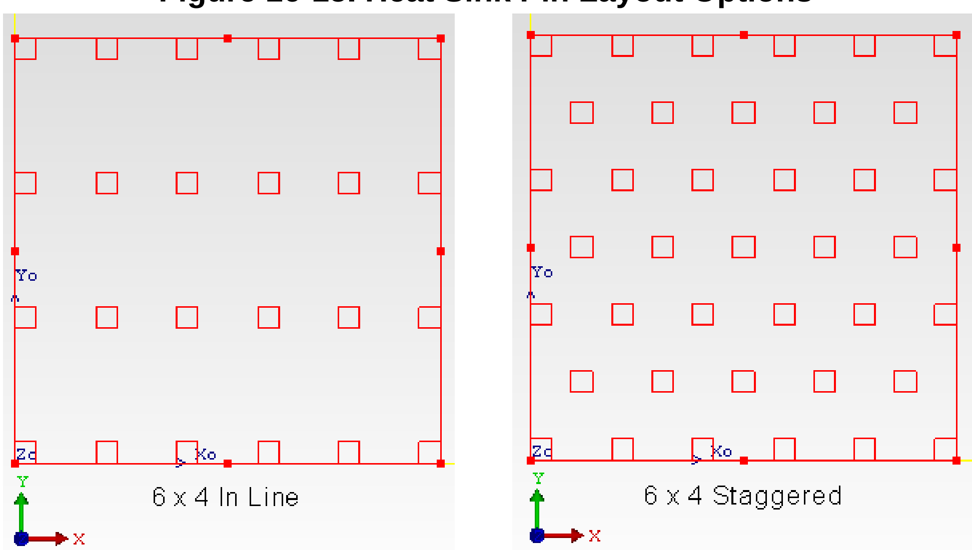


There are two style options: Uniform or Full Taper. Each pin is always represented by a uniform column. If the pins are tapered, they are represented by a uniform column having the same thermal mass, with the effective width being the mean value over the height, see "Fin Style" on page 182.

Pin Fin Arrangement

There are two pin layout options: In Line or Staggered. Figure 16-15 shows the In Line and Staggered arrangements when there are 6 pins specified in the Xo direction and 4 pins in the Yo direction.

Figure 16-15. Heat Sink Pin Layout Options



Heat Sink Property Sheet

To access: Select a Heat Sink SmartPart in the model.

Use this property sheet to configure a heat sink.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Construction tab.	See "Heat Sink Property Sheet Construction Tab" on page 188.
Internal Fins tab.	Not applicable to Pin Fin heat sinks. See "Heat Sink Property Sheet Internal Fins Tab" on page 189.
End Fins tab.	Not applicable to Pin Fin heat sinks. See "Heat Sink Property Sheet End Fins Tab" on page 190.
Pin Geometry tab.	Not applicable to Plate Fin heat sinks. See "Heat Sink Property Sheet Pin Geometry Tab" on page 191.
Pin Arrangement tab.	Not applicable to Plate Fin heat sinks. See "Heat Sink Property Sheet Pin Arrangement Tab" on page 192.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.

Heat Sink Property Sheet Construction Tab

To access: Select a Heat Sink SmartPart, then select the **Construction** tab.

Use this property sheet tab to change the specification of a plate or pin fin Heat Sink SmartPart. The Heat Sink SmartPart supports plain fin designs which are formed by extrusion, casting, or bonding.

Objects

Field	Description
Length (Xo), Width (Yo), Thickness (Zo)	The dimensions of the heat sink base.
Type	Two types of fin can be specified: Plate Fin or Pin Fin. The property sheet tabs available depend on which is selected,
Modeling Method	Two types of modeling method are available: Detailed or Compact. Compact models are best suited to heat sinks in forced convection flow, where radiative losses are not a significant contribution.
Method of Fabrication	The base to fin fabrication method: Extruded/Cast or Bonded.
Groove Depth and Groove Width	(Bonded) Sets the measurements of the grooves cutout in the base of the heat sink to affix the fins.
Fin Material	(Bonded) The fin material does not have to be the same as the base material, and can be selected here, or a new material created.
Adhesive	(Bonded) Select the adhesive material that fixes the fins to the base, or create a new material.

Related Topics

Creating Heat Sinks

Heat Sink Base Properties

Heat Sink Fabrication

Heat Sink Property Sheet Internal Fins Tab

To access: Select a Heat Sink or EDA Heat Sink SmartPart, then select the **Internal Fins** tab. This tab is only available if Plate Fin (Type) has been selected in the **Construction** tab.

Use this property sheet tab to define the number, size, and layout of the internal (not the end) plate fins on a heat sink.

Objects

Field	Description
Number of Internal Fins	An integer field for the total number of fins (excluding the end fins).
Fin Height	A floating point field for the fin height. Internal fins are all the same height.
Center Gap	The center gap size.
High Side Fin Inset and Low Side Fin Inset	Set the fins offset distance from the edge of the heat sink base.
Fin Style	• Uniform — requires only the width to be defined. • Full Taper — requires both the base width and tip width to be defined.
Width	(Uniform) The width of a uniform fin.
Base Width	(Full Taper) The width of a tapered fin at its base.
Tip Width	(Full Taper) The width of a tapered fin at its tip.
Cells Between Fins	The number of grid cells between fins, see "Heat Sinks and Gridding" on page 176. The default is 3.
Cells Within Fins	The number of grid cells within the body of a fin, see "Heat Sinks and Gridding" on page 176. The default is 2.

Usage Notes

Tapered fins are modeled as uniform fins with the same thermal mass.

Related Topics

Fin Properties Applicable to Plate and Pin Fins

Fin Properties Applicable to Plate Fins

Heat Sink Property Sheet End Fins Tab

To access: Select a Heat Sink or EDA Heat Sink SmartPart, then select the **End Fins** tab. This tab is only available if Plate Fin (Type) has been selected in the **Construction** tab.

Use this property sheet tab to define the end fins of a plate fin heat sink. In addition, an optional bracket can be attached to end plate fins.

Objects

Field	Description
Left and Right Offset	Defines distances of the end fins from the edges of the base plate.
Left Fin/Right Fin Non-Standard	“Non-Standard” being different from the internal fins.
Fin Height	(Non-Standard) Enables an end fin height to be defined that is different from the height of the internal fins.
Fin Style	(Non-Standard) • Uniform — requires only the width to be defined. • Full Taper — requires both the base width and tip width to be defined.
Width	(Non-Standard and Uniform) The width of a uniform end fin. This may be different from the internal fins.
Base Width	(Non-Standard and Full Taper) The width of a tapered end fin at its base. This may be different from the internal fins.
Tip Width	(Non-Standard and Full Taper) The width of a tapered end fin at its tip. This may be different from the internal fins.
End Fin Bracket	(Non-Standard) Enables a bracket to be added to the end fin.
Foot Length and Foot Thickness	(Non-Standard and End Fin Bracket) The dimensions of the bracket.

Related Topics

Fin Properties Applicable to Plate Fins

Heat Sink Property Sheet Pin Geometry Tab

To access: Select a Heat Sink or EDA Heat Sink SmartPart, then select the **Pin Geometry** tab.
This tab is only available if Pin Fin (Type) has been selected in the **Construction** tab.

Use this property sheet tab to define the heat sink pin shape, style, and dimensions.

Objects

Field	Description
Pin Shape	The cross-sectional shape can be either Rectangular or Circular.
Pin Style	The style of the pins can be either Uniform or Full Taper.
Length and Width	(Rectangular and Uniform) Length is the size in the Xo direction. Width is the size in the Yo direction.
Base Length, Base Width, Tip Width	(Rectangular and Full Taper) Length is the size in the Xo direction. Width is the size in the Yo direction.
Diameter	(Circular and Uniform) The diameter of a uniform pin fin.
Base Diameter and Tip Diameter	(Circular and Full Taper) The base and tip diameters of a tapered pin fin.
Pin Height	A floating point field for the pin height. Pin fins are all the same height.
Modeling Level	(Circular) Approximate to a curved surface using 4 (default), 8 or 12 Facets.

Usage Notes

Tapered fins are modeled as uniform fins with the same thermal mass.

Related Topics

Fin Properties Applicable to Plate and Pin Fins

Fin Properties Applicable to Pin Fins

Heat Sink Property Sheet Pin Arrangement Tab

To access: Select a Heat Sink or EDA Heat Sink SmartPart, then select the **Pin Arrangement** tab. Note that this tab is only available if Pin Fin (Type) has been selected in the **Construction** tab.

Use this property sheet tab to define how the heat sink pins are arranged over the base.

Objects

Field	Description
Pin Arrangement	Pins can be In Line or Staggered.
Pins in Xo and Yo Direction	The number of pins in each direction.
Xo and Yo Center Gap	The dimension of a center gap in each direction.
Cells Between Pins	The number of grid cells between pins, see "Heat Sinks and Gridding" on page 176. The default is 3.
Cells Within Pins	The number of grid cells within the body of a pin, see "Heat Sinks and Gridding" on page 176. The default is 2.

Related Topics

Fin Properties Applicable to Plate and Pin Fins

Fin Properties Applicable to Pin Fins

Chapter 17
Holes

Use Hole SmartParts to create rectangular-shaped ventilation gaps, more complex-shaped holes can be created using MCAD Bridge. Optionally, a hole can be filled and can have a defined resistance to flow.

Creating Holes

Holes can only be created as children of Cuboid SmartParts or children of Walls of Enclosure SmartParts.

Note

Holes that are 0.01% or less than the smallest side of the parent object's face are ignored by the grid, the solver, and the decompose command, and are only displayed in wireframe rendering mode.

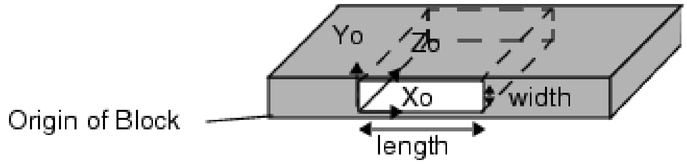
Procedure

1. Select a cuboid or enclosure wall in the data tree.
2. In the New Object Palette, choose Project Manager Create or Drawing Board Create and double-click the **Hole** icon.

Results

The origin of the hole, relative to the origin of the cuboid or enclosure wall, is defined by the Position fields in the **Location** tab.

Figure 17-1. Hole Position and Dimensions



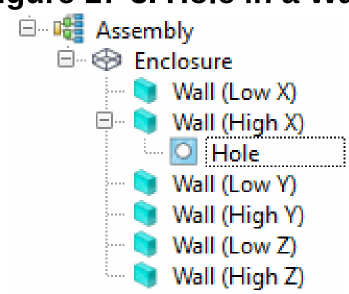
When a hole is added as a child of a cuboid, the combination of cuboid and hole is transformed into a Block with Holes SmartPart.

Figure 17-2. Hole in a Cuboid

The new hole is one-tenth the size of the cuboid X-Y face and located at the origin of the parent block. Any number of holes can be added to the new block.

When a hole is added as a child of an enclosure wall, the hole is one-tenth the size of the enclosure wall.

Figure 17-3. Hole in a Wall



Related Topics

Modeling Holes

Creating Complex Shaped Holes

Treatment of Vents

Hole Property Sheet

Modeling Holes

How to copy, locate, and resize holes.

Copying Holes

Repeated holes can be created by:

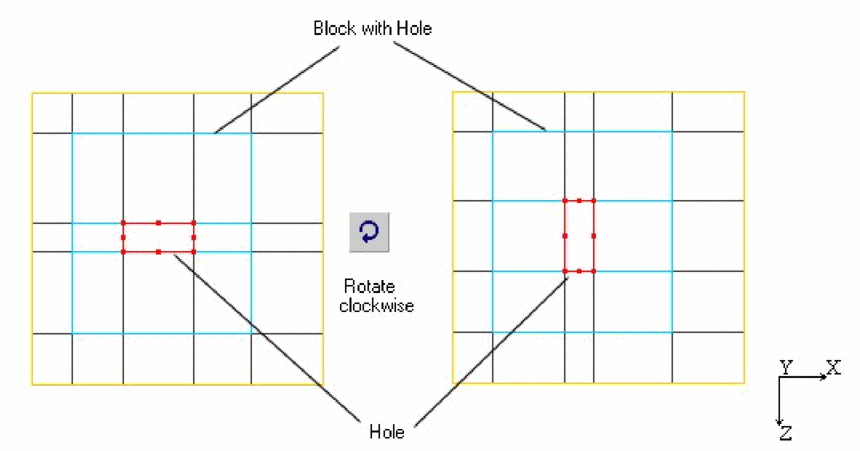
- Copying and pasting, using the **Edit** menu.
- Patterning, using **Edit** > **Pattern**, see Pattern Creation Dialog Box in the *Simcenter Flotherm User Guide*.

Locating and Resizing Holes

If you need to re-locate and/or resize a hole, you can edit the hole in the Drawing Board by dragging its boundaries.

Also, you can rotate a hole in the plane parallel to the axis of the parent plane. To rotate a hole, select it in the Drawing Board and click either the **Rotate Clockwise** or **Rotate Anti-Clockwise** icon, see Figure 17-4.

Figure 17-4. Locating and Resizing Holes in the Drawing Board



Related Topics

Creating Holes

Creating Complex Shaped Holes

Treatment of Vents

Hole Property Sheet

Creating Complex Shaped Holes

You can create more complex holes by transferring geometry to MCAD Bridge, modifying it there, then transferring it back.

For example, if it is essential to create a round hole, you could create a coincident cuboid and cylinder and transfer the geometry to MCAD Bridge where the shapes could be subtracted to create an assembly representing a block with a round hole. The resultant assembly is then transferred back to the Project Manager.

Procedure

1. Create a cuboid or collapsed cuboid using the New Object panel.
2. Create a cylinder, with the required number of facets, superimposed over the location of the hole.

3. Transfer the geometry to MCAD Bridge using **External** > **Import Project Geometry** in MCAD Bridge.
4. In MCAD Bridge, set Current Selection Mode to MCAD Body and select the cuboid, then the cylinder, and select **Tools** > **Subtract Bodies**.
5. Decompose the resultant object using **Tools** > **Decompose** which calls the Decompose dialog.
6. Transfer the geometry back to the Project Manager using **Tools** > **Transfer Assembly**.

Related Topics

Creating Holes

Modeling Holes

Treatment of Vents

Hole Property Sheet

Treatment of Vents

Vents can be modeled by specifying the flow boundaries over holes created in enclosures or blocks.

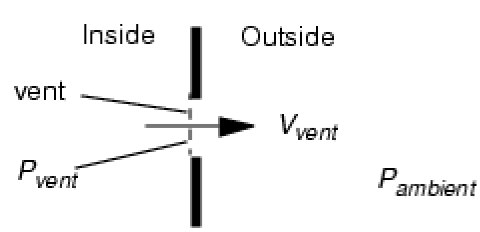
This can be done with either:

- a known stagnation pressure at an inlet, or
- a known external static pressure at an outlet.

The program determines the mass flow rate and the velocity boundary value.

In general, as a boundary condition, we need to prescribe the static pressure P_{vent} at the plane just inside the vent. We presume that the surroundings are a fluid reservoir with zero velocity, and known ambient static pressure $P_{ambient}$.

Outflow Vents



At outflow vents we want to impose a loss in P_{total} between the vent and the reservoir ambient of:

$$\frac{1}{2}\rho V_{vent}^2$$

which is well established empirically. From this we can deduce P_{vent} as follows:

$$P_{total}\text{at vent} = P_{vent} + \frac{1}{2}\rho V_{vent}^2$$

$$P_{total}\text{at ambient} = P_{ambient}$$

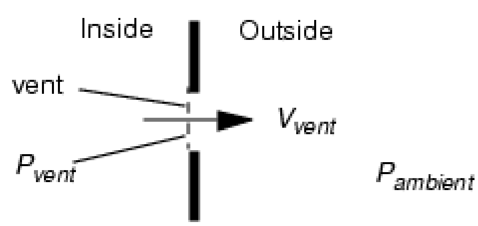
(because $V_{ambient}$ is zero)

We want:

$$P_{\text{total vent}} - P_{\text{total ambient}} = \frac{1}{2}\rho V_{vent}^2$$

Hence $P_{vent} = P_{ambient}$ as set in Simcenter Flotherm.

Inflow Vents



For inflows, we assume loss free acceleration from the zero velocity ambient in the reservoir to the vent. This implies that:

$$P_{total\ vent} = P_{total\ ambient}$$

Hence:

$$P_{vent} + \frac{1}{2}\rho V_{vent}^2 = P_{ambient}$$

so

$$P_{vent} = P_{ambient} - \frac{1}{2}\rho V_{vent}^{2}$$

as set in Simcenter Flotherm.

This means that P_{vent} is lower than $P_{ambient}$. This does not imply a loss, it is a consequence of the acceleration between ambient and vent.

Comments on Vents

In most cases there will be additional losses at the vent because of grilles, slats, and so on. These are treated as an additional loss (in total pressure) between vent and ambient, set as vent resistance.

For inflow, there may in fact be some losses between ambient and vent, rather than zero loss as we suppose here. The difficulty is that these losses will depend on the geometric configuration of the vent and the surroundings, and are impossible to estimate in general. Also, they will usually be swamped by the effects of grilles, slats, and so on Hence we make the simple consistent assumption of zero loss as the standard boundary condition. Additional losses, if known, and if significant, can be added as a vent resistance.

Finally, it is of course possible to extend the Simcenter Flotherm solution outside the vent, to include a portion of the ambient, at some expense in solution time. If this is done, then none of these assumptions are relevant.

Related Topics

Creating Holes

Modeling Holes

Creating Complex Shaped Holes

Hole Property Sheet

Hole Property Sheet

To access: Select a Hole SmartPart in the model.

Use this property sheet to configure a hole.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Construction tab.	See "Hole Property Sheet Construction Tab" on page 200.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.

Hole Property Sheet Construction Tab

To access: Select a Hole SmartPart, then select the **Construction** tab.

Use this property sheet tab to change holes added to a cuboid block or enclosure wall and, where necessary, replace the hole with a specified material or resistance.

Objects

Field	Description
Length (<*axis*>) and Width (<*axis*>)	The axes of the Length and Width are dependent on the direction of the hole. For example, for a hole in the Zo direction of a Block with Holes, or in a "Z" wall of an Enclosure, they are Length (Xo) and Width (Yo). For a hole in the Xo direction of a Block with Holes, or in an "X" wall of an Enclosure, they are Length (Yo) and Width (Zo). The dimensions set the area of the hole which will traverse the full distance of the block, see Figure 17-1. By default, the dimensions are one-tenth of that of the block.
Replace With	By default, holes are open spaces, that is, they are filled with air, but they can be replaced with a material or a resistance. For a description of the handling of vents, see "Treatment of Vents" on page 196.
Material	(Material) Select or create a Material attribute.
Resistance	(Resistance) Select or create a Resistance attribute.
Modeling Level	(Material or Resistance) When replacing holes with materials or defined resistances, then an additional modeling level can be set: • Block Set — the default, inherited from the parent property sheet (Block or Wall). • Thick — is treated as a 3D object • Thin — is treated as a collapsed object The Thick and Thin options allow you to set the thickness of the replacement section.
Position	(Thick or Thin) Locates the replacement section either on the edge or the middle of the wall. The choices are Low-Face, Mid-Face or High-Face.
Thickness	(Thick or Thin) Sets the thickness of the replacement section.

Related Topics

Creating Holes

Modeling Holes

Creating Complex Shaped Holes

Treatment of Vents

Block With Holes Property Sheet

Thick (3D) and Thin (2D) Walls, Holes, and Sloping Blocks

Chapter 18
Monitor Points

Use Monitor Point SmartParts when you want to monitor variables (usually temperature) at specific positions in the solution domain.

Although they are not physical objects, that is, they have no dimensions, monitor points are treated as part of the geometry structure and are displayed in the drawing board as cross-hairs, of a fixed size to aid visibility.

Monitor Point Locations and Default Names

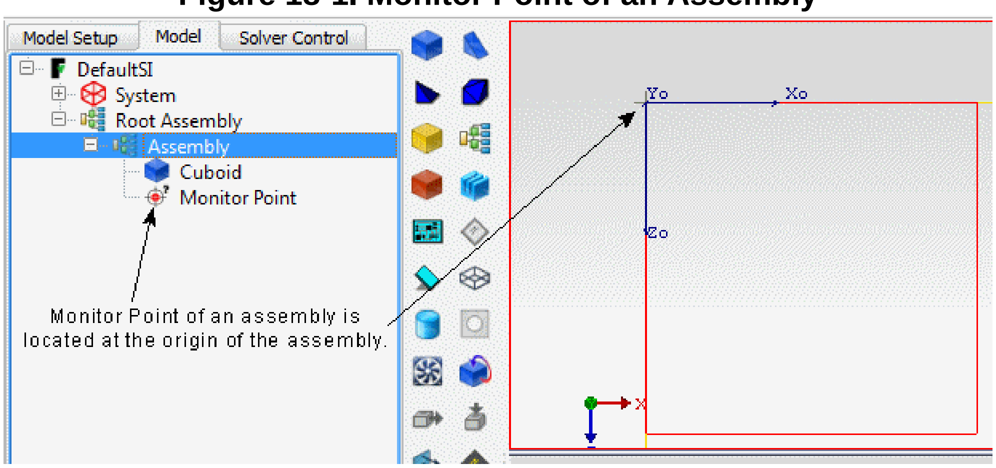
The default location and name of a monitor point depends on what was selected in the data tree before it was added.

- Assembly, Network Assembly or Network Node selected:

  The location is at the origin of either the parent assembly or the model (when local or absolute coordinates have been respectively set in the User Preferences dialog).

  The name is “Monitor Point” for the first instance, which is then suffixed with :<n> for subsequent instances.

Figure 18-1. Monitor Point of an Assembly

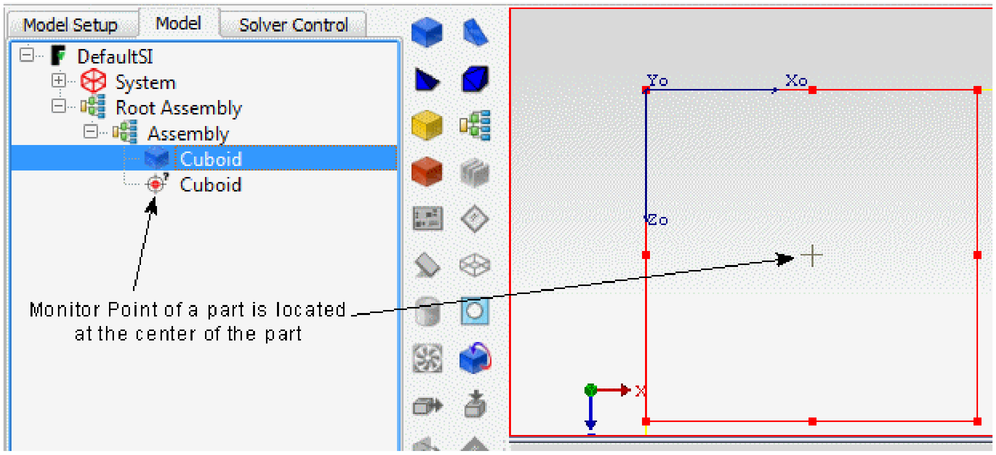


- Part, for example, a Cuboid or Region, selected:

  The location is at the center of the part.

  The name is the name of the parent, for example, “Cuboid” for the first instance, which is then suffixed with :<n> for subsequent instances.

Figure 18-2. Monitor Point of a Cuboid



Note

Assembly monitor points are moved when the assembly is moved, that is, they remain at the origin of the assembly. However, part monitor points, which are originally at the center of the part, *do not move with the part* unless they are co-selected before moving.

Monitor Points in the Drawing Board

When solid rendering is switched on, monitor points in centers of solids are only visible when they are selected.

Related Topics

Monitor Points Usage

Creating Monitor Points

Monitor Point Property Sheet

Monitor Points Usage

Monitor Points are used to check the variation of the field variables.

The variation of field variables is plotted using the Profiles window. This is useful for monitoring convergence and is fundamental to transient analyses as it is through monitor points that you observe how values of variables change with time.

Related Topics

Solution Monitoring and Profile Plots [Simcenter Flotherm User Guide]

Creating Monitor Points

Creating Monitor Points

Add monitor points at locations at which you want to check the variation of the field variables.

Procedure

1. Select an object to assign the monitor point.

   This determines the monitor point's default location, which can be subsequently changed.

2. Click the **Monitor Point** icon in the New Object Palette.

Results

The new monitor point appears in the data tree as a child of the object, for example, see Figure 18-1 and Figure 18-2.

In the particular case of adding a monitor point to a TEC, two monitor points are added, see "Adding a TEC With Hot and Cold Side Monitor Points" on page 316.

Related Topics

Monitor Point Locations and Default Names

Monitor Points Usage

Monitor Point Property Sheet

Monitor Point Property Sheet

To access: Select a Monitor Point in the model.

Use this property sheet to rename or change the position of a monitor point, or to deactivate or hide the monitor point.

Objects

Object	Description
Name, Position, De-active, and Hide.	See "Generic Property Sheet Location Tab" on page 28.

Related Topics

Monitor Point Locations and Default Names

Monitor Points Usage

Creating Monitor Points

Chapter 19
Network Assemblies, Network Nodes and Network Cuboids

The Network Assembly SmartPart can be used as a generalized thermal resistance for modeling a range of electronic parts, including multi-die and stacked-die packages.

Network Assembly Modeling

A network assembly is an assembly of objects with surfaces that behave as nodes in a network of thermal resistors.

The Compact Component SmartPart (see "Compact Components" on page 45) is tailored more specifically for packages including both resistor network and level two interconnect definitions.

Tip

If your project contains one or more network assemblies and is very slow to converge, try activating the Network Assembly Block Correction solver option, see Solver Control Tab in the *Simcenter Flotherm User Guide*.

Related Topics

Network Assembly Structure

Network Assembly Procedures

Examples of Network Assemblies

Network Assembly Property Sheet

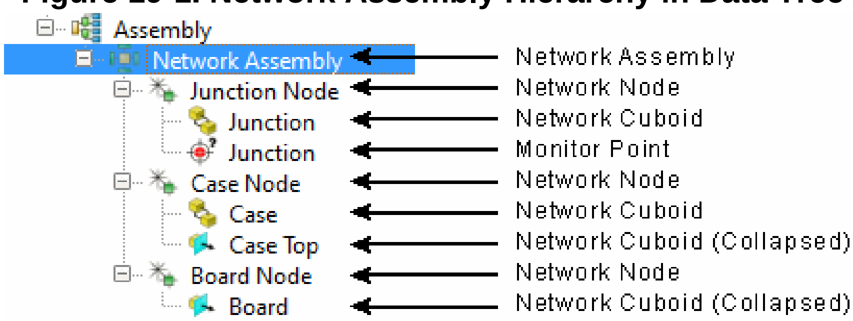
Network Assembly Structure

The complete structure of a network assembly is represented in the data tree as a hierarchical group of nodes and cuboids.

The hierarchy is as described below and shown in Figure 19-1.

- A top-level parent Network Assembly. This is a zero-sized object located at the origin of its parent assembly.
- Child Network Nodes, to which heat sources can be added.
- Grandchild Network Cuboids, which define the physical shape of the assembly.
- Grandchild Network Collapsed Cuboid, which connect the nodes to the surrounding CFD grid cells.
- Monitor points, for tracking node temperatures.

Figure 19-1. Network Assembly Hierarchy in Data Tree



Uncollapsed and collapsed cuboids perform different tasks:

- Use 3D network cuboids to define the region of space in which the nodal temperature is displayed, as well as to block the flow.
- Use 2D collapsed network cuboids to connect to the CFD environment, that is, use them to define the external surface associated with a node. 2D collapsed network children make the node a *peripheral* node as opposed to an *internal* node.

For modeling examples of network assemblies, see “Examples of Network Assemblies” on page 214.

Related Topics

Network Assembly Modeling

Network Assembly Procedures

Examples of Network Assemblies

Network Assembly Property Sheet

Network Assembly Procedures

This section describes how to create network assemblies and how to define node resistance and capacitance values.

Creating Network Assemblies

Create network assemblies by adding Network Nodes to Network Assemblies, then Network Cuboids to Network Nodes.

Procedure

1. Select an assembly in the data tree and add a Network Assembly SmartPart.
2. If appropriate, localize the grid inside the Network Assembly by checking Localize in the **Location** tab of the Network Assembly Property Sheet.

   This is useful for obtaining detailed nodal information for complicated networks, for example multi-chip models with several junctions.
3. Select the new Network Assembly and add a Network Node SmartPart.
4. Repeat Step 2 until all required Network Nodes have been added.
5. Select a Network Node and:
   - If appropriate, add a Network Cuboid to represent a 3D physical representation.
   - If required, add a Monitor Point to track temperature in grid cells.
   - If appropriate, add a Network Cuboid to define the external surface(s) of the node. This will need to be collapsed to be 2D, see “Collapsing 3D Cuboids to 2D Planes” on page 23. The data tree icon changes, see Figure 19-1 on page 208.
6. Repeat Step 4 for all nodes.

   So far you have created the hierarchical structure of the network assembly.
7. Select the Network Assembly and use the Network Assembly Property Sheet to define the resistances between pairs of nodes and the capacitances for the nodes, as appropriate. Refer to “Defining Resistances Between Nodes” on page 211 and “Defining Capacitance at Nodes” on page 212.
8. Where appropriate, select a Network Node and use the **Attachments** tab of the Network Node Property Sheet to add a Thermal attribute.

By default, Network Nodes have no power attached. The Network Node thermal attributes are restricted to the fixed temperature and fixed power options. Attaching a transient profile to the thermal attribute enables the power or temperature to vary over time.

9. Where appropriate, select a Network Cuboid and use the **Attachments** tab of the Network Cuboid Property Sheet to add Surface, Radiation, and Grid Constraint attributes.

    - Surface attributes attached to outer collapsed 2D Network Cuboids connect the nodes to the surrounding CFD cells.
    - The radiative option is restricted to Single Radiating only because Network Cuboids are isothermal.

10. Resize, rotate, and move the Network Cuboids to physically model the network assembly.

    Notes:

    a. Precedence rules apply to Network Cuboids, see Context Rules for Network Assemblies and Heat Pipes in the *Simcenter Flotherm User Guide*.

    b. Network Nodes cannot be selected and re-located independently.

Related Topics

Examples of Network Assemblies

Network Assembly Structure

Defining Resistances Between Nodes

The thermal resistances between pairs of nodes in a network assembly are specified in table format.

Prerequisites

- The Network Assembly must have the relevant Network Nodes attached for them to be selectable in the Resistance Table.

Procedure

1. Open the **Resistance** tab of the Network Assembly property sheet.
2. Select the units of resistance that you are going to enter.
3. Make sure that Read Only is unchecked.

    The Resistance Table is underneath this check box and consists of three columns which identify two nodes and define the resistance between them.

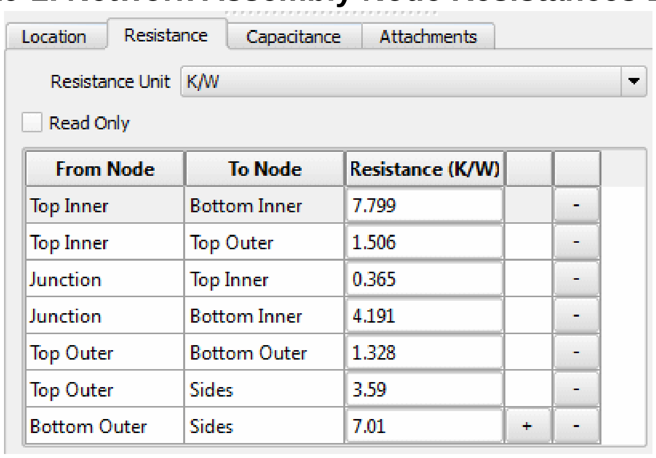
4. Click the selection arrow of a cell in the From Node column and choose the start node from the dropdown menu.
5. Click the selection arrow of the corresponding To Node column and choose the end node.
6. Click the selection arrow of the corresponding Resistance cell and edit the default value.

   By default, the resistance is set to 1.0×10^{10} K/W.
7. Continue, as required, to add or delete resistances by clicking the "+" or "–" button respectively.

Results

Figure 19-2 shows an example of a completed resistance table.

Figure 19-2. Network Assembly Node Resistances Example



Defining Capacitance at Nodes

The thermal capacitances at nodes in a network assembly are specified in table format.

Prerequisites

- The Network Assembly must have the relevant Network Nodes attached for them to be selectable in the Capacitance Table.

Procedure

1. Open the **Capacitance** tab of the Network Assembly property sheet.
2. Select the units of capacitance that you are going to enter.
3. Make sure that Read Only is unchecked.

   The Capacitance Table is underneath this check box and consists of two columns for identifying the node and setting its capacitance value.

4. Click the selection arrow of a cell in the Node column and choose the node from the dropdown menu.
5. Click the selection arrow of the corresponding Capacitance cell and edit the default value.

   By default, the capacitance is set to 1.0×10^{-10} J/K.
6. Continue, as required, to add or delete capacitances by clicking the "+" or "–" button respectively.

Examples of Network Assemblies

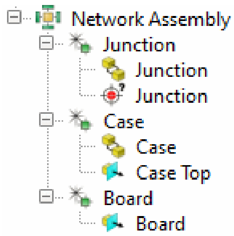
Examples of Network Assemblies are given in the following sections.

Dynamic 2-Resistor Model

In this example, the Network Assembly includes three nodes: Junction, Case, and Board.

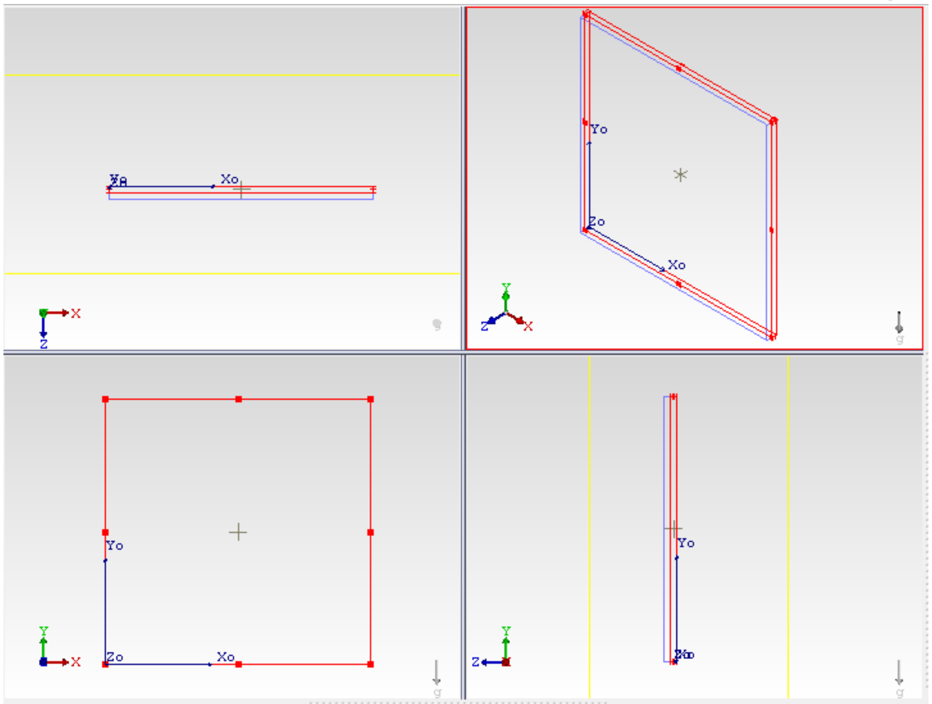
The external surface of the Case node is included as a collapsed cuboid on the top of the Case cuboid. The Board node is included as a collapsed cuboid on the bottom of the Junction node (see Figure 19-3).

Figure 19-3. 2-Resistor Model in the Data Tree



The drawing board views of the model are shown in Figure 19-4, where the Junction node is selected. In this model the junction and case nodes are set to have equal thickness. The cuboids representing the Junction and Case nodes are geometrically identical to those created by the Compact Component SmartPart, see Figure 4-8 on page 51.

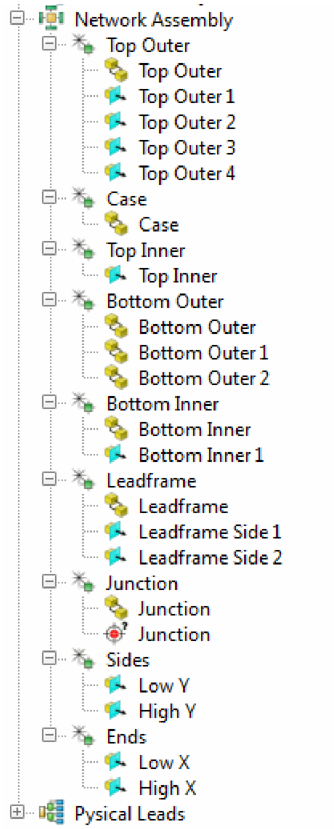
Figure 19-4. 2-Resistor Network Assembly Model in the Drawing Board



DELPHI-Style DCTM

The example given here is for a Dual-Leaded package with one Junction node.

Figure 19-5. DELPHI-Style DCTM Model in the Data Tree



In this case, the Network Assembly consists of the package body, constructed as three layers, as described below.

- Top Outer is the top layer of the assembly, in which the Case node punches a hole. The external connection from Top Outer is represented using 4 collapsed cuboids surrounding Top Inner, also represented as a collapsed cuboid.
- Top Inner is the surface in the center of the top of the package, below which is a cuboid representing the ‘case’ node. The Case node below Top Inner represents the case top center temperature, as measured by a thermocouple under JEDEC test conditions and is not part of a normal DELPHI model. It is not exposed.
- Leadframe is the central layer, represented by a cuboid in which the Junction, also represented by a cuboid punches a hole. The Leadframe node is used to display the temperature of the leads, and the external connections to the physical leads are represented with collapsed cuboids where the physical leads meet the Leadframe. The physical leads are external to the Network Assembly, providing a conduction path from the Leadframe to the board.
- Bottom Outer is the bottom layer of the assembly, in which the Bottom Inner node punches a hole. Bottom Inner extends along the entire length of the bottom of the package, representing an exposed thermal slug. The external connection to Bottom Inner

is represented as a collapsed cuboid. The external connections to Bottom Outer are the bottom surface either side of Bottom Inner

- The Sides node contains the surfaces above the physical leads, represented with collapsed cuboids.
- The Ends node contains the ends of the package, represented with collapsed cuboids.

The case temperature will lie between the Junction and Top Inner temperatures. In the Compact Component SmartPart this is calculated by specifying a factor α to indicate its weighting towards the junction:

$$T_{case} = \alpha \times T_{junction} + (1 - \alpha) \times T_{Top\ Inner}$$

In the Network Assembly, the Case would be solved as part of the network, using resistances of:

$$\alpha \times 1.0E{+}10$$

and

$$(1 - \alpha) \times 1.0E{+}10$$

to connect it to the chosen Junction and Top Inner respectively. This enables the Case node to reach the desired temperature without introducing an additional heat flow path within the network.

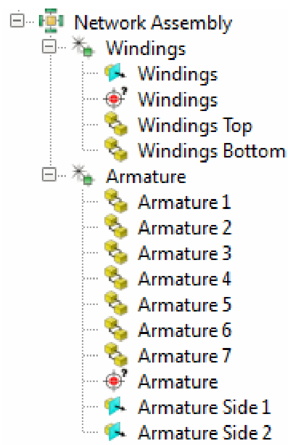
Geometrically this is identical to what would be created with the Compact Component SmartPart.

Transformer

The transformer model has two network nodes that represent the windings and armature.

This shown as a data tree in Figure 19-6.

Figure 19-6. Transformer Model in the Data Tree



Both the armature and the windings have Thermal attributes attached and are monitored with monitor points.

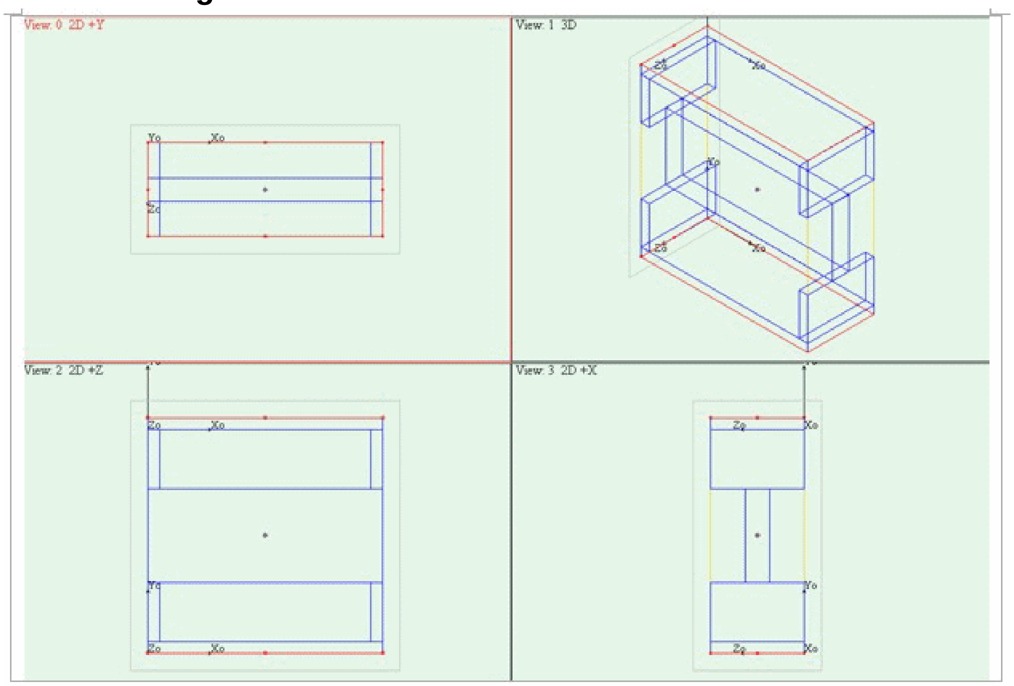
In this case the physical model created is pseudo-detailed, so the armature looks realistic.

The order of precedence rule is invoked, such that the windings are represented as a single cuboid which is overwritten by the armature where this passes through the center of the windings.

The cuboid representing the windings has been hidden, as have the collapsed cuboids representing this external node.

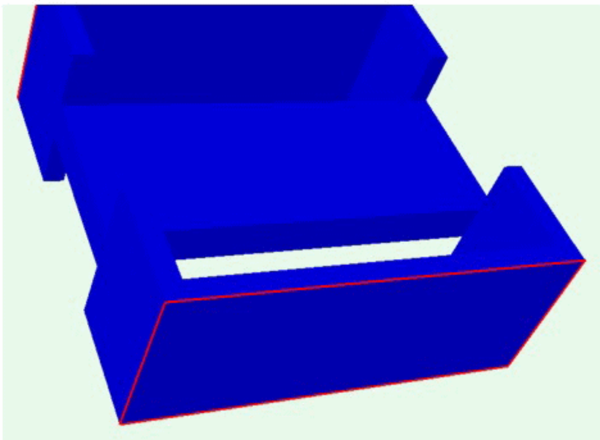
The drawing board views of the model are shown in Figure 19-7. The windings have been hidden and the two sides of the armature forming the external node are highlighted.

Figure 19-7. Transformer Model in Wireframe



The corresponding image of the armature in solid rendering is shown in Figure 19-8.

Figure 19-8. Transformer Model in Solid Rendering



Network Assembly Property Sheets

Network Assemblies are defined by Network Assembly, Network Node and Network Cuboid property sheets.

Network Assembly Property Sheet

To access: Select a Network Assembly SmartPart in the model.

Use this property sheet to configure a network assembly.

Description

Inter-node resistances and node capacitances are entered in separate tables.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Resistance tab.	See "Network Assembly Property Sheet Resistance Tab" on page 222.
Capacitance tab.	See "Network Assembly Property Sheet Capacitance Tab" on page 223.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.

Network Assembly Property Sheet Resistance Tab

To access: Select a Network Assembly SmartPart, then select the **Resistance** tab.

Use this property sheet tab to set the resistance values between the nodes for the selected Network Assembly.

Objects

Field	Description
Resistance Unit	Specifies the units of thermal resistance. The units can be K/W (default), K/mW or °F hr/Btu.
Read Only	Check to prevent the Resistance Table from being modified.
Resistance Table	Use the table to define resistances between pairs of nodes.

Related Topics

Defining Resistances Between Nodes

Network Assembly Property Sheet Capacitance Tab

To access: Select a Network Assembly SmartPart, then select the **Capacitance** tab.

Use this property sheet tab to set the thermal capacitance for each network node for the selected Network Assembly.

Objects

Field	Description
Capacitance Unit	Specifies the units of thermal capacitance. The units can be J/K (default) or mJ/K.
Read Only	Check to prevent the Capacitance Table from being modified.
Capacitance Table	Use the table to define the capacitances at nodes.

Related Topics

Defining Capacitance at Nodes

Network Node Property Sheet

To access: Select a Network Node SmartPart in the model.

Use tis property sheet to configure a network node.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.

Related Topics

Network Assembly Structure

Examples of Network Assemblies

Network Cuboid Property Sheet

To access: Select a Network Cuboid SmartPart in the model.

Use this property sheet to configure a network cuboid.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.

Related Topics

Network Assembly Structure

Examples of Network Assemblies

Chapter 20
PCBs

Use the PCB SmartPart to create basic models of non-conducting or conducting PCBs.

Board layout information can be imported into Simcenter Flotherm, refer to ECAD File Formats in the *Simcenter Flotherm User Guide*.

Modeling PCBs

Non-conducting PCBs are normally used in forced convection systems. Conducting PCBs are normally used for natural convection cases

Non-Conducting PCBs

Non-Conducting PCBs only require the setting of the proportion of the heat dissipated locally to the air on the side that the component is defined, and the surface roughness.

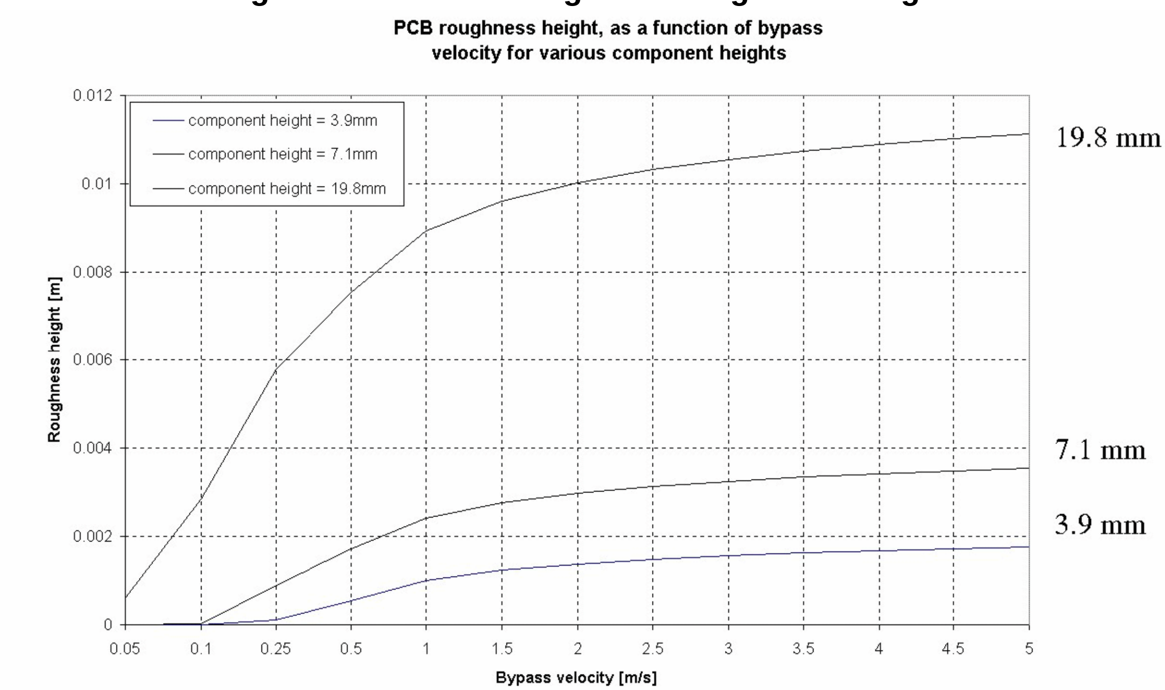
The roughness height is used to represent the effective surface roughness which should correspond to the average height of the components, and so on, above the surface of the PCB.

Calculating PCB Roughness Height

One of the parameters for Non-Conducting PCBs is roughness height. This information is required since the PCB behaves as a rough surface in turbulent flow.

The following graph provides a 'best fit' for the PCB roughness height obtained from a set of experimental test cases.

Figure 20-1. Calculating PCB Roughness Height



The mean component height shown in the PCB Property Sheet Summary Data Tab is an area averaged value, but the parameter shown in the graph above is not area averaged. The value required for the graph can be found by dividing the mean height by the fractional coverage of components on the board using the information presented in the PCB Property Sheet Summary Data Tab.

For example, for the top of the board, the component height is calculated as:

$$Top\ component\ height = \frac{100 \times Top\ mean\ height}{Top\ \%\ coverage}$$

The bypass velocity is the mean velocity flowing through the card slot. The estimate used can be checked at the end of the solution by placing a collapsed region at the entrance to the card slot. The Tables application window can then be used to determine the volume flow-rate through the mean flow region. Dividing this number by the cross sectional area of the card slot results in the mean velocity.

Conducting PCBs

Model PCB SmartParts as Conducting when conduction along a board is significant. You will need to specify the board composition.

Conducting PCBs require setting the material composition.

A conducting PCB is represented as an orthotropic conductivity block (that is, a block with conductivity modeled in three directions) with properties associated with the dielectric and conductor materials used.

Conducting, therefore, requires you to specify the board composition by one of the following:

- A value for the percentage conductor in the PCB.
- The total mass of the board.
- By defining the individual layers of the board. The conductivity of the board is then calculated from the thickness and thermal conductivity of each layer set up.

Simcenter Flotherm uses one of the following settings for calculating the conductivity of the PCB:

- Percentage Conductor By Volume
- Board Mass
- Layer Definition

Percentage Conductor By Volume

The Percentage Conductor is used to calculate the in-plane and normal plane conductivity as follows:

$$K_{in} = \left(\frac{A}{100} \times K_{cnd}\right) + \left(1 - \frac{A}{100}\right) \times K_{die}$$

$$\frac{1}{K_{thro}} = \frac{\frac{A}{100}}{K_{cnd}} + \frac{\left(1 - \frac{A}{100}\right)}{K_{die}}$$

where A is the Percentage Conductor value, K_{in} is the in-plane conductivity, K_{thro} is the normal (through-plane) conductivity, K_{cnd} is the conductor conductivity and K_{die} is the dielectric conductivity.

Board Mass

The specified mass value and board volume are used to determine the total density of the board ρ_T and the individual volume of the conductor is determined from:

$$\rho_T = \frac{(\rho_{cnd} \times vol_{cnd}) + (\rho_{die} \times vol_{die})}{vol_T}$$

and

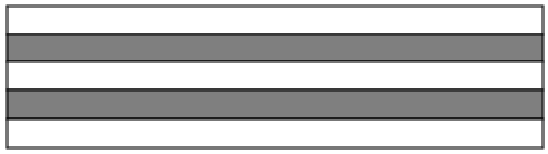
$$vol_T = vol_{die} + vol_{cnd}$$

These equations are used to determine the appropriate volume for the conductor within the board, and therefore, the value for the % conductor. The values for the in-plane and through conductivities can then be determined using the equations (1) and (2) above.

Layer Definition

A PCB is typically a layered composite of copper foil and a glass-reinforced polymer (FR-4) as shown in Figure 20-2.

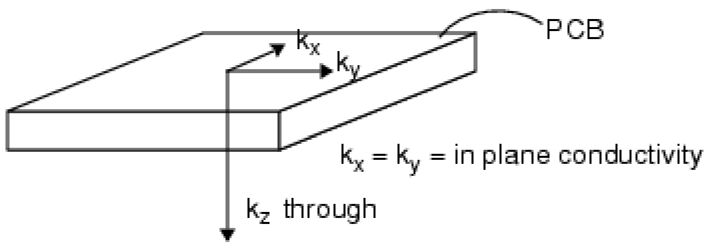
Figure 20-2. Example of PCB Layers



The conductivity for a board defined by individual layers is calculated from the given values for the thickness and thermal conductivity of each layer.

Heat Transfer Calculations

The PCB is represented as a block with conductivity modeled in-plane (K_{in}) and through the thickness of the plane (K_{thro}).



The equations for calculating each of these conductivities, given the values for the thickness and thermal conductivity of each layer is:

$$K_{in} = \left(\sum_{i=1}^{n} K_i t_i\right) \Big/ \left(\sum_{i=1}^{n} t_i\right)$$

$$K_{thro} = \left(\sum_{i=1}^{n} t_i\right) \Big/ \left(\sum_{i=1}^{n} \frac{t_i}{K_i}\right)$$

where:

K_i is the conductivity of a layer calculated from the %-age cover value

t_i is its thickness

See "Percentage Conductor By Volume" on page 230 for how Simcenter Flotherm uses the %-age cover conductor to calculate conduction.

Consistency Checking of Layers

The consistency of the layers is checked when the main PCB consistency check is run as the PCB is updated.

Creating PCBs

PCB SmartParts are created as children of assemblies. By default, a non-conducting PCB is created, one-tenth the size of the solution domain in the X- and Y-directions.

Note

Use EDA Bridge to construct detailed models of mother boards with the ability to import library components.

Procedure

1. Select an assembly in the data tree.
2. In the New Object Palette, choose Project Manager Create or Drawing Board Create and double-click the PCB icon.

Results

Components can be added as SmartParts, either as PCB Components or Compact Components.

Related Topics

Modeling PCBs

Defining the Board Layout [Simcenter Flotherm EDA Bridge User Guide]

PCB Property Sheet

PCB Property Sheet

To access: Select a PCB SmartPart in the model.

Use this property sheet to configure a PCB.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Construction tab.	See "PCB Property Sheet Construction Tab" on page 234.
Summary Data tab.	See "PCB Property Sheet Summary Data Tab" on page 236.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.

PCB Property Sheet Construction Tab

To access: Select a PCB SmartPart, then select the **Construction** tab.

Use this property sheet tab to define the PCB construction. The main categorization is whether the board is conducting or non-conducting.

Objects

Field	Description
Length Xo and Width Yo	The dimensions of the board.
Thickness Zo	(Conducting) When the board is non-conducting, thickness is not used and is, therefore, deactivated.
Modeling Level	• Non-Conducting — normally used for forced convection system level modeling where the PCB is a small part of the overall model. See "Non-Conducting PCBs" on page 228. • Conducting — for natural convection cases and forced convection systems when conduction along a board is significant (default). See "Conducting PCBs" on page 229.
Material Composition	(Conducting) Defines how conduction is calculated. • Percentage Conductor By Volume (default) • Board Mass • Layer Definition See "Conducting PCBs" on page 229.
% Conductor By Volume	(Conducting and Percentage Conductor By Volume) The default value is 10 %. See "Percentage Conductor By Volume" on page 230.
Board Mass	(Conducting and Board Mass) See "Board Mass" on page 230.
Number of Conducting Layers	(Conducting and Layer Definition) Use to define the layers of conductor in a PCB. A maximum of 32 layers is allowed. See "Layer Definition" on page 230.
Define Conducting Layer	(Conducting and Layer Definition) The layer number, numbered from 1 to the number specified in the field above.
Layer Thickness	(Conducting and Layer Definition) The thickness of the selected layer (in Define Conduction Layer field).
% Layer Coverage	(Conducting and Layer Definition) The percentage coverage of conductor material on the selected layer (in Define Conduction Layer field).
Dielectric Material	(Conducting) Not shown if % Conductor By Volume = 100. The dielectric material of the PCB board.
Conductor Material	(Conducting) The conductor material of the PCB board.

Field	Description
% Heat Dissipated Directly To Air	(Non-Conducting) The heat dissipation for individual components is set by specifying their size and power in the PCB Component Property Sheet. The heat dissipation is input as a percentage. The remainder of the heat is then dissipated to the air on the opposite side of the PCB to the defined component.
Top Roughness Height	(Non-Conducting) The roughness height enables the effects of the surface friction caused by components, and so on. Enter a value to represent the effective surface roughness which should correspond to the average height of the components, and so on, above the surface of the PCB. By default, the surface of the PCB is assumed to be hydrodynamically smooth with roughness height set to zero. See “Non-Conducting PCBs” on page 228.
Bottom Roughness Height	(Non-Conducting) See above.
Keypoint Component Height	(Non-Conducting) Check to maintain the keypoint placed at the mean height of components. Uncheck to remove the keypoint.

Related Topics

Modeling PCBs

Creating PCBs

PCB Property Sheet Summary Data Tab

To access: Select a PCB SmartPart, then select the **Summary Data** tab.

Use this property sheet tab to view the calculated effects of components added to the PCB.

Objects

Field	Description
Total Power	The total power output from all the components.
Top Mean Height	The mean height of the components averaged over the entire top surface of the PCB.
Top % Cover	The percentage of the top surface of the PCB covered by components.
Bottom Mean Height	The mean height of the components averaged over the entire bottom surface of the PCB.
Bottom % Cover	The percentage of the bottom surface of the PCB covered by components
In Plane Conductivity	The calculated in-plane conductivity, based on the definitions in the PCB Property Sheet Construction Tab. Also see "Heat Transfer Calculations" on page 231.
Normal Conductivity	The calculated normal conductivity.

Related Topics

Modeling PCBs

Creating PCBs

Chapter 21
PCB Components

You can use the Component SmartPart (also known as the PCB Component SmartPart) to model a PCB component as a planar or 3D heat source, or the heat can be smeared over the board.

Modeling PCB Components

PCB components can be represented by adding PCB Component SmartParts to PCB SmartParts.

PCB Component SmartParts are owned by the parent PCB and model the overall thermal effects of a component modeled discretely or smeared over the board.

If you know the type of component and its thermal resistance values, you can model the component as a resistor network package using the Compact Component SmartPart, see "Compact Components" on page 45.

To model components, you define the power output, location, and size of the component, whether it is to be applied over the board or represented discretely.

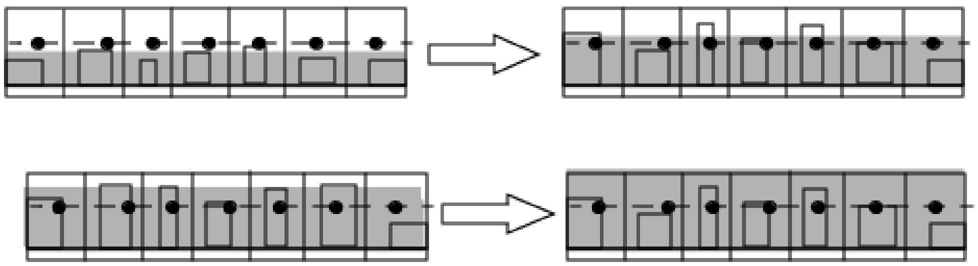
The component can be patterned to speed construction, the resultant arrays of components can be lumped to remove the need for additional grid to represent each component individually

Optionally, you can input resistance values to define a 2-Resistance compact model of the component. If you want to model array and peripheral lead packages, use Compact Components.

Component Blockage Effect

For greater accuracy when modeling the blockage effect, if the mean height of the components are found to be greater than 50% of the width of the grid cell, then the component is automatically expanded to fill the grid cell (Figure 21-1).

Figure 21-1. Component Blockage Effect



Component Heat Dissipation

There are a number of options for modeling the heat dissipation of components. These are applied differently, depending on the board specification.

Where the board is non-conducting, the heat is split between the component side and the other side by the percentage heat dissipated directly to air.

Where the board is conducting, the program will calculate the conduction through the board.

The heat from individual components can be applied either over the whole board or restricted to the discrete component's location. Restricting to the component's location is important if local board temperatures and the temperatures of the air around the board are required.

Where the blockage effects of individual components is considered essential, then the component can be represented as a solid block.

Related Topics

Creating PCB Components

PCB Component Property Sheet

Creating PCB Components

PCB Components are created as rectangular-shaped objects attached on one of the sides of the parent PCB board.

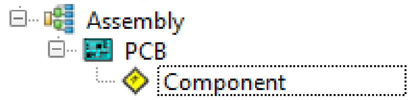
Procedure

1. Select a PCB in the data tree.
2. In the New Object Palette, choose Project Manager Create or Drawing Board Create and double-click the **Component** icon.

Results

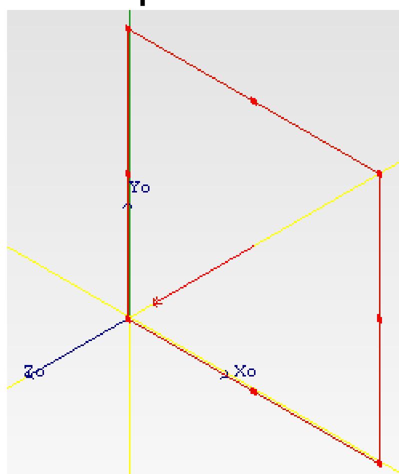
By default, a discrete component is created, one-tenth the size (in the X- and Y-directions) of the parent PCB.

Figure 21-2. PCB Component SmartPart in Data Tree



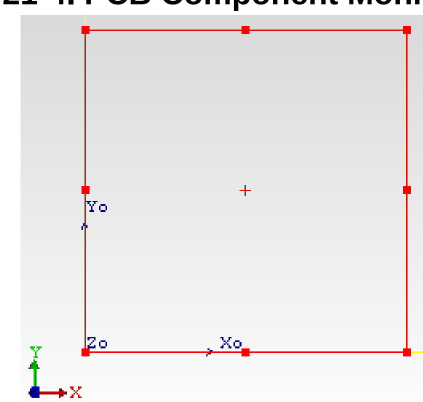
The source direction arrow is displayed in the drawing board, see Figure 21-3.

Figure 21-3. PCB Component Source Direction Arrow



Each PCB Component SmartPart has an embedded fixed monitor point in its center, this is displayed in the drawing board as shown in Figure 21-4.

Figure 21-4. PCB Component Monitor Point



Related Topics

Modeling PCB Components

PCB Component Property Sheet

PCBs

PCB Component Property Sheet

To access: Select a PCB Component SmartPart in the model.

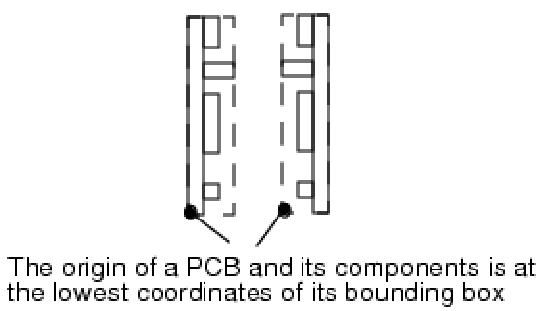
Use this property sheet to configure a PCB component.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28. The x- and y-coordinates locate the origin of the component with respect to the low coordinate corner of the bounding box of the PCB and its components, see Figure 21-5. It is not possible to lift the component above the board. It is, however, possible to locate a component off the board.
Construction tab.	See "PCB Component Property Sheet Construction Tab" on page 241
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.

Usage Notes

Figure 21-5. Origin of a PCB



PCB Component Property Sheet Construction Tab

To access: Select a PCB Component SmartPart, then select the **Construction** tab.

Use this tab to edit the component selected in the data display and, if required, create a pattern of new components based on this component.

Objects

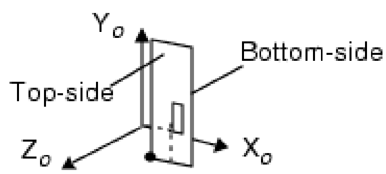
Field	Description
Power	Defines the power output for the component and each component created as a result of Patterning.
Length (Xo), Width (Yo) and Height (Zo)	Defines the size of the component.
Side of Board	Sets the attachment side for the component: Top or Bottom. Top faces the higher co-ordinate direction, as shown in Figure 21-6.
Modeling Option	Defines how heat dissipation is modeled: • Apply over Board — smears the heat that the component produces over the entire board as a planar source. • Discrete Footprint — heat dissipation is restricted to the component location as a planar source. • Solid Component — the component is modeled as a solid 3D cuboid.
Component Material	(Solid Component) Attach a material representation of the thermal parameters for the combined component material properties.The default is Typical Plastic Package. When using resistances to define the model, the material attached to the component should have a high thermal conductivity so that it does not significantly add to the resistance of the component. If thermal resistances are not known for the component, these can be left at zero and a pseudo material attached to the component to approximate its internal thermal resistance.
Junction-Board	(Solid Component) The thermal resistance between the junction and the board. Use this and the following two fields to input values of resistances used to define a compact model of the component. The 2-Resistance model is a popular compact model and consists of Junction-Board and Junction-Case Top resistances. Values for these resistances can be determined experimentally using JEDEC standard techniques. FloTHERM PACK (https://www.mentor.com/products/mechanical/flotherm/flotherm-pack/) is capable of generating Junction-Board and Junction-Case Top resistances for a variety of package styles.

Field	Description
Junction-Case Top	(Solid Component) The thermal resistance between the junction and the case top.
Junction-Sides	(Solid Component) The thermal resistance between the junction and the sides of the case.
Pattern Number in Xo and Yo	Sets up a 2D array of components over the board by defining the number of components in the two directions, Xo and Yo. The pattern starts from (and includes) the currently selected component and then adds components in the high coordinate directions as shown in Figure 21-7.
Pattern Pitch in Xo and Yo	The intervals between the patterned components as shown in Figure 21-8.
Lump Components	Allow arrays of minor components to be represented without incurring the additional grid needed to represent each component individually, see Figure 21-9.

Usage Notes

Setting Position and Size

Figure 21-6. Top and Bottom With Respect to Coordinate System



Patterned Components

Figure 21-7. Pattern Origin

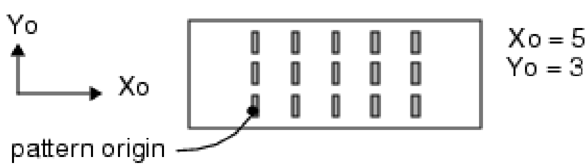
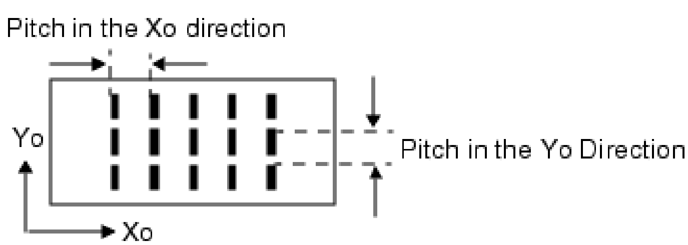
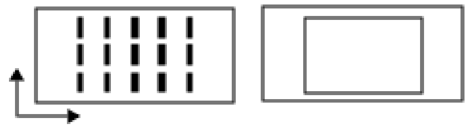


Figure 21-8. Pattern Pitch



Lumped Components

Figure 21-9. Lumped PCB Components



Related Topics

Modeling PCB Components

Creating PCB Components

Chapter 22
Perforated Plates

Use the Perforated Plate SmartPart to model perforated plates.

Resistance Models

Perforated plates are modeled in Simcenter Flotherm using a Perforated Plate SmartPart. Perforated Plate SmartParts are created as children of assemblies.

There are three resistance models for perforated plates: Automatic, Standard, and Advanced.

Automatic Resistance Model

The Automatic resistance model selects the correct coefficients in the resistance formula based on the free area ratio provided by the hole calculation or manual setting in the property sheet.

This model is valid for all flow regimes from laminar to fully turbulent, and is the default model.

Standard Resistance Model

This is the same as the Standard Planar Resistance model implemented for the Resistance Attribute.

Use this model when you know the loss coefficient for the perforated plate and the flow is fully turbulent.

Advanced Resistance Model

This is the same as the Advanced Planar Resistance model implemented for the Resistance Attribute.

Use this model when you know the loss coefficient for the perforated plate and the flow is transitional or laminar.

Related Topics

Perforated Plate Properties

Effects of Radiation

Planar Resistances [Simcenter Flotherm Project Attributes Reference Guide]

Perforated Plate Property Sheet

Perforated Plate Properties

The properties of Perforated Plate SmartParts are described in the following subsections and in the property sheet description.

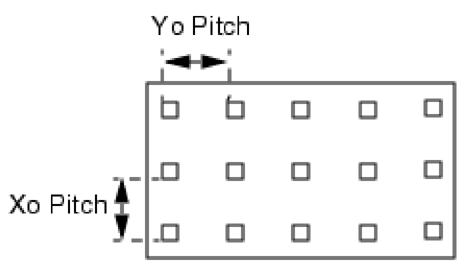
Coverage

Hole Coverage can either be:

- Defined by a single Free Area Ratio (the ratio of the holes to the total area across the plate) or
- Calculated from a defined array of holes where the hole pitch, shape, and size are specified.

The pitch sets the spacing between the holes, and is the distance between the start edges of each hole in the Xo and Yo directions.

Figure 22-1. Hole Pitches

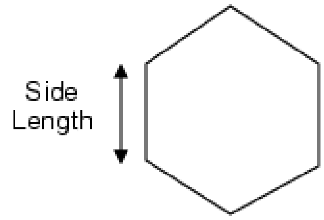


Hole Types

When using the Calculate Coverage option you can choose from three possible hole shapes.

- Square, defined by the side length.
- Round, defined by the diameter.
- Hexagonal, defined by the side length as shown in Figure 22-2.

Figure 22-2. Side Length of Hexagonal Holes

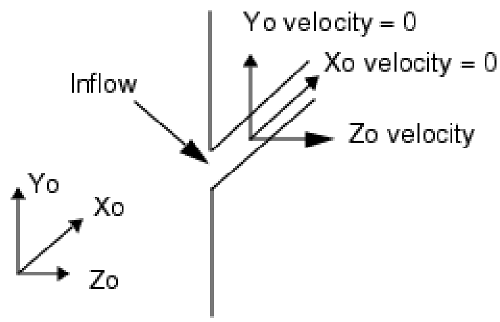


Straightening Flow

The flow straightening option is intended to capture the jetting effect caused by flow accelerating through holes in a perforated plate, or similar.

When active, the air velocity components parallel to the plate (that is in the Xo and Yo directions) are set to 0 (zero). You can specify straightening on either the high or low side of the plate, Figure 22-3 shows the flow straightened for the high side of the plate.

Figure 22-3. Straightened Flow



Related Topics

Resistance Models

Effects of Radiation

Perforated Plate Property Sheet

Effects of Radiation

Infrared radiation and solar radiation are treated separately.

Perforated plates are always totally transparent to infrared radiation. No radiation attribute is available.

Perforated plates block solar radiation in proportion to their Free Area Ratio, for example, a perforated plate with a Free Area Ratio of 0.7 blocks 30% of incident solar radiation.

Related Topics

Resistance Models

Perforated Plate Properties

Perforated Plate Property Sheet

Perforated Plate Property Sheet

To access: Select a Perforated Plate SmartPart in the model.

Use this property sheet to configure a perforated plate.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Construction tab.	See "Perforated Plate Property Sheet Construction Tab" on page 250.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.
Groups tab.	See "Generic Property Sheet Groups Tab" on page 31.

Perforated Plate Property Sheet Construction Tab

To access: Select a Perforated Plate SmartPart, then select the **Construction** tab.

Use this property sheet tab to set up the modeling data for the perforated plate. There are three possible models: Automatic, Standard, and Advanced. The fields available for data entry depend on which model is chosen.

Objects

Field	Description
Size Xo and Yo	The size of the plate. Perforated plates are modeled as 2D objects.
Plate Specification when Resistance Model = Automatic	
Coverage	Calculate — calculate the Free Area Ratio from hole data. Define — specify the Free Area Ratio.
Defined Free Area Ratio	(Define) Enter a value for the Free Area Ratio. No other data is required.
Calculated Free Area Ratio	(Calculate) The Free Area Ratio value calculated from the hole shape, size, and pitch. Read Only.
Hole Type	(Calculate) Choose a hole shape: Square, Round, or Hexagonal
Side Length	(Calculate and Square or Hexagonal) The length of one of the sides of the square or hexagonal hole.
Diameter	(Calculate and Round) The diameter of the round hole.
Xo Pitch	(Calculate) The distance between holes in the Xo direction.
Yo Pitch	(Calculate) The distance between holes in the Yo direction.
Plate Specification when Resistance Model = Standard or Advanced	
Loss Coefficients Based On	Selects the velocity criteria used to define the loss coefficient when calculating the pressure drop across the resistance: • Approach Velocity — use the velocity of the fluid as it approaches the resistance. • Device Velocity — use the velocity of the fluid through the device. • Accelerated — use the velocity of the fluid through the device plus the recoverable pressure drop. Use this option when a high velocity through the device is likely to cause significant entrainment on exit. If using this option, you should have a refined grid near the exit. For further information, see Accelerated Model in the *Simcenter Flotherm Project Attributes Reference Guide*.
Defined Free Area Ratio	(Device Velocity or Accelerated) Set the Free Area Ratio to represent the proportion of the modeled area (perpendicular to that direction) that is open.

Field	Description
Loss Coefficient	(Standard) The loss coefficient as described for the Planar Resistance Standard treatment of a Resistance attribute.
Length Scale, A Coefficient, B Coefficient and Index	(Advanced) The parameters as described for the Planar Resistance Advanced treatment of a Resistance attribute.
Straighten Flow	Remove velocity components parallel to plate.
High Side/Low Side	(Straighten Flow) The side of the plate where flow is straightened.

Related Topics

Perforated Plate Properties

Planar Resistances [Simcenter Flotherm Project Attributes Reference Guide]

Resistance Models

Chapter 23
Power Maps

The Power Map SmartPart uses HyperLynx data to model Joule heating.

Working With Power Maps

Power Maps are created from CSV files exported from Mentor Graphics HyperLynx® PI (Power Integrity).

The HyperLynx CSV files contain PCB geometry information and power values. There are two applications of these files:

- Calculation of Joule Heating from the geometry and power data within the CSV file.
- Calculation of Joule Heating from the geometry data only within the CSV file, the power being defined separately by Potential Source attributes on PCB layers.

Power Maps are subject to a number of rules, described here, along with advice on how to avoid problems.

Power Values

- Power Map powers can be overwritten by solid geometry (cuboid, prism, and so on) lower in the Project Manager data tree. In such cases, no power is applied because of the Power Map.
- Cells containing powers from multiple Power Maps use the sum of the powers present in that CFD cell.

Materials

- Power Map materials can be overwritten where multiple Power Map bounding boxes overlap.

  The material attached to the overwriting Power Map is used throughout the overwriting volume. As the Power Map material is usually Copper (Pure), it is usually the case that Copper (Pure) replaces Copper (Pure).

Fully Overwritten Power Maps are Not Supported

- Simcenter Flotherm does not support Power Maps that are fully overwritten by other Power Maps, and issues a “fully overwritten” Error message. However, partially overwritten Power Maps are supported, therefore, rearranging the project hierarchy so that the largest Power Maps are higher in the data tree should help to avoid such Error messages.

Surface Temperature Plots

- Surface temperature plots for models with multiple Power Maps work as expected provided that none of the Power Map solid is overwritten by another Power Map bounding box.
- If any solid portion of a Power Map is overlapped by the bounding box of another Power Map, then the surface temperature plots are either incomplete or displayed in incorrect areas of the model *if such plots are created for individual Power Maps*.
- If any solid portion of a Power Map is overlapped by the bounding box of another Power Map, then the correct surface temperature plots are created *if all Power Maps in the model are included in the plot*.

Related Topics

Modeling Joule Heating

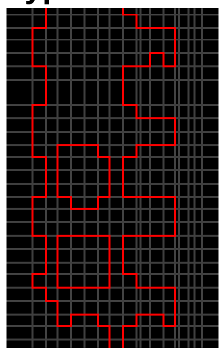
Power Map Properties

Use Power Map SmartParts to predict the temperatures of (high) current carrying copper nets in PCBs.

Power Map Grid Constraints

Two grid constraints are attached to the power map object to ensure that thermally critical parts, for example, necked traces, are keypointed, see Figure 23-1:

Figure 23-1. Keypointed Power Map Layer

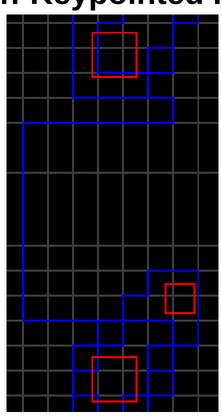


- A grid constraint, named <*power map object name*>_xy, is attached to the Xo- and Yo-Directions to keypoint the layer geometry. These settings are determined by the power map x and y cell dimensions, which are specified in the CSV file.
- A grid constraint, named <*power map object name*>_z, is attached to the Zo-Direction to keypoint all of the active layers.

Electrical Vias

Power map electrical vias are not keypointed with the power map grid constraints, see Figure 23-2.

Figure 23-2. Non-Keypointed Power Map Vias



Calculations on cells containing vias or sections of vias are as follows:

- Where there are grid cells overlapping conducting and background materials, the thermal conductivity, density, and specific heat of the cells are calculated using the relative volumes of conducting and background materials.

- Where more than one via is enclosed by a cell, the vias are combined as a single via with diameter and plating thickness selected to conserve dielectric and conductor areas within the cell.

Power Map Material

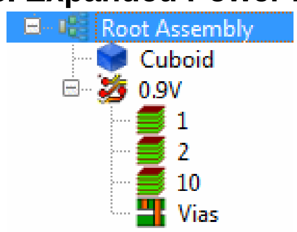
A default material of Copper (Pure) is attached to the power map object. The power map is, in effect, modeled as a combination of copper cuboids, each cuboid having an associated power source. To include power density data in your solution to view these power sources, check the Power Density check box in the Stored Variables section of the **Model Setup** tab.

Power Map Realization

The data extracted from a power map CSV file is used to build an object which can be viewed in the data tree and GDA. The CSV file is saved to the PDProject directory, and renamed by adding a number to the file extension.

A Power Map object appears in the data tree as shown in the example of Figure 23-3.

Figure 23-3. Expanded Power Map Object



Each active layer (named “1”, “2” and “10” in the example) can be independently selected, as can the vias. When selected, they are highlighted in the GDA.

The default name of the power map object and the default names of the layers are taken from the CSV file.

Summary Information

Summary information (**Window** > **Show Summary**) for power map objects includes:

- The total power dissipation, which is a summation of the discrete values in the CSV file.
- The overall size of the layers.

Summary information for each layer and the vias includes:

- The overall size of the layer/vias.

Related Topics

Modeling Joule Heating From Power Map Geometry and Power Data

Joule Heating Analysis

Modeling Joule Heating

Power Maps exported from HyperLynx can be used to model Joule heating in PCBs.

Modeling Joule Heating From Power Map Geometry and Power Data

Geometry and power data from imported HyperLynx files is used to model the effects of Joule heating.

Restrictions and Limitations

- You cannot save a power map to a library.
- Power maps are excluded from exported PDML files, but are included in "PACK File (no results)" files.
- Power maps are realized in a different way than other objects in Simcenter Flotherm, especially in de-keypointed situations. In such cases, the power map geometry does not snap out to fill the CFD cell, instead a material merge is made with the background solid of the CFD cell.
- If Potential Sources are defined on Power Map solid material (see "Specifying Sources on Power Map Layers" on page 259), then any power defined within the Power Map SmartPart (power data from HyperLynx PI) is ignored.

Prerequisites

- A CSV file exported from Mentor Graphics HyperLynx® PI (Power Integrity).

Procedure

1. Create an assembly.
2. Add an enclosing body, for example, a Cuboid or Non-Conducting PCB, to the assembly, and attach an appropriate electrical insulating material, for example, FR4.
3. Select the assembly then attach a Power Map object. A file browser opens for you to import a valid power map file. The resulting Power Map object is copied into the assembly. The default material is Copper (Pure).
4. The Power Map object should be in the correct X/Y relative location in the drawing board, but you will have to position it correctly in the Z-direction. Adjust the size and position of the solid object(s) (for example Cuboid(s)) to enclose the power map.
5. Localize the Power Map object grid to reduce the number of cells, and the solution time.

6. To view calculated Power Density scalar variable values in the results, check the Power Density check box in the Stored Variables section of the **Model Setup** tab.

7. Run a Sanity Check then run the solver.

Results

The Project Manager in Analyze mode can display surface temperature plots showing the heating effect due to Joule heating calculated from the imported power map. If Power Density was stored, then Power Density scalar plots can be created.

The Power Maps results table includes the maximum, minimum, and mean power map temperatures, and the applied power map power.

The Geometry Model legacy table includes total Heat Specified and the Heat Applied values obtained from the CSV file.

Related Topics

Power Map Properties

Joule Heating Analysis

Working With Power Maps

Power Map Property Sheet

Specifying Sources on Power Map Layers

The following procedure uses geometry data from imported files, but power is determined from Potential Sources attached to PCB layers.

Restrictions and Limitations

- When Sources are applied to Power Map geometry, then all internal Power Map power data is ignored.
- Restrictions and Limitations described under Modeling Joule Heating With Potential Source Attributes in the *Simcenter Flotherm Project Attributes Reference Guide*.

Procedure

1. Add a Power Map as described in "Modeling Joule Heating From Power Map Geometry and Power Data" on page 258, but do not solve the model.

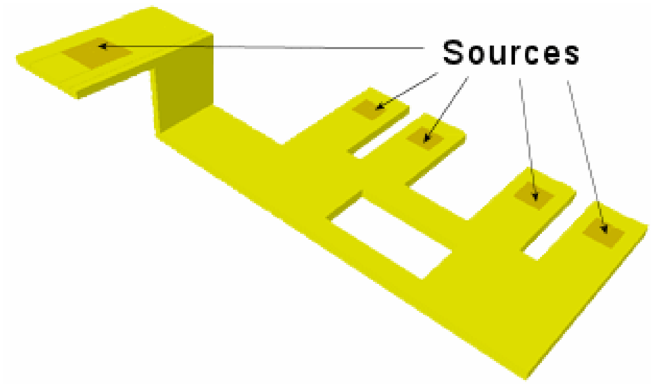
2. Switch on Joule Heating at the **Model Setup** tab.

3. Add Source SmartParts to the Power Map Layers and/or Vias. Typically these will used to set fixed voltages and current sources and sinks at the ends of tracks.

Tip

If you select a Power Map Layer or Via before adding a Source SmartPart, then the Source dimensions will match the bounding box dimensions of the selected object. The Source SmartPart can then be resized.

Figure 23-4. Sources on Power Map Layers



4. Define Potential Source attributes as necessary and attach these to the Source SmartParts. See Source Attributes in the *Simcenter Flotherm Project Attributes Reference Guide*.

5. Check that the correct Electrical Resistivity values are defined. The default Material attribute attached to Power Maps is Copper (Pure).

6. Run a sanity check to ensure the Sources are adequately defined and that there is electrical connectivity between them.

7. Solve the model.

Results

Refer to the Results section of Modeling Joule Heating With Potential Source Attributes in the *Simcenter Flotherm Project Attributes Reference Guide*.

Related Topics

Power Map Properties

Joule Heating Analysis

Working With Power Maps

Power Map Property Sheet

Power Map Property Sheet

To access: Select a Power Map in the model.

Use this property sheet to configure a power map.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28. Use this tab to change the location of the power map. After importing, the power map object will be in the correct X/Y relative location but will need to be positioned correctly in the Z-direction.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.

Related Topics

Modeling Joule Heating From Power Map Geometry and Power Data

Power Map Layer Property Sheet

Power Map Vias Property Sheet

Power Map Layer Property Sheet

To access: Select a Power Map Layer in the model.

Use this property sheet to define the name of the Power Map Layer or to hide the Layer.

Objects

Object	Description
Name	Enter the name of the layer.
Hide	Check this check box to hide the layer.

Related Topics

Modeling Joule Heating From Power Map Geometry and Power Data

Power Map Property Sheet

Power Map Vias Property Sheet

Power Map Vias Property Sheet

To access: Select a Power Map Vias in the model.

Use this property sheet to define the name of the Power Map Vias or to hide the Vias.

Objects

Object	Description
Name	Enter the name of the vias.
Hide	Check this check box to hide the vias.

Related Topics

Modeling Joule Heating From Power Map Geometry and Power Data

Power Map Layer Property Sheet

Power Map Property Sheet

Chapter 24
Racks

Use Rack SmartParts to model Data Center racks.

Modeling Racks

The Rack SmartPart can represent any row-based or room-based Data Center rack.

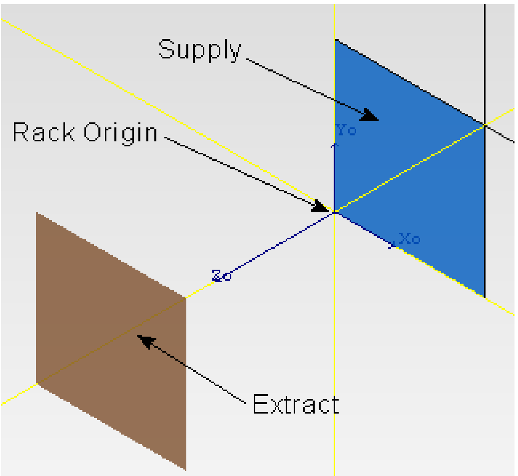
The main component when modeling a Rack SmartPart, in addition to the positions of the air supplies and extracts, is the control of the airflow on the faces of the rack.

Adjustment of the airflow rate can be based on the temperature at the inlet (extract).

Rack SmartParts are only allowed to add heat to a system, they cannot remove heat.

Each supply and extract of a rack is represented by a planar surface. For identification, a supply is shown on the drawing board in light blue, and an extract in brown, see Figure 24-1.

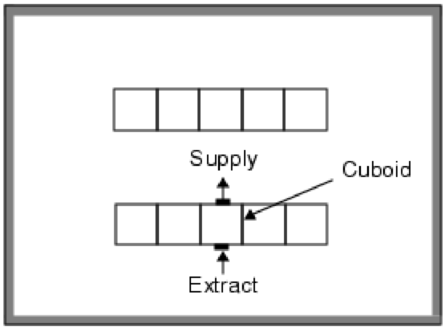
Figure 24-1. Rack SmartPart in the Drawing Board



To ensure that the rack is included in the simulation, the supplies and extracts must sit on the edges of a cuboid, see Figure 24-2.

Note
When using a rack on the edge of the solution domain or a cutout, ensure that this is not coincident with an open boundary. Either make the boundary a symmetry plane or add a collapsed cuboid on the edge, on which to place the rack extract or supply.

Figure 24-2. Arrangement of a Rack in a Data Center



It is important that the air cannot move directly from Extract to Supply in the model, therefore, you must introduce a primitive cuboid block to represent the portion of the calculation domain occupied by the rack.

Transfer of Stratification

For rack configurations set front-to-back (that is, where supply and extract are parallel (in the same plane)) and with a single extract and single supply, the solver supports data persistence for the vertical profile (in the direction aligned with gravity) of scalar quantities only: temperature and concentration if activated. The supply and extract may be slightly different in size and may be offset. This supports the transfer of any stratification in the flow as it passes from the inlet to the outlet of the rack. Note, however, that transfer of stratification is not supported in any other geometric configuration of the Rack SmartPart.

Related Topics

Creating New Racks

Extracts

Supplies

Rack Property Sheet

Creating New Racks

When Racks are created, they have Extract and Supply SmartParts added as children.

Procedure

1. Select an assembly in the data tree.

2. In the New Object Palette, choose Project Manager Create or Drawing Board Create and double-click the **Rack** icon.
3. By default, racks have no structure. A cuboid can be colocated with a rack's bounding box by first selecting the rack and then adding a cuboid.

Results

The **Groups** tab, used for assigning racks to Cold and Hot Aisle groups, does not appear unless you have set the modeling to Calculate Capture Indexes. Cooler groups are only used when a cooler airflow is controlled by a remote rack temperature.

A Rack is created with an Extract and a Supply (see Figure 24-3), which are separate objects with their own property sheets. More Supplies and Extracts can be added to the Rack as children, but there must always be at least one supply and extract for each rack.

Figure 24-3. Rack SmartPart in the Data Tree

The bounding box of a rack can be selected and moved, but not resized, because the overall size is determined by the sizes of the extract and supply entities that the bounding box contains.

The display of a rack in the drawing board is dependent on the data tree. For example, collapsing the rack in the data tree displays the bounding box in the drawing board, whilst expanding the rack in the data tree displays the supply and extract in the drawing board.

When the rack is expanded, the supply and extract become visible and can be separately edited.

As the supply and extract components of a rack are always attached to a solid, they are not transferred to MCAD Bridge and hence not included in outputs to external MCAD format files.

Related Topics

Modeling Racks

Extracts

Supplies

Creating New Recirculation Devices

Rack Property Sheet

Rack Property Sheet

To access: Select a Rack SmartPart in the model.

Use this property sheet to configure a rack.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Construction tab.	See "Rack Property Sheet Construction Tab" on page 269.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.
Groups tab.	See "Generic Property Sheet Groups Tab" on page 31.

Related Topics

Extract Property Sheet

Supply Property Sheet

Rack Property Sheet Construction Tab

To access: Select a Rack SmartPart, then select the **Construction** tab.

Use this property sheet tab to change the specification of a Rack SmartPart.

Objects

Field	Description
Power Dissipation	The power dissipation of the rack. The default value is 3000 W.
Flow Type	The flow rate of air passing through the rack can be defined by: • Volume Flow Rate / Power — Enter the volumetric flow rate as a function of the power. • Temperature Change — The airflow is determined from the equation: $$\dot{M} = \frac{Q}{C_p \times \Delta T}$$ where: • Q is the specified Power Dissipation. • ΔT is the specified Temperature Change. • C_p is the specific heat of air. • Volume Flow Rate — The default setting. Enter a total volume flow rate. This total flow rate is divided, proportionally by area, between all the supply entities, and is similarly divided between all the extract entities. • Curve — Define a characteristic curve by setting the volume flow rate and static flow at any number of points along the curve.
Volume Flow Rate	(Curve) The volume flow rate at 0 (zero) pressure. Defines one of the end points of the characteristic curve.
Pressure	(Curve) The pressure at 0 (zero) volume flow rate. Defines one of the end points of the characteristic curve.
Fan Curve Chart	(Curve) Click **Click to Edit** to open the Fan Curve Chart Dialog Box.
Airflow Adjustment	Check this setting if you want the airflow rate through the rack to be adjusted by a percentage factor if the extract temperature exceeds a specified reference value. The airflow adjustment is based solely on the average temperature across all defined extracts and operates on all four settings of Flow Type.
Critical Temperature	(Airflow Adjustment) Temperature at which the airflow adjustment Factor comes into effect.

Field	Description
Factor	(Airflow Adjustment) Percentage increase of airflow rate at the critical temperature. When Flow Type is set to Curve, the effect of this value is similar to the influence of a fan derating factor (however, fan derating reduces the airflow by specifying a fraction).

Related Topics

Modeling Racks

Creating New Racks

Chapter 25
Recirculation Devices

Use Recirculation Device SmartParts to model heat exchangers, CRAC units and blowers.

Modeling Recirculation Devices

The recirculation device is a convenient way to represent systems such as fan coil units that recirculate air by extracting it from the enclosure, then conditioning and returning it.

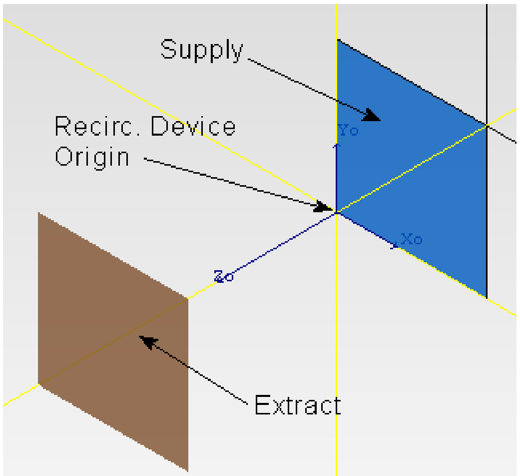
The recirculation device is modeled in three stages:

1. Extract a flow from the solution domain at a specified flow rate.
2. Subject this extracted flow to cooling, or heating.
3. Re-supply this cooled, or heated, flow at a different location.

The recirculation device is defined by specifying the type and thermal properties of the flow through it, and the location of the air supply(ies) and extract(s).

Each supply and extract of a recirculation device is represented by a planar surface. For identification, a supply is shown in the drawing board in blue, and an extract in brown. Figure 25-1 shows this coloring in solid rendering.

Figure 25-1. Recirculation Device SmartPart in the Drawing Board

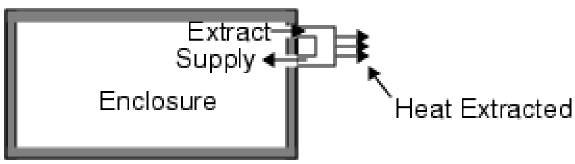


To ensure that the recirculation device is included in the simulation, the supplies and extracts of the recirculation device must sit on the edge of cuboids, cutouts, or the domain boundary.

External Recirculation Devices

External recirculation devices are completely outside the calculation domain. An example of an external cooling circuit is shown in Figure 25-2.

Figure 25-2. Example of an External Recirculation Device



To model an external recirculation device, the following are the only specifications required:

- The degree of heating or cooling.
- The supply and extract locations on the domain boundaries.
- The extract flow rate.

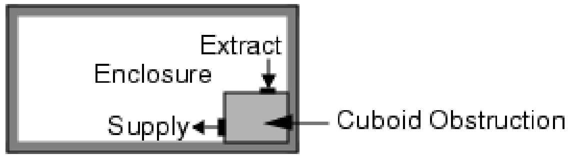
Note

When using a recirculation device on the edge of the solution domain or a cutout, ensure that this is not coincident with an open boundary. Either make the boundary a symmetry plane or add a collapsed cuboid on the edge, on which to place the recirculation device extract or supply.

Internal Recirculation Devices

Internal recirculation devices, have their extracts and supplies fully located within the calculation domain, for example, see Figure 25-3.

Figure 25-3. Example of an Internal Recirculation Device



It is important that the air cannot move directly from extract to supply in the model so an obstruction has to be used when the extract and supply terminals are not on the boundaries of the domain. For example, you could add a cuboid block (as shown in Figure 25-3) to represent the portion of the calculation domain occupied by the heater/cooler.

Related Topics

Example Models Using Recirculation Devices

Extracts

Supplies

Creating New Recirculation Devices

Recirculation Device Properties

Example Models Using Recirculation Devices

The following subsections give examples of using recirculation devices to model heat exchangers.

Heat Exchanger Simple Model

The simple heat exchanger model is that of a heat exchanger where the air passing through the recirculation device can be regarded as in contact with a secondary having a specified temperature.

The heat flux extracted from the secondary (added to the solution) is calculated as:

$$H(T_{secondary} - T_{extract})$$

where H is the user-set heat transfer coefficient (in W/degree difference for SI units) and $T_{secondary}$ is the user-set secondary temperature.

The program calculates the supply temperature of the air by a simple heat balance which yields:

$$T_{supply} = T_{extract} + \frac{H(T_{secondary} - T_{extract})}{(mass\ flow) \times (specific\ heat)}$$

If the heat transfer coefficient (H) of the heat exchanger is known to depend strongly upon temperature, you may proceed as follows:

- To start, set H according to your best estimate of what you expect the temperature within the coil to be.
- After obtaining a solution, check the temperature at the air extract (using the Tables application window), and use this to improve your estimate of the working temperature within the exchanger.
- Reset H according to the known dependence on temperature and repeat the calculation.
- Make further adjustments as necessary.

Following the standard ε-NTU method terminology, the rate at which heat is extracted (negative) from the primary fluid (air) is given by:

$$\varepsilon C_{min}(T_{liquid,inlet} - T_{air,inlet})$$

where:

ε is the heat exchanger effectiveness

C_{min} is the smaller of either (mass flow rate $\times$ $c_p)_{air}$ or (mass flow rate $\times$ $c_p)_{liquid}$

c_p is the specific heat of either fluid.

The following interpretations can be made:

$$H = \varepsilon C_{min}$$

$$T_{coolant} = T_{liquid,\ inlet}$$

$$T_{extract} = T_{air,\ inlet}$$

Therefore, given the ε-NTU performance curve for a heat exchanger, you can iteratively update H and $T_{coolant}$ (following the above replacement rules) until computed values no longer change significantly with further iterations.

Related Topics

Example Models Using Recirculation Devices

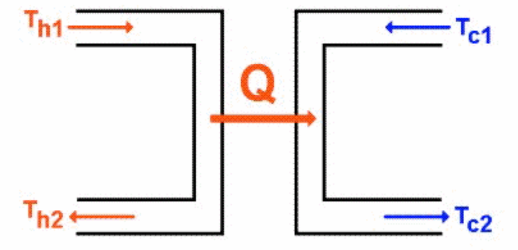
Heat Exchanger LMTD Model

The LMTD model is similar to the Simple Exchanger model except that the fluid on the secondary side is not at a fixed temperature.

In this case you also have to specify the fluid type on the secondary side, the volume flow rate of the secondary fluid, and whether the flow is parallel or counter flow.

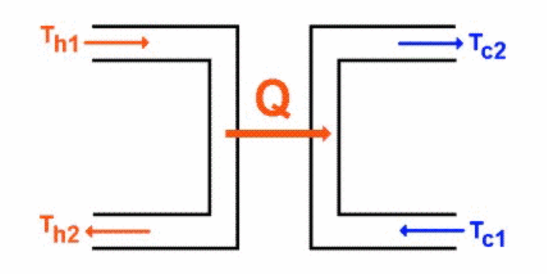
In *parallel flow*, the fluids on either side of the heat exchanger are flowing in the same direction, that is, they enter together and leave together as shown in Figure 25-4.

Figure 25-4. Parallel Flow



In *counter-current flow*, the fluids are flowing in the opposite directions, that is, they enter at the opposite ends and leave at the opposite ends as shown in Figure 25-5.

Figure 25-5. Counter-Current Flow



The heat transferred from the hot side to the cold side is given by the following expression:

$$Q = H\Delta T_{lm}$$

where:

H is the heat transfer coefficient (W/K).

ΔT_{lm} is the log mean temperature difference as described below.

The flow of heat on each side of the heat exchanger may be related to the temperature change of the fluids on the hot side (T_h) and the cold side (T_c), by the following expressions:

$$Q = (\dot{m}C_p)_c(T_{c2} - T_{c1})$$

$$Q = (\dot{m}C_p)_h(T_{h1} - T_{h2})$$

The log mean temperature difference is given by the following expressions:

For the parallel flow:

$$\Delta T_m = \frac{[(T_{h1} - T_{c1}) - (T_{h2} - T_{c2})]}{\ln[(T_{h1} - T_{c1})/(T_{h2} - T_{c2})]}$$

For the counter-current flow:

$$\Delta T_{lm} = \frac{[(T_{h1} - T_{c2}) - (T_{h2} - T_{c1})]}{\ln[(T_{h1} - T_{c2})/(T_{h2} - T_{c1})]}$$

Related Topics

Example Models Using Recirculation Devices

Computer Room Air Conditioning (CRAC) Devices

The CRAC Heat Exchanger models Computer Room Air Conditioning Devices (CRACs), for example, to model data centers.

The CRAC unit extracts air at a temperature, $T_{extract}$, and uses a limited power, Q, to try and cool or heat the extracted air to a user specified temperature, $T_{desired}$. The resupplied air is delivered at a temperature, T_{supply}. The following are user input variables:

- Maximum Power (Q_{max}), the CRAC units maximum available power to cool or heat the extracted air.
- Desired Temperature, the desired temperature of the resupplied air.

The change in temperature, ΔT, between the extracted air and the resupplied air is given by:

$$\Delta T = T_{supply} - T_{extract}$$

The desired change in temperature, $\Delta T_{desired}$ is given by:

$$\Delta T_{desired} = T_{desired} - T_{extract}$$

The change in temperature, ΔT, is limited by the CRAC units maximum available power, Q_{max} giving a maximum possible temperature change:

$$\Delta T_{max} = \frac{Q_{max}}{\dot{m} C_p}$$

where:

$\dot{m}$ is the specified mass flow rate of the extracted air.

C_p is the specific heat capacity of the extracted air.

The temperature of the resupplied air, T_{supply} is therefore given by:

$$T_{supply} = \begin{cases} T_{desired} & \text{if } (|\Delta T_{desired}| \le \Delta T_{max}) \\ T_{desired} + \Delta T_{max} & \text{if } (\Delta T_{desired} > \Delta T_{max}) \\ T_{desired} - \Delta T_{max} & \text{if } (\Delta T_{desired} < \Delta T_{max}) \end{cases}$$

Related Topics

Example Models Using Recirculation Devices

Modeling Blowers

To model blowers, in addition to specifying the recirculating flow, you add a surrounding casing and define the shear profile.

Procedure

1. As an example, create a cuboid, the full length and height but only 70% the width of the blower.
2. Place a recirculation device on the cuboid, locating the Extract and Supply on the surface of the cuboid.
3. Create the hub inlet using a Cylinder SmartPart and place over the extract to block it.
4. Create the outer diameter of the inlet using prisms.
5. Set up the fan curve (that is, flow rate versus pressure characteristic). To define the flow, select Non-Linear Fan Curve and click **Fan Curve**, then set the characteristics of the flow in the expected operating range. Refer to "Defining a Fan Characteristic Curve" on page 144.
6. For the Supply, choose a Flow Dependent shear model.

Results

Consider the simple representation of a micro-blower as shown in modeled in Figure 25-6.

Figure 25-6. Modeled Blower

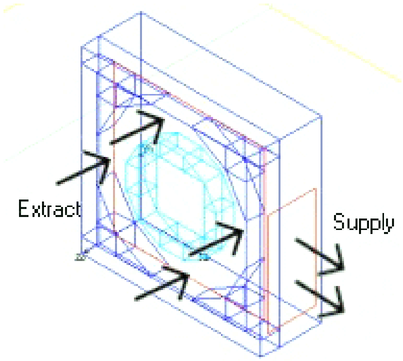
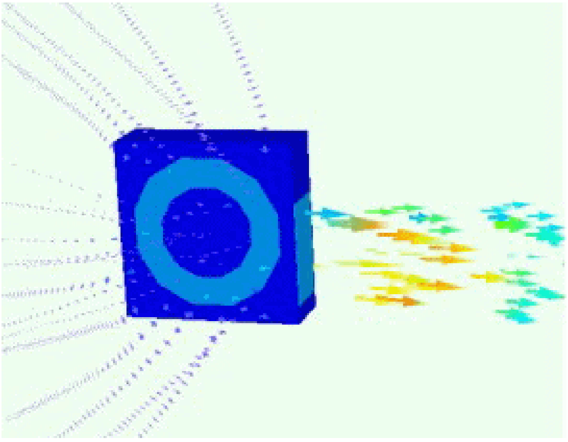


Figure 25-7, captured in Analyze mode, shows the modeled flow.

Figure 25-7. Flow Through Blower



Related Topics

Modeling Recirculation Devices

Creating New Recirculation Devices

When Recirculation Devices are created, they have Extract and Supply SmartParts added as children.

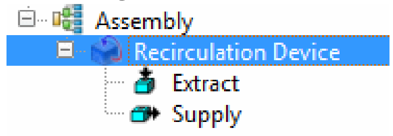
Procedure

1. Select an assembly in the data tree.
2. In the New Object Palette, choose Project Manager Create or Drawing Board Create and double-click the **Recirculation** icon.

Results

A Recirculation Device is created with an Extract and a Supply, which are separate objects with their own property sheets. More Supplies and Extracts can be added to the Recirculation Device as children, but there must always be at least one supply and extract for each recirculation device. You cannot delete (Cut) a supply or extract if only one exists.

Figure 25-8. Recirculating Device SmartPart in the Data Tree



For many electronic models, specifying a temperature rise or power dissipation is sufficient to define the heating or cooling; however, more sophisticated heat exchanger options are available to model interaction with a coil heat exchanger.

The bounding box of a recirculation device can be selected and moved, but not resized, because the overall size is determined by the sizes of the extract and supply entities that the bounding box contains.

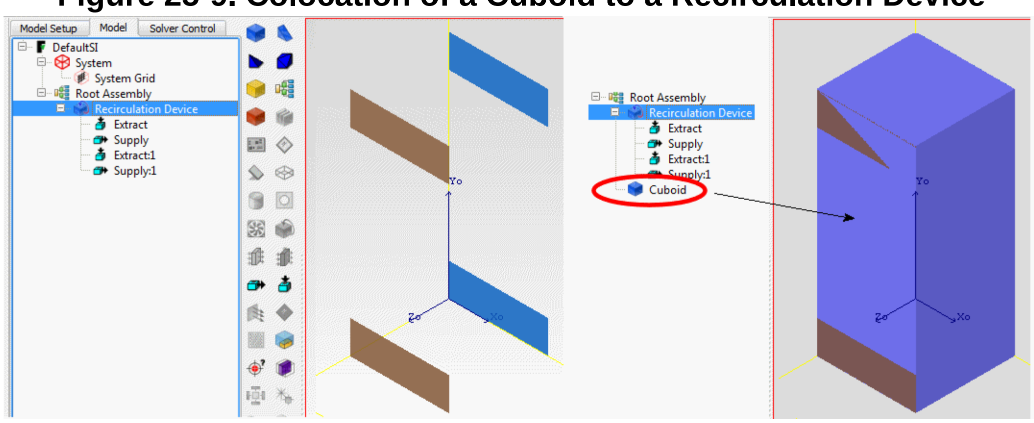
The display of a recirculation device in the drawing board is dependent on the data tree. For example, collapsing the recirculation device in the data tree displays the bounding box in the drawing board, whilst expanding the recirculation device in the data tree displays the supply and extract in the drawing board.

When the recirculation device is expanded, the supply and extract become visible and can be separately edited.

Colocation, the placement of several entities in a single location, occurs when a cuboid is created either after or before a recirculation device has been created, provided the first object has been selected when you create the second object.

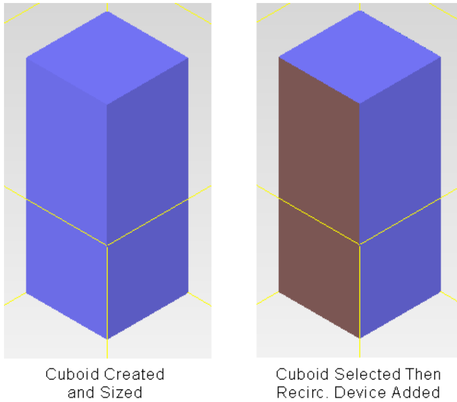
When a recirculation device is a model of a computer center cubicle it is usual to model the cubicle body as a Cuboid SmartPart. If you have created a recirculation device, say with multiple supplies and extracts then, if you select the recirculation device in the Project Manager tree and click the Cuboid SmartPart, a cuboid is created that matches the size of the recirculation device's bounding box. That is, the surfaces of the cuboid will, where possible, be colocated with the supplies and extracts, for example, see Figure 25-9.

Figure 25-9. Colocation of a Cuboid to a Recirculation Device



Similarly, if a cuboid is created first, resized to represent the cubicle, then selected before a recirculation device is added, the extract and supply of the recirculation device will be colocated on the surfaces of the cuboid and be the full size of the cuboid faces in the default orientation of the recirculation device, see Figure 25-10.

Figure 25-10. Colocation of a Recirculation Device to a Cuboid Showing Full-Face Extract



As the supply and extract components of a recirculation device are always attached to a solid or to the boundary of the solution domain, they are not transferred to MCAD Bridge and hence not included in outputs to external MCAD format files.

Related Topics

Modeling Recirculation Devices

Extracts

Supplies

Recirculation Device Properties

Recirculation Device Properties

The properties of Recirculation Device SmartParts are described in the following subsections and in the property sheet description.

Flow Rate

The flow rate of the air passing through the recirculation device, is defined using one of four methods.

The following are the methods:

- Volume Flow Rate
- Linear Fan Curve
- Non Linear Fan Curve
- Normal Velocity at Supply

Note

When either of the fan curve options are selected, the change in pressure (dP) used to compute the volume flow rate is calculated by averaging over all the extract and supply instances.

Thermal Properties of Recirculation Devices

The cooling (or heating) of the air which passes through a recirculation device can be defined by as a temperature change, as a heat extraction rate, as a heat exchanger, or as a CRAC unit.

- As a Temperature change:

  You specify the temperature change (ΔT) as an increase (positive) or a decrease (negative). The temperature at which the air is supplied back to the enclosure is:

  $$T_{supply} = T_{extract} + \Delta T$$

  where $T_{extract}$ is the mean temperature calculated by the program for the air adjacent to the extract.

- As a Heat extraction rate:

  You specify the amount of heat extracted (or supplied) from the air passing through the recirculation device. The temperature at which the air is supplied back to the enclosure is:

$$T_{supply} = T_{extract} + \frac{Rate\ of\ heat\ extraction}{(mass\ flow) \times (specific\ heat)}$$

  Extracted heat should be set as a negative quantity. The *specific heat* of the air is set in Fluid property attached to the system. The extract temperature is the temperature determined by the program adjacent to the extract.

- As a Heat Exchanger, for modeling simple or LMTD heat exchangers:

  See "Heat Exchanger Simple Model" on page 274 and "Heat Exchanger LMTD Model" on page 275.

- As a CRAC, for computer air conditioning devices:

  See "Computer Room Air Conditioning (CRAC) Devices" on page 277.

Where a conceptual knowledge of the device is available, the Temperature Change or Heat Input/Extraction Rate options provide simple models.

If more is known, for say a fan-coil unit, then use the Heat Exchanger option.

Thermostats for Transient Cases

When running a transient calculation, a thermostat can be activated to control the flow or heating/cooling from the recirculation device.

The location of the sensor(s) is specified using the monitor point functionality. Mass flow on either side of the recirculation device can be switched on and off by the thermostat effectively activating and deactivating flow and heat transfer boundary conditions depending on whether the device is heating or cooling the side the thermostat is attached to.

The solution domain where the thermostat is placed and into which the recirculation device is supplying air is termed the control side, while the other side of the heat exchanger where no CFD solution is carried out is termed the secondary side.

Thermostat Modeling Advice

Consider systems where:

T_{sensor} represents the temperature at which the thermostat location.

T_{on} is the temperature at which the heat exchanger switches on.

T_{off} is the temperature at which the heat exchanger switches off.

For the heat exchanger providing cooling:

- If $T_{sensor} > T_{on}$ the flows are on.
- If $T_{sensor} < T_{off}$ flows are off.
- For $T_{off} < T_{sensor} < T_{on}$ then this is dead band and the device remains unchanged from previous state.

  In this case, T_{on} is always greater than T_{off} where:

  $T_{on} = T_{control} + T_{deadband}/2$

  $T_{off} = T_{control} - T_{deadband}/2$

  $T_{deadband}$ is limited to a minimum of 1°C for stability.

For the heat exchanger providing heating:

- If $T_{sensor} < T_{on}$ the flows are on.
- If $T_{sensor} > T_{off}$ flows are off.
- For $T_{off} > T_{sensor} > T$on then this is dead band and the device remains unchanged from the previous state.

  In this case, T_{off} is always greater than T_{on} where:

  $T_{on} = T_{control} - T_{deadband}/2$

  $T_{off} = T_{control} + T_{deadband}/2$

  $T_{deadband}$ is limited to a minimum of 1°C for stability.

Fan Relaxation

With no fan relaxation is set (**Solver Control** tab) then a recirculation device with fan curves may cause the solution to diverge.

A fan relaxation of 0.7 is typically required to ensure good convergence of a recirculation device with a fan curve attached.

Related Topics

Modeling Recirculation Devices

Creating New Recirculation Devices

Recirculation Device Property Sheet

Recirculation Device Property Sheet

To access: Select a Recirculation Device SmartPart in the model.

Use this property sheet to configure a recirculation device.

Objects

Object	Description
Location tab.	See “Generic Property Sheet Location Tab” on page 28.
Construction tab.	See “Recirculation Device Property Sheet Construction Tab” on page 287.
Thermostat tab.	See “Recirculation Device Property Sheet Thermostat Tab” on page 289.
Attachments tab.	See “Generic Property Sheet Attachments Tab” on page 30.
Groups tab.	See “Generic Property Sheet Groups Tab” on page 31.

Recirculation Device Property Sheet Construction Tab

To access: Select a Recirculation Device SmartPart, then select the **Construction** tab.

Use this property sheet tab to change the specification of a heating or refrigeration circuit SmartPart. By optionally adding a fan curve and a shear output profile, you can model blowers.

Objects

Field	Description
Flow Type	The flow rate of air passing through the recirculation device can be defined by: • Volume Flow Rate • Linear Fan Curve • Non Linear Fan Curve • Normal Velocity at Supply
Volume Flow Rate	(Volume Flow Rate) The total volume flow rate. This total flow rate is divided, proportionally by area, between all the supply entities, and is similarly divided between all the extract entities. (Linear and Non-Liner Fan Curve) The volume flow rate at 0 (zero) pressure. Defines one of the end points of the linear or characteristic curve.
Pressure	(Linear and Non-Liner Fan Curve) The pressure at 0 (zero) volume flow rate. Defines one of the end points of the linear or characteristic curve.
Fan Curve Chart	(Non-Liner Fan Curve) Click **Click to Edit** to open the Fan Curve Chart window, see Fan Curve Chart Dialog Box.
Velocity	(Normal Velocity at Supply) The flow velocity normal to the supply outlet.
Thermal Property	Defines how cooling (or heating) is determined (see "Thermal Properties of Recirculation Devices" on page 282). • Temperature Change • Heat Input/Extraction Rate • Heat Exchanger • CRAC
Temperature Change	(Temperature Change) The temperature at which the air is supplied back to the enclosure is: $T_{supply} = T_{extract} + \Delta T$ where $T_{extract}$ is the mean temperature calculated by the program for the air adjacent to the extract.

Field	Description
Heat Input/ Extraction Rate	(Heat Input/Extraction Rate) The temperature at which the air is supplied back to the enclosure is: $$T_{supply} = T_{extract} + \frac{Rate\ of\ heat\ extraction}{(mass\ flow) \times (specific\ heat)}$$ Extracted heat should be set as a negative quantity. The *specific heat* of the air is set in Fluid property attached to the system. The extract temperature is the temperature determined by the program adjacent to the extract.
Exchanger Model	(Heat Exchanger) Two models of heat exchanger are available: Simple or LMTD.
Temperature	(Heat Exchanger and Simple) Refer to "Heat Exchanger Simple Model" on page 274.
Heat Transfer Coefficient	
Fluid	(Heat Exchanger and LMTD) Refer to "Heat Exchanger LMTD Model" on page 275.
Secondary Volume Flow Rate	
Temperature	
Heat Transfer Coefficient	
Configuration Type	
Maximum Power	(CRAC) The CRAC unit's maximum available power to cool or heat the extracted air.
Desired Temperature	(CRAC) The desired change in temperature between the extracted air and the resupplied air.

Related Topics

Recirculation Device Properties

Modeling Recirculation Devices

Modeling Blowers

Example Models Using Recirculation Devices

Recirculation Device Property Sheet Thermostat Tab

To access: Select a Recirculation Device SmartPart, then select the **Thermostat** tab.

For transient cases only. Use this property sheet to define the location and controls for a thermostat which enables the flows on either side of a recirculation device to be controlled by a temperature sensor.

Objects

Field	Description
Thermostat (Transient Solutions Only)	Check to incorporate a thermostat into a recirculation device and activate the data entry fields.
Control Temperature	The reference $T_{control}$ temperature. The heat exchanger providing cooling is one where $T_{control}$ is greater than that of T_{inlet} of the secondary side. Conversely, a heat exchanger providing heating is one where $T_{control}$ is less than that of T_{inlet} of the secondary side.
Dead Band Temperature	The dead band is the temperature range around the control temperature within which the thermostat takes no action. This exists in any on/off controller to prevent continuous switching on/off of heating or cooling and is distributed evenly on either side of the control temperature.
Position Xo, Yo, Zo	A monitor point will be located at this location. If the sensor is on the solved side, then the flow is switched off, and all heat transfer and flow ceases. However, if the sensor is on the non-solved side, only the heat transfer ceases. As the sensor is to be checked only once during a time step, then the temperature values used will always be the previous time step values. This avoids instabilities during the convergence of a time step.
Apply Thermostat to	Sets the location of the thermostat: • Control Side — in the solution domain where the heat exchanger is supplying the air • Secondary Side — where no CFD solution is carried out.

Related Topics

Thermostats for Transient Cases

Chapter 26
Regions

Region SmartParts define spaces in the model.

Region Usage

Use regions to capture values, refine the grid or create new projects from existing projects.

Use regions to:

- Capture summary values for 2D (collapsed) or 3D (uncollapsed) spaces. These summary values can be viewed in the regions tables when in Analyze mode, or set up as output variables in the Command Center.
- Refine the grid in areas not occupied by geometry. See "Grid Refinement Using Regions" on page 293.
- Create new projects based on the part of the model enclosed within the selected region.

  See Creating a Zoom-In Project in the *Simcenter Flotherm User Guide*.

Creating a Region

By default, regions define volumes, but they can be collapsed to create plane (2D) regions.

Restrictions and Limitations

- A plane region must have a row of cells on either side to display a full set of results. If a plane region is located on the edge of the solution domain, the results in the direction normal to the solution domain are reported as zero.

Procedure

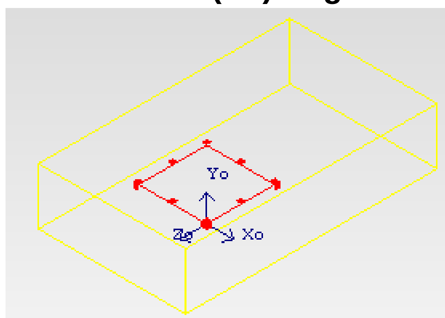
1. Select the assembly to hold the region, then click the **Volume Region** icon.

2. A volume region is created 1/10th the size of the solution domain.
3. To create a plane (2D) region, select the collapse direction and enter start coordinates relative to the model origin, and its size.

Results

Figure 26-1 shows a plane region which is collapsed on the Yo High Face.

Figure 26-1. Plane (2D) Region Example

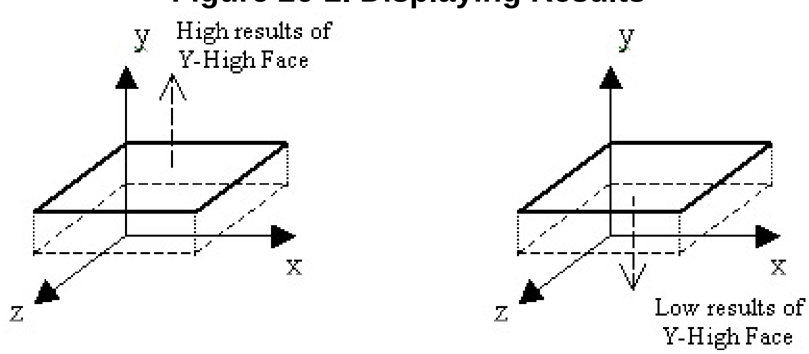


Displaying Results Over a Region

After solution, the mean flow results as well as the mean, minimum, and maximum values for any variable, can then be tabulated over this new region when in Analyze mode.

For each planar face of a region, the results will show the High results, which will represent the results in the first row of cells in the positive direction of the plane normal to the face, and the Low results, which will represent the results in the other direction.

Figure 26-2. Displaying Results



Related Topics

Region Mean Flows Results Table [Simcenter Flotherm User Guide]

Multi-Fluid Regions

A volume region can be used to define a space which contains an alternative fluid to the base fluid (as a Fluid attribute attachment). This can be used, for example, to create a water cooled heat sink.

The restriction on the use of alternative fluids, is that the volume containing each fluid must be fully enclosed by solid geometry, that is, you cannot have a fluid-to-fluid interface. Up to 100 multi-fluid regions can be used.

To create a fluid region, attach the required Fluid attribute to it using the **Attachments** tab of the Region property sheet.

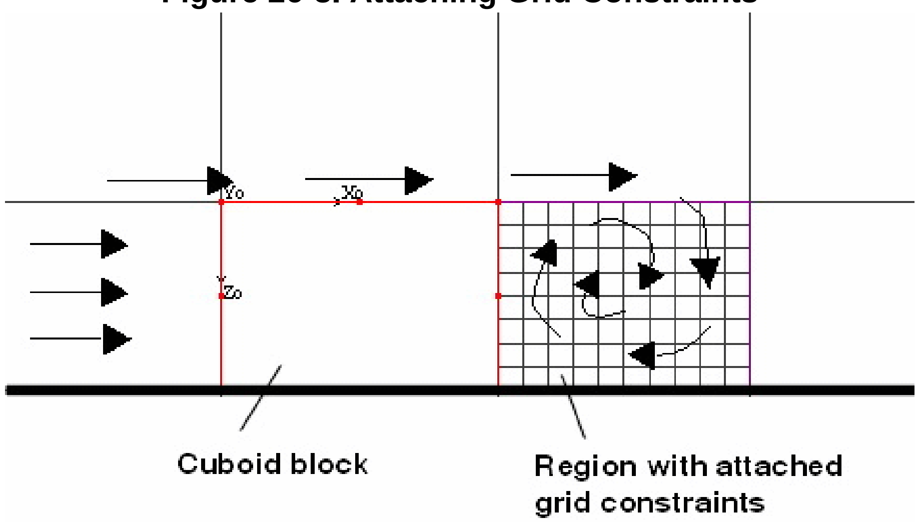
After attaching the new fluid, within the defined volume region any space which would normally be the base fluid will contain the alternative attached fluid.

Grid Refinement Using Regions

When there are locations within the flow where the grid is not locally dependent on the geometry, regions can be added with attached grid constraints to refine the grid at the location.

This is shown in Figure 26-3.

Figure 26-3. Attaching Grid Constraints



Region Property Sheet

To access: Select a Region SmartPart in the model.

Use this property sheet to configure a region.

Objects

Object	Description
Location tab.	See “Generic Property Sheet Location Tab” on page 28.
Attachments tab.	See “Generic Property Sheet Attachments Tab” on page 30.

Chapter 27
Resistances

Use Resistance SmartParts for setting the resistance to the airflow distributed across a finite volume or plane (collapsed resistance).

Resistance Applications

You can define volume or planar (2D) resistances.

An example of a volume resistance is the distributed resistance to flow provided by a region occupied by a large number of cables.

An example of a planar resistance is the resistance to airflow caused by the presence of a grille, mesh, or other partial obstruction to the flow that does not occupy a significant volume.

Related Topics

Creating a Resistance

Rotating a Collapsed Resistance

Resistance Property Sheet

Creating a Resistance

To create a new resistance, you first create the resistance geometry and then attach the resistance attribute.

Procedure

1. To create the resistance geometry, add a Resistance SmartPart.

   A cuboid shape one-tenth the size of the solution domain is created. Resistances are represented in the drawing board by a yellow outline.

2. Change the location and shape by re-selecting the object and dragging its boundaries in the drawing board or use property sheet. You can also collapse the resistance using a setting on the property sheet.
3. Resistance SmartParts are, by default, open spaces. Attach a Resistance attribute to define the resistance properties.

Related Topics

Resistance Applications

Rotating a Collapsed Resistance

Resistance Attributes [Simcenter Flotherm Project Attributes Reference Guide]

Resistance Property Sheet

Rotating a Collapsed Resistance

In addition to the 90 degree-step rotation that can be applied to geometry (for example, Geometry > Rotate Clockwise), collapsed resistances can be rotated more accurately using the Resistance property sheet.

Restrictions and Limitations

- Rotation is not possible about the same axis as the collapse direction.
- Rotation is not possible if using Absolute coordinates (User Preferences dialog box).

Procedure

1. In the Resistance property sheet **Location** tab, select a face to be collapsed to.

   The resistance becomes a 2D object and the property sheet **Rotation** tab is opened.
2. In the **Rotation** tab, select an About axis and enter the angle of rotation in the Rotate field.

Results

Figure 27-1 and Figure 27-2 show the results of applying a 60 degree rotation to an X-axis-collapsed resistance about the Y- and Z-axis respectively.

Figure 27-1. Collapsed Resistance: 60 Degree Rotation About the Y-Axis

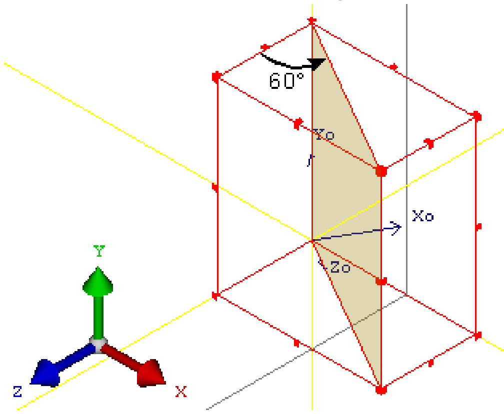
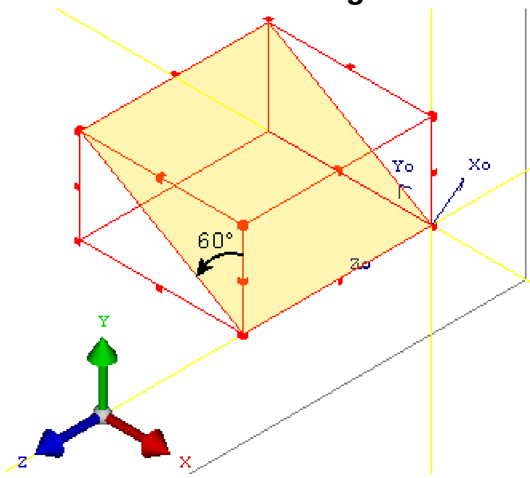


Figure 27-2. Collapsed Resistance: 60 Degree Rotation About the Z-Axis



Related Topics

Resistance Applications

Creating a Resistance

Resistance Property Sheet

Resistance Property Sheet

To access: Select a Resistance SmartPart in the model.

Use this property sheet to configure a resistance.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Rotation tab.	See "Resistance Property Sheet Rotation Tab" on page 299.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.

Resistance Property Sheet Rotation Tab

To access: Only available for collapsed resistances and when Local coordinates are selected.
Select a Resistance SmartPart in the model and, in the **Location** tab, select a collapse face.

Use this property sheet to define an angle of rotation of a collapsed resistance.

Objects

Field	Description
Rotate	The angle of rotation must be between -90 and 90 degrees. A positive angle of rotation is a clockwise rotation when viewed along the axis from the origin in the positive direction.
About	The axis of rotation: • None — no axis, that is, ignore the angle of rotation. • The two axes that are not the Collapse axis. For example, if the resistance has been collapsed to a Yo face, then rotation is possible about the Xo or Zo axis.

Related Topics

Resistance Applications

Creating a Resistance

Rotating a Collapsed Resistance

Chapter 28
Sloping Blocks

Use Sloping Block SmartParts to represent cuboids with surfaces not aligned with the coordinate axes.

Sloping Block Properties

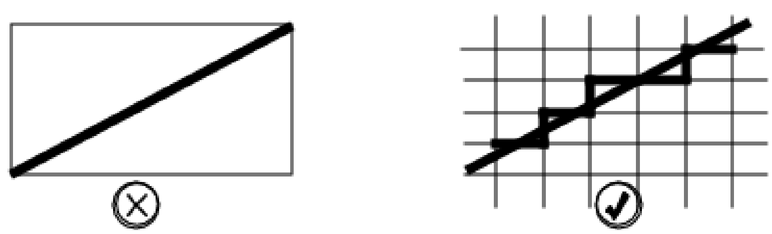
Sloping blocks are rotated cuboids with the ends cut perpendicular to the end of the block.

Sloping blocks can be Thick or Thin. Thin blocks (collapsed 2D representations) are modeled by default.

Thin Sloping Blocks

Thin Sloping Blocks are 2D representations which only allow conduction through the plane and are the default. When creating a Thin Sloping Block it is important to ensure that there is sufficient grid to capture the way it cuts through the grid, see Figure 28-1.

Figure 28-1. Recommended Grid for of a Sloping Block



Thick Sloping Blocks

Thick Sloping Blocks are full 3D representations constructed of several cuboids and prisms to achieve the desired geometry. Also, Thick Sloping Blocks allow conduction through and in the plane of the block.

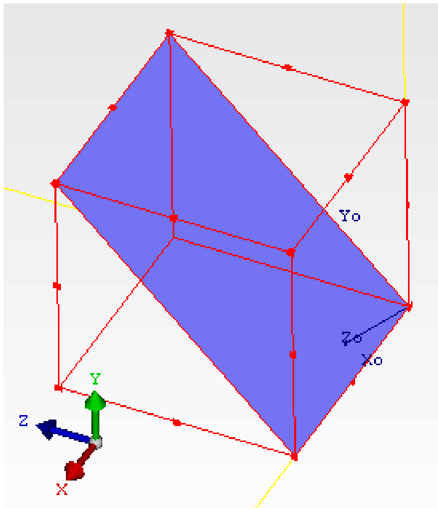
Care should be used when modeling relatively thin objects, since a Thick Sloping Block may produce small cells locally and, as a result, create cells with a high aspect ratio which will at best increase solution time, or it may make convergence difficult.

For thin objects use the Thin option, unless the thickness is critical to the airflow or the internal thermal distribution is of great importance.

Sloping Block Size and Slope Angle

The dimensions of the sides of the block are set relative to its own local coordinate system, see Figure 28-2.

Figure 28-2. 3D View of Sloping Block in Drawing Board

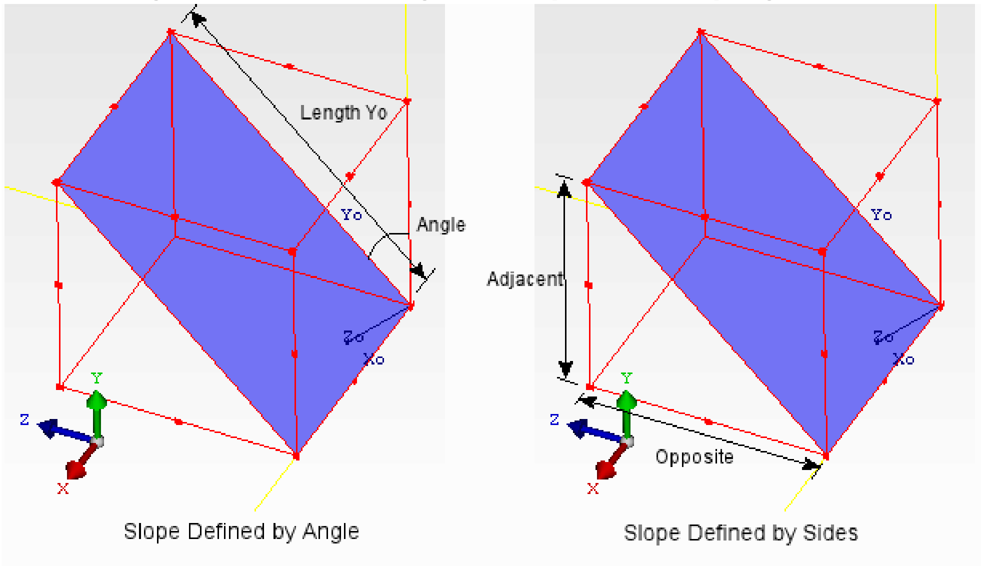


If a Sloping Block is drawn by dragging downwards, the local block origin is located at the higher Y system coordinates.

There are two methods of setting the slope: by Angle or by Sides, refer to Figure 28-3.

- Defining a Sloping Block by Angle requires the local Yo dimension, and the angle between the local y-axis (Yo) and the solution domain y-axis (Y).
- Defining a Sloping Block by Sides requires the local Yo dimension, and the angle between the local y-axis (Yo) and the solution domain y-axis (Y).

Figure 28-3. Defining the Slope of a Sloping Block



Caution

An angle of less than 10° from the horizontal or vertical will cause an excessive grid size.

Related Topics

Creating Sloping Blocks

Sloping Blocks and Solar Radiation

Sloping Block Property Sheet

Creating Sloping Blocks

The default Sloping Block is a thin angled plate with a 45 degree slope which has a smooth, non-conducting surface.

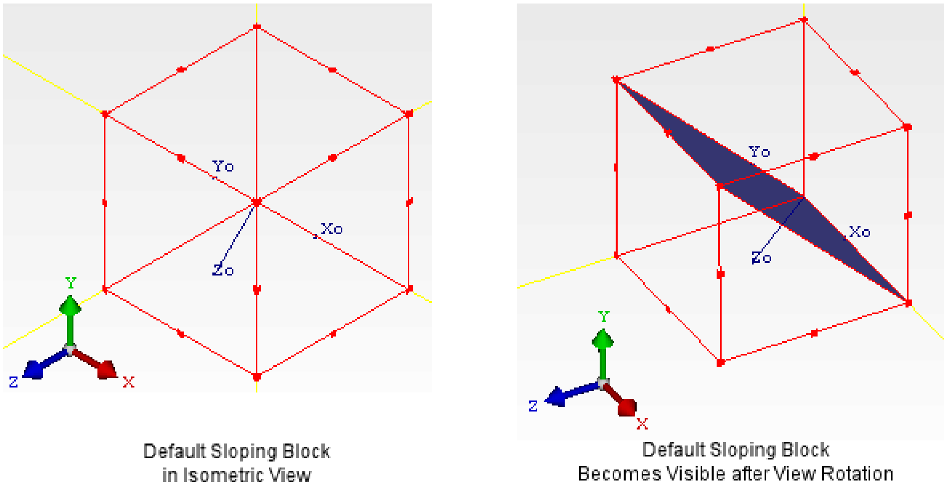
Procedure

1. Select an assembly in the data tree.
2. In the New Object Palette, choose Project Manager Create or Drawing Board Create and double-click the **Sloping Block** icon.

Results

In Isometric View, other than the local Zo axis, the default Sloping Block is not visible as it lies in the plane of view. To view the default Sloping Block, set Solid rendering and rotate the view, see Figure 28-4.

Figure 28-4. Isometric Views of Default Sloping Block



Related Topics

Sloping Block Properties

Sloping Blocks and Solar Radiation

Sloping Block Property Sheet

Sloping Blocks and Solar Radiation

Thin Sloping Blocks are totally opaque to radiation, they absorb no solar radiation.

Thick Sloping Blocks are made of prisms and cuboids, therefore, it is likely that solar absorption will not be correctly accounted for. See "Primitives and Solar Radiation" on page 86 for information on how cuboids and prisms are handled differently.

Related Topics

Sloping Block Properties

Creating Sloping Blocks

Sloping Block Property Sheet

Sloping Block Property Sheet

To access: Select a Sloping Block SmartPart in the model.

Use this property sheet to configure a sloping block.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Construction tab.	See "Sloping Block Property Sheet Construction Tab" on page 306.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.

Sloping Block Property Sheet Construction Tab

To access: Select a Sloping Block SmartPart, then select the **Construction** tab.

Use this property sheet tab to define a cuboid whose surfaces are not parallel to the coordinate axes. A Sloping Block is assumed to be a rotated cuboid with the ends cut perpendicular to the end of the block.

Objects

Field	Description
Modeling Level	• Thin — only allow conduction through the plane (default). • Thick — allow conduction through and in the plane of the block.
Width Xo	The local X dimension, that is, the width of the block.
Define Using	There are two options for defining the length and angle of slope. • Angle — Sets the angle of inclination from the line of the surrounding box. The program calculates the opposite and adjacent lengths on confirmation of the settings. • Sides — Sets the opposite and adjacent side lengths. The program calculates the angle of slope on confirmation of the settings.
Length Yo and Angle	(Angle) Length in Local Yo direction and angle between Yo and Y axes, refer to Figure 28-3.
Opposite and Adjacent	(Sides) Dimensions in opposite and adjacent directions, refer to Figure 28-3.
Thickness Zo	The thickness of the block.

Related Topics

Sloping Block Properties

Thick (3D) and Thin (2D) Walls, Holes, and Sloping Blocks

Chapter 29
Sources

Use Source SmartParts to simulate the velocity, pressure, and temperature over a planar area or volume.

Source Usage

You can use sources to simulate the source of heat because of the dissipation of electrical power within a volume, and a planar (or collapsed) source can be used to set a source of heat that is distributed uniformly over a surface.

Sources can also be used to simulate the source of electrical potential and current when calculating Joule heating.

The Source SmartPart must have an attached Source attribute that defines the type of source.

The default size of a source is one-tenth the size of the solution domain is created. Sources can be collapsed to represent a planar surface.

Sources are represented in the drawing board by a brown outline.

Related Topics

Source Property Sheet

Source Attributes [Simcenter Flotherm Project Attributes Reference Guide]

Source Property Sheet

To access: Select a Source SmartPart in the model.

Use this property sheet to configure a source.

Objects

Object	Description
Location tab.	See “Generic Property Sheet Location Tab” on page 28.
Attachments tab.	See “Generic Property Sheet Attachments Tab” on page 30.

Chapter 30
Supplies

Supply SmartParts are children of Recirculation Device, Rack, and Cooler SmartParts.

Modeling Supplies

Supplies are physically defined by their dimensions, free area ratio and supply direction, which can be normal, angled, or sheared.

Note

The supply location may be on the solution domain boundary for an external device. For internal devices, they must be positioned on the surfaces of a region blanked out by a primitive cuboid as described under "Recirculation Device Property Sheet" on page 286.

Rotating Supplies

The plane of a selected supply can be rotated independent of its parent. Rotating a parent device rotates all its child supplies and extracts.

Overlapping Supplies

If two or more supplies from *different* parent devices overlap, their flow rates are additive.

If two or more supplies from *the same* parent device overlap, they override, but in such a way that the total mass flow required by that device is maintained.

Grid Constraints on Supplies

Grid constraints are available to supplies but localized grid is only available to the parent device.

Turbulent Supplies

If the LVEL K-Epsilon turbulence model has been set, then the values of Turbulence Kinetic Energy and Turbulent Dissipation Rate can be set for the values of the incoming fluid. If left at the default value of zero, an estimate is made based on the flow rate calculated for the average fan speed.

Related Topics

Rotating Objects [Simcenter Flotherm User Guide]

Grid Constraints [Simcenter Flotherm User Guide]

Supply Property Sheet

Recirculation Devices

Extracts

Supply Property Sheet

To access: Select a Supply SmartPart in the model.

Use this property sheet to configure a supply.

Objects

Object	Description
Location tab.	See “Generic Property Sheet Location Tab” on page 28.
Construction tab.	See “Supply Property Sheet Construction Tab” on page 312.
Attachments tab.	See “Generic Property Sheet Attachments Tab” on page 30.

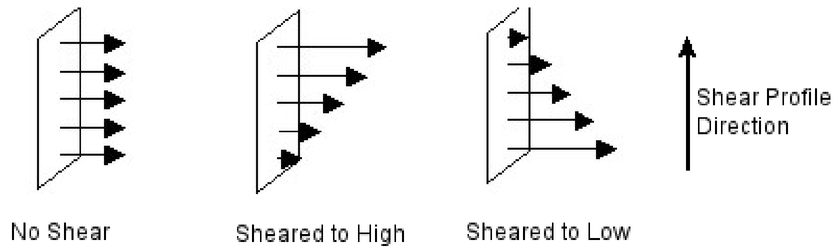
Supply Property Sheet Construction Tab

To access: Select a Supply SmartPart, then select the **Construction** tab.

Use this property sheet tab to define the supply of a recirculation device.

Objects

Field	Description
Size Xo, Yo	Sets the dimensions of the rectangular area representing the supply, that is, the extent of the supply in the coordinate direction indicated (see Usage Notes).
Free Area Ratio	A measure of any obstruction to the flow through the device, for example, a grille across the flow path. The Free Area Ratio sets the ratio of the actual free area to the total area across the device (see Usage Notes). For example, an obstruction blocking 25% of the flow path is represented by 0.75.
Supply Direction	Defines the inclination of the emerging flow: • Normal — Normal to the plane of the supply. • Angled — Inclined to the normal direction. • Sheared — According to a shear profile. If a Sheared supply is detected as being partially blocked, the supply is set to Normal to solve the case.
Turbulent Kinetic Energy	(Turbulence Model set to LVEL K-Epsilon in **Model Setup** tab) Sets the turbulent kinetic energy for the incoming fluid. If left at the default value, an estimate is made based on the flow rate calculated for the average fan speed.
Turbulent Dissipation Rate	(Turbulence Model set to LVEL K-Epsilon in **Model Setup** tab) Sets the turbulent dissipation rate for the incoming fluid. If left at the default value, an estimate is made based on the flow rate calculated for the average fan speed.
Direction XoN, YoN, ZoN	(Angled) Input the three components of the direction vector. See “Angled Flow” on page 132.
Shear Direction	(Sheared) Xo — The shear profile is in the Xo direction. Yo — The shear profile is in the Yo direction.

Field	Description
Maximum Velocity Location	(Sheared) • High — The maximum velocity is at the high end of the shear profile direction. • Low — The maximum velocity is at the low end of the shear profile direction. 
Shear Model	(Sheared) • Constant — The shear profile is constant. • Flow Dependent — The shear profile is determined by the fan characteristic curves. The operating point is found on the FlowRate-Pressure curve, and then from this operating point the efficiency is determined from the FlowRate-Efficiency curve. The operating efficiency is then used to determine the shear profile.

Usage Notes

Dimensions

The dimensions should be set so as to give an area equivalent to that of the actual supply region, for example, if the regions are round ducts, the length of the side of an equivalent square is given by: $(\sqrt{\pi}/2) \times diameter\_of\_duct$

Free Area Ratio

Where the Free Area Ratio is used to represent a significant obstruction, care should be taken to define a sufficiently fine grid perpendicular to the air terminal to be able to capture the entrainment caused by the low pressure next to the obstructions.

Related Topics

Modeling Supplies

Chapter 31
TECs

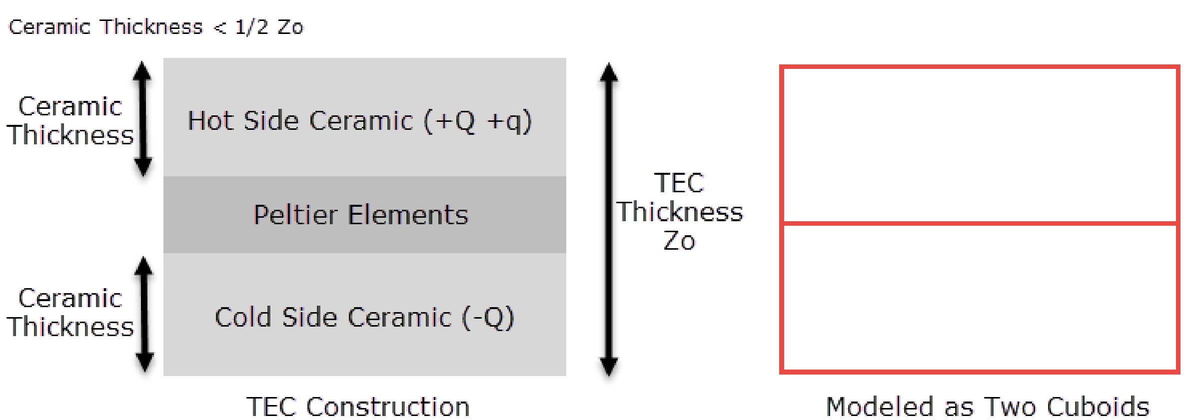
Use TEC SmartParts to model thermoelectric coolers.

TEC Construction

A TEC is represented in the GUI as two abutting cuboids: one each for the hot and cold sides.

The two cuboids are required to ensure there are two rows of cells to hold the hot side and cold side temperatures.

In reality, a TEC can be thought of a being composed of three layers: two outer ceramic layers and an inner layer of Peltier elements. For the purposes of the model image this is simplified as two layers, however, the thickness of the outer ceramic layers is defined in the property sheet. This is identical for both layers but is not shown in the model image. The ceramic thickness must be less than half of the total thickness of the TEC.



Adding a TEC With Hot and Cold Side Monitor Points

A pair of monitor points can be added in a single operation.

Procedure

1. Select an assembly and click the TEC SmartPart icon.

   You have the option of adding a pair of monitor points, one for each side of the TEC.

2. To add a pair of monitor points, select the TEC and click the Monitor Point SmartPart icon.

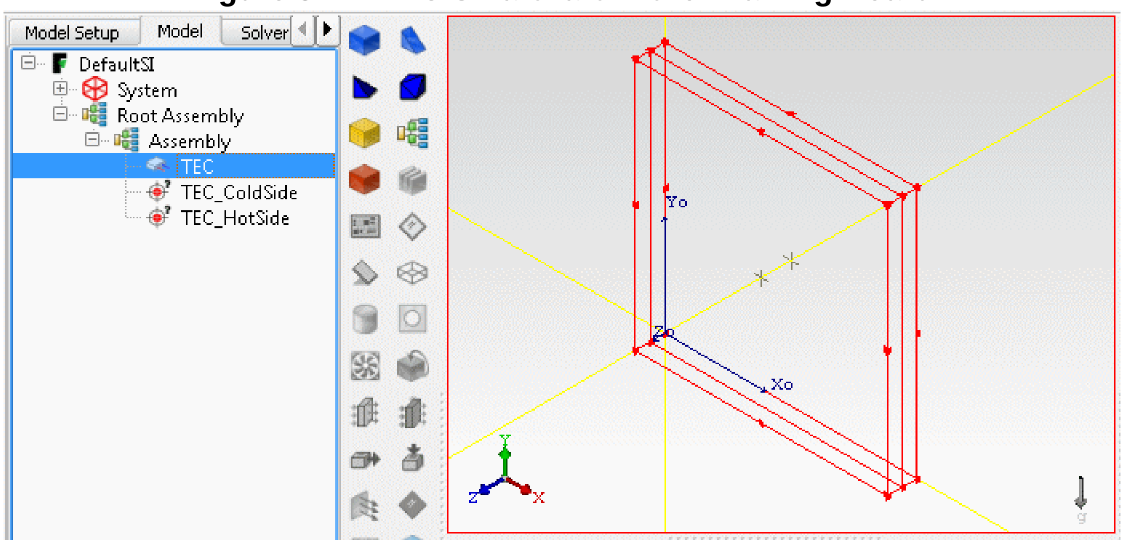
Results

Figure 31-1 shows the assembly in the drawing board. Two monitor points are added by default to report the hot and cold side temperatures. The Hot Side is in the High Zo plane.

The TEC_HotSide monitor point is placed just below the top surface of the hot side.

The TEC_ColdSide monitor point is placed just above the bottom surface of the cold side.

Figure 31-1. TEC SmartPart in the Drawing Board



Related Topics

TEC Property Sheet

Modeling TECs

TECs are most commonly modeled sitting between the top of a component and heat sink.

A definition of the contact resistance (because of the attachment method, that is, glue) between the component and the TEC, and the TEC and the heat sink may be required.

The amount of heat the TEC pumps, the additional heat the TEC generates and the hot and cold side temperatures are the output parameters of interest.

TEC Arrays

TECs can be placed side by side to build an array of TECs. This might be done under situations where a large surface area needs to be cooled.

TEC Stacking

TECs can be stacked. Under these circumstances heat is pumped through all TECs sequentially.

Using the Command Center to Determine the Operating Current

A common goal is to determine the operating current required to achieve the desired performance, that is, limit component junction temperature. The operating current can be determined using the Command Center.

Procedure

1. Vary the operating current as a DoE.
2. Solve each scenario.
3. Use the RSO feature to instantly determine the required current.

   To do this, set the cost function as a target value of maximum junction temperature and using the RSO to provide an indication of the current required for the cost function to be zero, that is, Tj = Tj_max.

TEC Sizing

The relationship between TEC size (length and width, but not height) and its thermal performance is of interest to the designer.

There are two ways of that this relationship can be determined:

- Swap a TEC with another held in the library. This is a more common approach.
- Alter the size of the TEC using the Command Center, using linear functions to ensure the TEC is always centered on the component and to potentially alter the characterization data as a linear function of the size (if appropriate).

TECs in Application Windows

The following subsections give information about TEC SmartParts represented in Results Tables, the Command Center and MCAD Bridge

TEC Results Data in Results Tables

The TEC SmartPart results are displayed in tables in Analyze mode.

TEC SmartPart heat pumped, heat added, dT, and maximum hot side temperature data are available after solution in the TEC table.

Input/Output Settings for the Command Center

Input variables for the Command Center:

- Absolute Location
- Activated
- Localized Grid
- Grid Constraints
- Parametric Data / Operational Current

Output variables for the Command Center:

- Heat Pumped
- Heat Added
- dT
- Maximum Hot Side Temperature

MCAD Bridge Representation

When a model is transferred to MCAD Bridge, each TEC SmartPart is represented as two cuboid bodies, one for the hot side and one for the cold side.

TEC Property Sheet

To access: Select a TEC SmartPart in the model.

Use this property sheet to configure a TEC.

Description

The **Hot Side 1** and **Hot Side 2** tabs are used to enter data at two hot side temperatures. This data is usually available from manufacturer's data sheets.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Construction tab.	See "TEC Property Sheet Construction Tab" on page 320.
Hot Side 1 tab.	See "TEC Property Sheet Hot Side Tabs" on page 321.
Hot Side 2 tab.	See "TEC Property Sheet Hot Side Tabs" on page 321.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.

TEC Property Sheet Construction Tab

To access: Select a TEC SmartPart, then select the **Construction** tab.

Use this property sheet tab to define the general operation and structure of the selected TEC SmartPart.

Objects

Field	Description
Size Xo, Yo, and Zo	The default size is 40 mm × 40 mm × 4.7 mm.
Operational Current	The operational current (the current through the device). The default current is 2.5 A.
Ceramic Thickness	The thickness of the hot and cold side ceramic cuboids. This must be less than half the Zo size. The default thickness is 0.75 mm.
Ceramic Material	The material of the hot and cold side ceramic cuboids. The default material is Alumina (Typical).

Related Topics

TEC Construction

Modeling TECs

TEC Property Sheet Hot Side Tabs

To access: Select a TEC SmartPart, then select either the **Hot Side 1** or **Hot Side 2** tab.

Use this property sheet tab to enter manufacturer's data for a specific hot side temperature. There are two tabs for two sets of data at two temperatures.

Objects

Field	Description
Temperature	Hot Side Temperature at which the characteristic data applies.
Q Max	Maximum pumped heat at the hot side temperature.
Delta T Max	Maximum temperature drop at the hot side temperature.
I Max	Maximum current strength at the hot side temperature.
V Max	Maximum voltage at the hot side temperature.

Usage Notes

There are usually only two hot-side temperatures quoted in the manufacturer's data, but often several currents. The input data uses two currents: the maximum and a nominal current intended to be close to the expected operational current in the user's application.

Related Topics

TEC Construction

Modeling TECs

Chapter 32
EDA SmartParts

Model objects transferred from EDA Bridge.

What are EDA SmartParts?

EDA SmartParts are SmartParts that are created in Simcenter Flotherm when a board model is transferred from EDA Bridge.

Refer to Transferring an EDA Bridge Model to Simcenter Flotherm in the *Simcenter Flotherm EDA Bridge User Guide*.

EDA SmartParts cannot be created directly in Simcenter Flotherm.

Table 32-1 lists the EDA SmartParts and their source objects.

Table 32-1. EDA SmartParts

Icon[1]	EDA SmartPart	Source Object in EDA Bridge
	EDA Boards	MotherBoard or DaughterBoard
	EDA Cans	Can
	EDA Components	Component (Rectangular) or Cylindrical Component
	EDA Electrical Vias Assemblies	Electrical Vias Assembly
	EDA Electrical Vias	Electrical Vias
	EDA Functional Groups	Functional Group
	EDA Heat Sinks	Heatsink
	EDA Layer Assemblies	Layers
or	EDA Layers	Layer (Metallic or Dielectric)
	EDA Potting Compound	Potting Compound
	EDA Thermal Vias	Thermal Via

1. A black border signifies that the Lightweight option is switched on. This is the default setting after transfer from EDA Bridge.

Extending EDA SmartParts

To simplify the data tree, EDA SmartParts have the Lightweight option switched on by default. Use this procedure to extend EDA SmartParts into their constituent SmartParts and primitives.

Restrictions and Limitations

- EDA Layer Assembly and EDA Electrical Vias Assembly SmartParts do not have the Lightweight option.

Procedure

1. Select an EDA SmartPart.

   The **Location** tab of the property sheet opens by default.

2. Uncheck the Lightweight check box.

Results

The EDA SmartPart becomes expandable in the data tree and the black bounding box is removed from the icon. Note there is no change to the representation of the SmartPart in the GDA.

Any changes made in property sheets are coordinated to maintain consistency between the SmartPart and its constituent objects.

EDA Boards

MotherBoards and DaughterBoards transferred from EDA Bridge.

Modeling EDA Boards

The EDA Board SmartPart is equivalent to the MotherBoard or DaughterBoard modeling object in EDA Bridge.

An EDA Board can have EDA DaughterBoards, EDA Functional Groups, EDA Components, EDA Cans, and EDA Potting Compound.

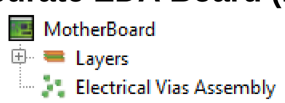
The properties of an EDA Board depend the originating board. The options in the **Construction** tab of the property sheet depend on whether the board was a MotherBoard or a Daughter Board in EDA Bridge, see “EDA Board Property Sheet Construction Tab” on page 330.

Accurate EDA Boards

When the modeling level of the board is set to Accurate in EDA Bridge, the EDA Board has:

- An EDA Layer Assembly child, see “EDA Layer Assemblies” on page 350.
- An EDA Electrical Vias Assembly, see “EDA Electrical Vias Assemblies” on page 340.

Figure 32-1. Accurate EDA Board (Lightweight On)



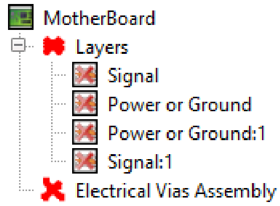
Trend EDA Boards

When the modeling level is set to Trend in EDA Bridge, the EDA Board is treated as orthotropic, there is no EDA Layer Assembly or EDA Electrical Vias Assembly.

Layers and Electrical Vias Assemblies are transferred, but by default they are hidden and ignored, that is, they take no part in the solution.

To show the ignored geometry of a Trend board, check the Show Ignored Geometry check box in the **Project Manager** tab of the User Preferences dialog box (F11).

Figure 32-2. Example of a Trend EDA Board Showing Ignored Geometry



If you want to include these objects in the solution, you can unignore the ignored geometry by unchecking the Ignore Geometry check box in the **Location** tab of the property sheet.

Extended EDA Board

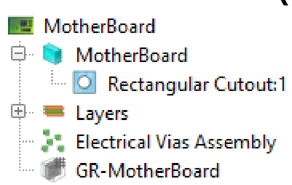
When Lightweight is switched off (Figure 32-3), an EDA Board SmartPart extends to:

- A Block with Holes SmartPart, see "Blocks With Holes" on page 39.

  If a Rectangular Cutout was attached to the board in EDA Bridge, this is transferred as an attached Hole SmartPart, see "Holes" on page 193.

- A Volume Region SmartPart with attached Grid Constraint attributes.

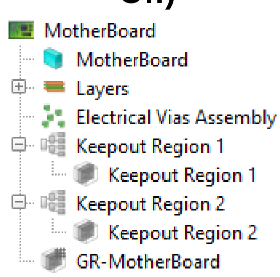
Figure 32-3. Accurate EDA Board (Lightweight Off)



Placement Keepout Regions

When transferred boards have placement keepout regions, these are made visible when the Lightweight is turned off for the EDA Board. The keepout regions are assemblies with a child Volume Region SmartPart that provides an indication of the position and size of the keepout region. The assemblies are hidden by default.

Figure 32-4. Accurate EDA Board Showing Two Keepout Regions (Lightweight Off)



Related Topics

MotherBoards [Simcenter Flotherm EDA Bridge User Guide]

DaughterBoards [Simcenter Flotherm EDA Bridge User Guide]

Rectangular Cutouts [Simcenter Flotherm EDA Bridge User Guide]

Placement Keepout Regions [Simcenter Flotherm EDA Bridge User Guide]

EDA Board Property Sheet

To access: Select an EDA Board SmartPart in the Project Manager.

Use this property sheet to view and change properties of an EDA Board.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Construction tab.	See "EDA Board Property Sheet Construction Tab" on page 330.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.
Notes tab.	See "Generic Property Sheet Notes Tab" on page 34.

EDA Board Property Sheet Construction Tab

To access: Select an EDA Board SmartPart, then select the **Construction** tab.

Use this property sheet to view the size, rotation, and material of the board. If the board is a DaughterBoard, then you can also view the properties of the mounting to the MotherBoard.

Description

The properties depend on the type of originating board: MotherBoard (see Table 32-2) or DaughterBoard (see Table 32-3).

Objects

Table 32-2. Motherboard Construction Properties

Object	Description
Modeling Level	The modeling level of the board: • Trend − The board is treated as a single orthotropic board and heat sinks are modeled as compact. • Accurate − Each layer is modeled explicitly and heat sinks are modeled as detailed.
Size Xo	The overall dimensions of the board.
Size Yo	
Size Zo	
Conductor Material	The conductor material of layers of the board.
Dielectric Material	The material of the board, and the dielectric material for any layers of the board.
In Plane Conductivity	(Trend modeling) The effective material properties of the board.
Normal Conductivity	
Density	
Specific Heat	

Table 32-3. DaughterBoard Construction Properties

Object	Description
Mount Type	How the board is aligned with its MotherBoard: • Parallel − The board is parallel with the MotherBoard. • Perpendicular X/Y Low/High − The board is perpendicular to the MotherBoard.
Standoff	(Parallel mount) The distance between the board and its MotherBoard.

Table 32-3. DaughterBoard Construction Properties (cont.)

Object	Description
Board Offset	(Perpendicular mount) The distance from the edge of the board to the surface of the MotherBoard.
Connector Height	(Perpendicular mount) The height of the connector used to mount the board onto its MotherBoard.
Connector Width	(Perpendicular mount) The width of the connector used to mount the board onto its MotherBoard. The length of the connector is always equal to the width of the board.
Board Side	The board location with respect to its MotherBoard: • Top − Over the top of the Motherboard. • Bottom − Under the bottom of the MotherBoard.
Z Rotation	The rotation of the board about the Z-axis: None, 90 Degrees, 180 Degrees, or 270 Degrees.
Size Xo	The overall dimensions of the board.
Size Yo	
Size Zo	
Connector Material	(Perpendicular mount) The material of the connector used to mount the board onto its MotherBoard.
Conductor Material	The conductor material of layers of the board.
Dielectric Material	The material of the board, and the dielectric material for any layers of the board.

Related Topics

MotherBoard Property Sheet [Simcenter Flotherm EDA Bridge User Guide]

DaughterBoard Property Sheet [Simcenter Flotherm EDA Bridge User Guide]

EDA Cans

Cans transferred from EDA Bridge.

Modeling EDA Cans

The EDA Can SmartPart is equivalent to the Can modeling object in EDA Bridge.

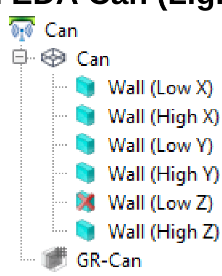
An EDA Can is defined by its dimensions, material, and orientation on the board, see "EDA Can Property Sheet Construction Tab" on page 334.

Extended EDA Can

When Lightweight is switched off (Figure 32-5), an EDA Can SmartPart extends to:

- An Enclosure SmartPart, see "Enclosures" on page 113. There is no Low Z wall as this is the side of the can that is mounted on the board.
- A Volume Region SmartPart with attached Grid Constraint attributes.

Figure 32-5. EDA Can (Lightweight Off)



Related Topics

Cans [Simcenter Flotherm EDA Bridge User Guide]

EDA Can Property Sheet

To access: Select an EDA Can SmartPart in the Project Manager.

Use this property sheet to view and change properties of an EDA Can.

Objects

Object	Description
Location tab.	See “Generic Property Sheet Location Tab” on page 28.
Construction tab.	See “EDA Can Property Sheet Construction Tab” on page 334.
Attachments tab.	See “Generic Property Sheet Attachments Tab” on page 30.
Notes tab.	See “Generic Property Sheet Notes Tab” on page 34.

EDA Can Property Sheet Construction Tab

To access: Select an EDA Can SmartPart, then select the **Construction** tab.

Use this property sheet to view the location, size, and material of a Can.

Objects

Object	Description
Size Xo	The outer dimensions of the can.
Size Yo	
Size Zo	
Thickness	The thickness of the can walls.
Z Rotation	The rotation of the can about the Z-axis: None, 90 Degrees, 180 Degrees, or 270 Degrees.
Board Side	The side of the board on which the can is mounted: Top or Bottom.
Material	The can material.

Related Topics

Can Property Sheet [Simcenter Flotherm EDA Bridge User Guide]

EDA Components

Rectangular and Cylindrical Components transferred from EDA Bridge.

Modeling EDA Components

The EDA Component SmartPart is equivalent to the Rectangular Component or Cylindrical Component modeling object in EDA Bridge.

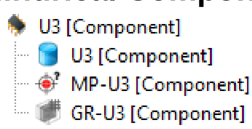
An EDA Component depends on the source component in EDA Bridge.

Cylindrical Components

When Lightweight is switched off (Figure 32-6), the EDA Component SmartPart extends to:

- A cylinder.
- A monitor point.
- A Volume Region SmartPart with attached Grid Constraint attributes.

Figure 32-6. EDA Cylindrical Component (Lightweight Off)

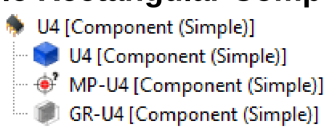


Simple Rectangular Components

When Lightweight is switched off (Figure 32-7), the EDA Component SmartPart extends to:

- A cuboid.
- A monitor point.
- A Volume Region SmartPart with attached Grid Constraint attributes.

Figure 32-7. EDA Simple Rectangular Component (Lightweight Off)



2-Resistor Rectangular Components

When Lightweight is switched off, the EDA Component SmartPart extends to:

- Either:
    - A Compact Component SmartPart, as shown in Figure 32-8, see "Compact Components" on page 45, or
    - A Network Assembly SmartPart, as shown in Figure 32-9, see "Network Assemblies, Network Nodes and Network Cuboids" on page 207.
- A Volume Region SmartPart with attached Grid Constraint attributes.

Figure 32-8. EDA 2-Resistor Rectangular Component as a Compact Component (Lightweight Off)

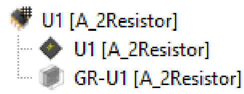
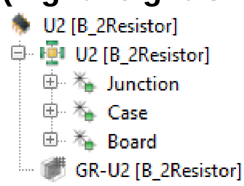


Figure 32-9. EDA 2-Resistor Rectangular Component as a Network Assembly (Lightweight Off)



Detailed Rectangular Components

Detailed components can be powered or unpowered. A powered detailed component has power values defined in the **Construction** tab of the property sheet.

When Lightweight is switched off, the EDA Component SmartPart extends to an assembly and a Volume Region SmartPart with attached Grid Constraint attributes. The assembly defines the structure of the component comprises child assemblies and geometry objects. Extended powered detailed components have their power defined by Source SmartParts with attached Source attributes and/or cuboids with attached Thermal attributes.

DELPHI Resistor Rectangular Components

When Lightweight is switched off, the EDA Component SmartPart extends to:

- Either:
    - A Compact Component SmartPart, as shown in Figure 32-10, see "Compact Components" on page 45, or
    - A Network Assembly SmartPart, as shown in Figure 32-11, see "Network Assemblies, Network Nodes and Network Cuboids" on page 207.

- A Volume Region SmartPart with attached Grid Constraint attributes.

Figure 32-10. EDA DELPHI Resistor Rectangular Component as a Compact Component (Lightweight Off)

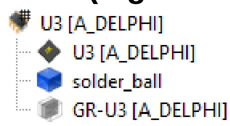
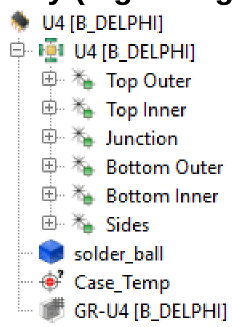


Figure 32-11. EDA DELPHI Resistor Rectangular Component as a Network Assembly (Lightweight Off)

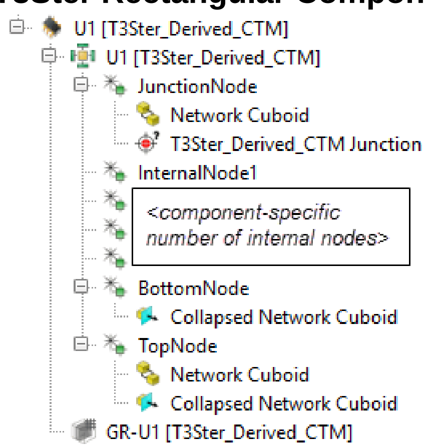


T3Ster Rectangular Components

When Lightweight is switched off (Figure 32-12), the EDA Component SmartPart extends to:

- A Network Assembly SmartPart, see "Network Assemblies, Network Nodes and Network Cuboids" on page 207.
- A Volume Region SmartPart with attached Grid Constraint attributes.

Figure 32-12. EDA T3Ster Rectangular Component (Lightweight Off)



Related Topics

Rectangular Components [Simcenter Flotherm EDA Bridge User Guide]

Cylindrical Components [Simcenter Flotherm EDA Bridge User Guide]

EDA Component Property Sheet

To access: Select an EDA Component SmartPart in the Project Manager.

Use this property sheet to view and change properties of an EDA Component.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Construction tab.	See "EDA Component Property Sheet Construction Tab" on page 339.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.
Notes tab.	See "Generic Property Sheet Notes Tab" on page 34.

EDA Component Property Sheet Construction Tab

To access: Select an EDA Component SmartPart, then select the **Construction** tab.

Use this property sheet to view a component and its position on the board.

Objects

Object	Description
Reference Designator	Identifies the component.
Package Name	Identifies the package to which the component belongs.
Part Number	Identifies the component part number.
Power	Select a unit of power.
Power Table	Power values are editable, however, table rows cannot be added or removed.
EDA Component Type	Cylindrical, Simple, 2-Resistor, Detailed Powered, Detailed Unpowered, DELPHI Resistor, T3Ster.
Z Rotation	The rotation of the component about the Z-axis: None, 90 Degrees, 180 Degrees, or 270 Degrees.
Size Xo	Component dimension in the X-direction.
Size Yo	Component dimension in the Y-direction.
Size Zo	Component dimension in the Z-direction.
Board Side	Location of the component on the board: Top or Bottom.
Tc Max	(2-Resistor, DELPHI Resistor, Detailed, or T3Ster types) Maximum case temperature.
Tj Max	(2-Resistor, DELPHI Resistor, Detailed, or T3Ster types) Maximum junction temperature.
Component Material	(Cylindrical or Simple) The component material.

Related Topics

Cylindrical Component Property Sheet [Simcenter Flotherm EDA Bridge User Guide]

Component Property Sheet [Simcenter Flotherm EDA Bridge User Guide]

EDA Electrical Vias Assemblies

Electrical Vias Assemblies transferred from EDA Bridge.

Modeling EDA Electrical Vias Assemblies

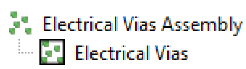
The EDA Electrical Vias Assembly SmartPart is equivalent to the Electrical Vias Assembly modeling object in EDA Bridge.

Note

EDA Electrical Vias Assemblies are ignored if the board has been transferred with the Trend modeling level, but they can be unignored and as described in "Trend EDA Boards" on page 326.

EDA Electrical Vias Assemblies have an attached Material attribute, inherited from the Dielectric Material of the parent EDA Board.

The SmartPart is a holder for any EDA Electrical Vias, if they were transferred. See "Modeling EDA Electrical Vias" on page 342.



The EDA Electrical Vias Assembly SmartPart does not have the Lightweight option.

EDA Electrical Vias Assembly Property Sheet

To access: Select an EDA Electrical Vias Assembly SmartPart in the Project Manager.

Use this property sheet to view and change properties of an EDA Electrical Vias Assembly.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.
Notes tab.	See "Generic Property Sheet Notes Tab" on page 34.

EDA Electrical Vias

Electrical Vias transferred from EDA Bridge.

Modeling EDA Electrical Vias

The EDA Electrical Vias SmartPart is equivalent to the Electrical Vias modeling object in EDA Bridge when Electrical Vias are applied to a whole board.

Note

When Electrical Vias are applied to a single dielectric layer, they are represented by a Hole SmartPart, see “Modeling EDA Layers” on page 353.

EDA Electrical Vias are child objects of an EDA Electrical Vias Assembly.

EDA Electrical Vias represents an array of electrical vias, modeled as a single block.

EDA Electrical Vias have an attached Material attribute, named “Electrical Vias”. This is a bulk material with aggregate properties calculated from the geometry of the array. The thermal conductivity is orthotropic because of via geometry, and is equal in the X and Y directions but different in the Z direction.

Extended Electrical Vias

When Lightweight is switched off (Figure 32-13), an EDA Electrical Vias SmartPart extends to:

- <*n*> cuboids with an attached material attribute.

  Where <*n*> = <*number of layers*> − 1

  The cuboids are placed between the layers of the board.

Figure 32-13. EDA Electrical Vias (Lightweight Off)

Related Topics

Electrical Vias [Simcenter Flotherm EDA Bridge User Guide]

EDA Electrical Via Property Sheet

To access: Select an EDA Electrical Via SmartPart in the Project Manager.

Use this property sheet to view and change properties of EDA Electrical Vias.

Note

The Lightweight check box must be unchecked in the **Location** tab of the parent Electrical Vias Assembly property sheet to view Electrical Vias.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.
Notes tab.	See "Generic Property Sheet Notes Tab" on page 34.

EDA Functional Groups

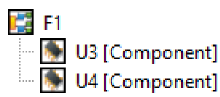
Functional Groups transferred from EDA Bridge.

Modeling EDA Functional Groups

The EDA Functional Group SmartPart is equivalent to the Functional Group modeling object in EDA Bridge.

An EDA Functional Group (Figure 32-14 is an example) contains EDA Components that belong to the functional group.

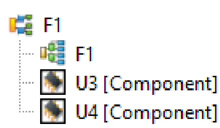
Figure 32-14. EDA Functional Group (Lightweight On)



Extended EDA Functional Group

When Lightweight is switched off, an EDA Functional Group SmartPart extends to an assembly with an attached Material attribute, inherited from the Dielectric Material of the parent EDA Board.

Figure 32-15. EDA Functional Group (Lightweight Off)



Related Topics

Functional Groups [Simcenter Flotherm EDA Bridge User Guide]

EDA Functional Group Property Sheet

To access: Select an EDA Functional Group SmartPart in the Project Manager.

Use this property sheet to view and change properties of an EDA Functional Group.

Objects

Object	Description
Location tab.	See “Generic Property Sheet Location Tab” on page 28.
Construction tab	See “EDA Functional Group Property Sheet Construction Tab” on page 346.
Attachments tab.	See “Generic Property Sheet Attachments Tab” on page 30.
Notes tab.	See “Generic Property Sheet Notes Tab” on page 34.

EDA Functional Group Property Sheet Construction Tab

To access: Select an EDA Functional Group SmartPart, then select the **Construction** tab.

Use this property sheet to view the identity of a Functional Group.

Objects

Object	Description
Function Name	Identifies the function.
Reference Designator	Identifies the functional group.
Total Power	The combined power output of the functional group. This is a manually inserted number inherited from EDA Bridge.

Related Topics

Functional Group Property Sheet [Simcenter Flotherm EDA Bridge User Guide]

EDA Heat Sinks

Heatsinks transferred from EDA Bridge.

Modeling EDA Heat Sinks

The EDA Heat Sink SmartPart is equivalent to the Heatsink modeling object in EDA Bridge.

The EDA Heat Sink SmartPart properties are similar to those of the Heat Sink SmartPart. The differences being:

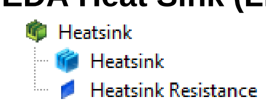
- The following properties are defined in the **Construction** tab of the property sheet, transferred from EDA Bridge:
  - The heat sink material.
  - The rotation of the heat sink about the Z-axis.
  - The interface material.
- The **Attachments** tab of the property sheet does not contain Material, Surface Exchange, nor Radiation attributes.

Extended EDA Heat Sink

When Lightweight is switched off (Figure 32-16), an EDA Heat Sink SmartPart extends to:

- A Heat Sink SmartPart, see “Heat Sinks” on page 173, or a cuboid if the heat sink has no fins.
- A collapsed cuboid, representing the interface material.

Figure 32-16. EDA Heat Sink (Lightweight Off)



EDA Heat Sink Property Sheet

To access: Select an EDA Heat Sink SmartPart in the Project Manager.

Use this property sheet to view and change properties of an EDA Heat Sink.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Construction tab.	See "EDA Heat Sink Property Sheet Construction Tab" on page 349.
Internal Fins tab.	(Heat Sink with Fins) The content is the same as for a Heat Sink SmartPart, see "Heat Sink Property Sheet Internal Fins Tab" on page 189.
End Fins tab.	(Heat Sink with Fins) The content is the same as for a Heat Sink SmartPart, see "Heat Sink Property Sheet End Fins Tab" on page 190.
Pin Geometry tab.	(Heat Sink with Pins) The content is the same as for a Heat Sink SmartPart, see "Heat Sink Property Sheet Pin Geometry Tab" on page 191.
Pin Arrangement tab.	(Heat Sink with Pins) The content is the same as for a Heat Sink SmartPart, see "Heat Sink Property Sheet Pin Arrangement Tab" on page 192.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.
Notes tab.	See "Generic Property Sheet Notes Tab" on page 34.

Related Topics

Heatsink Property Sheet [Simcenter Flotherm EDA Bridge User Guide]

EDA Heat Sink Property Sheet Construction Tab

To access: Select an EDA Heat Sink SmartPart, then select the **Construction** tab.

Use this property sheet to view the dimensions and construction of a heat sink.

Description

Most of the properties are the same as for a Heat Sink SmartPart, see "Heat Sink Property Sheet Construction Tab" on page 188, only those that are different are described below.

Objects

Object	Description
Z Rotation	The rotation of the heat sink about the Z-axis: None, 90 Degrees, 180 Degrees, or 270 Degrees.
Heat Sink Material	Material of the heat sink. **Note:** Heat Sink SmartParts have this defined by an attached Material attribute.
Interface Type	How the interface material is specified: • By Material. • By Value.
Interface Material	(By Material) The material of the interface.
Value	(By Value) The value of thermal resistance of the interface.

Related Topics

Heatsink Property Sheet [Simcenter Flotherm EDA Bridge User Guide]

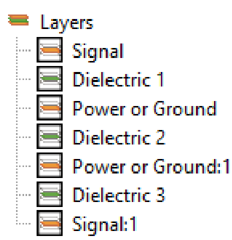
EDA Layer Assemblies

The layers container transferred from EDA Bridge.

Modeling EDA Layer Assemblies

The EDA Layer Assembly SmartPart is equivalent to Group Layers in EDA Bridge.

The SmartPart is a holder for any EDA Layers, see “Modeling EDA Layers” on page 353.



The EDA Layer Assembly SmartPart does not have the Lightweight option.

EDA Layer Assembly Property Sheet

To access: Select an EDA Layer Assembly SmartPart in the Project Manager.

Use this property sheet to view and change properties of an EDA Layer Assembly.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Construction tab.	See "EDA Layer Assembly Property Sheet Construction Tab" on page 352.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.
Notes tab.	See "Generic Property Sheet Notes Tab" on page 34.

EDA Layer Assembly Property Sheet Construction Tab

To access: Select an EDA Layer Assembly SmartPart, then select the **Construction** tab.

Use this property sheet to view whether the layers in the assembly are equally spaced or not, and the offset distances of the layers.

Objects

Object	Description
Equispaced	Yes or No.
Offset Table	A table of offset distances (the distance from the top of the board to the top of the layer) for each layer.

Related Topics

Group Layer Property Sheet [Simcenter Flotherm EDA Bridge User Guide]

EDA Layers

Individual board layers transferred from EDA Bridge.

Modeling EDA Layers

The EDA Layer SmartPart is equivalent to the Layer modeling object in EDA Bridge.

Note

EDA Layers are ignored if the board has been transferred with the Trend modeling level, but they can be unignored as described in "Trend EDA Boards" on page 326.

The Layer Type in EDA Bridge can be set to Metallic or Dielectric.

An EDA Layer is defined by its thickness and materials, see "EDA Layer Property Sheet Construction Tab" on page 356.

%Coverage and Processed

The %Coverage and Processed fields in the **Construction** tab of the EDA Layer property sheet indicate how the layer will be represented.

- Processed = No

  The layer is represented as a smeared thermal conductivity, either entered manually or calculated from the import of an ECAD file.

  The %Coverage field reports the smeared copper content of the layer.

- Processed = Yes

  Layer patches have been created, either manually or by image processing in EDA Bridge. The layer patches are collapsed in the data tree and shown when Lightweight is switched off.

  In this case, the %Coverage field only includes the copper that has not been captured in layer patches, and that copper is smeared across the area of the layer excluding any layer patches.

Extended EDA Layer

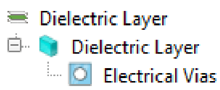
When Lightweight is switched off, an EDA Layer SmartPart extends to:

- A Block with Holes SmartPart, see "Blocks With Holes" on page 39.

- If it was a metallic layer in EDA Bridge and a Layer Patch was attached, this is transferred as a child Hole SmartPart, see Figure 32-17.
- If it was a dielectric layer in EDA Bridge and an Electrical Vias was attached, this is transferred as a child Hole SmartPart, see Figure 32-18.
- If a Rectangular Cutout was attached to the parent board in EDA Bridge, this is transferred as a child Hole SmartPart.

Figure 32-17. EDA Metallic Layer (Lightweight Off)

Figure 32-18. EDA Dielectric Layer (Lightweight Off)



Related Topics

Layers [Simcenter Flotherm EDA Bridge User Guide]

Electrical Vias [Simcenter Flotherm EDA Bridge User Guide]

EDA Layer Property Sheet

To access: Select an EDA Layer SmartPart in the Project Manager.

Use this property sheet to view and change properties of an EDA Layer.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Construction tab.	See "EDA Layer Property Sheet Construction Tab" on page 356.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.
Notes tab.	See "Generic Property Sheet Notes Tab" on page 34.

EDA Layer Property Sheet Construction Tab

To access: Select an EDA Layer SmartPart, then select the **Construction** tab.

Use this property sheet to view the properties of a board layer.

Objects

Object	Description
Layer Type	Metallic or Dielectric.
Thickness	The thickness of the layer.
Conductor Material	The conductor material of the layer.
Dielectric Material	The dielectric material of the layer.
%Coverage	The area-weighted averaged proportion of conductor material in the layer. When Processed = Yes, this field reports the conductor content not processed into layer patches or electrical vias, see "%Coverage and Processed" on page 353.
Processed	Yes or No. If Yes, then layer patches (in metallic layers) or electrical vias (in dielectric layers) are present and are shown in the data tree when Lightweight is switched off.
In Plane Conductivity	Calculated values, based on materials and coverage.
Normal Conductivity	
Density	
Specific Heat	

Related Topics

Layer Property Sheet [Simcenter Flotherm EDA Bridge User Guide]

EDA Potting Compound

Potting Compound transferred from EDA Bridge.

Modeling EDA Potting Compound

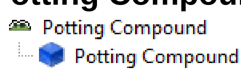
The EDA Potting Compound SmartPart is equivalent to the Potting Compound modeling object in EDA Bridge.

Extended EDA Potting Compound

When Lightweight is switched off (Figure 32-19), an EDA Potting Compound SmartPart extends to:

- A cuboid, with the following attached attributes:
    - A Material attribute inherited from the potting compound material defined in EDA Bridge.
    - If the Material attribute references a Surface attribute, then a corresponding Surface attribute.
    - A Radiation attribute of a single radiating surface.

Figure 32-19. EDA Potting Compound (Lightweight Off)



Related Topics

Potting Compound [Simcenter Flotherm EDA Bridge User Guide]

EDA Potting Compound Property Sheet

To access: Select an EDA Potting Compound SmartPart in the Project Manager.

Use this property sheet to view and change properties of an EDA Potting Compound.

Objects

Object	Description
Location tab.	See “Generic Property Sheet Location Tab” on page 28.
Construction tab.	See “EDA Potting Compound Property Sheet Construction Tab” on page 359.
Attachments tab.	See “Generic Property Sheet Attachments Tab” on page 30.
Notes tab.	See “Generic Property Sheet Notes Tab” on page 34.

EDA Potting Compound Property Sheet Construction Tab

To access: Select an EDA Potting Compound SmartPart, then select the **Construction** tab.

Use this property sheet to view the location, size, and material of potting compound.

Objects

Object	Description
Size Xo	Dimension in the local X-direction.
Size Xo	Dimension in the local Y-direction.
Size Xo	Height of the top of the potting compound from the board surface.
Z Rotation	The rotation of the potting compound about the Z-axis: None, 90 Degrees, 180 Degrees, or 270 Degrees.
Board Side	Top or Bottom.
Material	Material of the potting compound.

Related Topics

Potting Compound Property Sheet [Simcenter Flotherm EDA Bridge User Guide]

EDA Thermal Vias

Thermal Vias imported from EDA Bridge.

Modeling EDA Thermal Vias

The EDA Thermal Via SmartPart is equivalent to the Thermal Via modeling object in EDA Bridge.

An EDA Thermal Via is a child of the component to which it is attached.

An EDA Thermal Via represents an array of thermal vias, but is modeled as a single block.

This aggregate thermal properties are calculated from the geometry of the array. The thermal conductivity is biaxial because of via geometry, that is, it is equal in the X and Y directions (In Plane Conductivity) but different in the Z direction (Normal Conductivity).

Extended EDA Thermal Via

When Lightweight is switched off (Figure 32-20), an EDA Thermal Via SmartPart extends to:

- <*n*> cuboids.

  Where <*n*> = <*number of layers*> − 1

  The cuboids have an attached Material attribute, named “ThermalVia”. This is a bulk material with aggregate properties calculated from the geometry of the array.

  The cuboids are placed between the layers of the board.

Figure 32-20. EDA Thermal Via (Lightweight Off)

Related Topics

Thermal Vias [Simcenter Flotherm EDA Bridge User Guide]

EDA Thermal Via Property Sheet

To access: Select an EDA Thermal Via SmartPart in the Project Manager.

Use this property sheet to view and change properties of EDA Thermal Vias.

Objects

Object	Description
Location tab.	See "Generic Property Sheet Location Tab" on page 28.
Construction tab.	See "EDA Thermal Via Property Sheet Construction Tab" on page 362.
Attachments tab.	See "Generic Property Sheet Attachments Tab" on page 30.
Notes tab.	See "Generic Property Sheet Notes Tab" on page 34.

EDA Thermal Via Property Sheet Construction Tab

To access: Select an EDA Thermal Via SmartPart, then select the **Construction** tab.

Use this property sheet to view the construction properties of a staggered or unstaggered array of PCB thermal vias.

Objects

Object	Description
Diameter	The outer drilled diameter of a via hole.
Plating Thickness	The thickness of the plating inside the vias.
Pitch in X	The distance between via centers in the X-direction.
Pitch in Y	The distance between via centers in the Y-direction.
Number in X	The number of vias in the X-direction.
Number in Y	The number of vias in the Y-direction.
Staggered	• No − An unstaggered array. • Yes − A staggered array.
Plating Material	Copper (Pure) or Copper (Aluminized).
Fill Material	None or Solder.
In Plane Conductivity	The calculated thermal conductivity in the plane of the board.
Normal Conductivity	The calculated thermal conductivity in the Z-direction.
Density	The calculated averaged density of the volume occupied by the bounding box of the thermal vias.
Specific Heat	The calculated averaged specific heat of the volume occupied by the bounding box of the thermal vias.

Related Topics

Thermal Via Property Sheet [Simcenter Flotherm EDA Bridge User Guide]

Index

— S —

— T —

— V —