Simcenter™ Flotherm™ Project Attributes Reference Guide

Software Version 2021.1

SIEMENS

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
Attributes

Attributes can be attached to modeling objects to assign thermally significant properties to the objects or, in the case of grid constraints, to improve the resolution of the results.

Attribute Types

The following paragraphs summarize the usage of the attribute types available in Simcenter™ Flotherm™ software.

- Ambient attributes define conditions outside the domain.
- Fluid attributes define the surrounding cooling or heating fluid.
- Grid Constraint attributes define minimum grid requirements over and near to objects.
- Material attributes define the thermal properties of a material such as thermal conductivity, density, and specific heat.
- Radiation attributes define radiative exchange of heat. Radiation effects are, by default, switched off.
- Resistance attributes define resistance to fluid flow.

- Source attributes define heat sources. By default, source objects have no thermal output, therefore, a source attribute must be attached.
- Surface attributes define surface finishes. Surface attributes can be attached directly to objects or via a material attribute.
- Surface Exchange attributes define solid-to-fluid heat transfer. These attributes are only used in exceptional circumstances.
- Thermal attributes define thermal output.
- Transient attributes define functions describing time-dependent variables.

Project and Library Attributes

Attributes can be classified as project attributes and library attributes depending on how they are stored.

- Project attributes are stored with a project and are only available to objects in that project.
- Library attributes are stored in a project-independent library and are available to all projects. When a library attribute is attached to an object in the project then it becomes a project attribute.

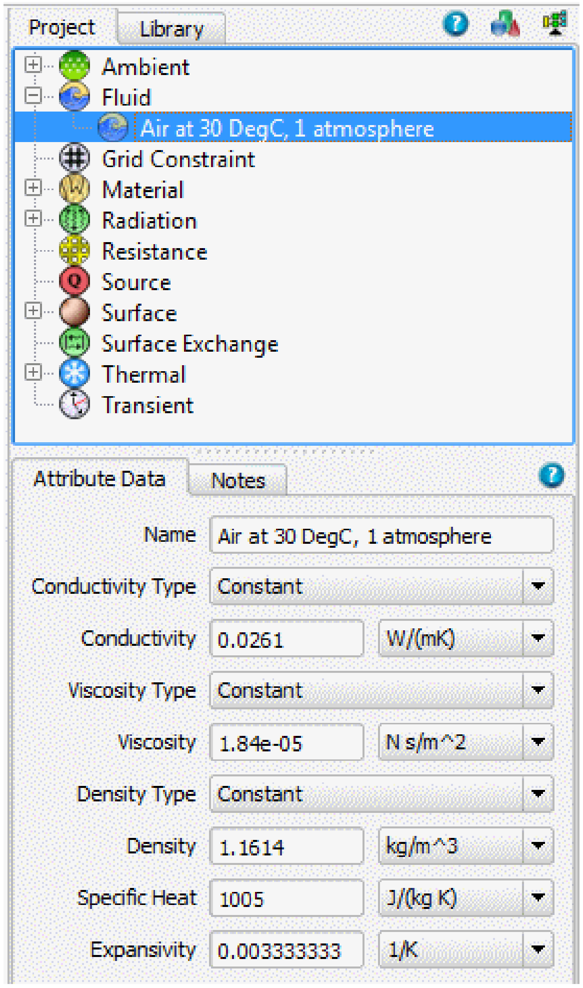
Project Attributes

The attributes known to the current project, that is, those attributes created in or copied to the project, are displayed in the **Project** (Attributes) tab.

If the **Project** tab is not visible, then press F7.

The project attributes are organized by attribute type, in a tree structure.

Figure 1-1. Project Attributes Tab



Library Attributes

Sets of attributes are provided in the Simcenter Flotherm_Libraries folder of the Library as part of the installation. Each attribute type is organized in a separate subfolder.

The Library is listed in the **Library** tab. This tab is shared in a pane with the **Project** tab. If the **Library** tab is not visible, then press F7.

Project attributes can be saved to user libraries for use on other projects.

Related Topics

Libraries [Simcenter Flotherm User Guide]

Adding an Attribute From a Library to the Project

Saving a Project Attribute to a Library

Attribute Operations

Attribute Attachment to Modeling Objects

Attributes attached to objects are shown in the **Attachments** tab of an object's property sheet.

Figure 1-2. Attachments Tab

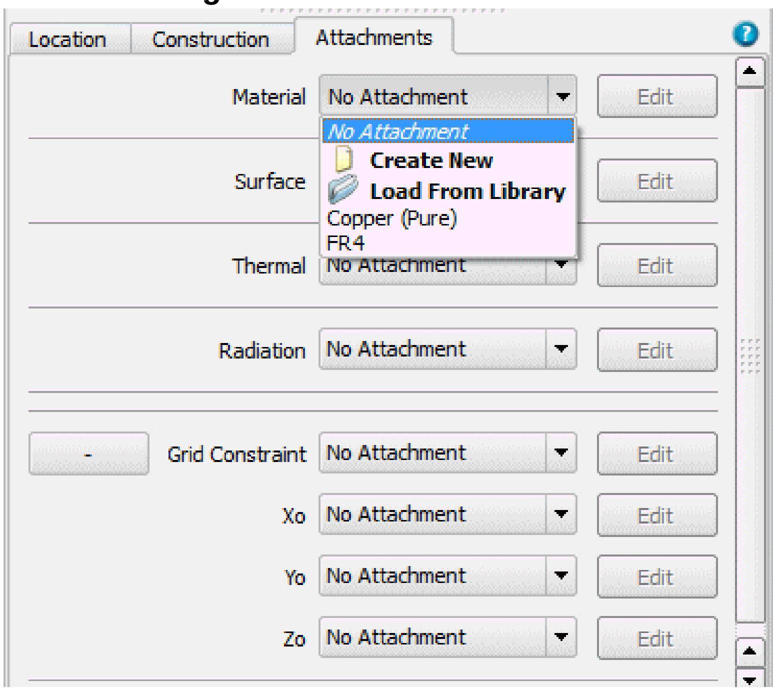


Table 1-1 lists attribute types and the modeling objects to which they can be attached.

Table 1-1. Attribute Types That can be Attached to Objects

Attribute Type	Objects
Ambient	Cutout, Overall Solution Domain, Fixed Flow.
Fluid	Region.
Grid Constraint	All geometry objects.
Material	Assembly, Prism, Cuboid, Tet, Inverted Tet, Heat Sink, PCB, Sloping Block, Enclosure, Cylinder, Block with Holes.
Radiation	Prism, Cuboid, Tet, Inverted Tet, Heat Sink, PCB, Sloping Block, Enclosure, Cylinder, Block with Holes.
Resistance	Resistance, Fan.
Source	Source.
Surface[1]	Prism, Cuboid, Tet, Inverted Tet, Sloping Block, Cylinder, Compact Component.
Surface Exchange	Prism, Cuboid, Tet, Inverted Tet, Heat Sink, Sloping Block, Enclosure, Block with Holes.

Table 1-1. Attribute Types That can be Attached to Objects (cont.)

Attribute Type	Objects
Thermal	Prism, Cuboid, Tet, Inverted Tet, Cylinder, Sloping Block, Enclosure, Block with Holes.
Transient[2]	As for Ambient, Source, and Thermal attributes.

1. Which can also be attached to a Material property.
2. Attached via Ambient, Resistance, Thermal, and Source attributes.

Attribute Operations

Attributes are managed using the **Attachments** tab of object property sheets, and the Project Attributes and Library trees.

Creating a New Unattached Attribute

You can create an attribute without having to attach it to an object. The attribute is then available within the project for future use.

Procedure

1. Open the Project Attributes tree.
2. Right-click on the attribute type node and select **Create <*attribute type*>**, for example, **Create Material**.

   The new attribute is listed in the tree.

Creating a New Attribute for an Object

New attributes created from an object property sheet are attached to the object.

Procedure

1. Select an object from the data tree and open the **Attachments** tab of the property sheet.
2. For the relevant attribute type, open the dropdown list and select **Create New**.

   The new attribute is highlighted in the Project Attributes tree and its property sheet is open.

3. Define the Name and other properties of the attribute using the attribute property sheet.

Related Topics

Adding an Attribute From a Library to the Project

Adding an Attribute From a Library to the Project

Library attributes added to project attributes are available for attaching to objects.

Procedure

Use one of the following methods:

If you want to...	Do the following:
Add a library attribute to the project from the Library tree (Quick method).	• Double-click the attribute in the library tree. If an object is currently selected that accepts that attribute type, then the attribute will be added to the object at the same time. **Note:** This generally only applies to attributes that are attached via the object's **Attachments** tab. The exceptions are the System and Cutout attributes attached to the **Boundaries** tab.
Add a library attribute to the project from the Library tree (Slow method).	1. Select the attribute in a library tree. 2. Either right-click and select **Transfer To Project**, or click the **Transfer To Project** icon in the header of the Project/Library pane.
Add a library attribute to the project from the object property sheet.	1. Select an object from the data tree and open the **Attachments** tab of the property sheet. 2. For the relevant attribute type, open the dropdown list and select **Load From Library**. The library tree is filtered to only show branches that contain attributes of the selected type. 3. Double-click the attribute in the library tree. 4. Click **Clear Filter** to be able to view all library branches.

Results

Once an attribute has been loaded to the project, then it is selectable from the **Attachments** tab dropdown list, and from the Project Attributes tree.

Related Topics

Attaching an Attribute to an Object

Project and Library Attributes

Saving a Project Attribute to a Library

Attaching an Attribute to an Object

Attributes must be attached to objects for them to take effect.

Attributes can be attached from the Project Attributes tree or directly from the Library tree.

Restrictions and Limitations

- For a list of attributes and the objects to which they can be attached, see "Attribute Attachment to Modeling Objects" on page 14.

Procedure

Use one of the following methods:

If you want to...	Do the following:
Add one or more attributes directly to an object.	1. Press **F7** if the Project/Library pane is not open. 2. Select one or more valid attributes in the Project Attributes tree or a single attribute in the Library tree. Multiple-select of library attributes is not allowed. 3. Drag and drop the attribute(s) onto the object in the data tree.
Add a project attribute to an object via the object's property sheet. **Note:** To add a library attribute to the project from the object property sheet, see "Adding an Attribute From a Library to the Project" on page 17.	1. Open the **Attachments** tab of the object's property sheet. 2. Select from the available project attributes listed in the attachments dropdown list.

Attaching an Attribute to a Surface

Surface, Surface Exchange and Radiation attributes can be attached to surfaces of primitive geometry (cuboids, prisms, tets, and inverted tets).

Procedure

1. When attachment to a surface is possible there is an expand "+" button in the **Attachments** tab of the object property sheet.
2. Expand the attachment options.

   The surfaces of the object are listed separately.
3. Select attributes for the surfaces as required.

   Caution

   Attributes are attached with respect to the coordinate system of the root assembly. If the root assembly has been rotated, you must take care to ensure you select the correct face. See "Coordinate Systems" in the *Simcenter Flotherm User Guide* for an explanation of local and system coordinates.

Finding an Attached Attribute by Name

Highlight objects in the project data tree that have a particular attached attribute.

Procedure

1. Open the Find dialog box.
2. Select Common.
3. Of the available Criteria, select the relevant attribute type, for example, Material.
4. Add this to the Criteria list.

   The operators are "is" and "is not". The list of parameters include No Attachment and all project attributes.
5. Select "is" and the named attribute that you are searching for.

   For example, Material — is — Copper (Pure).
6. Click **Find**.

   Objects that have the attribute attached are highlighted in the data tree.

Related Topics

Find Dialog Box [Simcenter Flotherm User Guide]

Detaching an Attribute

You must detach an attribute before it can be deleted.

Procedure

1. Find all objects that have the attribute attached.
2. For each object, open the **Attachments** tab of the object property sheet.
3. Select either **No Attachment** or a different attribute from the attachments dropdown list.

Related Topics

Finding an Attached Attribute by Name

Deleting Unattached Project Attributes (Flushing)

Deleting Unattached Project Attributes (Flushing)

Flushing unattached project attributes clears out any unused attributes from the Project Attributes tree.

Procedure

1. Open the Project Attributes tree.
2. Right-click an attribute type node.
3. Select either **Flush <attribute type>** to only delete unattached attributes of that type, or **Flush All** to delete all unattached attributes.

Related Topics

Detaching an Attribute

Copying a Project Attribute

You can make a copy of an attribute, which can then be edited, to avoid having to create an attribute from scratch.

Procedure

1. Open the Project Attributes tree.
2. Right-click an attribute node and select **Copy**.

   A copy of the attribute is created under the same node. The copy has the same name and properties as the original.
3. To avoid confusion, it is recommended that the copied attribute is renamed.

Deleting an Attribute

You can delete attributes from the Project attributes tree or from the Library tree.

Restrictions and Limitations

- You can only delete attributes that are not attached to project objects.

Procedure

1. Open the Project Attributes tree or Library tree.
2. Right-click the attribute and select **Delete**.

   If the attribute is in a library, a confirmation dialog is opened. Click **Yes** to confirm deletion.

Related Topics

Detaching an Attribute

Saving a Project Attribute to a Library

A project attribute saved to a library ensures that the attribute is available for other projects.

Procedure

1. Open the Library tree.
2. Select a Library folder.
3. Open the Project Attributes tree.
4. Right-click an attribute and select **Save To Selected Library**.

Results

Expand the Library to view the copied attribute.

Related Topics

Adding an Attribute From a Library to the Project

Project and Library Attributes

Changing the Name of an Attribute

In addition to changing the name in the attribute property sheet you can also change the name by direct editing.

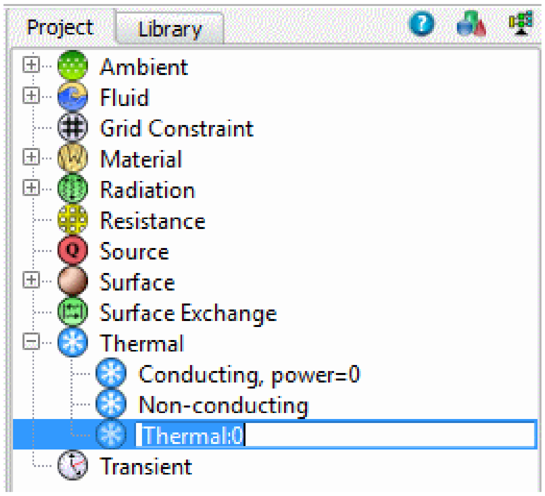
Restrictions and Limitations

- You cannot change the names of the “Conducting, power=0” and “Non-conducting” thermal attributes.

Procedure

Double-click the attribute in the Project Attributes tree.

The name becomes editable.



Chapter 2
Ambient Attributes

The ambient conditions can be refined for a particular face of the solution domain, or any cutout on it, by attaching an Ambient attribute.

Note

Ambient conditions attached to faces are attached with respect to the coordinate system of the root assembly. If the root assembly is rotated, then make sure that the ambient is attached to the correct face.

Ambient Conditions

The default ambient conditions are those set in the Global System Settings section of the **Model Setup** tab. These are the datum pressure (Pressure), fluid temperature (Default Ambient Temperature) and temperature of a remote radiating source (Default Radiant Temperature).

Ambient Temperature

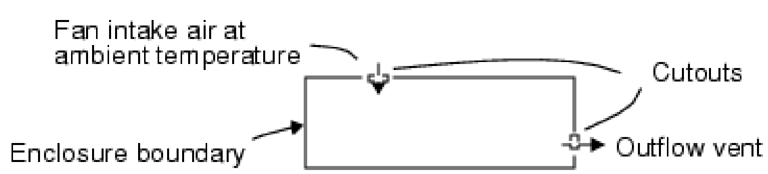
The ambient temperature is, by default, the temperature of the fluid, usually air, that flows into the solution domain. The ambient temperature is also used to calculate heat flow through enclosure walls at the solution domain boundary.

Ambient Temperature and Inflows

The temperature of the air that flows into the domain through an intake fan or vent, or across a free boundary is by default taken as the ambient temperature. For any individual fan or vent, it is possible to override this by setting a different intake temperature by locating a coincident cutout and applying a local ambient temperature using an Ambient attribute. For a fixed flow, you need to attach an Ambient attribute.

This use is illustrated in Figure 2-1 for an enclosure.

Figure 2-1. Ambient Temperature and Inflows

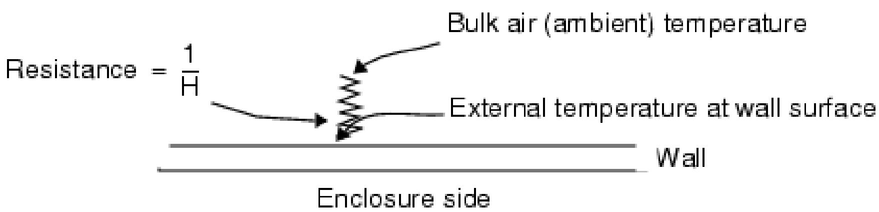


At an outflow (fan or vent), the air that leaves the enclosure leaves with a within-enclosure temperature calculated by the program. Therefore at outflows, the external ambient temperature has no affect.

Ambient Temperature and Heat Flux Through Walls

The ‘bulk’ temperature of the fluid that is on the outside of external walls is, by default, taken as the ambient temperature. This external temperature is of course needed for the program to calculate the heat flux through the wall. The bulk temperature referred to is the fully-mixed air temperature some way away from the wall (Figure 2-2). (The temperature at the exterior surface of a wall differs from the bulk temperature and drives the heat flux by the temperature difference in accordance with the user-specified value of the external heat transfer coefficient (H).)

Figure 2-2. Ambient Temperature and Heat Flux Through Walls

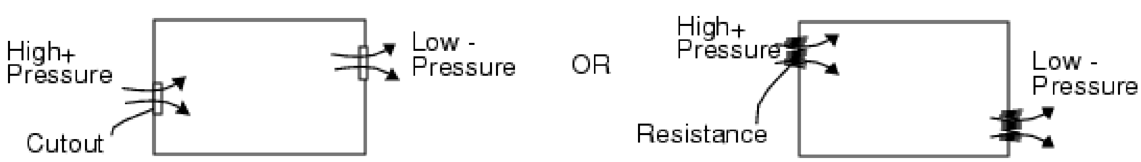


Ambient Pressure

The pressure setting can be used to define boundary conditions for an enclosure with a known pressure difference from inlet to outlet.

Two examples are shown in Figure 2-3.

Figure 2-3. Inlet and Outlet Pressure Defining Boundary Conditions



This can be used to define the wind-induced flow.

Turbulence

When the KE model is active you can define turbulence by setting the Turbulent Kinetic Energy, *K*, and the Turbulent Dissipation Rate, *e*.

From these the turbulent viscosity μ_t is computed.

The equations below are provided as guidance for setting *K* and *e*.

$$K = 10^{-3} \times (estimated\ velocity)^2$$

$$e = \frac{0.1643 \times [10^{-3} \times (estimated\ velocity)^2]^{\frac{3}{2}}}{l_i}$$

where:

$l_i = 0.1 \times (normal\ inlet\ area)^{1/2}$

Heat Transfer Coefficient

The value for *H* can be specified according to the standard free and forced convection correlations found in most heat transfer text books.

For example, for forced convection and Reynolds numbers (Re_L) less than 5×10^5, the Nusselt number (Nu_L) can then be determined from:

$Nu_L = 0.664 Re_L^{0.5} \times Pr^{0.33}$

The heat transfer coefficient, *H*, can be derived from:

$H = (Nu \times k)/L$

where *k* is the conductivity of air and *L* is an appropriate length scale.

External Velocity

The external velocity values are used to specify the magnitude and direction of the free stream, external to the system being modeled.

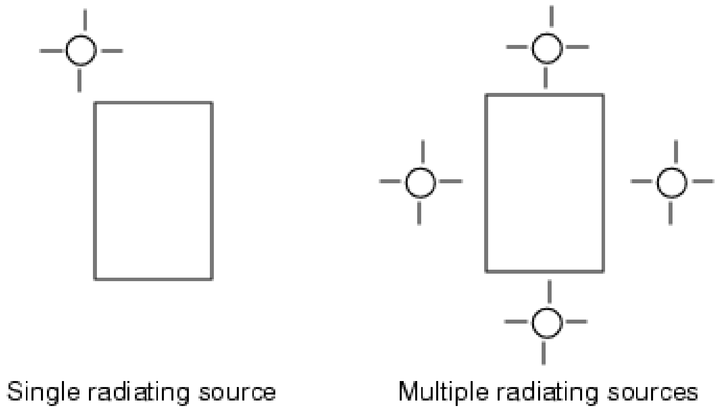
This may be appropriate for modeling the environmental flow (for example, flow around the system) where the value of the velocity should be applied on the open boundaries defined on the overall solution domain.

Radiant Temperature

The radiant temperature is used when there are more than one remote radiating sources to model.

By default, there is one remote radiating source whose temperature is set in the Global System Settings section of the **Model Setup** tab. You can define more than one remote radiating source (Figure 2-4) by attaching ambients with different radiant temperatures.

Figure 2-4. Radiant Temperature Sources



Related Topics

Ambient Attribute Property Sheet

Ambient Attribute Property Sheet

To access: Select or edit an Ambient attribute.

Use this property sheet to set the ambient conditions that prevail outside the overall solution domain.

Description

Values for external pressure, temperature, turbulence, heat transfer coefficient and external velocity are applied to the faces (either individually or to all faces) of the domain boundary or any selected cutouts on the domain boundary.

Objects

Field	Description
Name	Identifies the attribute.
Gauge Pressure	Sets the pressure for the fluid external to the selected domain or cutout face, relative to the datum Pressure set in the **Model Setup** tab.
Ambient Temperature	Sets the temperature for the fluid external to the selected domain or cutout face. This value overwrites the Default Ambient Temperature set in the **Model Setup** tab.
Radiant Temperature	Sets the temperature of a remote radiating source. The radiant temperature set here takes precedence over the Default Radiant Temperature set in the **Model Setup** tab.
Turbulent Kinetic Energy	(Turbulence Model set to LVEL K-Epsilon in **Model Setup** tab) Sets the turbulent kinetic energy for the fluid external to the selected domain or cutout face. 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 fluid external to the selected domain or cutout face. If left at the default value, an estimate is made based on the flow rate calculated for the average fan speed.
Heat Transfer Coefficient	Sets the Heat Transfer Coefficient for any solid surfaces coincident with the selected domain or cutout face. The Heat Transfer Coefficient value for a solution domain (System attribute) is ignored when there are no solids touching the edge of the solution domain, for example, when the geometry is surrounded by air. See "Heat Transfer Coefficient" on page 26.
External Velocity	Sets the ambient wind velocity for the fluid external to the selected domain or cutout face. See "External Velocity" on page 26.
Radiant Transient	Choose or set-up a transient attribute to allow the radiant temperature to vary as a function of time. See "Transient Attributes" on page 131.

Field	Description
Ambient Transient	Choose or set-up a transient attribute to allow the ambient temperature to vary as a function of time. See "Transient Attributes" on page 131.

Related Topics

Ambient Conditions

Model Setup Tab [Simcenter Flotherm User Guide]

Chapter 3
Control Attributes

Control attributes can only be attached to Control objects and define sets of power versus temperature curves for different frequencies.

Frequency-Based Power Control

The Control attribute enables power to be controlled as the temperature hits specific values.

The Control attribute defines a set of curves of power versus temperature where each curve is designated a frequency.

Each curve has a defined minimum temperature, which is shown as a blue rectangle on the curve, and a defined high temperature, shown as a red rectangle.

Single Monitor Point Control

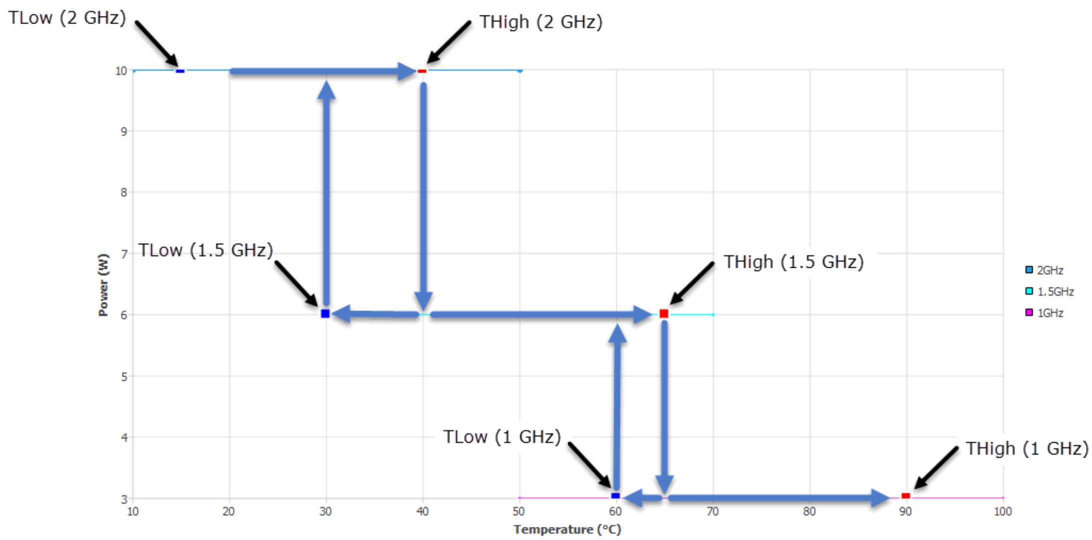
The power delivered depends on the monitor point temperature, and on which curve is being followed. When the maximum temperature on a curve is reached, then the frequency, and hence the power, is decreased. When the minimum temperature on a curve is reached, then the frequency, and hence the power, is increased.

With reference to the curves shown in Figure 3-1, if the ambient temperature is 20 degC then 10 W of power will be applied. This continues following the 2 GHz curve, until the temperature reaches 40 degC, whereupon the power is dropped to 6 W (1.5 GHz curve).

What happens next depends on whether the temperature rises or falls.

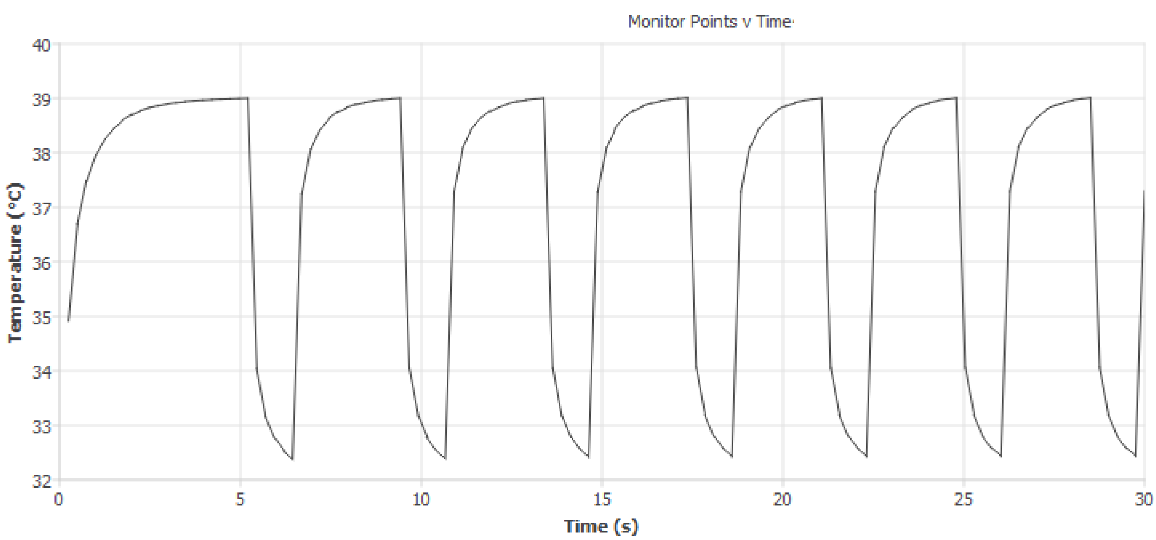
- If the temperature continues to rise until it reaches 65 degC then the power will be reduced again, this time down to 3 W (moving down to the 1 GHz curve).
- If the temperature falls from 40 degC down to 30 degC then the power will increase to 10 W (moving up to the 2 GHz curve).

Figure 3-1. Application of Power as the Temperature Varies



The overall effect of this is that temperature rises can be controlled by reducing power when a set maximum temperature is reached, and increasing power when a set minimum temperature is reached. Figure 3-2 shows this in practice (on a different model) where a THigh of 39 degC has been set on a high control curve, and a TLow of 32.5 degC has been set on a lower control curve. The temperature oscillates between these two values as different power is activated.

Figure 3-2. Transient Monitor Point Profile Plot Demonstrating Controller Switching



- When temperatures lie outside the ranges of the power curves then the power values used are those defined at the ends of the defined range. Power curves are not extrapolated based on their gradient.
- If the temperature is below TLow at the end of a time step, then the next highest frequency is used at the next time step, otherwise the current frequency is used.

- If the temperature is above THigh at the end of a time step, then the next lowest frequency is used at the next time step, otherwise the current frequency is used.

Multiple Monitor Point Control

When there are multiple monitor point children of a controller, the switching between frequencies depends on the choice of THigh Frequency Switching Criteria in the **Construction** tab of the Controller SmartPart property sheet: Any Monitor Point or All Monitor Points.

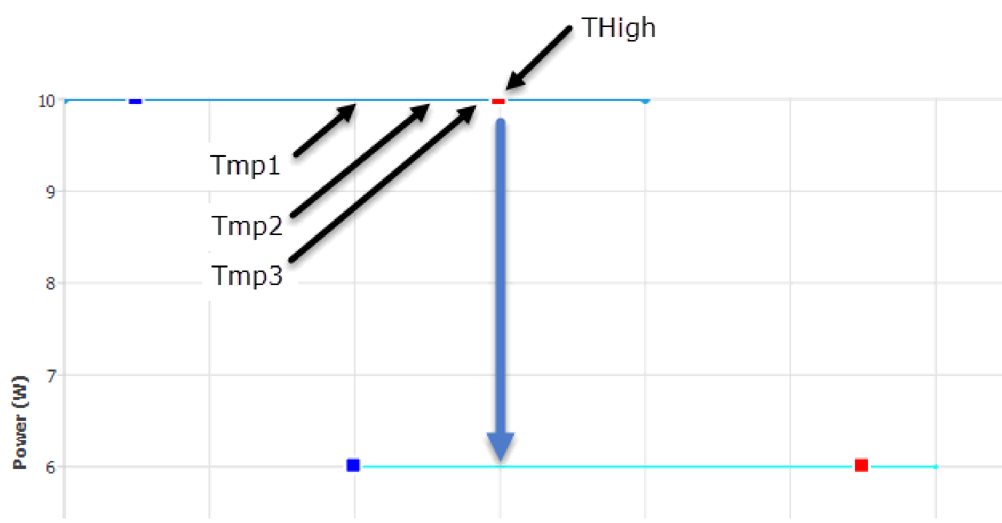
Temperatures Near THigh

Consider the case when all of the monitor point temperatures are below THigh. This is shown in Figure 3-3, where Tmp1, Tmp2 and Tmp3 are the temperatures of three monitor points, MP1, MP2 and MP3 and a power of 10 W is being applied.

If the temperatures rise towards THigh, then the power will drop to 6 W at these events:

- When any one of the monitor point temperatures exceeds THigh if the THigh Frequency Switching Criteria is set to Any Monitor Point.
- Only when all of the monitor point temperatures exceed THigh if the THigh Frequency Switching Criteria is set to All Monitor Points.

Figure 3-3. Multiple Monitor Point Temperatures Below THigh

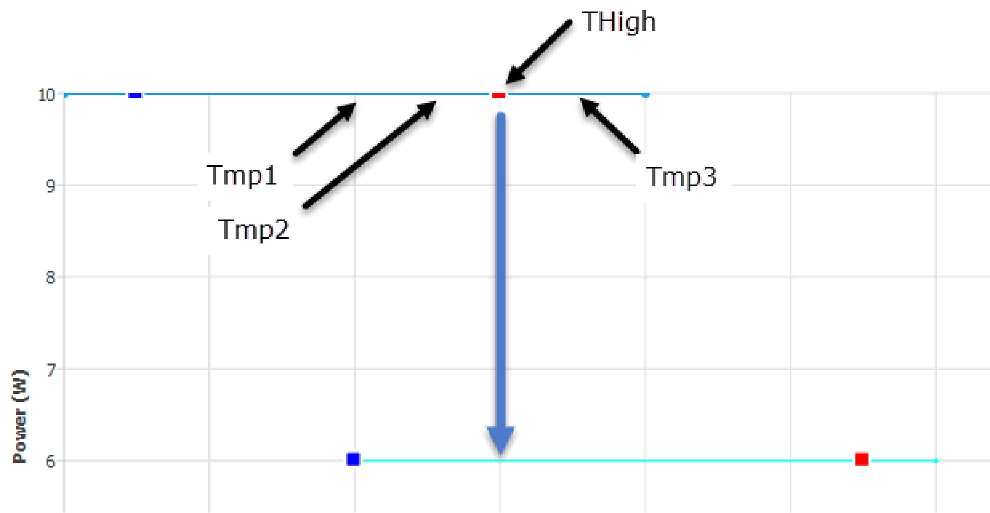


Now consider the case when the monitor point temperatures are either side of THigh. This is shown in Figure 3-4.

The power will drop to 6 W at these events:

- At the next time step if the THigh Frequency Switching Criteria is set to Any Monitor Point.
- When all of the monitor point temperatures exceed THigh if the THigh Frequency Switching Criteria is set to All Monitor Points.

Figure 3-4. Multiple Monitor Point Temperatures Each Side of THigh

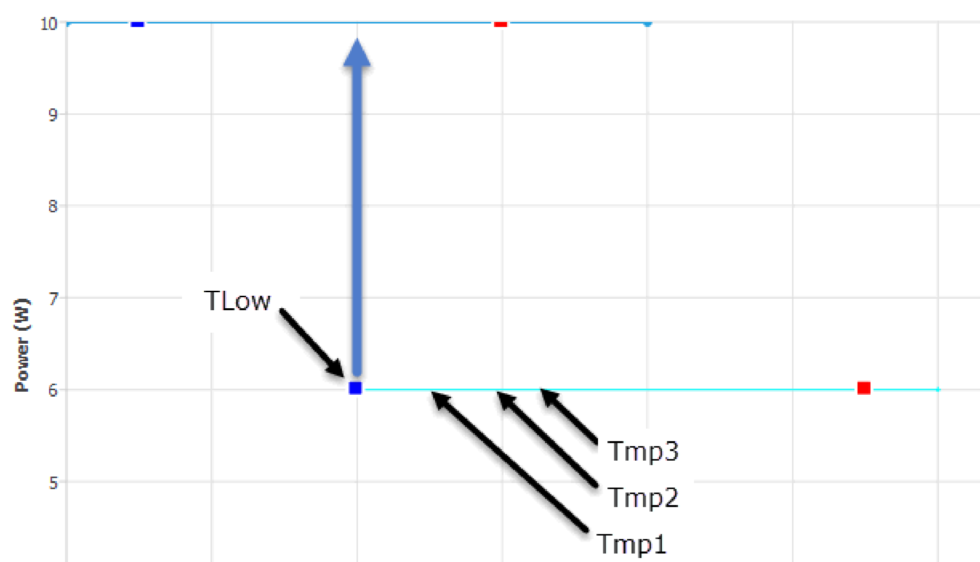


Temperatures Near TLow

Consider the case when all of the monitor point temperatures are above TLow. This is shown in Figure 3-5, where Tmp1, Tmp2 and Tmp3 are the temperatures of three monitor points, MP1, MP2 and MP3 and a power of 6 W is being applied.

If the temperatures fall towards TLow, then the power will rise to 10W only when all of the monitor point temperatures fall below TLow, irrespective of the THigh Frequency Switching Criteria setting.

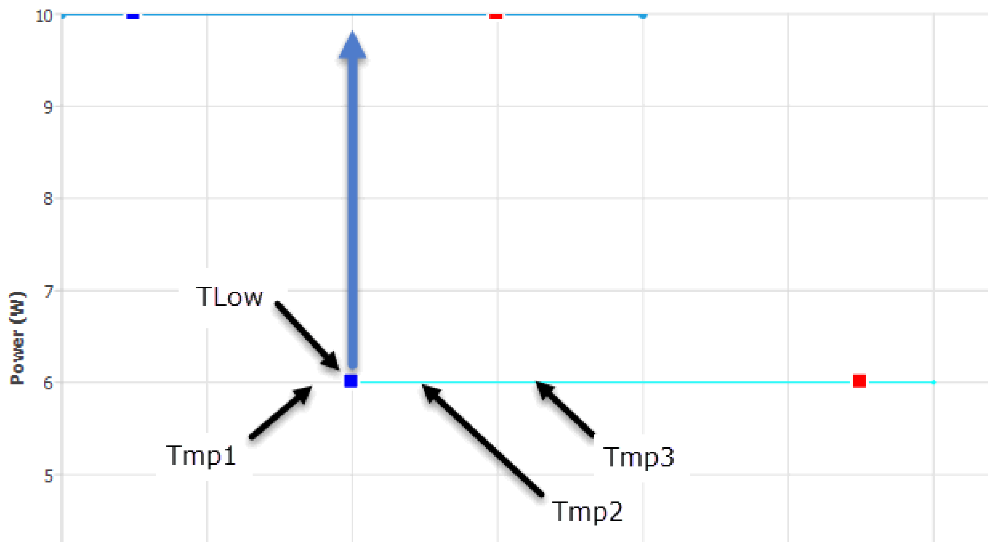
Figure 3-5. Multiple Monitor Point Temperatures Above TLow



Now consider the case when the monitor point temperatures are either side of TLow. This is shown in Figure 3-6.

As before, the power will rise to 10W only when all of the monitor point temperatures fall below TLow, irrespective of the THigh Frequency Switching Criteria setting.

Figure 3-6. Multiple Monitor Point Temperatures Each Side of TLow



Related Topics

Controllers [Simcenter Flotherm SmartParts Reference Guide]

Control Attribute Power vs Temperature CSV File

A comma-separated text data file.

Use this file to set up a frequency-based power source which varies with temperature.

Format

The file must conform to the following formatting and syntax rules:

- The units are those currently set by the chart.
- 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.

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

Parameters

- temperature

  Temperature at which the power dissipation is known.

- power

  Power dissipation at the temperature.

Related Topics

Control Attribute Power vs Temperature Chart

Control Summary Chart CSV File

A comma-separated text data file.

Use this file to export multiple control curves from a control attribute definition.

Format

The file format is multiple columns of data separated by a comma. Each power curve is defined by a pair of columns:

```
<frequency_1>,,<frequency_2>,,...
Min Temperature,<min_temp_1>,Min Temperature,<min_temp_2>,...
Max Temperature,<max_temp_1>,Max Temperature,<max_temp_2>,...
Temperature,Power,Temperature,Power,...
<temperature_1>,<power_1>,<temperature_2>,<power_2>,...
<temperature_1>,<power_1>,<temperature_2>,<power_2>,...
<temperature_1>,<power_1>,<temperature_2>,<power_2>,...
...
```

Parameters

- frequency_n

  Frequency identifier for curve n.

- min_temp_n

  Minimum temperature for curve n.

- max_temp_n

  Maximum temperature for curve n.

- temperature_n

  Temperature at which the power dissipation is known.

- power_n

  Power dissipation at the temperature.

Examples

Output from a three-curve summary chart:

```
1.00E+09,,1.50E+09,,2.00E+09,
Min Temperature,60,Min Temperature,35,Min Temperature,15
Max Temperature,90,Max Temperature,65,Max Temperature,40
Temperature,Power,Temperature,Power,Temperature,Power
50,3,30,6,10,10
100,3,70,6,50,10
```

Viewed in a spreadsheet:

	A	B	C	D	E	F	G
1	1.00E+09		1.50E+09		2.00E+09		
2	Min Temperature	60	Min Temperature	35	Min Temperature	15	
3	Max Temperature	90	Max Temperature	65	Max Temperature	40	
4	Temperature	Power	Temperature	Power	Temperature	Power	
5	50	3	30	6	10	10	
6	100	3	70	6	50	10	
7							

The first curve is defined by columns A and B, the second curve by columns C and D, and the third curve by columns E and F.

Related Topics

Control Summary Chart

Control Attribute Property Sheet and Charts

Descriptions of the property sheet and power versus temperature charts used when defining control attributes.

Control Attribute Property Sheet

To access: Select or edit a Control attribute

Use this property sheet to define control curves for frequency-based power control.

Objects

Object	Description
Name	Identifies the control attribute.
Frequency Unit	Choose which frequency units you want to use to designate control (power vs temperature) curves: Hz, KHz, MHz, or GHz.
Control Summary	Click **Click To View** to open the Control Summary Chart.
Control Summary table	Use to add control curves to the attribute. Each row of the table corresponds with a curve. • Frequency (<*frequency unit*>) − Used to identify the control curve. • TLow − Minimum temperature. • THigh − Maximum temperature. • Curve Defined − Changes state from no to yes when a Power vs Temperature Chart has been defined.
Power vs Temperature Chart	This button is not active until you have defined the temperature limits of at least one control curve and then selected a row. Click **Click To Edit** to open the Power vs Temperature Chart for the selected control curve.

Related Topics

Control Summary Chart

Control Attribute Power vs Temperature Chart

Control Summary Chart

To access: Open a Control attribute property sheet then click the Control Summary **Click To View** button.

Use this chart to examine the set of defined control curves, and to export the data points of those curves.

Description

Low and high temperature thresholds are shown on control curves by blue and red squares respectively

Objects

Object	Description
Export CSV File	Click to save the values of the chart in CSV format.

Related Topics

Control Attribute Property Sheet

Control Summary Chart CSV File

Control Attribute Power vs Temperature Chart

To access: Open a Control attribute property sheet, select a row of the Control Summary table, then click the Power vs Temperature Chart **Click To Edit** button.

Use this chart to plot a power control curve for a particular frequency. The title bar of the chart displays the frequency.

Objects

Object	Description
Temperature	The units of temperature.
Power	The units of power.
Table containing a row for each power value	• Temperature − Temperature at which the power dissipation is known. • Power − Power dissipation at the temperature.
THigh	Maximum temperature, as defined in the Control Summary table of the property sheet. Shown on the chart by a vertical red line.
TLow	Minimum temperature, as defined in the Control Summary table of the property sheet. Shown on the chart by a vertical blue line.
Import CSV File	Click this button to import temperature and power values from a CSV File.

Related Topics

Control Attribute Property Sheet

Control Attribute Power vs Temperature CSV File

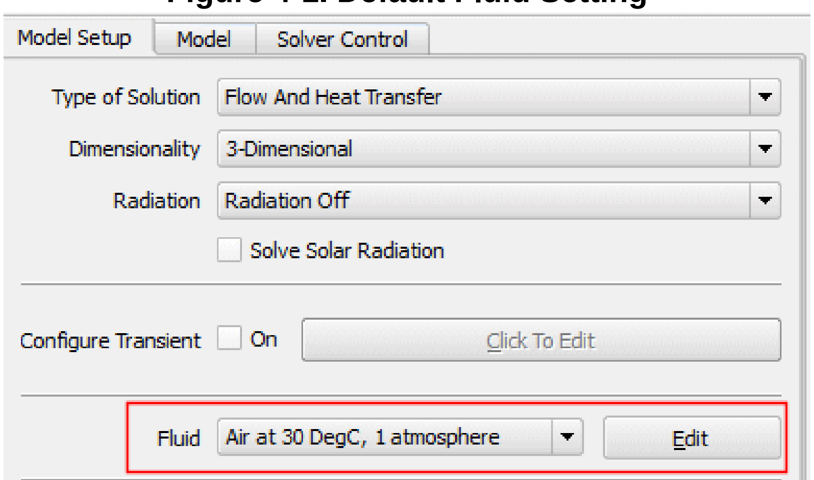
Chapter 4
Fluids

In addition to the model default fluid (usually air), other fluids can be defined within the model.

Default Fluid

The default fluid used to represent the circulating fluid (normally air) is defined in the **Model Setup** tab by a Fluid attribute.

Figure 4-1. Default Fluid Setting



Other Fluids in the Model

In addition to a default fluid, other fluids can be defined in the model as Fluid attributes attached to volume Region SmartParts to create multi-fluid regions.

Related Topics

Multi-Fluid Regions [Simcenter Flotherm SmartParts Reference Guide]

Modeling of Fluids

Modeling of fluids includes setting values for the conductivity, viscosity, density, and specific heat. The conductivity, viscosity, and density values can be set as constant or temperature-dependent.

Fluid Thermal Conductivity

Thermal conductivity in fluids is dependent on whether flow is laminar or turbulent.

Laminar Conductivity

When Laminar is selected in the Turbulence section of the **Model Setup** tab, the molecular conductivity alone governs the conduction (that is, diffusive transport) of heat in the fluid.

Turbulent Conductivity

When Turbulent is selected in the Turbulence section of the **Model Setup** tab, the molecular conduction of heat is augmented by turbulent exchange which is generally much greater than the molecular conduction. The program automatically models this by augmenting the molecular conductivity by a turbulent conductivity which is calculated as:

$$k = (\textit{turbulent viscosity}) \times (\textit{specific heat})/0.9$$

where 0.9 is the default value for the turbulent Prandtl number.

The following have to be set in the property sheet in order for the thermal conductivity above to be used:

- The Conductivity Type is Constant.
- The Viscosity Type is Constant.

If either of these conditions is not met, the conductivity is computed as:

$$k = (\textit{laminar viscosity}) \times (\textit{specific heat})/0.7$$

where 0.7 is the default value (that is, air value) for the laminar Prandtl number.

Temperature Dependent Conductivity of Fluids

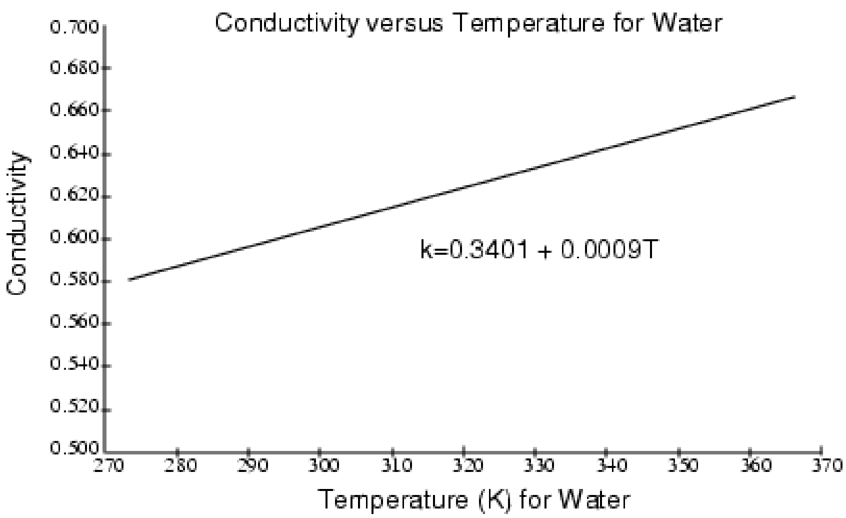
Temperature-dependency of conductivity is rarely required for dilute gases such as air.

Caution

You should not set Temperature Dependent values unless advised to do so by Mentor Graphics.

However, some liquids have a more critical dependence between temperature and conductivity. For example, water has the variation shown in Figure 4-2 for the temperature range 270-to-370K.

Figure 4-2. Temperature Dependent Conductivity



To apply this variation with temperature, set the Fluid attribute property sheet values to:

- Conductivity Type = Temperature Dependent
- Conductivity = 0.58 W/(mK)
- Coefficient = 0.0009 W/(m K^2)
- T ref = 273 K

Viscosity of Fluid Flow

Viscosity is dependent on whether flow is laminar or turbulent.

Laminar Viscosity

When Laminar Flow is selected in the Turbulence section of the **Model Setup** tab, the molecular viscosity alone is responsible for the friction between the fluid elements and also between the fluid and solid surfaces with which it is in contact.

Turbulent Viscosity

When Turbulent Flow is selected in the Turbulence section of the **Model Setup** tab, the molecular friction is augmented by turbulent exchange both within the fluid and between the fluid and the solid surfaces. The program automatically models this by augmenting the molecular viscosity by a turbulent viscosity (determined from the settings made in the Turbulence section of the **Model Setup** tab), thereby modeling the enhanced 'friction' between fluid elements. For friction between the fluid and solid surfaces in turbulent flow, the program calculates friction factors from the wall functions.

For a description of the model used, see Turbulence Viscosity Model in the *Simcenter Flotherm Background Theory Reference Guide*.

Temperature Dependent Viscosity

Use a temperature-dependent viscosity when the temperature range is expected to result in a significant viscosity variation.

This option is rarely required for a dilute gas, but may be useful for a liquid. In this case, the viscosity value defined in the property sheet is the value at the reference temperature.

Fluid Density

Generally, a constant fluid density is assumed throughout the domain of integration, however, if the density variations because of temperature or molecular weight are thought to be significant, then the Ideal Gas Law can be invoked.

Constant Density

For a density type of Constant, the density of the fluid is taken as constant throughout the domain of integration and is represented by one number. Its default value is 1.1614 kg m^{-3} which is the density of dry air at 30°C at atmospheric pressure.

Ideal Gas Law

For a density type of Ideal Gas Law, the density of the fluid in each cell is calculated as:

$$\rho = \frac{(molecular\ weight) \times (p + datum\ pressure)}{R \times (T_{absolute})}$$

where:

- R is the Universal Gas Constant, 8314.4 N m/(kmol K)
- *datum pressure* is set in the Global System Settings section of the **Model Setup** tab. The default values for these datum values are STP conditions, 1.0133×10^5 Pa and 273.15K respectively.
- p denotes the in-cell value of the pressure which is relative to the *datum pressure*. The value of p is determined by the program.
- $T_{absolute}$ is the local in-cell value of temperature on the absolute scale, for example, 20°C becomes 273.15 + 20 = 293.15K.

The default value for the *molecular weight* is that of air, 28.996 g/mol, but this value can be re-set in the property sheet.

Density Modeling Considerations

In most cases the constant density model is adequate, even when temperature and pressure variations are present.

The Ideal Gas Law model should be used when volumetric expansion because of temperature increase is significant. A volumetric expansion of 20% or more is significant. This occurs when local air temperatures in a system exceed 60°C above air-intake conditions.

The constant density model can be used when you do not expect local air temperatures to exceed 60°C above air-intake conditions.

Buoyancy effects of temperature variations of only a few per cent often cannot be ignored. If you have selected the constant density model and activated gravity, the program uses the Boussinesq approximation.

Refer to Model Setup Tab - Gravity Section in the *Simcenter Flotherm User Guide* for further details.

The Ideal Gas Law model uses the datum pressure specified in the Global System Settings section of the **Model Setup** tab, which, by default, is the atmospheric value. The pressure field which the program calculates will be relative to this datum pressure. This fact must be taken into account at fixed-pressure boundaries for what must be set at these boundaries is the relative pressure, not the absolute pressure. Relative pressure values at boundaries will often be zero given that the default value of the datum pressure is one standard atmosphere.

Modeling Expansivity

The expansivity of the fluid is used when gravity is ‘on’ in the expression for the Boussinesq approximation for the buoyancy force.

Expansivity is not used when gravity is not activated, or if gravity is activated but the Ideal Gas Law is selected. In these cases the Boussinesq approximation is replaced by the exact formula.

However, the common situation for electronic enclosure simulations is that gravity is activated and the constant density model is used, in which case the Boussinesq formula is used and the expansivity then enters into the calculation.

The expansivity of liquids, for example, water, is very different from that of air. At 20°C the expansivity of water is 2.1×10^{-4} K^{-1}.

Note

For water, the Boussinesq approximation is only valid for a total temperature range of just a few degrees.

Related Topics

Regions [Simcenter Flotherm SmartParts Reference Guide]

Fluid Attribute Property Sheet

Fluid Attribute Property Sheet

To access: Select or edit a Fluid attribute.

Use this property sheet to define the properties of a fluid.

Description

The property sheet defines the conductivity, laminar viscosity, density, specific heat and expansivity for a particular fluid type, which may be used to represent the bulk fluid in the model.

Objects

Field	Description
Name	Identifies the attribute.
Conductivity Type	• Constant — the default, sets a constant value for the thermal conductivity of the fluid. • Temperature Dependent — expands the property sheet to allow the setting of the conductivity to vary according to a linear function of temperature.
Conductivity	Constant conductivity value, or value at a reference temperature if conductivity is temperature-dependent. Default value is 0.0261 W m^{-1} K^{-1}, which is the thermal conductivity of air at 20°C.
Coefficient	(Temperature Dependent Conductivity) *Coeff* in the following equation: $k = Value + Coeff \times (T - T_{ref})$ where *Value* is the conductivity at T_{ref}, and T is the temperature of the bulk fluid.
T Ref	(Temperature Dependent Conductivity) The reference temperature, T_{ref}, in the above equation.
Viscosity Type	• Constant — the default, sets the molecular viscosity of the fluid to a constant value. • Temperature Dependent — expands the property sheet to allow the setting of the molecular viscosity of the fluid to vary according to a linear function of temperature.
Viscosity	Constant viscosity value, or value at a reference temperature if viscosity is temperature-dependent. Default value is 2.1896×10^{-5} N s m^{-2}, which is the dynamic viscosity of air at STP.
Coefficient	(Temperature Dependent Viscosity) *Coeff* in the following equation: $\mu = Value + Coeff \times (T - T_{ref})$ where *Value* is the viscosity at T_{ref}, and T is the temperature of the bulk fluid.

Field	Description
T Ref	(Temperature Dependent Viscosity) The reference temperature, T_{ref}, in the above equation.
Density Type	• Constant — sets the fluid density to a constant value throughout the domain of integration. • Ideal Gas Law — is only needed when the density variations because of temperature or molecular weight are significant. It activates the ideal gas law requiring as data the gas molecular weight, defaulting to atmospheric air. A different fluid density for each grid cell is calculated according to the ideal gas law. This option can be expensive computationally and is rarely required in Simcenter Flotherm applications. The Ideal Gas Law can only be applied to gaseous coolants.
Density	(Constant Density) The default setting is for a constant value of 1.1614 kg m^{-3}, which is the density of dry air at 30°C at atmospheric pressure.
Molecular Weight	(Ideal Gas Law) The molecular weight in units of g/mol. The default is 28.966 g/mol.
Specific Heat	Only used when the program is solving for temperature. It sets a constant value for the specific heat (constant pressure) of the fluid throughout the integration domain. The default setting is 1005 J kg^{-1} K^{-1}, which is the specific heat of air at STP.
Expansivity	(Constant Density) Only used when gravity is on. It sets the volumetric (that is, cubic) expansivity of the fluid. For an ideal gas, the expansivity is the reciprocal of the absolute temperature. The default value for the expansivity is set to 1/300 K^{-1}. This value may be re-set for constant density in the above dialog.

Related Topics

Modeling of Fluids

Chapter 5
Grid Constraints

You can apply grid constraints to individual objects using these attributes.

Grid constraints applied to the overall solution domain are described under Spatial Solution Grid in the *Simcenter Flotherm User Guide*.

Grid Constraint Considerations

Grid constraints allow you to attach minimum grid requirements to an item of geometry to ensure adequate grid coverage wherever it is located in the calculation domain.

If a grid constraint is applied that requires more coverage than the current grid provides, then the program adds more grid cells. However, if the grid constraint specifies less coverage than actually exists, then no action is taken. The grid constraint does not remove grid, it only ensures adequate coverage.

Note

Grid constraints always try and save a uniform grid over the object to which they are applied, taking into account any existing grid.

Minimum Grid Size

You can specify a Minimum Size (previously called Keypoint Tolerance) to make sure that grid cells smaller than this value are never created.

The default minimum size of the system grid is, in scientific e notation, 1e-05 m (that is, 1/100000th of the default solution domain size). This system (or base) grid minimum size is always in effect, therefore, in a default model, all geometry greater than 10 microns will have grid lines.

If this system minimum size value is increased beyond the size of any cell that would have otherwise been created, then one of the grid lines defining that cell will be removed. In reality, the grid line is never created during the grid calculation process. The minimum size is a tolerance value that is referenced throughout the grid generation process.

There is also the ability to define and activate a minimum size in a grid constraint.

Note

A Minimum Size specified in a Grid Constraint only takes effect if it is less than the Minimum Size specified as the System Grid. For example, if you set a 1 mm minimum size on an assembly in a default grid setup, no grid lines will be removed because the minimum grid size is 1e-05 m at the system grid level.

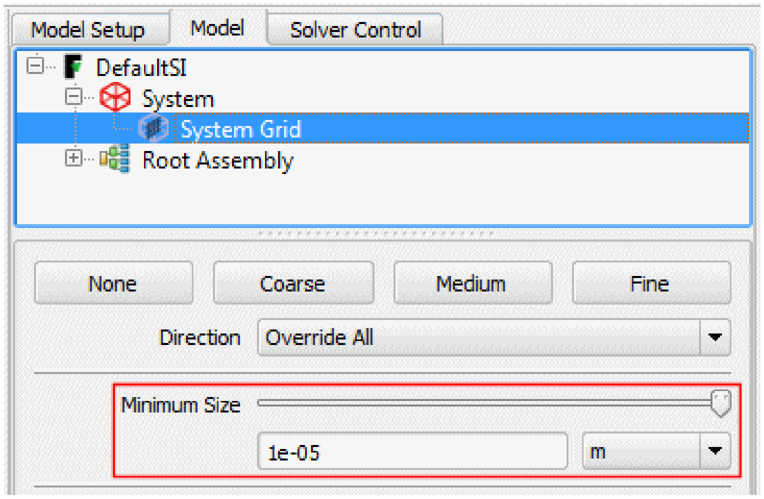
Setting Up a Minimum Grid Size

Minimum grid size is set at system and object levels.

Procedure

1. Set an appropriate minimum size at the system grid level using the System Grid property sheet, see Figure 5-1.

Figure 5-1. Minimum System Grid Size



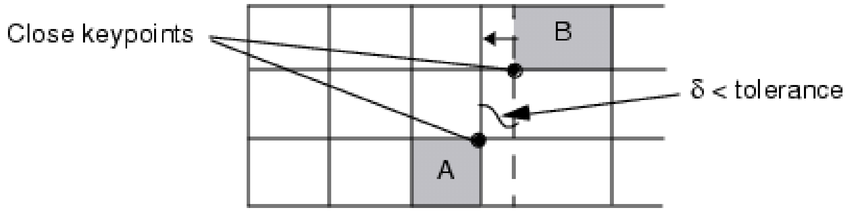
2. Identify objects or assemblies that require a finer grid resolution.
3. Create and attach a grid constraint with the Minimum Size set to either:
   - The default cell size of 1e-05 m to ensure all geometry is resolved (protected), or
   - A more efficient (for example, slightly larger) value, but a value smaller than that defined in the System Grid property sheet.

Minimum Size and Objects in Close Proximity

If two keypoints are closer than the specified tolerance, in a particular direction, then they are treated as one.

This is illustrated in Figure 5-2, where, adding Cuboid B to a model containing Cuboid A would create keypoints closer than the Minimum Size; therefore, Cuboid B expands to snap to the existing grid line, so preventing the creation of a very small grid cell.

Figure 5-2. Keypoints Closer Than Minimum Size Constraint

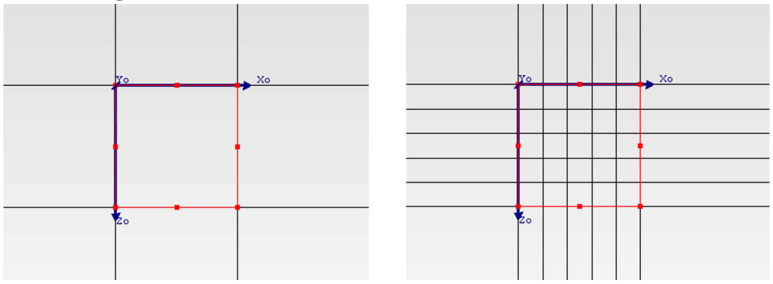


Minimum Number of Grids

Consider applying the grid constraints to a basic cuboid. By default, the program aligns the grid lines along the cuboid boundary.

Applying a minimum cell number of 5 ensures that, in areas of no interference from grids generated from other objects, four additional cells are added, as shown in Figure 5-3.

Figure 5-3. Minimum Number Constraint

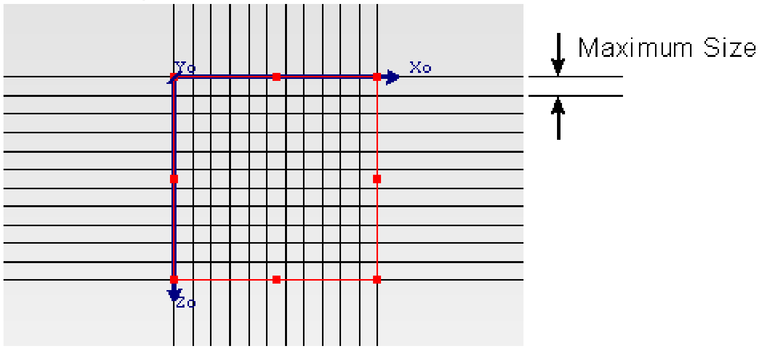


Any grid interference is taken into account, and if sufficient grid already exists, then no action is taken.

Maximum Grid Size

When setting a grid constraint to a maximum grid size, if there are grid cells covering the object that are larger than the maximum grid size, the grid cell density is increased so that the grid spacing is decreased to agree with the grid constraint size.

Figure 5-4. Maximum Size Constraint



Grid Inflation

Use grid inflation to add extra grid cells to the sides of objects, thereby enabling closer monitoring near surfaces.

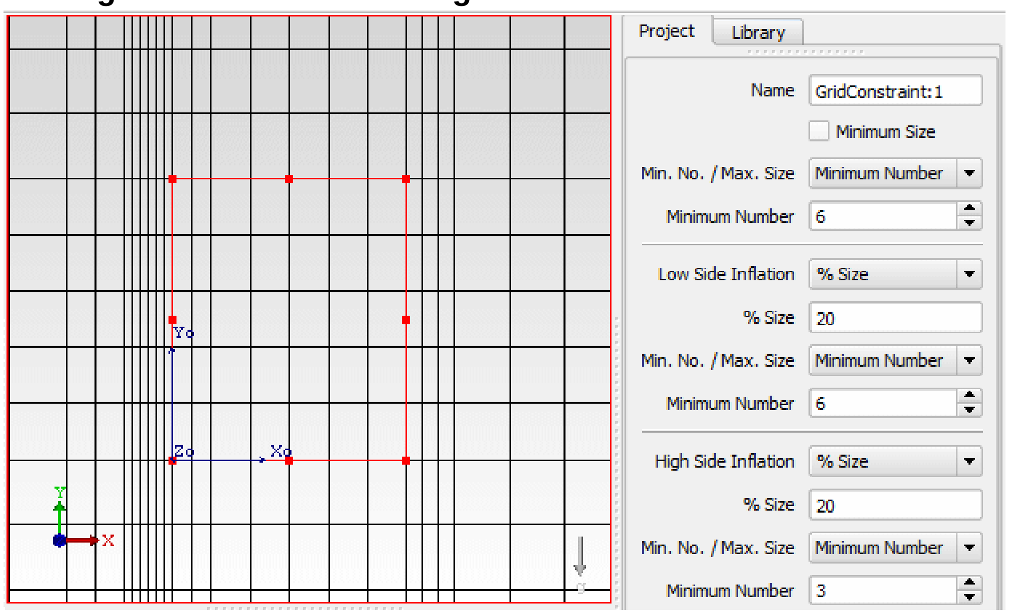
Default Inflation

If inflation is not specified to a side of a selected object, then, when you click the **Inflate Grid** icon or choose **Grid** > **Inflate**, a default inflation of 5% on the high and low coordinate side length, with a Minimum Number of 1 is applied by creating and attaching a grid constraint to the object. The name of the grid constraint is the object name appended with an underscore and letter, for example Cuboid:1_a.

Applied Inflation

Figure 5-5 shows an example of a cuboid where a Minimum Number of grid cells is set to 6, plus additional Inflation cells of 6 on the low side and 3 on the high side for 20% of the cuboid X-direction length.

Figure 5-5. Inflation on High and Low Sides of X Direction



Overlapping Grid Constraints

Where objects with attached grid constraints are coincident, the stricter constraint takes precedence.

Grid constraints on objects are applied in the hierarchical order of the objects in the Project Manager, with any grid constraints on the solution domain satisfied last.

Related Topics

Grid Constraint Property Sheet

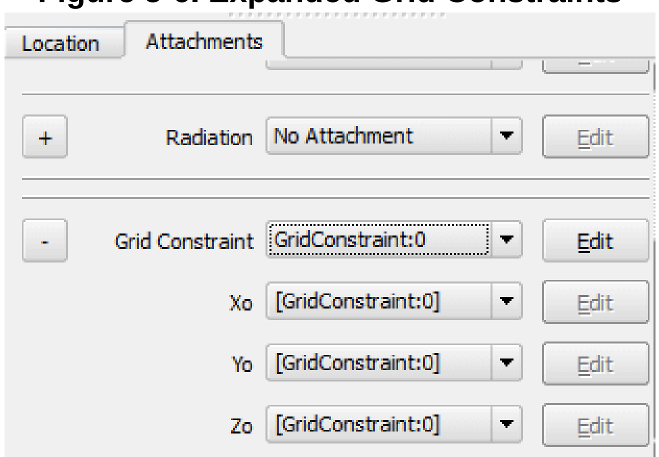
Applying Grid Constraints to Geometry

Grid constraints define the minimum grid requirement across selected geometry, that is, objects, assemblies, and cutouts on the solution domain.

Procedure

1. Select the geometry in either the data tree or drawing board.
2. Open the **Attachments** tab of the geometry property sheet. To apply grid constraints in different directions, expand the Grid Constraint, see Figure 5-6.

Figure 5-6. Expanded Grid Constraints



3. Make sure that the grid is visible in the drawing board view(s): key G.
4. Either select an existing Grid Constraint attribute or create a new one.

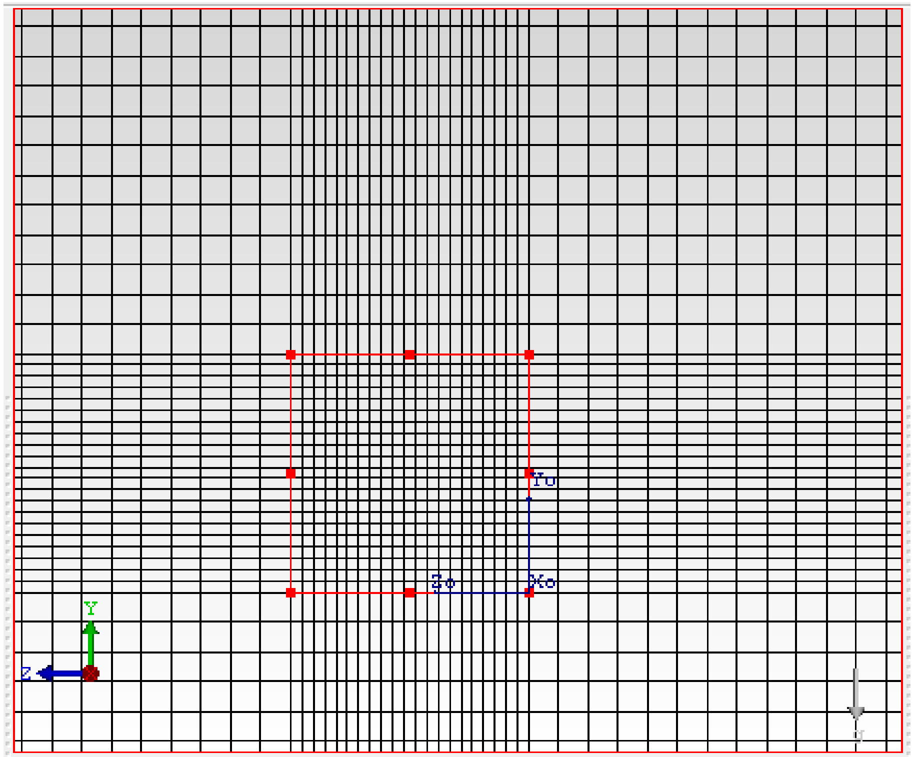
   The Grid Constraint property sheet is displayed and the drawing board view is updated to show the effect of the grid constraint.
5. If necessary, edit the Grid Constraint property sheet.

   As you make modifications to the property sheet you will be able to see the effect in the drawing board.

Results

As an example, Figure 5-7 shows the effect of attaching a grid constraint to a cuboid. In this case a Maximum Size of grid has been set.

Figure 5-7. Maximum Size Grid Constraint Attached to Cuboid



Maintaining Grid Constraints Between Items

Use this procedure when you want to maintain a number of grid cells between a pair of items.

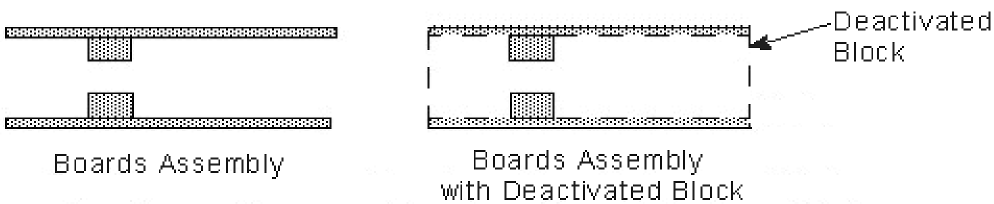
Procedure

1. Add a block to the assembly containing the boards and apply a grid constraint to this block.
2. Deactivate the block so that only the grid associated with it remains.

Results

When the board assembly is moved, then both of the boards and the deactivated block move, maintaining the grid, see Figure 5-8.

Figure 5-8. Maintaining Grid Constraints Between Items



Related Topics

Grid Constraint Property Sheet

Grid Constraint Considerations

Grid Constraint Property Sheet

To access: Select or edit a Grid Constraint attribute.

Use this property sheet to define a grid constraint that can be attached to modeling objects.

Description

The settings made here specify the minimum grid requirements wherever they are applied and will take preference over the overall grid controls at the same location.

Objects

Field	Description
Name	Identifies the attribute.
Minimum Size	Activates Minimum Size. If not active (default), the grid tolerance is inherited from the grid containing the object.
Minimum Size	(Minimum Size) The tolerance (distance) above which keypoints are treated as distinct. This value is also used as the minimum size for the grid spaces created by any object to which the constraint is attached. The value is ignored if the values for the current grid space are smaller. See "Minimum Grid Size" on page 54.
Min. No / Max Size	Controls the number of grid cells according to two options: Minimum Number and Maximum Size.
Minimum Number	(Minimum Number) The minimum number of grid cells allowed over the current object or assembly for the given direction, see "Minimum Number of Grids" on page 56.
Maximum Size	(Maximum Size) The maximum size of grid cells allowed over the current object or assembly in the given direction.See "Maximum Grid Size" on page 56.
Low/High Side Inflation	You can select % Size or Size to define the extent of the grid constraint in the High or Low coordinate direction. The axis direction is specified when you attach the constraint to an object, see "Applying Grid Constraints to Geometry" on page 58. See also "Grid Inflation" on page 57.
% Size	(% Size) The inflation, that is, the extent of the grid constraint beyond the object, as a percentage value of the side length.
Size	(Size) The inflation as an absolute value.
Min. No / Max Size	(% Size > 0) The number of grid cells within the inflation.
Minimum Number	(% Size > 0 and Minimum Number) The minimum number of grid cells within the inflation.

Field	Description
Maximum Size	(% Size > 0 and Maximum Size) The maximum size of grid cells within the inflation.

Related Topics

Applying Grid Constraints to Geometry

Grid Constraint Considerations

Chapter 6
Materials

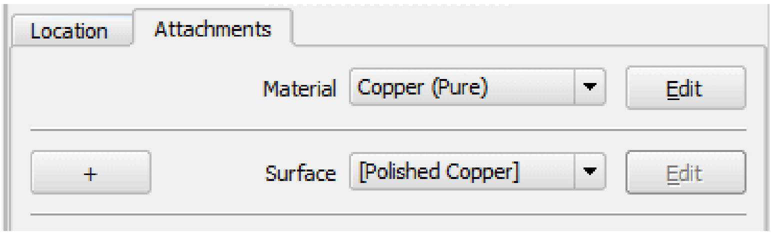
Material properties can be applied to most modeling objects by direct attachment of a Material attribute in the **Attachments** tab of the object's property sheet.

Inherited Surface Attachments

A Surface attribute can be attached to a Material attribute by selection in the Material Attribute property sheet.

If this has been done then the Surface attribute will be marked as inherited from the Material attribute, using brackets as shown in Figure 6-1, when the Material is attached to an object. You can override this inherited attribute by selecting a different Surface attribute.

Figure 6-1. Material Attachment With Inherited Surface Attachment



Material Properties

In addition to specifying the density and specific heat of a material, there is provision for specifying either constant or temperature-dependent thermal conductivity, electrical resistivity and, for solar radiation calculations, the transparency of the material.

Material Thermal Conductivity

The thermal conductivity for the material can be defined as a constant or temperature-dependent for isotropic, orthotropic, or bi-axial materials. X Conductivity, Y Conductivity, Z Conductivity, and Fluid Conductivity scalar fields are available for post-processing in Analyze mode.

Temperature Dependent Conductivity

In cases where the thermal conductivity of a material varies with temperature, you can specify a table of values, either by manual input or CSV file upload. The variation is displayed as a curve.

Note

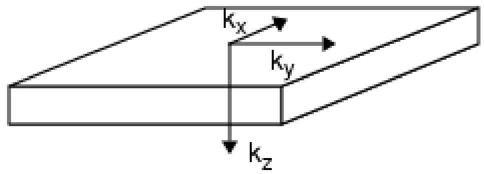
The following objects do not support temperature-dependent conductivity: prisms, SmartParts that contain prisms (Cylinders and Thick Sloping Blocks), collapsed (2D) cuboids, and thin-walled enclosures. If a material with a temperature-dependent conductivity definition is attached to any of these object types then an ERROR is generated during sanity checking.

Orthotropic Conductivity

The conductivity is different in each direction for the material. The local coordinates of the object (cuboid, sloping block) are used. However, you must be cautious when attaching an orthotropic conductivity to other SmartParts (such as enclosures, cylinders, and so on) and review the results with care, see "Advice on SmartParts and Orthotropic Conduction" on page 65.

The Orthotropic Conduction option enables you to define different conductivity values, X Conductivity (kx), Y Conductivity (ky), and Z Conductivity (kz), in the three local coordinate directions: x, y, and z.

Figure 6-2. Orthotropic Conductivity in a Cuboid



This can be useful in correctly describing the heat transfer characteristics of composite materials. A typical PCB containing ground and signal layers will have In Plane Conductivities (kx and ky) several times greater than the Normal Conductivity (kz).

For example, a typical PCB containing two 1oz signal and two 1oz power planes with a thickness of 1.92 mm, would have values in the order of 18 $Wm^{-1}K^{-1}$ for the In Plane Conductivities and 0.6 for the Normal Conductivity.

Biaxial Conductivity

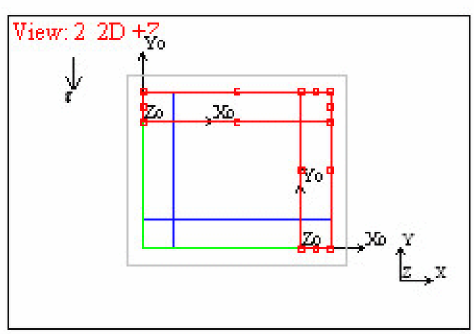
For biaxial materials, the in-plane conductivity (kx or ky) is uniform, but the normal (through-plane) conductivity (kz) value can be different. The local coordinates of the object are used.

Advice on SmartParts and Orthotropic Conduction

If applying orthotropic conductivity to SmartParts, then decompose the SmartPart and let the local coordinates Xo, Yo, and Zo be displayed.

The kx, ky, kz settings in the Material property sheet are applied to the Xo, Yo, and Zo directions respectively (Figure 6-3).

Figure 6-3. Orthotropic Conductivity in an Enclosure



In Figure 6-3 an enclosure is decomposed. ky would be the normal (through-plane) conductivity for the upper wall, but would be in-plane for the right wall. If it is really intended to work with orthotropic conductivities in the walls, then some more materials have to be defined with proper permutations of kx, ky, and kz.

Caution

Strange results will arise from unintended attachment of orthotropic conductivities to SmartParts.

Electrical Resistivity

The electrical resistivity of the material can be defined as a constant or as temperature-dependent.

Temperature Dependent Electrical Resistivity

The electrical resistivity varies according to a linear function of temperature:

$$\rho(T) = Value + Coeff\,(T - T_{ref})$$

where

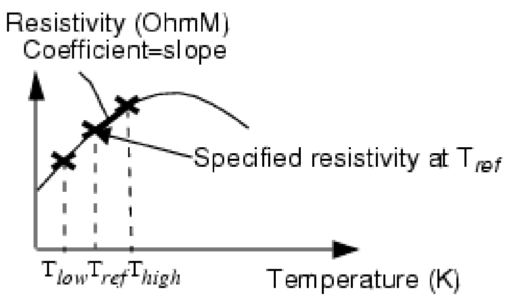
Value, *Coeff*, and T_{ref} are values taken from standard material data books.

T is the temperature in the grid cells.

In addition, T_{low} and T_{high} settings limit the temperature range over which the variation is allowed to apply. Outside these temperature limits the resistivity will be held constant at the levels calculated for the T_{low} and T_{high} values.

Note the following:

- The Electrical data is only required.
- This feature has two possible options: Constant Current or Constant Voltage.
- For Constant Current, the resistivity will be determined from the slope of the curve at T_{ref}.



For Constant Voltage, a conductance will be used which varies linearly with temperature. This will have the same value and slope as 1/resistivity at T_{ref}. The T_{low} and T_{high} cutoffs will be applied as for the constant current case.

Transparent Material

By default, all materials are opaque and all solar radiation is either reflected or absorbed.

By activating Transparent Material you can model the transmission, reflection, and absorption of solar radiation through transparent materials using one of two models: Model Type 0 and Model Type 1.

Model Type 0 Transparent Materials

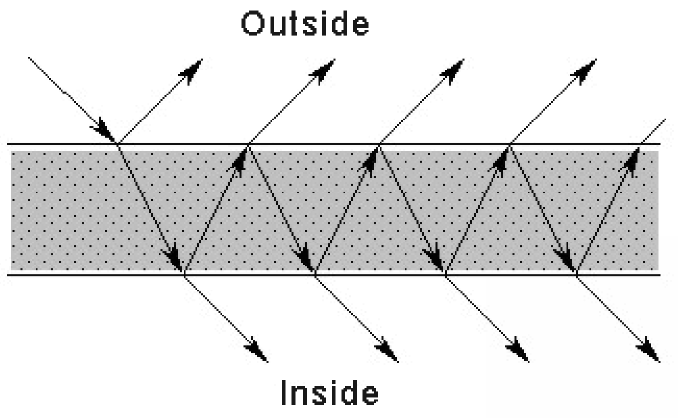
The Solar Absorption Coefficient, a, and Refractive Index, n, are required to calculate reflection and absorption through transparent materials using Model Type 0.

- The Solar Absorption Coefficient (SAC) is a material property that describes the amount of solar radiation absorbed per unit length of material. The actual proportion of the heat absorbed will, therefore, be a fraction of this value dependent on the thickness of the glass and the angle at which the sun's rays strike the glass and are subsequently transmitted.
- The Refractive Index (RI) is defined as the ratio of the speed of light passing through a vacuum to the speed at which it passes through the material in question. This property is important because it determines what proportion of the solar radiation is reflected at a surface where there is a change in refractive index. It also changes the angle at which the solar travels through the glass from the incident angle, therefore affecting the amount of solar radiation absorbed.

These two parameter values are normally available from material manufacturers and, providing that the material is used in isolation (that is, it is not part of a double or triple glazing system, and so on) and it is not tinted or coated, these will be sufficient in calculating the correct solar absorption, transmission, and reflectance.

The SAC and RI allow the software to calculate how much of the solar gain is transferred into the building and how much is reflected away from the glazing as well as determining how much of the solar radiation is used heating the glazing. The calculation represents the transmission of solar radiation through a single layer of transparent material allowing for all the internal reflections as shown in Figure 6-4.

Figure 6-4. Reflections in Transparent Materials



All the beams on the bottom of the layer are added together to obtain the total amount of solar transmitted, which gives the correct solar intensity behind the single layer of glazing. The reflections on the top of the layer are not traced further, therefore this part of the solar radiation is removed from the model. This is appropriate where the glazed element is on the edge of the domain, however, it should be noted that where the glazing is internal this will introduce a second order error.

Unless represented in detail (normally computationally impracticable, although localized grid make this approach more realistic), multiple layers of transparent materials, for example, glass with coatings, or double/triple glazing, cannot be simulated using this model because of the significant grid required. To model in detail, the multiple layer windows with air gap between glass layers can be represented using multiple layers of glass with a layer of air in between, but the multiple reflections between two layers of glass are ignored.

To represent more complex transparent materials (for example, materials that are part of glazing systems with many thin layers, such as coatings) or double or triple glazing where there is insufficient grid an equivalent SAC and RI value for the specific angle of incidence will need to be derived. Contact Mentor Graphics customer support if you need information on how to derive these values.

Model Type 1 Transparent Materials

Transmittance and Reflectance values are required to calculate reflection and absorption through transparent materials using Model Type 1.

- Transmittance is the fraction of direct solar radiation that is transmitted through the glass system.
- Reflectance is the fraction of direct solar radiation that does not penetrate the external surface of the glass. It should also account for the effects of shading, as well as tinted and coated surfaces.

Note

This value for Reflectance takes precedence over the Solar Reflectivity value in the Surface Attribute Property Sheet. An exception is for a collapsed cuboid, where the Solar Reflectivity figure in the Surface Attribute dialog takes precedence over the value defined here.

The solar value behind the glass material will be equal to the transmittance multiplied by the solar value in front of the glass.

The solar energy, A, absorbed by the glass is:

$$A = 1 - T - R$$

where T is the Transmittance and R is the Reflectance.

Reflected Solar Energy

In both transparency models, the solar energy reflected from the surface is not accounted elsewhere, which means the energy will be missing from the system.

Related Topics

Material Attribute Property Sheet

Surface Attribute Property Sheet

Phase Change Material (PCM)

The Material attribute definition includes the option to model phase change between solid and liquid.

The PCM feature is only relevant when solving transient analyses, and is for encapsulated material applications as the solver does not consider any volume change of the material, and does not consider the melted state of the material to be a liquid in the analysis.

Latent heat and melt temperature values are used by the solver to ensure enthalpy is accounted for.

Within PCM objects, the solver fully considers the latent heat of melting within each time step. With this method, the user does not need to ensure that melting events are keypointed by the time grid.

To help convergence of PCM models, it is recommended that you:

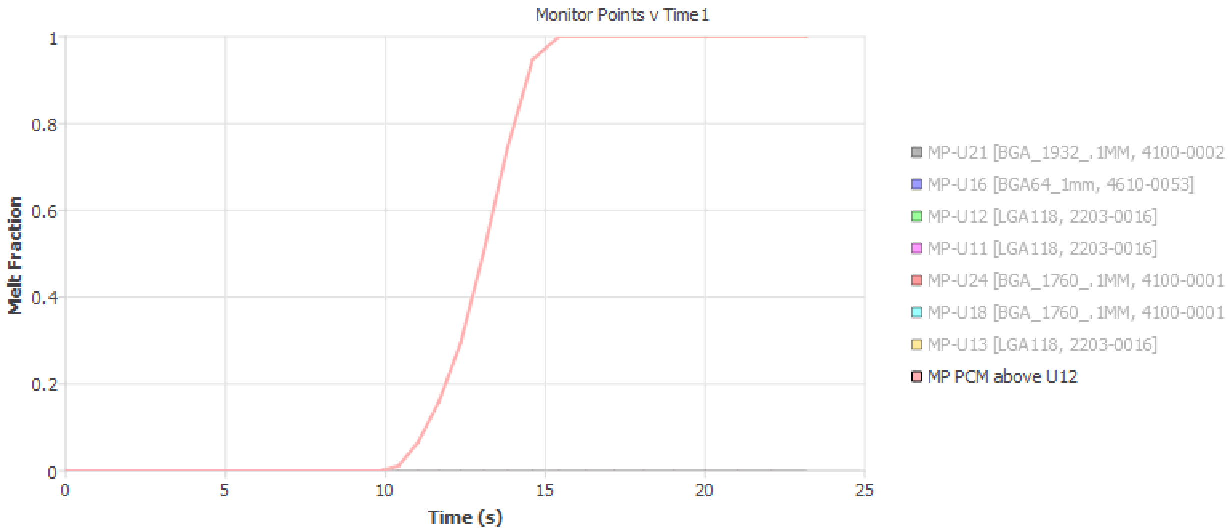
- Activate the double-precision solver (**Solver Control** tab).
- Increase the number of outer iterations (**Solver Control** tab) from the default 500 to allow convergence for all time steps.

Material attribute definitions for some common phase change materials are provided in the installed library under Materials > Phase Change Materials.

When PCM is switched on, the Melt Fraction scalar variable is calculated. This is a measure of the proportion of liquid to solid, where 1 = all liquid and 0 = all solid.

The Melt Fraction can be selected as a variable in monitor point profile plots.

Figure 6-5. Melt Fraction Profile Plot



After solving, isosurface plots of the Melt Fraction can be used to visualize the melting front within the PCM.

Conductivity Temperature Chart CSV File Format

A comma-separated text data file.

Use this file to set up a thermal conductivity profile which varies with temperature.

Format

A Conductivity Temperature Chart CSV File must conform to the following formatting and syntax rules:

- The units are those currently set by the chart.
- 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.

```
<temperature>,<conductivity>
<temperature>,<conductivity>
...
```

Parameters

- temperature

  A temperature at which you want to define a thermal conductivity.
- conductivity

  The thermal conductivity value at that temperature.

Related Topics

Conductivity Temperature Chart

Material Attribute Property Sheet

To access: Select or edit a Material attribute.

Use this property sheet to define the thermal conductivity, density, specific heat of a material. You can also specify the properties that determine solar radiation transmission, reflection, and absorption by the material.

Description

The heat transfer from a block is calculated from the temperature distribution within the block. Such a calculation requires you to set the material properties of the block, namely its specific heat, thermal conductivity and density.

Objects

Field	Description
Name	Identifies the attribute.
Thermal Conductivity Type	Choose from: • Isotropic − Defined by a single parameter or table. • Orthotropic − by a set of three (X, Y and Z) parameters. • Biaxial − by two (axial and plane) parameters.
Input Method	(Isotropic) Defines the isotropic conductivity as: • Single Value − A single value. • Table − A temperature-dependent curve.
Conductivity	(Isotropic and Single Value) Value is constant in all directions.
Conductivity Temperature Chart	(Isotropic and Table) Click **Click To Edit** to open the Conductivity Temperature Chart to specify temperature-dependence of the thermal conductivity.
X, Y, and Z Input Method	(Orthotropic) Defines the orthotropic conductivities as: • Single Value − A single value. • Table − A temperature-dependent curve.
X, Y, and Z Conductivity	(Orthotropic and Single Value) Value is definable in each direction. The local coordinates of the object are used.
X, Y, and Z Conductivity Temperature Chart	(Orthotropic and Table) Click **Click To Edit** to open the X, Y, or Z Conductivity Temperature Chart to specify temperature-dependence of the thermal conductivity.
In Plane Input Method, Normal Input Method	(Biaxial) Defines the biaxial conductivities as: • Single Value − A single value. • Table − A temperature-dependent curve.

Field	Description
In Plane Conductivity	(Biaxial and Single Value for In Plane Input Method) Specifies the thermal conductivity in the plane orthogonal to the local Z axis (that is, in the local X-Y plane).
Normal Conductivity	(Biaxial and Single Value for Normal Input Method) Specifies the thermal conductivity along the local Z axis.
In Plane Conductivity Temperature Chart, Normal Conductivity Temperature Chart	(Biaxial and Table) Click **Click To Edit** to open the In Plane or Normal Conductivity Temperature Chart to specify temperature-dependence of the thermal conductivity.
Electrical Resistivity	Sets the electrical resistivity for the material as Constant or Temperature Dependent.
Resistivity	Constant resistivity value, or the value at a reference temperature if resistivity is temperature-dependent. Joule heating is only calculated when this value is between 1e-30 and 1e+10 Ohm m. For all other values, including zero (the default for new materials), the material is considered to be dielectric.
Coefficient,	(Temperature Dependent resistivity) *Coeff* in the following equation: $\rho(T) = Value + Coeff\,(T - T_{ref})$ where the *Value* is the resistivity at T_{ref}.
T ref	(Temperature Dependent resistivity) The reference temperature, T_{ref}, in the above equation.
T low and T high	(Temperature Dependent resistivity) These values limit the temperature range over which the temperature-dependent resistivity equation (above) applies.
Transparent Material	Switches on transparency.
Transparency Model	Select either Model Type 0 or Model Type1.
Absorption Coeff	(Transparent Material and Model Type 0) The Solar Absorption Coefficient of the material.
Refractive Index	(Transparent Material and Model Type 0) The Refractive Index of the material.
Transmittance	(Transparent Material and Model Type 1) The Transmittance of the material.
Reflectance	(Transparent Material and Model Type 1) The Reflectance of the material.

Field	Description
Density	Sets a constant value for the material density. The default value, 1, should be changed. If it remains unchanged and a transient solution is set then an information message (INFO I/09078) is generated at a sanity check. A separate message is issued for each attached material where the density has been left to the default value, and the material name is output as part of the message. A solution will still be possible.
Specific Heat	Sets a constant value for the material specific heat. The default value, 1, should be changed. If it remains unchanged and a transient solution is set then an information message (INFO I/09078) is generated at a sanity check. A separate message is issued for each attached material where the specific heat has been left to the default value, and the material name is output as part of the message. A solution will still be possible.
Phase Change	Activates calculations for a phase change material (PCM). See "Phase Change Material (PCM)" on page 69.
Liquid Specific Heat	(Phase Change) Sets a constant value for the specific heat when the PCM is in the liquid state. The default value, 1, should be changed. If it remains unchanged and a transient solution is set then an information message (INFO I/09078) is generated at a sanity check. A separate message is issued for each attached material where the liquid specific heat has been left to the default value, and the material name is output as part of the message. A solution will still be possible.
Latent Heat	(Phase Change) The latent heat of fusion of the PCM.
Melt Temperature	(Phase Change) The melting point of the PCM.
Melt Temperature Band	(Phase Change) The temperature band over which melting occurs. The band is centered around the Melt Temperature. The default value for new materials is 1 degC. By default, values less than 0.1 degC are treated as 0.1 degC. You can change this minimum bandwidth value using the FLOPCM_MELT_BAND_MINIMUM environment variable
Surface	Sets the surface conditions of the material (that is, roughness, emissivity, flow resistance, rippling, and color representation in the GDA). **Note:** The direct attachment of a Surface attribute to an object takes precedence over the selections made here (apart from the exception of collapsed cuboids).

Related Topics

Material Properties

Conductivity Temperature Chart

Conductivity Temperature Chart

To access: Open a Material Attribute property sheet, select a Table input method for a thermal conductivity definition, then click the Conductivity Temperature Chart **Click to Edit** button.

Use this chart to set a thermal conductivity versus temperature profile.

Note

The chart name depends on from where it was opened, for example, "X Conductivity Temperature Chart" or "In Plane Conductivity Temperature Chart".

Objects

Field	Description
Temperature	The units of temperature values entered into the table.
Conductivity	The units of thermal conductivity values entered into the table.
Import CSV File	Click to open a file browser to select a valid CSV file.
A single table containing a row for each temperature.	
Temperature	A temperature at which a thermal conductivity value is known.
Conductivity	The thermal conductivity value at that temperature.

Related Topics

Conductivity Temperature Chart CSV File Format

Chapter 7
Radiation Attributes

Radiation attributes can be applied to surfaces of solid objects for when modeling radiation effects.

Radiation Model

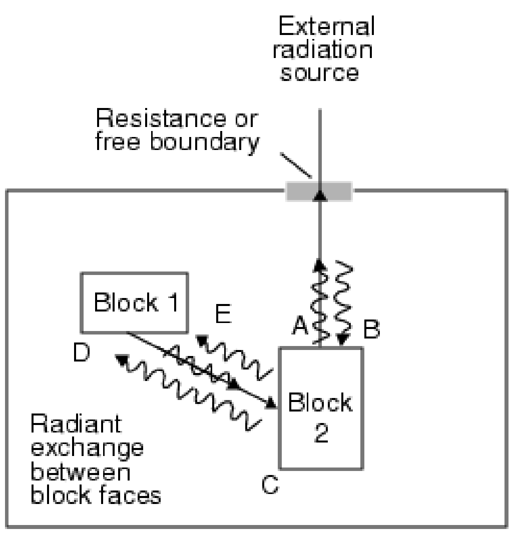
The radiation model defines the surfaces which exchange radiant energy.

In Simcenter Flotherm, a radiant surface can exchange radiant energy with:

- Sources outside of the integration domain, and/or
- Between surfaces of distinct blocks within the integration domain.

The two types of radiative exchange are illustrated in Figure 7-1 where radiation attributes are attached to Faces AB, DE, and AC. Face AB exchanges radiant energy with a remote external radiation source, and Face DE exchanges radiative energy with Face AC.

Figure 7-1. Radiation Model



The grid cells in the solid blocks adjoining the Faces AB, AC, and DE are heated or cooled depending on whether there is a net influx of radiant energy or a net efflux of radiant energy.

External Radiation Sources

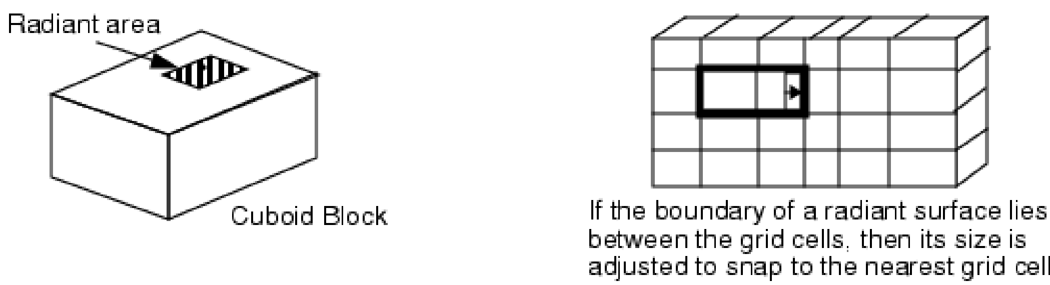
A surface flagged as radiating will exchange radiant energy with the external radiant temperature as set in the Global System Settings section of the **Model Setup** tab if one of the following conditions holds:

- The surface is positioned on the edge of the domain.
- The surface has an unobstructed path between either a free (that is, open) boundary or resistance placed on a domain boundary.

Subdivided Radiation

If the subdivided tolerance is less than any of the grid cells, then the program will create as many subdivisions as there are cells.

Figure 7-2. Subdivided Surface Tolerance



Symmetry and Radiation

For radiation, the Symmetry settings for Faces in the System property sheet **Boundaries** tab mean that every object within the domain has a mirror image on the other side of the symmetry face, and as a result, we consider the radiation between all user defined, radiating, solid objects and all their mirror images. The exchange factors are automatically modified to add the total effect into the analysis.

Radiation Model Theory

Refer to Radiation Model in the *Simcenter Flotherm Background Theory Reference Guide*.

Related Topics

Radiation Attribute Property Sheet

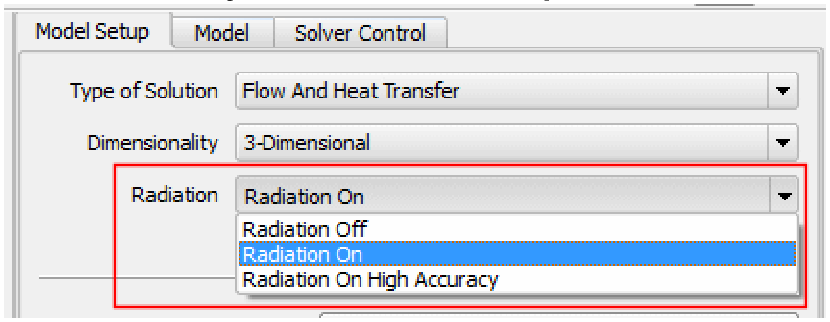
Modeling Radiation Effects

To model radiation, you must switch on radiation modeling, select radiation surfaces and set emissivity values for those surfaces. Radiation effects are not modeled by default.

Procedure

1. Switch on Radiation in the **Model Setup** tab. By default, objects do not radiate in a Simcenter Flotherm model.

Figure 7-3. Radiation Options



2. Attach a Radiation attribute to selected surfaces depending on the location of the sources:

If you want to...	Do the following:
Model radiative exchange with a source outside the integration domain.	1. Select the radiant surface (or face) of the block to interchange radiant energy. 2. Represent the external radiant source by setting its temperature in the Global System Settings section of the **Model Setup** tab.
Model radiative exchange within an enclosure.	• Select the radiant faces of the cuboids expected to exchange radiative energy with one another.

3. Use Surface attributes to set emissivity values for each radiative surface.

Related Topics

Radiation Attribute Property Sheet

Surface Attributes

Radiation Attribute Property Sheet

To access: Select or edit a Radiation attribute.

Use this property sheet to specify the level of detail of modeling radiative effects.

Note

Emissivity values are specified in Surface attributes.

Objects

Field	Description
Name	Identifies the attribute.
Surface	Sets the radiative modeling level: • Non-Radiating — no radiation from the face. • Single Radiating — represent the face as a single radiant face. • Sub-divided Radiating — represent the face as many radiant surfaces. Not available for cylinders, sloping blocks, prisms, tets, and inverted tets.
Subdivided Surface Tolerance	(Sub-divided Radiating) Sets the length of an edge of a radiant surface subdivision, subject to the restriction that a radiant surface may never be smaller than a face of a grid cell.
Minimum Area Considered	Indicates the area of the smallest radiant surface. Radiant surfaces smaller than this value are treated as non-radiating.

Usage Notes

Radiative conditions can be applied to:

- Individual faces of cuboids, prisms, tets, inverted tets, or Heat Sink, PCB, Enclosure, Blocks With Holes, Cylinders, and Sloping Block SmartParts.

Related Topics

Radiation Model

Modeling Radiation Effects

Chapter 8
Resistance Attributes

By default, Resistance SmartParts are open spaces with no resistance to flow, therefore, a Resistance attribute must be attached to the SmartPart to define a resistance to flow.

Pressure Drop Modeling

The formula for modeling the pressure drop across either planar or volume resistance depends on the selection of Velocity type and Treatment.

- Velocity type can be Approach, Device, or Accelerated.
- Treatment can be Standard or Advanced.

The following subsections describe the different models for and how these models are set up in the Resistance attribute property sheet. The density of the fluid is set in the Fluid attribute property sheet, see "Fluid Attribute Property Sheet" on page 50.

Planar Resistances

Examples of planar resistances include grilles, meshes, and other partial obstruction to the flow that does not occupy a significant volume.

Planar resistances can be calculated from Standard or Advanced formulae.

Planar Resistance Standard Treatment

The standard method of calculating the pressure drop for losses across a planar resistance is:

$$\Delta P = \left(\frac{f}{2}\right) \times (density) \times (velocity)^2$$

where:

f is the Loss Coefficient.

density is the Density of the fluid.

velocity is the approach, device or accelerated velocity.

The following are the respective settings in the Resistance Attribute Property Sheet:

- Resistance Type = Planar
- Loss coefficient Based On = Approach Velocity, Device Velocity or Accelerated

- Formula Type = Standard
- Loss Coefficient = the value correlated with the selected velocity.
- When using the Device Velocity or Accelerated options, then the Free Area Ratio should be set to represent the proportion of the modeled area that is actually open in practice.

  For example, if the resistance represents a perforated plate which has holes over 30% of its area, then the Free Area Ratio should be set to 0.3. This is used by the program to calculate the actual velocity (rather than the bulk velocity) and so is able to calculate the correct pressure drop.

Planar Resistance Advanced Treatment

The advanced method of calculating the pressure drop for losses across a planar resistance is deduced as follows:

$$\Delta P = \left(\frac{f}{2}\right) \times (density) \times (velocity)^2$$

but now *f* is modeled by the formula:

$$f = \frac{A}{Re} + \frac{B}{(Re)^{index}}$$

where *Re* is the local Reynolds number calculated from:

$$Re = \frac{[density \times length\ scale \times velocity]}{viscosity}$$

Note: The advanced model reduces to the standard model when:

$A = 0$

$B = f$, where *f* is the desired loss coefficient, and

$index = 0$

The following are the respective settings in the Resistance Attribute Property Sheet:

- Resistance Type = Planar
- Loss coefficient Based On = Approach Velocity, Device Velocity or Accelerated
- Formula Type = Advanced

- Length scale
- A coefficient
- B coefficient
- Index
- For Device Velocity or Accelerated, set the Free Area Ratio to represent the proportion of the modeled area (perpendicular to that direction) that is open.

Examples of Planar Resistances

This example provides for using a planar resistance where the pressure drop is linear with velocity.

If the pressure drop is known to be linear in velocity (for example, laminar flow) then:

$$\Delta P = K \times velocity$$

where *K* is a known loss coefficient for the laminar flow device.

The equation used for the advanced representation by Simcenter Flotherm is:

$$\Delta P = \left(\frac{A}{Re} + \frac{B}{Re^{index}}\right) \times \frac{1}{2} \times density \times velocity^2$$

Equating this with laminar pressure drop gives:

$$K = \left(\frac{A}{Re} + \frac{B}{Re^{index}}\right) \times \frac{1}{2} \times density \times velocity$$

Therefore we set $B = 0$, so now:

$$K = \frac{A}{Re} \times \frac{1}{2} \times density \times velocity$$

Since the velocity and density terms appear in the equation for *Re*, this gives us:

$$A = \frac{2 \times length\_scale \times K}{viscosity}$$

where the length scale is taken from the calculation for the Reynolds Number.

This approach can be used for any known pressure drop/velocity relationship resulting in appropriate values for both A and B.

Setting a Planar Resistance as a Solar Shade

Use the following settings to model a planar resistance as a solar shield.

Procedure

Set the following in the Resistance attribute property sheet:

- Resistance Type = Volume.
- Loss Coefficients Based On = Device Velocity.
- Formula Type = Standard.
- Free Air Ratio = transmittance value.
- Loss Coefficient = 0 (zero). If a non-zero value is entered, then solar shading is not considered.

Related Topics

Resistance Attribute Property Sheet

Volume Resistances

Calculation of volume resistance.

Examples of volume resistances to flow include regions occupied by large numbers of cables and the fins of a heat sink.

Volume resistances can be calculated from Standard or Advanced formulae.

Volume Resistance Standard Treatment

The standard method of calculating the pressure drop for losses across a volume resistance is deduced as follows:

$$\frac{dP}{dL} = \left(\frac{f}{2}\right) \times (density) \times (velocity)^2$$

where:

f is the loss coefficient per unit length.

density is the density of the fluid.

velocity is either the approach, device or accelerated velocity.

The calculation is based on the pressure drop per unit length and can be set separately for each of the three coordinate directions.

The following are the respective settings in the Resistance Attribute Property Sheet:

- Resistance Type = Volume
- Loss coefficient Based On = Approach Velocity, Device Velocity or Accelerated
- Formula Type = Standard
- Direction (one of the coordinates, Xo, Yo, or Zo), and the following for each of the three coordinate directions:
    - Loss coefficient *f* per unit length correlated with the selected velocity for each of the three coordinate directions.
    - For Device Velocity or Accelerated, the Free Area Ratio.

The program will use the density set in the "Fluid Attribute Property Sheet" on page 50.

Volume Resistance Advanced Treatment

The advanced method of calculating the pressure drop for losses across a volume resistance is deduced as follows:

$$\frac{dP}{dL} = \left(\frac{f}{2}\right) \times (density) \times (velocity)^2$$

but now *f* is modeled by the formula:

$$f = \frac{A}{Re} + \frac{B}{(Re)^{index}}$$

where *Re* is the local Reynolds number calculated from:

$$Re = \frac{[density \times length\ scale \times velocity]}{viscosity}$$

The Advanced model reduces to the Standard model when:

- $A = 0$
- $B = f$, where *f* is the desired loss coefficient, and

- *index* = 0

The calculation is based upon the pressure drop per unit length and can be set separately for each of the three coordinate directions.

The following are the respective settings in the Resistance Attribute Property Sheet:

- Resistance Type = Volume
- Loss coefficient Based On = Approach Velocity, Device Velocity or Accelerated
- Formula Type = Advanced
- Direction (one of the coordinates), and the following for each of the three coordinate directions:
    - Length Scale
    - A Coefficient per unit length
    - B Coefficient per unit length
    - Index
    - For Device Velocity or Accelerated, the Free Area Ratio.

A worked example of an Advanced resistance model is available as a download from Support Center, see “Downloading Resistance Examples” on page 89.

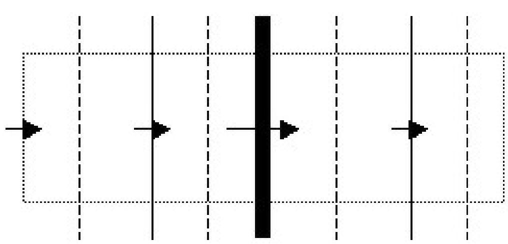
Accelerated Model

The Accelerated model uses an area factor within the calculations to reduce the cell area in the plane of the resistance.

The Accelerated model enables Simcenter Flotherm to account for the acceleration of flow through the resistance and its subsequent deceleration.

Consider a unidirectional flow passing through an accelerated resistance. The accelerated resistance is shown in Figure 8-1 as a thick black line.

Figure 8-1. Unidirectional Flow Through an Accelerated Resistance



The dashed lines represent the boundaries of the momentum cells, and the regular lines represent the boundaries of the pressure cells. Arrows represent flow direction.

The following assumptions are made about the model:

- The resistance has an area factor of $\lambda<1$
- Viscous effects (including turbulence) are negligible.
- The flow is unidirectional with velocity *v*.
- The resistance is zero.

The accelerated model reduces the cell area(s) in the plane of the resistance. Therefore, if the geometrical Area is *A* then the reduced area is $A\lambda$. In this example the normal velocity in and to the plane of the resistance is v/λ. This is designed to simulate the acceleration through the resistance and the subsequent deceleration.

Consider the momentum balance in the momentum cell straddling the resistance. This has to balance the local pressure difference multiplied by the area *A*, therefore, the following equation is true:

$$\rho A v\left[\frac{v+\frac{v}{\lambda}}{2}\right]-\rho A\frac{v}{\lambda}\left[\frac{v+\frac{v}{\lambda}}{2}\right] = A\Delta P$$

Note the use of the upwind areas. This practice ensures conservation of total pressure for unidirectional flows.

Division by *A* and simplification of the above equation produces the following expression for the pressure drop in the first cell downwind of the resistance:

$$\frac{1}{2}\rho v^2\left[1-\frac{1}{\lambda^2}\right] = \Delta P$$

In the second cell downwind of the resistance, pressure instantly recovers to its upstream value. This is somewhat artificial but this method does qualitatively capture the pressure drop downstream of a constriction and the consequent in-flow of fluid from the sides.

When the resistance has a non-zero value pressure differences are additive.

Accelerated Resistances in Fans

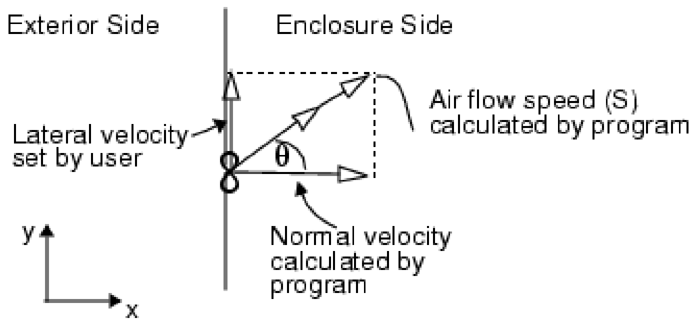
The aforementioned information applies equally to the Fan SmartPart. This functionality is activated when a Free Area Ratio is specified as less than 1 in the Fan property sheet.

External Angled Resistance

When external angled resistance is modeled, the resistance uses the direction cosines of the flow coming into the enclosure.

This is illustrated for the X-Y plane in Figure 8-2.

Figure 8-2. Direction of Angled Resistance



The x-direction cosine is Cos(θ).

The y-direction cosine is Cos(π/2 − θ) = Sin(θ).

For example, property sheet settings of XoN = 0, YoN = 1 and ZoN = 1 are normalized to 0, 1/√2 and 1/√2, that is, 0, Cos 45° and Sin 45°.

The program checks the positive/negative sense of the direction and always sets it to conform with the inward normal.

Downloading Resistance Examples

Examples of resistance generators can be downloaded from the Related Downloads section of Support Center. These worked examples were formerly called Web Parts.

Procedure

1. Log on to Support Center.
2. Go to the Simcenter Flotherm product page.
3. Choose **Downloads** from the menu bar.
4. Click on the **Related Downloads** tab, then select **Simcenter Flotherm Utilities**.

   There are two worked examples for resistances: AdvancedResistance and AngledThickResistance
5. Click on a worked example to save it as a zip file.

To save all of the worked examples, click on **All**.

Resistance Attribute Property Sheet

To access: Select or edit a Resistance attribute.

Use this property sheet to define the attributes of a flow resistance which can be attached to a resistance primitive present in the solution domain.

Objects

Field	Description
Name	Identifies the attribute.
Resistance Type	• Planar — Use this setting when attaching the resistance attribute to a collapsed Resistance SmartPart. This models a resistance to fluid flow across a plane. • Volume — Use this setting when attaching the resistance attribute to an uncollapsed Resistance SmartPart. This models a resistance to fluid flow which is distributed over a finite volume.
Loss Coefficients Based On	The velocity criteria used to define the loss coefficient when calculating the pressure drop across the resistance: • Approach Velocity — the velocity of the fluid as it approaches the resistance. • Device Velocity — the velocity of the fluid through the device. • Accelerated — the velocity of the fluid through the device plus the recoverable pressure drop. This option should be used when the high velocity through the device is likely to cause significant entrainment on exit. Care should be taken to have a refined grid near the exit. See "Accelerated Model" on page 87.
Formula Type	Sets the mathematical model for the resistance. There are two options for both the planar and volume resistance types: • Standard — ΔP is proportional to V^2, that is, for most turbulent flow devices. • Advanced — ΔP is *not* proportional to V^2, for fluid filters and most laminar transitional flows.
Direction	(Volume resistance type) Set the coordinate direction in which the values apply.
Free Area Ratio	(Device Velocity and Accelerated) The ratio of the free area to the total area of the constricting device in the coordinate direction.
Loss Coefficient	(Standard) Refer to the pressure drop formulae described under Planar Resistance Standard Treatment and Volume Resistance Standard Treatment.

Field	Description
A and B Coefficients, Length Scale and Index	(Advanced) Refer to the pressure drop formulae described under Planar Resistance Advanced Treatment and Volume Resistance Advanced Treatment.
External Angled Resistance	Only used if a collapsed resistance is located on an external domain boundary. Sets the direction the incoming flow is turned as it passes through the resistance: • Normal — the angle of flow is normal to the boundary. • Angled — the angle of flow is inclined to the normal.
XoN, YoN, and ZoN	(Angled resistance) The three components of the direction vector which the program normalizes to create the resultant flow direction, see "External Angled Resistance" on page 89.
Transparent to Radiation	Only applicable when the Loss Coefficient value is non-zero, and then the resistance is always opaque to radiation unless this option is checked. The compact heat sink generates a volume resistance which is transparent to radiation and therefore does *not* block radiation.

Usage Notes

If a Volume Resistance attribute is attached to a collapsed Resistance SmartPart, then a Warning is output during a sanity check and the Resistance SmartPart is deactivated by the solver.

Related Topics

Pressure Drop Modeling

Chapter 9
Source Attributes

By default, source objects have no thermal effects. A source attribute must be attached to a source object.

Source Types and Options

The available source types are pressure, velocity (in an axis direction) and temperature.

You can also specify a potential source type if Joule Heating is enabled.

Pressure Sources

You can use a pressure source to activate a mass flow, however, the recommended method of introducing mass flow is to use Fans or Fixed Flow Devices.

When a Pressure source has been set, the other variables, that is, the velocities and the temperature, become subordinate to it.

You have a choice on how a Pressure source is defined, as described below.

Pressure Source Defined by Source/Volume

A prescribed inflow (positive) or outflow (negative) of fluid per unit volume.

Pressure Source Defined by Source/Area

A prescribed inflow (positive) or outflow (negative) of fluid per unit area.

Pressure Source Defined as a Total Source

A prescribed inflow that occurs over the entire plane.

The total value is applied over the Source SmartPart to which the Source attribute is attached. The Source SmartPart may be a volume or a plane (collapsed).

Pressure Source Defined as a Linear Source

The program uses the following equation:

$$source = Coefficient \times (Value - P)$$

where:

source is:

- For planar sources, the linear source of mass flow per unit area of the planar region applied to each grid cell on the active side of the plane.
- For volume sources, the linear source of mass flow per unit volume.

P is calculated by program and is:

- For planar source, the pressure in the grid cells on the active side of the plane.
- For volume sources, the Mean pressure of the volume.

Coefficient is user-defined.

- The value will be multiplied by the volume or the area, depending upon whether the Source attribute is attached to a 3D (No Collapse) or 2D (Collapsed) Source SmartPart. For example, a user-defined value of 1 m s will be applied as 1 s/m^2 to a 3D source, and as 1 s/m to a 2D source.

Value is user-defined, and has the significance of an external pressure.

Related Topics

Source Attribute Property Sheet

Velocity Sources

You can use a velocity source to create a source of momentum.

You have a choice on how a Velocity source is defined, as described below.

Velocity Source Defined by Source/Volume

A source of momentum per unit volume, that is, a force acting on the fluid.

Velocity Source Defined by Source/Area

A source of momentum per unit area, that is, a force acting on the fluid.

Velocity Source Defined as a Total Source

A total force (in Newtons) acting on the fluid at the plane or volume, which the program will distribute uniformly over the plane or volume.

The total value is applied over the Source SmartPart to which the Source attribute is attached. The Source SmartPart may be a volume or a plane (collapsed).

Velocity Source Defined as a Linear Source

The program uses the following equations:

$$source = Coefficient \times (Value - u)$$

$$source = Coefficient \times (Value - v)$$

$$source = Coefficient \times (Value - w)$$

where:

source is:

- For planar sources, the linear source of momentum per unit area of the planar region in the x-, y-, and z-directions respectively.
- For volume sources, the linear source of momentum per unit volume of the volume region in the x-, y-, and z-directions respectively.

The program computes u, v, and w.

Coefficient is user-defined.

- The value will be multiplied by the volume or the area, depending upon whether the Source attribute is attached to a 3D (No Collapse) or 2D (Collapsed) Source SmartPart. For example, a user-defined value of 1 kg/s will be applied as 1 kg/(m^3 s) to a 3D source, and as 1 kg/(m^2 s) to a 2D source.

Value is user-defined. The default is zero which results in a resistance to fluid flow for each velocity resolute linearly proportional to the velocity resolute.

For the velocity resolutes parallel to the plane, the source is applied to the velocities adjacent to the plane on the attached side.

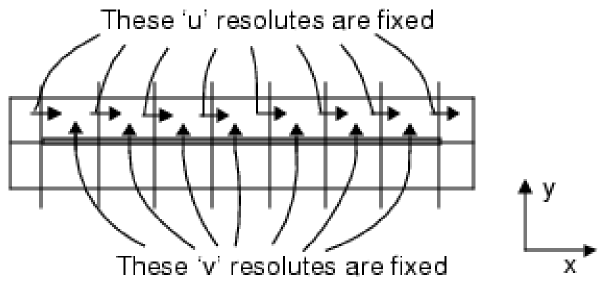
For the velocity resolutes perpendicular to the plane, the source is applied to the in-plane resolutes.

Linear dependence is the correct law of drag for laminar flow. It is also appropriate for the drag law for flow through a porous medium (for example, cotton wool) even when the flow outside is turbulent (D'Arcy's law).

Velocity Source Defined as a Fixed Source

Fixes velocities to the values set for the respective velocity components. Figure 9-1 shows which velocities are fixed for a plane of constant Y when the source is applied to the High Side of the plane.

Figure 9-1. Fixed Value Velocities in the Constant Y Plane



Related Topics

Source Attribute Property Sheet

Temperature Sources

You can use a temperature source to create a source of heat.

You have a choice on how a Temperature source is defined, as described below.

Temperature Source Defined by Source/Volume

A source (positive) or sink (negative) of heat per unit volume.

Temperature Source Defined by Source/Area

A source (positive) or sink (negative) of heat per unit area of the plane.

Temperature Source Defined as a Total Source

A total source of heat applied over the defined plane or volume which the program will distribute uniformly over the plane or volume.

The total value is applied over the Source SmartPart to which the Source attribute is attached. The Source SmartPart may be a volume or a plane (collapsed).

Temperature Source Defined as a Linear Source

The program uses the following equation:

$$source = Coefficient \times (Value - T)$$

where:

source is:

- For planar sources, the linear source of heat per unit area of the planar region, for each grid cell on the attached side.
- For volume sources, the linear source of heat per unit volume.

T is calculated by program and is:

- For planar sources, the temperature in the grid cells in the fluid on the attached side.
- For volume sources, the temperature in the grid cells that the source is active over.

Coefficient is user-defined.

- The value will be multiplied by the *volume* or the *area*, depending upon whether the Source attribute is attached to a 3D (No Collapse) or 2D (Collapsed) Source SmartPart. For example, a user-defined value of 1 W/K will be applied as 1 W/(m^3 K) to a 3D source, and as 1 W/(m^2 K) to a 2D source.

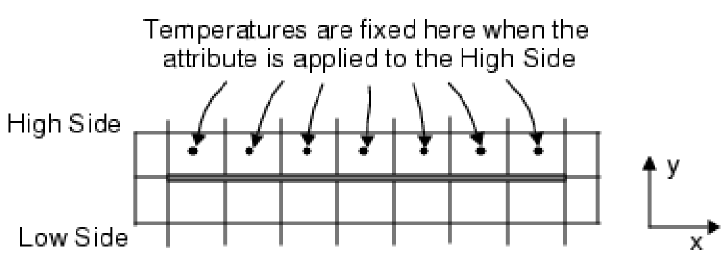
Value is user-defined.

Temperature Source Defined as a Fixed Source

Fixes the temperatures to the set value.

For planar sources, where the Source attribute is applied to a collapsed Source SmartPart on either the High Side or the Low Side, then the temperature is set in the cells adjoining the plane on the respective side of the plane. Figure 9-2 shows the situation of a High Side attachment of the Source attribute.

Figure 9-2. Fixed Value Temperatures for Planar Sources



Temperature Source Defined as a Non Linear Source

See "Defining a Non-Linear Power Variation With Temperature for a Source" on page 103.

Related Topics

Source Attribute Property Sheet

Potential Sources

A Potential source activates a source of electric current. Joule Heating must be enabled.

You have a choice on how a Potential source is defined, as described below. When a current is defined, it can be positive or negative.

Potential Source Defined by Source/Volume

An electric current per unit volume.

Potential Source Defined by Source/Area

An electric current per unit area.

Potential Source Defined as a Total Source

The total amount of electrical current.

The total value is applied over the Source SmartPart to which the Source attribute is attached. The Source SmartPart may be a volume or a plane (collapsed).

Potential Source Defined as a Fixed Source

An electric potential (voltage).

Related Topics

Source Attribute Property Sheet

Application of Transient Functions to Coefficients of Linear Sources

When transient functions are applied to sources, they can affect coefficients of linear sources.

By default, transient functions multiply the value, V_ϕ, of a defined linear source:

$$S_\phi = C_\phi(V_\phi - \phi)$$

The Apply Transient to Coefficient setting in the property sheet also applies the transient function to the coefficient, C_ϕ, thereby applying the transient functions to the total source S_ϕ. This can be useful for some transient applications, for example, Crenel Functions.

Modeling Joule Heating With Potential Source Attributes

Potential Source Attributes provide a means of specifying electrical conditions so that Joule heating can be calculated.

Restrictions and Limitations

- Potential sources must be electrically inter-connected, otherwise error message E/9106 is output.
- There must be at least one Potential source that is a Fixed Value, that is, there must be a fixed voltage boundary condition, otherwise error message E/9107 is output.
- Joule heating is only calculated for materials whose resistivity is between 1e-30 and 1e+10 Ohm m. For all other values, including zero (the default for new materials), the material is considered to be dielectric.

Procedure

1. Switch on Joule Heating at the **Model Setup** tab to enable Source attributes of Potential source types to be defined.
2. Define Potential Source attributes as necessary. Transient attributes can be used to modify the values if required.
3. Add Source SmartParts to the model at locations where the electrical voltage(s) and current(s) are known, and attach the respective Potential Source attributes to these Source SmartParts. Source SmartParts can be collapsed.
4. Ensure that the correct Electrical Resistivity values are defined in Material attributes that are attached to circuit geometry.
5. Optionally, define any required Electrical Resistance to Surface attributes that are used in the model.
6. Ensure that circuit geometry is connected, that is, that there are no “breaks” in the circuit.
7. Optionally, you can:
    - Set a Results vs Distance profile plot to display Potential vs Distance.
    - Add monitor points and select Potential variable values to be monitored in Monitor Points vs Iteration and Monitor Points vs Time (Transient solutions) plots.

    Potential residual curves are created by default when Joule Heating is switched on.
8. Solve the model.

Results

After solving, Joule Heating results data is available in Analyze mode.

- Data available in the GDA:
    - Additional Scalar fields:
        - Potential
        - X Current, the Current Density (Amps per unit area) component in the x-direction.
        - Y Current, the Current Density component in the y-direction.
        - Z Current, the Current Density component in the z-direction.
        - Mag Current, the Current Density magnitude.
        - Joule Heating, the Power Density due to Joule heating.
        - Electrical Resistivity

        Note

        Plots of Electrical Resistivity show dielectrics as having a value of 0 (zero).

    - Additional Vector field, Current Density.

    Tip

    When defining a particle plot in solid materials, use a low value for the Time-Step, for example, between 1e-07 and 1e-09.

- Data available from Tables when in Analyze mode:
    - In the Geometry Model table, Current and Voltage assigned to source objects, and Joule Heating Applied to all electrically conductive solid objects.
    - In the Cuboid Fluxes table, Current In, Current Out, and Current Net for faces of electrically conductive solids.
    - In the Solid Conductors Summary table, cumulative Joule Heat, Current In, Current Out, and Current Net for electrically conductive solids.
    - In Monitor tables, Potential values at each monitor point.

Examples

Figure 9-3 shows an example of a Joule Heating particle plot through solid conductors. The arrows represent Current Density vectors and are colored by Mag Current scalar values.

Figure 9-3. Current Density Vectors Used in a Particle Plot

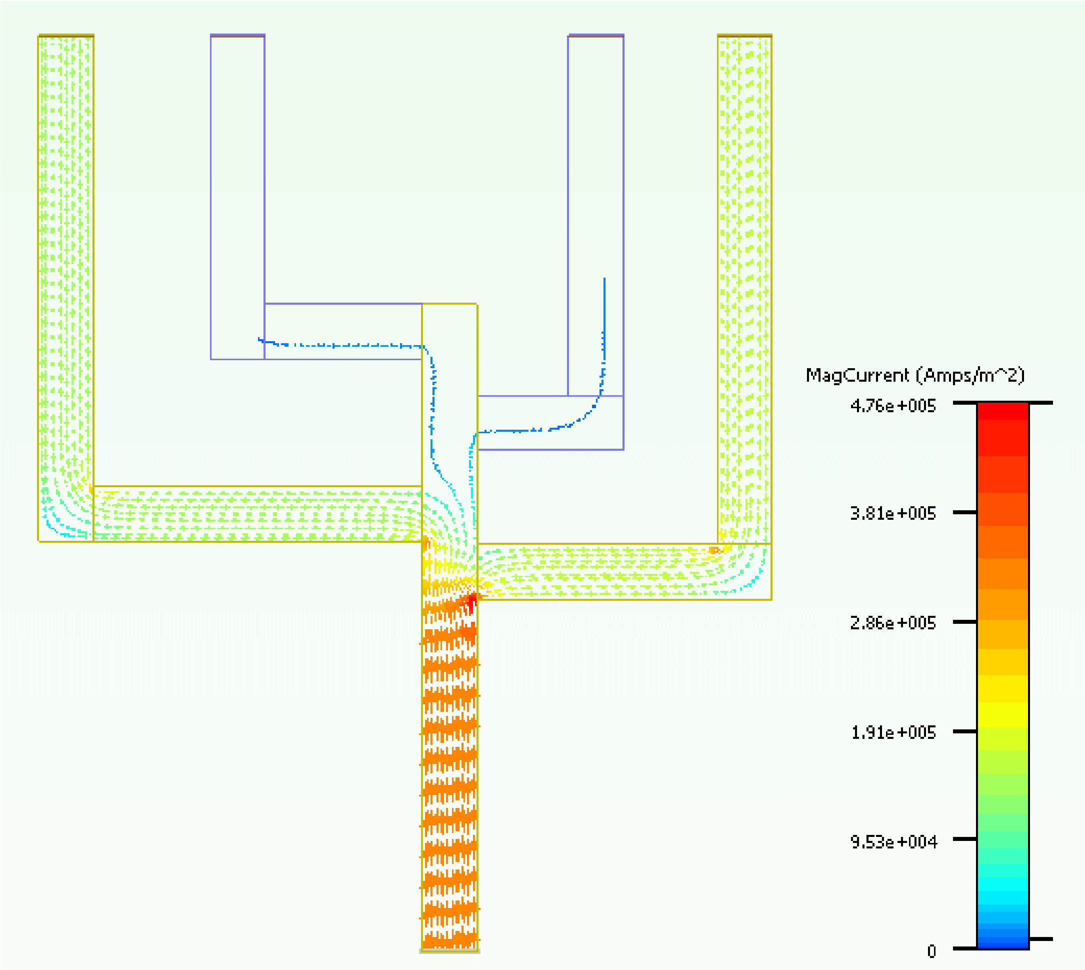
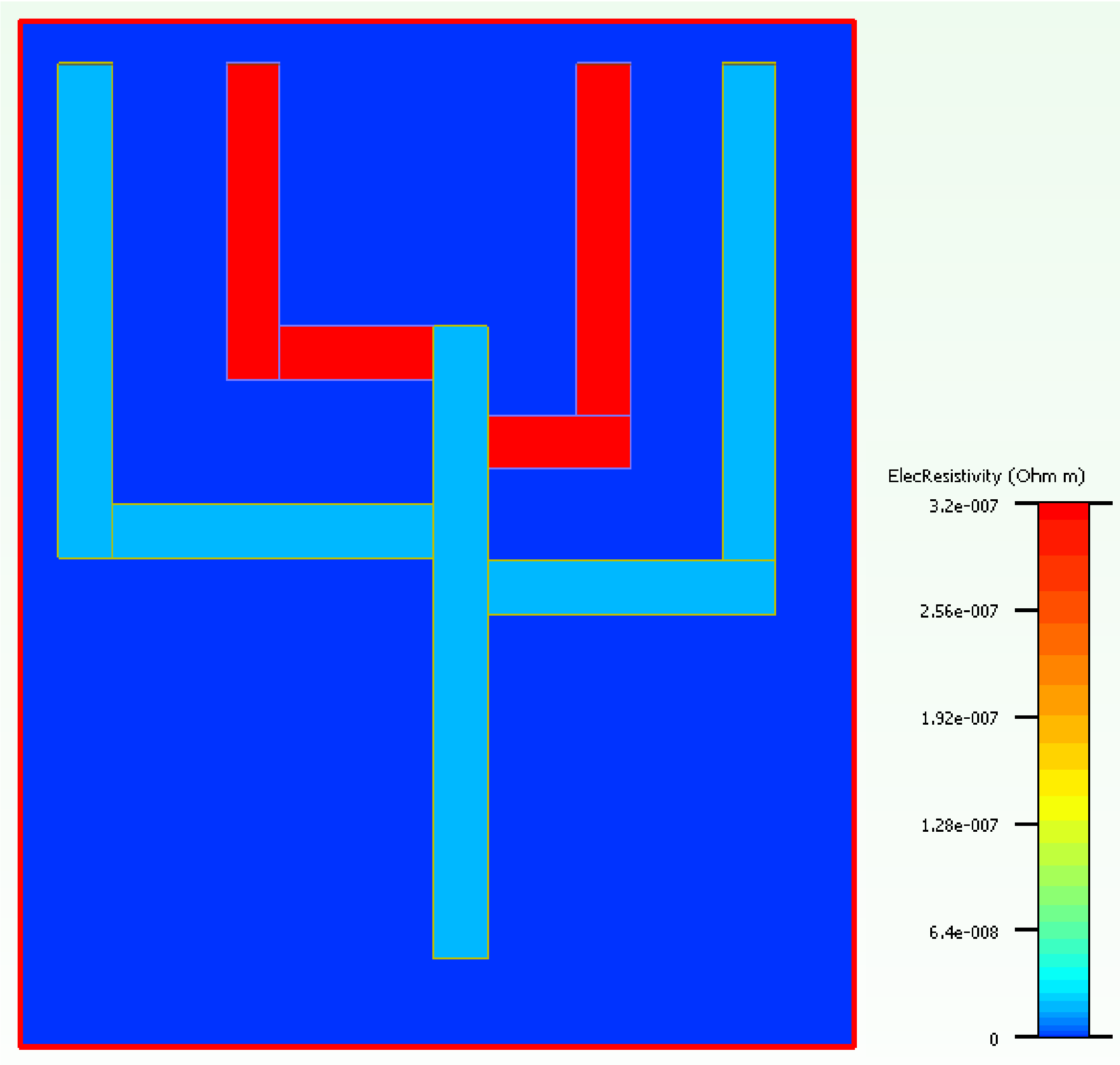


Figure 9-4 shows a planar plot of Electrical Resistivity for the same model. Non-conducting areas (dielectrics) are shown as 0.

Figure 9-4. Electrical Resistivity Plane Plot



Related Topics

Joule Heating Analysis

Materials

Surface Attributes

Defining a Non-Linear Power Variation With Temperature for a Source

A non-linear relationship between power and temperature can be defined either by importing a CSV file or adding points interactively.

Defining a non-linear source is applicable to sub-90 nm technologies where leakage power becomes a significant part of the overall power dissipation of a die. Leakage power is known to be a non-linear function of temperature.

Procedure

1. In the **Attribute Data** tab of the Source Attribute property sheet, make sure that the Activate check box is checked.
2. For Option, select Non Linear Source.
3. Click the Power vs Temperature Chart **Click to Edit** button.

   The Power vs Temperature Chart is opened.
4. Select the units for Temperature and Power.
5. 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. Add values for Temperature and Power. 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**.

6. Click **OK** to save the data and close the Power vs Temperature Chart.

Results

The Power vs Temperature data for the source is linearly interpolated to determine the correct power dissipation on a cell-by-cell basis for the cells that source is active over. Other cases are handled as follows:

- For temperatures falling outside the range of the data pairs, the power is linearly extrapolated from the last chart segment.
- If only one point is defined then that power value is used for all temperatures.
- If no points are defined then a warning is output. If this warning is ignored, the power dissipated is zero.

In the case of transient analyses, a transient function attached to the source is applied to the power dissipation value determined from the curve for each grid cell that the source is active over.

Note

You can edit points after they have been imported by a CSV file, but importing a file deletes any existing points in the chart.

Related Topics

Source Attribute Power vs Temperature Chart

Source Attribute Power vs Temperature Chart CSV File Format

Source Attribute Power vs Temperature Chart CSV File Format

A comma-separated text data file.

Use this file to set up a source attribute power which varies with temperature.

Format

A Power vs Temperature Chart CSV File must conform to the following formatting and syntax rules:

- The units are those currently set by the chart.
- 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.

```
<temperature>,<power_dissipation>
<temperature>,<power_dissipation>
...
```

Parameters

- temperature

  A temperature at which the power dissipation is known.

- power_dissipation

  The power dissipation at the temperature.

Related Topics

Source Attribute Power vs Temperature Chart

Defining a Non-Linear Power Variation With Temperature for a Source

Source Attribute Property Sheet

To access: Select or edit a Source attribute.

Use this property sheet to set prescribed fluxes or fixed values for the variables that you want to apply to the selected source.

Objects

Field	Description
Name	Identifies the attribute.
Source Type	Selects the solution variable to which the source is to be applied: • Pressure • X Velocity • Y Velocity • Z Velocity (3-Dimensional simulations only) • Temperature • Potential (Joule Heating enabled) See “Source Types and Options” on page 94.
Activate	Activates the property sheet options. When deactivated, the effects of the source are removed. By default, all sources are deactivated.
Option	How the solution variable value is defined: • Source/Volume • Source/Area • Total Source • Linear Source (not Potential source types) • Fixed Value (not Pressure source types) • Non Linear (Temperature source types only) See “Source Types and Options” on page 94.
Source/Volume	(Source/Volume) Use with 3D Source SmartParts.
Source/Area	(Source/Area) Use with 2D (collapsed) Source SmartParts.
Total Source	(Total Source) A total value for the source type.
Value	(Linear Source) The *Value* in the appropriate linear equation.
Coefficient	(Linear Source) The *Coefficient* in the appropriate linear equation. **Note:** As displayed, the unit appears to be context insensitive to its final usage, but the value will be multiplied by the *volume* or the *area* of the Source SmartPart, depending upon whether the SmartPart is Not Collapsed or Collapsed.

Field	Description
Apply Transient To Coefficient	(Linear Source) The transient function is applied to the *Coefficient* as well as the *Value* in a linear equation, see “Application of Transient Functions to Coefficients of Linear Sources” on page 99.
Fixed Value	(Fixed Value) A fixed value for the source type.
Power vs Temperature Chart	(Non Linear Source and Temperature) **Click to Edit** opens the Power vs Temperature Chart.
Transient Attribute	Only applicable for transient simulations. You can apply a transient function describing the variation of the source output with time. You can either select an existing transient function or create a new one.

Related Topics

Source Types and Options

Transient Attributes

Source Attribute Power vs Temperature Chart

Modeling Joule Heating With Potential Source Attributes

Source Attribute Power vs Temperature Chart

To access: Open a Source attribute property sheet, click the Activate check box, select an Option of Non Linear Source, then click the Power vs Temperature Chart **Click To Edit** button.

Use this to chart to define or examine the non-linear relationship between power and temperature of the source.

Objects

Field	Description
Temperature	The units of time values entered into the table.
Power	The units of power values 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 value	
Temperature	A temperature at which the power dissipation is known.
Power	The power dissipation at the temperature.

Related Topics

Defining a Non-Linear Power Variation With Temperature for a Source

Source Attribute Power vs Temperature Chart CSV File Format

Source Attribute Property Sheet

Chapter 10
Surface Attributes

Surface attributes are unusual in that they can be attached directly, or indirectly via a Material attribute, depending on the type of object.

Surface Attribute Attachment

Surface attributes include properties that are relevant to thermal modeling, and properties that control the appearance of solid objects when rendered as solids in the drawing board (such properties are purely visual and do not affect the solution).

Surface attribute definitions are applied either:

- Directly, by attachment of a Surface attribute.

  This is the case for solid blocks such as cuboids and compact components. Such Surface attributes can be applied to individual surfaces.

- Indirectly, by attachment of a Material attribute that references a Surface attribute.

  This is the case for heat sinks, PCBs, enclosures, and blocks with holes. The Surface attribute is applied via the Surface field in the Material Attribute Property Sheet. Such Surface attributes cannot be applied to individual surfaces.

Note

Surface finishes attached directly to objects take precedence over surfaces attached via the material properties, the only exception being for collapsed cuboids.

Surface Attribute Properties

Surface attribute properties are generally described in the Surface attribute property sheet description, further detail is provided here.

Roughness

The roughness model used is that appropriate to a roughness element distribution of the sand-grain type.

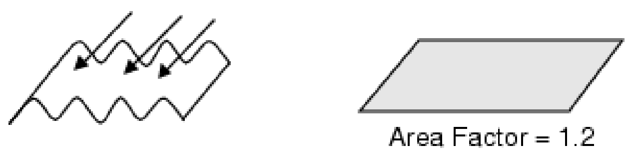
Area Factor

The friction and heat transfer effects of rippled surfaces, with ribs in the direction of the flow, can be specified by setting the Area Factor.

The Area Factor is a multiplier that equates the rippled surface to a flat surface area.

For example, Figure 10-1 shows how the friction and heat transfer effects caused by the rippling can be represented as being equivalent to a planar surface occupying a 20% larger area.

Figure 10-1. Equivalent Area Due to Rippling



You would typically use the Area Factor to represent ribbing on the fins of an extruded finned heat sink where the flow is along the corrugations.

Emissivity

The emissivity of a face can be defined by a Surface attribute, either directly, or by reference from a Material attribute. There is also a default value. The value that is used is determined from the following rules, in order of precedence:

1. If a Surface attribute is directly attached to a face, then the emissivity defined in that attribute is used.
2. If there is no Surface attribute directly attached to a face, then the emissivity value defined in the Surface attribute attached to the object material is used.
3. If there is no Surface attribute directly attached to a face, and no Surface attribute is attached to the object material, then an emissivity value of 1 (one) is used.

Note

If the Radiation attribute for the face has a Surface definition of Non-radiating, then the face will not radiate, regardless of what surface setting is defined for the face.

Solar Reflectivity

Defines how much of the solar energy reaching a surface is reflected instead of being absorbed between the solid-fluid interface.

Solar Reflectivity defined by the Reflectance field in the Material Attribute Property Sheet, when Transparent Material Model Type 1 is modeled, takes precedence. An exception is for a collapsed cuboid, where the Solar Reflectivity field in the Surface attribute property sheet takes precedence over the Reflectance field in the Material attribute property sheet.

Adding a Custom Color for Rendition of Surface Finishes in the Drawing Board

A palette of basic colors is available for selection. If these colors are not suitable, then you can create a new color and add it to the Custom Colors palette.

Procedure

1. Open the Modify Color dialog box.
2. Select an undefined color on the Custom Color palette, unless you want to modify an existing color.
3. Define the selected color. There are three methods:
   - Specifying the Hue/Saturation/Lightness values.
   - Specifying the RGB values.
   - Direct selection from a point in the Color Spectrum. The Lightness of the color is controlled using a slider bar on the right-hand side of the spectrum.
4. Click **Add to Custom Colors**.

   The color of the selected custom color changes to the newly defined color.
5. To save the changes and close the dialog box, click **OK**.

Results

The custom color will be available for selection when the Modify Color dialog box is reopened.

Related Topics

Modify Color Dialog Box

Surface Attribute Property Sheet

To access: Select or edit a Surface attribute.

Use this property sheet to define the property attributes of a surface finish and choose its display representation.

Objects

Field	Description
Name	Identifies the attribute.
Roughness	A measure of roughness, defined as the mean height of the roughness elements present on the surface of the object. By default, the surface is hydrodynamically smooth (zero roughness).
Rsurf-fluid	An additional surface-to-fluid thermal resistance which is added to the thermal resistance calculated by the wall function treatment.
Rsurf-solid	The surface-to-solid thermal resistance.
Emissivity	The surface emissivity required for calculations of radiative exchange in the radiation model to a value between 0 and 1.
Area Factor	A factor to use when modeling rippled surfaces.
Solar Reflectivity	The amount of reflection as a value between 0 (a non-reflecting surface) and 1 (a perfect mirrored surface). Note: this value is ignored in favor of the value of Reflectance defined in the Material attribute property sheet when Transparent Material Model Type 1 is modeled.
Electrical Resistance	Electrical resistance per unit area. This is only used when Joule Heating is switched on and an electrical current passes through the surface during the solution. See “Modeling Joule Heating With Potential Source Attributes” on page 100.
The following fields only define the “Solid” appearance of the surface type as it is displayed in the GDA.	
Color	The representative color for the surface. Colored surfaces provide recognition of different surface types in the model. Click the color box to open the Modify Color dialog box.
Shininess	A value between 0 (matt) and 1 (glossy).
Brightness	A value between 0 (no light reflected) and 1 (all light reflected).

Related Topics

Surface Attribute Attachment

Surface Attribute Properties

Adding a Custom Color for Rendition of Surface Finishes in the Drawing Board

Radiation Model

Modeling Radiation Effects

Solar Radiation [Simcenter Flotherm User Guide]

Modify Color Dialog Box

To access: Click the color box in the Surface Attribute property sheet.

Use this dialog box to add or modify colors for a surfaces with surface attributes when they are displayed in the drawing board.

Objects

Field	Description
Basic Colors palette	Predefined basic colors available for selection.
Custom Colors palette	Custom colors available for selection.
Color Spectrum selector and Lightness slider bar	The spectrum enables point selection of a color. The slider bar varies the lightness of the selected color.
Hue, Sat, Val.	Define a color in terms of its hue, saturation, and lightness.
Red, Green, Blue.	Define a color in terms of its red, green, and blue components.

Related Topics

Adding a Custom Color for Rendition of Surface Finishes in the Drawing Board

Surface Attribute Property Sheet

Chapter 11
Surface Exchange Attributes

Surface Attributes should only be used when you are absolutely certain about the settings being applied and under advice from Mentor Graphics Support.

Use With Care

By default, Simcenter Flotherm uses the wall functions to determine the heat transfer coefficient at the surface of an object as part of the standard solution. Surface exchange attributes overwrite this value by fixing the heat transfer coefficient to a surface or volume based method.

Using surface exchange attributes to specify heat transfer coefficients, and possibly reference temperatures also, is very advanced functionality which is not required in most electronics cooling simulations.

By specifying these parameters, instead of allowing the software to calculate them automatically, is *dangerous* as it pre-supposes a lot about the situation under study. In some cases this is justified but, unless the model very closely matches the set-up used to derive the specified values, it could be inaccurate.

Refer to Wall Functions in the *Simcenter Flotherm Background Theory Reference Guide*.

Modeling Surface Exchanges

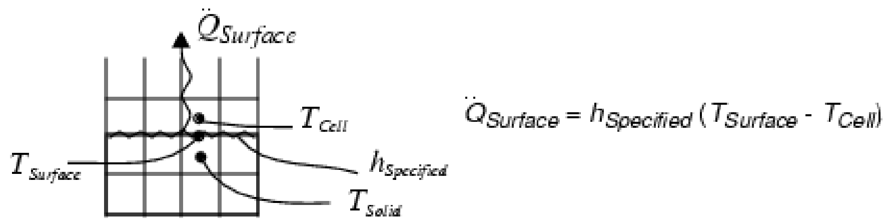
There are two methods of setting the surface exchange attribute: by setting only the heat transfer coefficient and allowing the reference temperature to be calculated (Calculated method), or by setting the heat transfer coefficient *and* a reference temperature (Specified method).

Users should approach fixing the heat transfer coefficient with some caution, however, it is strongly recommended that the Specified method be used rather than the Calculated method if this becomes necessary in a particular modeling situation.

Setting Only the Heat Transfer Coefficient

The Calculated Reference Temperature option uses the Simcenter Flotherm calculated temperature *T*Cell as the reference value.

Figure 11-1. Heat Flux Derived From Calculated Temperature

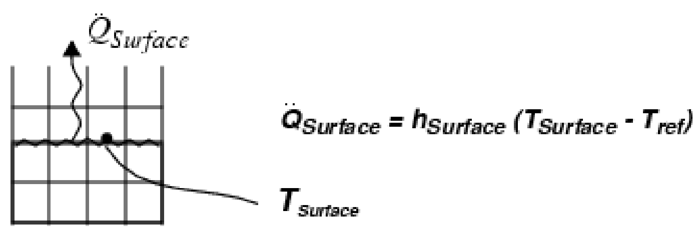


This option will only allow the software to determine the correct heat flux if the specified heat transfer coefficient has originally been calculated using a reference temperature equivalent to the local node value. Since this is rarely the case, Simcenter Flotherm offers the alternative of setting the heat transfer coefficient and the reference temperature.

Setting the Heat Transfer Coefficient and Reference Temperature

The Specified Reference Temperature option enables you to specify both the heat transfer coefficient and the reference temperature originally used to determine the coefficient.

Figure 11-2. Heat Flux Derived From Specified Reference Temperature



Since the reference temperature is normally a remote value well away from the object, this is a more flexible approach than setting only the heat transfer coefficient.

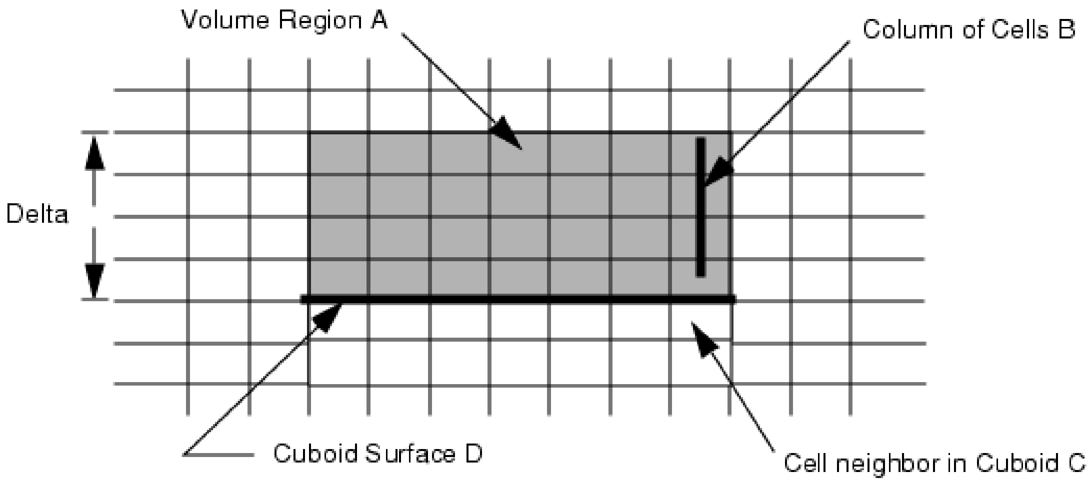
Volume Heat Transfer Method

The volume heat transfer method explained.

The volume heat transfer method is used to simulate heat transfer in a region attached to the surface of a cuboid, for example, the region of fins above a heat sink base.

With reference to Figure 11-3, the volume heat transfer works in a region (A) by linking heat sources between the column of cells (B) above the cuboid surface with the first neighbor cells (C) in the cuboid below the surface.

Figure 11-3. Volume Heat Transfer Method



Volume Heat Transfer Calculations

Volume heat transfer is calculated either using a constant heat transfer coefficient or by the interpolation of heat transfer coefficient from a chart (profile).

Constant Heat Transfer Coefficient

For Constant heat transfer coefficient types, the heat transfer for each cell in the columns above the surface is calculated from:

$$q = h\,A_w\,V\,(T_P - T_{REF})$$

where:

q is the heat transferred to the column of cells B (W)

h is the user specified heat transfer coefficient (W/m^2K)

A_w is the user specified wetted surface area per unit volume (m^{-1}).

T_P is the temperature of the first cell in the cuboid C (°C)

T_{REF} is the Reference Temperature. This is either user-specified or calculated from the cell temperature in the region B.

The corresponding heat source in the cuboid cells is given by:

$$Q = -\sum_{1}^{n} q$$

where n is the number of cells in the column in the region.

Varying Heat Transfer Coefficient

For Specified Profile heat transfer coefficient types, the total heat transfer from the cuboid to the fluid volume is calculated from:

$$Q_{tot} = \frac{T_{mean} - T_{in}}{R}$$

where:

T_{mean} is the mean temperature on the heat sink base.

T_{in} is the mean temperature of the flow entering the heat sink volume.

R is the thermal resistance at the mean speed of flow inside the heat sink volume. R is obtained from the Surface Exchange Chart, see "Defining an HTC Curve of Thermal Resistance vs Airflow Velocity" on page 121.

The heat transfer to individual cells in the heat sink volume is weighted according to the local temperature difference between the cell and the heat sink base:

$q = \alpha V(T_{base} - T_{cell})$

where:

q is the heat transfer to a fluid cell.

T_{base} is the heat sink base temperature below the fluid cell.

T_{cell} is the temperature in the fluid cell.

α is a scaling factor, which is calculated to give the correct total heat transfer, such that:

$$\sum_{1}^{N} q = Q_{tot}$$

As with the Constant Heat Transfer Coefficient, the heat source in the cuboid cells (that is, a cell in the heat sink base) is equal and opposite to the sum of the heat sources in the column of fluid cells above it.

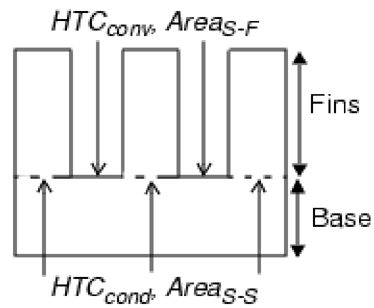
Compact Models

How to obtain a single heat transfer coefficient value, that can be applied to the compact model base cuboid.

The results from a detailed analysis of a heat sink can be used in the surface exchange attribute settings to create a compact model.

The convective and conductive heat transfer coefficients are listed in the Cuboid Fluxes table (a Solid Conductors results table shown in Analyze mode). The heat transfer coefficients are for the base cuboid of the detailed heat sink SmartPart at the interface between the base and the fins, see Figure 11-4.

Figure 11-4. Convective and Conductive Heat Transfer Coefficients at a Heat Sink Base



Use the following equation to obtain a single heat transfer coefficient value, $HTC_{combined}$ which is a weighted average, that can be applied to the compact model base cuboid:

$$HTC_{combined} = \frac{(HTC_{conv} \times Area_{S-F}) + (HTC_{cond} \times Area_{S-S})}{Area_{Base}}$$

where $Area_{Base}$ is the total base area.

Defining an HTC Curve of Thermal Resistance vs Airflow Velocity

A non-linear relationship between thermal resistance and airflow velocity can be defined either by importing a CSV file or adding points interactively.

Procedure

1. In the **Attribute Data** tab of the Surface Exchange Attribute property sheet, select a Heat Transfer Method of Volume and a Volume Coefficient Type of Specified Profile.
2. Click the Surface Exchange Chart **Click to Edit** button.

   The Surface Exchange Chart is opened.
3. Select the units for Velocity and Resistance.
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. Add values for Speed and Resistance. 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 Surface Exchange Chart.

Related Topics

Surface Exchange Chart

Surface Exchange Chart CSV File Format

Surface Exchange Chart CSV File Format

A comma-separated text data file.

Use this file to set up a varying thermal resistance with speed of flow.

Format

A Surface Exchange Chart CSV File must conform to the following formatting and syntax rules:

- The units are those currently set by the chart.
- The list can be any length.
- The values are in floating point format.
- There are no control limits on the start and end values as the internal solver extrapolates values outside the limits of the specified data.
- The file format is two columns of data separated by a comma.

```
<velocity>,<thermal_resistance>
<velocity>,<thermal_resistance>
...
```

Parameters

- velocity

  A velocity at which the thermal resistance is known.

- thermal_resistance

  The thermal resistance at that velocity.

Related Topics

Surface Exchange Chart

Defining an HTC Curve of Thermal Resistance vs Airflow Velocity

Surface Exchange Attribute Property Sheet

To access: Select or edit a Surface Exchange attribute.

Use this property sheet to attach a heat transfer coefficient to objects.

Description

Objects that can have heat transfer coefficients attached using this attribute include cuboids, prisms, tets, inverted tets, heat sinks, sloping blocks, enclosures, and blocks with holes.

The heat transfer coefficient can be attached either to the entire object or to individual surfaces.

Caution

This property sheet is not required for most electric cooling simulations and should only be used under advice from Mentor Graphics Mechanical Analysis Support.

Objects

Field	Description
Name	Identifies the attribute.
Heat Transfer Method	• Surface — the heat is transferred from solid surfaces. • Volume — the heat is transferred from a solid to a volume representing a complicated shape, for example, heat sink pins/fins.
Surface Heat Transfer Method	
Surface Coefficient Type	• Calculated — calculated by Simcenter Flotherm. • Specified — user-specified in the Specified Coefficient field.
Reference Temperature	(Specified Surface Coefficient Type) • Calculated — calculated by Simcenter Flotherm. • Specified — user-specified in the Temperature field.
Volume Heat Transfer Method	
Extent of Heat Transfer Delta	The height of the region above the base surface.
Volume Coefficient Type	The heat transfer coefficient can be specified as either Constant or Specified Profile.
Constant Coefficient	(Constant) A user-specified heat transfer coefficient, that is, h in the formula: $q = h\, A_w\, V\, (T_P - T_{REF})$
Wetted Area/ Volume Transfer Ratio	(Constant) The wetted surface area of the solid material per unit volume in the volume region, that is, A_w in the formula above.

Field	Description
Reference Temperature	(Constant) Specifies how the reference temperature is obtained. • Calculated — Simcenter Flotherm uses the local air temperatures for the reference value in determining the heat flux. • Specified.
Temperature	(Constant and Specified) A user-specified reference temperature.
Surface Exchange Chart	(Specified Profile) Click **Click To Edit** to open the Surface Exchange Chart to define a profile of resistance varying with speed.

Related Topics

Modeling Surface Exchanges

Volume Heat Transfer Method

Surface Exchange Chart

Volume Heat Transfer Calculations

Surface Exchange Chart

To access: Open a Surface Exchange attribute property sheet, select a Heat Transfer Method of Volume and a Volume Coefficient Type of Specified Profile, then click the Surface Exchange Chart **Click To Edit** button.

Use this to chart to define or examine the coefficient profile varying with speed.

Description

Use this feature for modeling complex heat sink designs in a compact way by directly using data that has been determined experimentally or analytically from spreadsheets or other specialized heat sink design tools. For simpler heat sink designs, there is an automatic compact modeling option for the Heat Sink SmartPart.

Take care to specify a physically realistic curve, since this chart effectively prescribes the amount of heat added to the fluid at a given flow rate through the heat sink volume.

If the thermal resistance is unrealistically low, it is possible to reach a situation where the amount of heat transferred into the fluid is more than can physically be removed by the flow through the heat sink volume. This may be indicated by the fluid temperature in the heat sink volume reaching or exceeding the heat sink base temperature, and will result in non-convergence of the solution.

Outside the limits of the specification data, the solver extrapolates from the first and last points with an assumed profile of resistance inversely proportional to speed. If this extrapolation is likely to result in an unphysical resistance, then you should ensure that the data covers the full range of speeds which might be encountered.

Objects

Field	Description
Velocity	The units of velocity values entered into the table.
Resistance	The units of thermal resistance values 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 resistance value	
Speed	A velocity at which the thermal resistance is known.
Resistance	The thermal resistance at that velocity.

Related Topics

Defining an HTC Curve of Thermal Resistance vs Airflow Velocity

Surface Exchange Chart CSV File Format

Surface Exchange Attribute Property Sheet

Chapter 12
Thermal Attributes

By default, cuboids, prisms, tets, inverted tets, cylinders, sloping blocks, enclosures, and blocks with holes are non-conducting. To thermally model these objects, you must attach thermal attributes.

Thermal Models

When thermal attributes are attached to objects they can represent the heat transfer from the object according to different models.

Fixed Temperature

The surface temperature of the object is set to a constant value. In addition, the in-block temperatures are also set to this value.

For Fixed Temperature thermal models, the heat transfer from the object to the fluid is calculated as:

$$H\,(T_{surface} - T_{fluid})$$

where:

T_{fluid} is the temperature of the fluid adjacent to the point in question.

$T_{surface}$ is the user set temperature which applies over the six faces of the block.

H is the heat transfer coefficient determined by two methods:

- The program calculates H by default from the wall functions, see Wall Functions in the *Simcenter Flotherm Background Theory Reference Guide*.
- The alternative is to set H directly using the Surface Exchange Attribute Property Sheet, which might be appropriate if the surface has some special features (for example, ribs or fins) for which an empirical value for H is known.

Fixed Heat Flow

A fixed flux of heat is set to flow uniformly into or out of all the faces of the block. In this thermal model, the in-block temperatures are not calculated and, therefore, remain at their initial values. If required, initial values for the block can be set using the Variable Solution Control section of the **Solver Control** tab.

Conduction

The temperature distribution within the block and the heat transfer through it are calculated. Such a calculation requires the user to set the material properties of the block, namely its thermal conductivity and, for time dependent calculations, its specific heat capacity and density. There is no Total Power option with conduction for enclosures.

Availability of Thermal Models

Not all thermal models are available to all objects.

Table 12-1. Thermal Modeling Options

Object	Heat Transfer			Transient Function
	Fixed Temperature	Fixed Heat Flow	Conduction	
Cuboid	Yes	Yes	Yes	Yes
Prism	Yes	Yes	Yes	Yes
Tet	Yes	Yes	Yes	Yes
Inverted Tet	Yes	Yes	Yes	Yes
Cylinder	Yes	No	Yes	Yes
Sloping Block	Yes	No	Yes	Yes
Enclosure	Yes	No	Yes[1]	No
Block with Hole	Yes	No	Yes	Yes

1. "Total Power" cannot be defined.

Related Topics

Thermal Attribute Property Sheet

Thermal Attribute Property Sheet

To access: Select or edit a Thermal attribute.

Use this property sheet to define the thermal modeling of solid objects, sloping blocks, enclosures, and blocks with holes.

Objects

Field	Description
Name	Identifies the attribute.
Thermal Model	Select a thermal model, see "Thermal Models" on page 127.
Total Power	(Conduction) The temperature distribution within the object and the heat transfer through it are calculated, see "Conduction" on page 128.
Fixed Temperature	(Fixed Temperature) Fixes the temperature of the attached object to be constant.
Power Specification	(Fixed Heat Flow) Select Power/Area or Total Power.
Power/Area	(Fixed Heat Flow and Power/Area) A fixed heat flow defined as a power across an area.
Total Power	(Fixed Heat Flow and Total Power) A fixed heat flow defined as a total power.
Transient Attribute	Only applicable when running transient solutions. Enables a transient function to describe the thermal output.

Related Topics

Thermal Models

Transient Attributes

Chapter 13
Transient Attributes

A Transient attribute can be set to vary with time, with temperature or with both.

Transient Attribute Attachment

Transient attributes are commonly attached to another attribute which they then modify.

The following are the attributes to which a transient can be attached:

- Thermal Attributes, where they are used to vary the thermal property.

- Ambient Attributes, where they are used to vary the ambient and/or radiant temperature.
- Source Attributes, where they are used to vary the value of a source.

An exception is that a transient attribute can be attached directly to a Fan SmartPart, where it varies the derating factor.

Transient attributes only take effect when a transient solution is run.

A Transient attribute acts as a multiplier on the attribute to which it is attached. If no multiplier is defined for a time period, irrespective of the location (start, middle, or end) during the total time period, then a multiplier of 0 (zero) is used.

Related Topics

Using Multipliers

Transient Variation With Time

Transient Variation With Temperature

Transient Analysis [Simcenter Flotherm User Guide]

Transient Variation With Time

For transient attributes that vary with time, you can define the variation as a profile or by function, where the function is created by combining instances of standard sub-function templates.

- Variation Defined by Profile

  Data points are defined from which a variation plot is interpolated.

- Variation Defined by Function

Related Topics

Transient Functions and Sub-Functions

Transient Functions and Sub-Functions

Functions are created by combing sub-functions. A set of mathematical sub-functions is provided.

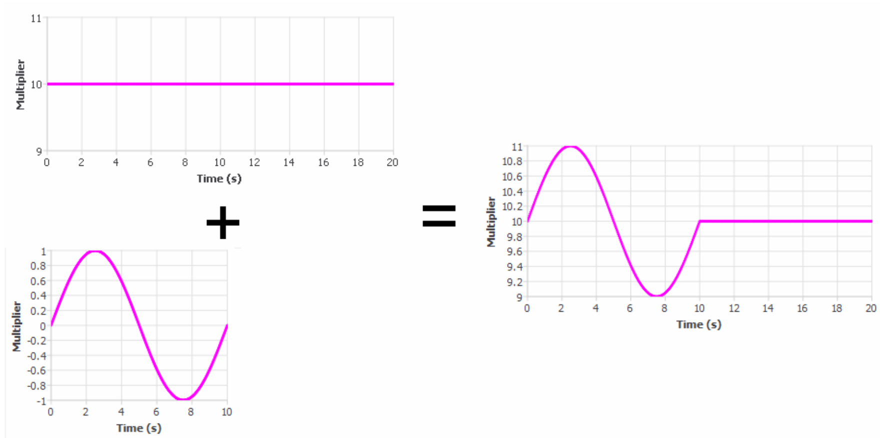
Sub-Function Overlap Combinations

When more than one sub-function is defined over the same time interval they are combined by addition or by multiplication.

- Addition of Sub-Functions

  Figure 13-1 shows the resultant function produced by the addition of a linear sub-function and a sinusoidal sub-function.

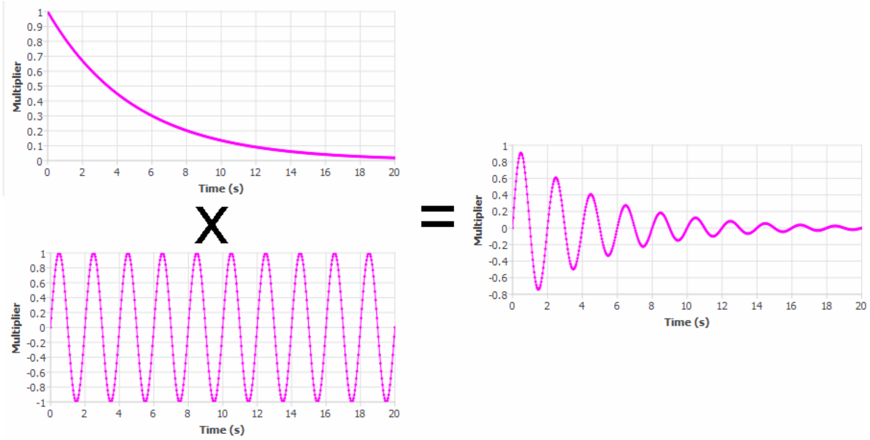
Figure 13-1. Addition of Sub-Functions



- Multiplication of Sub-Functions

  Figure 13-2 shows the resultant function produced by the multiplication of an exponential sub-function and a sinusoidal sub-function.

Figure 13-2. Multiplication of Sub-Functions



Note

Profiles defined by CSV files cannot be added or multiplied.

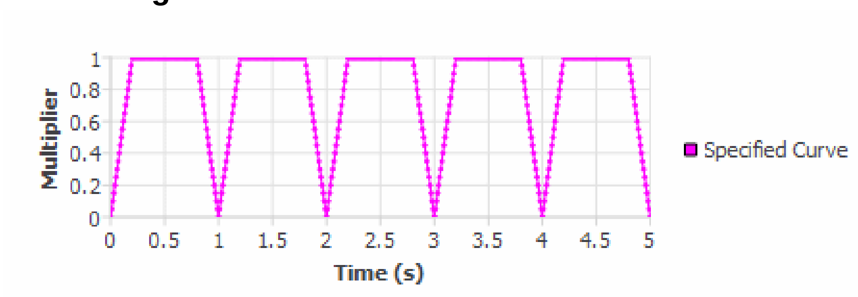
Periodic Functions

Periodic functions are defined by the Start Time of the period (T start) and the End Time of the period (T end).

Figure 13-3 shows a repeated pulse sub-function where T start = 0 and T end = 1 s.

The plot in the Transient Time Chart (View mode) or Function Time Chart defaults to show five cycles for the defined periodic function and does not show the cycles in relation to the defined overall length of the transient.

Figure 13-3. Periodic Pulse Sub-Function

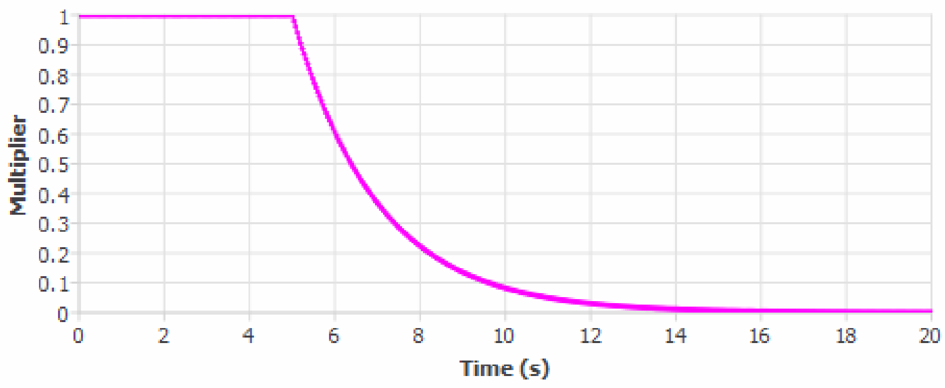


Sequential Functions

Transient function attributes can be made up of any number of elementary functions which may follow sequentially or overlap.

For example, modeling the switching off of a heat source, could be represented by the sequence shown in Figure 13-4.

Figure 13-4. Sequential Linear Followed by Exponential Decay



Transient Sub-Functions

The start and end points of all defined transient sub-functions are keypointed in the time grid to provide an accurate description of the overall function.

Linear Sub-Function

A linear sub-function is defined by:

$$F = m_0 + a\,(T - T_0)$$

where m_0 is the Baseline Value, a is a Coefficient and T_0 is the Baseline Time.

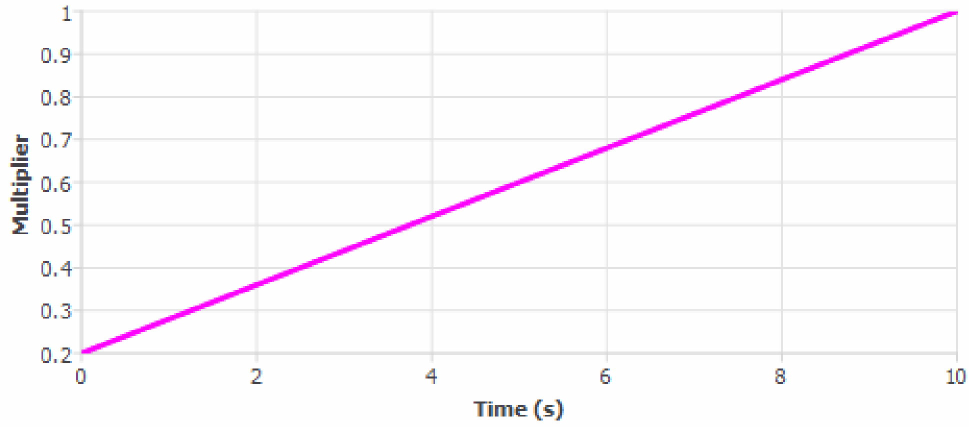
Figure 13-5 shows a linear sub-function, defined from 0 s to 10 s, with parameters:

Baseline Value = 0.2

Baseline Time = 0 s

Coefficient = 0.08 s^{-1}

Figure 13-5. Linear Sub-Function



Power Law Sub-Function

A power law sub-function is defined by:

$$F = m_0 + (a\,(T - T_0))^i$$

where m_0 is the Baseline Value, a is a Coefficient, T_0 is the Baseline Time and i is the Power Index.

Figure 13-6 shows a power law sub-function, defined from 0 s to 10 s, with parameters:

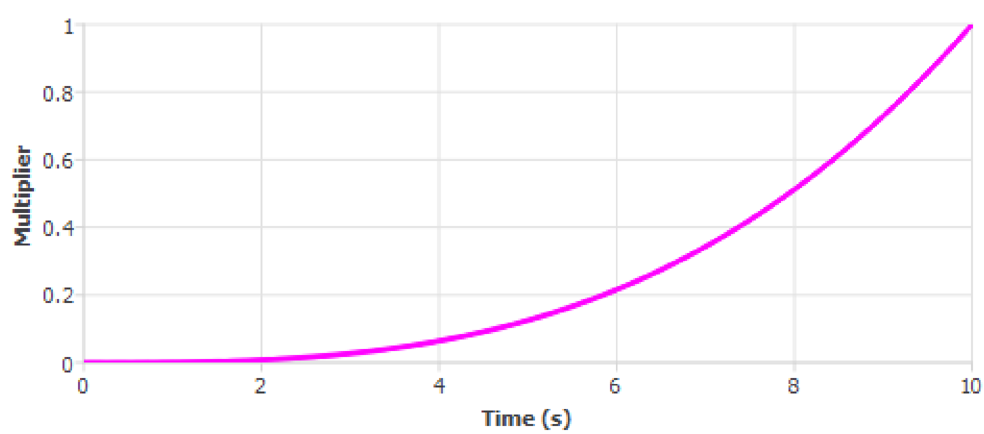
Baseline Value = 0

Baseline Time = 0 s

Coefficient = 0.1 s^{-1}

Power Index = 3

Figure 13-6. Power Law Sub-Function



Exponential Sub-Function

An exponential sub-function is defined by:

$$F = m_0 + (B \cdot \exp(A(T - T_0)))$$

where m_0 is the Baseline Value, A and B are coefficients and T_0 is the Baseline Time.

Figure 13-7 shows an exponential sub-function, defined from 0 s to 10 s, with parameters:

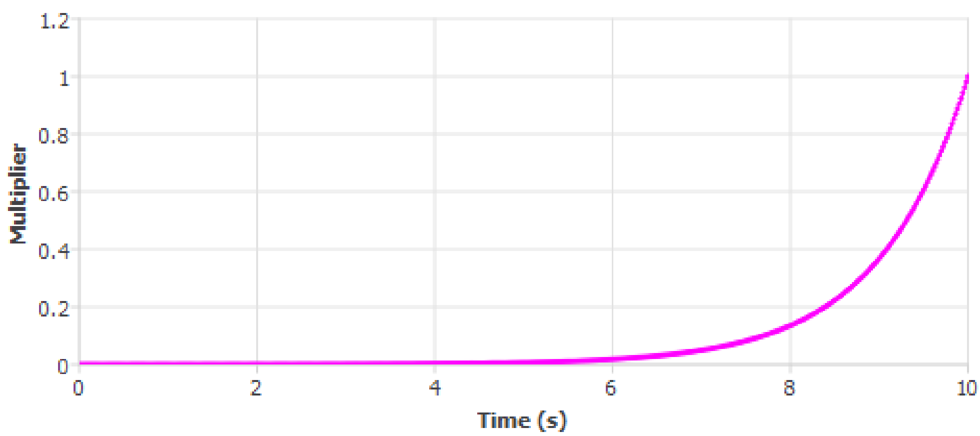
Baseline Value = 0

Baseline Time = 0 s

A Coefficient = 1.0 s^{-1}

B Coefficient = 0.0000454

Figure 13-7. Exponential Sub-Function



Sinusoidal Sub-Function

A sinusoidal sub-function is defined by:

$$F = m_0 + A \cdot \sin\left(\frac{T - T_0}{P} \cdot 2\pi\right)$$

where m_0 is the Baseline Value, A is the amplitude Coefficient, T_0 is the Baseline Time and P is the Period. Refer to Figure 13-8.

Figure 13-8. Sinusoidal Sub-Function Parameters

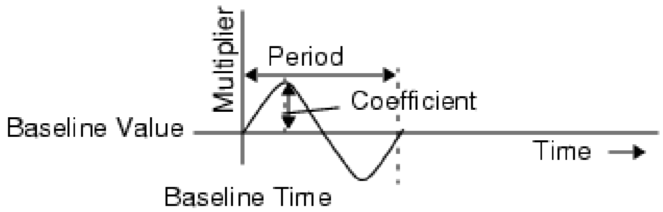


Figure 13-9 shows a sinusoidal sub-function, defined from 0 s to 10 s, with parameters:

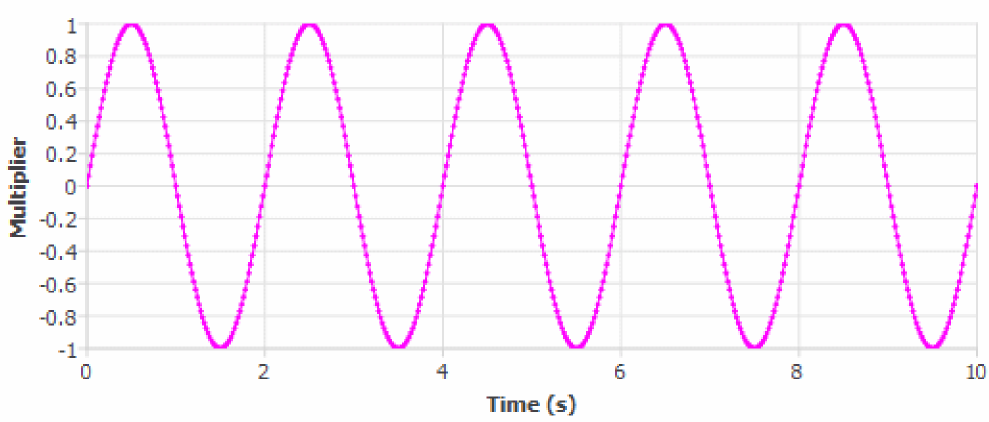
Baseline Value = 0

Baseline Time = 0 s

Coefficient = 1.0

Period = 2 s

Figure 13-9. Sinusoidal Sub-Function



Gaussian Sub-Function

A Gaussian sub-function is defined by:

$$F = A \cdot \exp\left(-\left(\frac{T-B}{C}\right)^2\right)$$

where A, B and C are coefficients, refer to Figure 13-10.

Figure 13-10. Gaussian Sub-Function Parameters

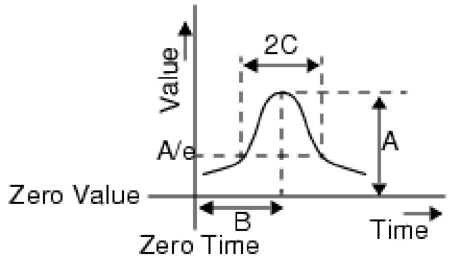


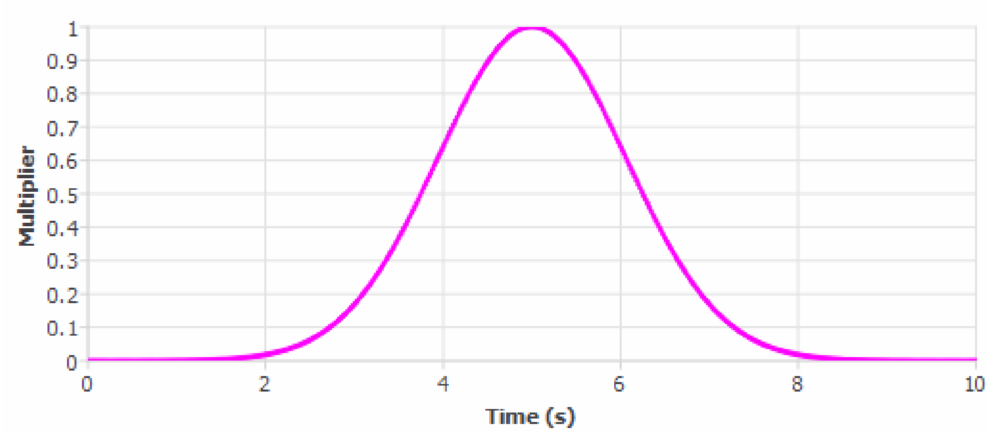
Figure 13-11 shows a Gaussian sub-function, defined from 0 s to 10 s, with parameters:

A Coefficient = 1

B Coefficient = 5 s

C Coefficient = 1.5 s

Figure 13-11. Gaussian Sub-Function



Pulse Sub-Function

A Pulse sub-function is defined by:

- Amplitude — the maximum multiplier value.
- Rise Time — the duration of the rising part of the pulse.
- High Time — the duration of the flat (top) part of the pulse.
- Fall Time — the duration of the falling part of the pulse.

Figure 13-12 shows a pulse sub-function, defined from 0 s to 5 s, with parameters:

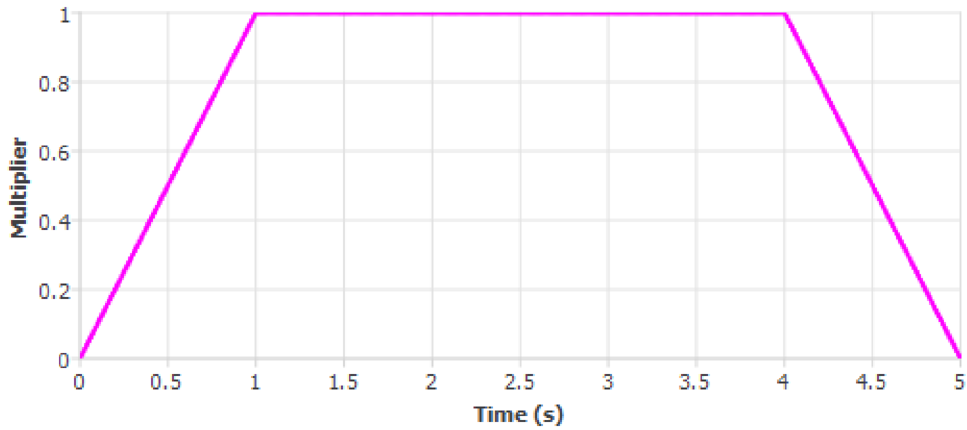
Amplitude = 1

Rise Time = 1 s

High Time = 3 s

Fall Time = 1 s

Figure 13-12. Pulse Sub-Function



Double Exponential Sub-Function

A double exponential sub-function is defined by:

$$F = A\left(\exp(-\frac{t}{B}) - \exp(-\frac{t}{C})\right)$$

where A is the asymptotic amplitude, B is the rise time and C is the decay time.

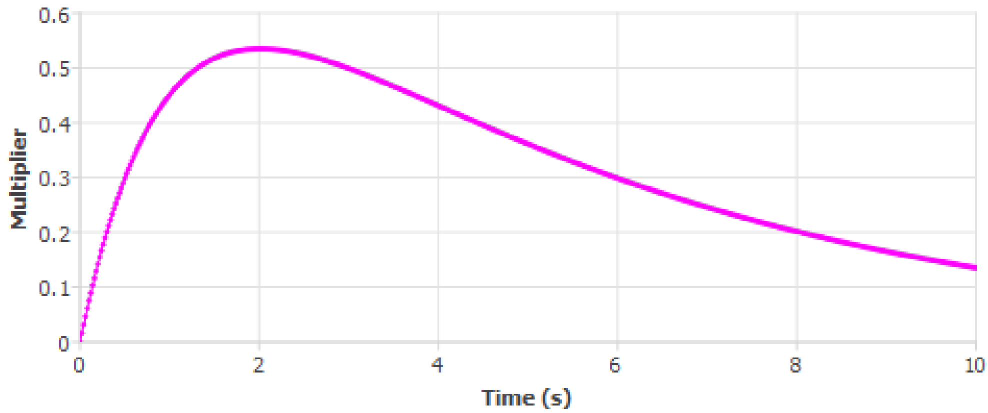
Figure 13-12 shows a double exponential sub-function, defined from 0 s to 10 s, with parameters:

Amplitude = 1

Decay = 1 s

Rise = 5 s

Figure 13-13. Double Exponential Sub-Function



Transient Variation With Temperature

Variations with temperature can be used to model systems whose power is derated when a component or case exceeds a specified temperature.

A variation with temperature and a variation with time can be both active. Such a setup can be used to model systems where power output is not constant over time and is derated if a temperature threshold is reached.

Variations with temperature are defined as profiles.

A monitor point is used to monitor the temperature at a particular location, for example, on a component or the surface of a case.

Related Topics

Defining a Transient Attribute Profile That Varies With Temperature

Transient Attribute Property Sheet Multiplier vs. Temperature Tab

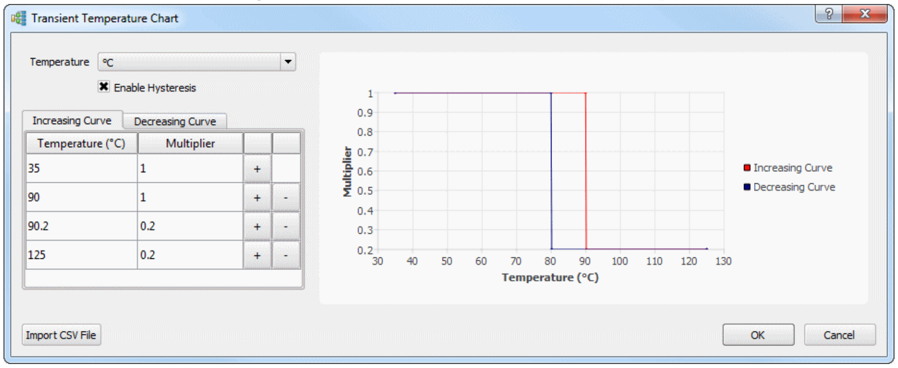
Hysteresis Loops

The use of hysteresis loops enables more realistic temperature control methods, commonly used in electronics, to be simulated.

Hysteresis is normally employed in a beneficial way to dampen dynamic systems that can become oscillatory. An example is in the case of thermostatic control, where, if you did not employ hysteresis, thermostats can switch on and off unnecessarily for small changes in temperature, shortening their own lives and adding a cost overhead by switching power on and off.

You can separately define responses to increasing and decreasing temperature to create a hysteresis loop, see Figure 13-14.

Figure 13-14. Hysteresis Loop



Caution

When initializing with results from a previous transient solution set, the available history data must be sufficient to determine the correct point on the hysteresis loop, that is, whether the temperature is increasing or decreasing. If there are not *a minimum* of two time step values available in the initializing solution set, then an error message will be generated.

Related Topics

Defining a Hysteresis Loop That Varies With Increasing and Decreasing Temperature

Transient Temperature Chart

Transient Attribute Procedures

Use these procedures to define how transient multipliers vary with time or temperature.

Using Multipliers

Although you can combine the attribute value and the multiplier in any number of ways, the simplest method is described.

Procedure

1. Set the value of the attribute to which the transient will be attached to be the maximum required.

   For example. if you want to vary a Thermal attribute up to a maximum of 5 W, then set the Thermal attribute to 5 W.

2. When you define the Transient attribute, make sure that the multiplier has a maximum value of 1.

Results

In this example, when the Multiplier is 0.5, the Thermal attribute will have a value of 2.5 W.

Alternatively, you could have set up the Thermal attribute to be a maximum of 1 W and set up a maximum multiplier of 5; however, the Transient attribute would be more difficult to use with other Thermal or Source attributes as you would have to factor in the ×5 multiplier.

There is a constraint on the value of each transient function of between -1e+15 and 1e+15.

Defining a Transient Attribute

Transients can vary with time or with the temperature from a monitor point.

Procedure

1. Create a new or open an existing Thermal, Source, or Ambient attribute.

2. At the Transient Attribute (or Ambient Transient) field, select **Create New**.

   A new Transient attribute is opened at the default **Multiplier vs. Time** tab.

3. Either leave the **Multiplier vs. Time** tab open to define a time-varying transient, or open the **Multiplier vs. Temperature** tab to define a temperature-varying transient.

4. Check Activate.

   More options open in the property sheet.

5. For a time-varying transient, select either Profile or Function as the Type depending on whether you want to specify the transient as a profile, that is, by defining points on a curve, or as a mathematical function. See Defining a Transient Attribute Profile That Varies With Time or Defining a Transient Attribute Function.

6. For a temperature-varying transient, select an Associate Monitor Point that will provide temperature values as input to the transient. If a suitable monitor point does not currently exist then you can add one and return to the list in the transient property sheet to make your selection.

   Temperature-varying transients can only be defined as a profile, see Defining a Transient Attribute Profile That Varies With Temperature.

Defining a Transient Attribute Profile That Varies With Time

You can specify data points either by direct entry in a chart table or by importing the point values from CSV files.

Procedure

1. In the **Multiplier vs. Time** tab of the Transient Attribute property sheet, select a Type of Profile.

2. Click the Transient Time Chart **Click to Edit** button.

   The Transient Time Chart is opened in Edit mode.

3. Select the units for Time.

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. Add values for Time and Multiplier. 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 Transient Time Chart.

6. To repeat the profile, see "Defining a Periodic Transient Attribute" on page 148.

Related Topics

Transient Attribute Property Sheet

Transient Time Chart CSV File Format

Defining a Transient Attribute

Using Multipliers

Transient Time Chart

Defining a Transient Attribute Profile That Varies With Temperature

You can specify data points either by direct entry in a chart table or by importing the point values from CSV files.

Procedure

1. In the **Multiplier vs. Temperature** tab of the Transient Attribute property sheet, make sure that the Activate check box is checked.

2. Click the Transient Temperature Chart **Click to Edit** button.

   The Transient Temperature Chart is opened.

3. Select the units for Temperature.

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

If you want to...	Do the following:
Enter data manually.	1. Add values for Temperature and Multiplier. 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 Transient Temperature Chart.

Related Topics

Transient Attribute Property Sheet

Using Multipliers

Defining a Transient Attribute

Transient Temperature Chart

Transient Temperature Chart CSV File Format

Defining a Hysteresis Loop That Varies With Increasing and Decreasing Temperature

Two profiles are created by fitting curves to two supplied sets of data points.

Procedure

1. In the **Multiplier vs. Temperature** tab of the Transient Attribute property sheet, make sure that the Activate check box is checked.
2. Click the Transient Temperature Chart **Click to Edit** button.

   The Transient Temperature Chart is opened.
3. Select the units for Temperature.

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

If you want to...	Do the following:
Enter data manually.	The simplest method is to enter one set of values (for example, the increasing curve set) as you would for a non-hysteresis curve, then, when you enable hysteresis, both curve tables will be populated with the same values. 1. Make sure that the Enable Hysteresis check box is unchecked and add values for Temperature and Multiplier as described in "Defining a Transient Attribute Profile That Varies With Temperature" on page 145. 2. Check the Enable Hysteresis check box. Note that both the Increasing Curve and Decreasing Curve tables are loaded with the values entered. 3. Edit the values in the table(s) as required.
Enter data by CSV file import.	1. Check the Enable Hysteresis check box. 2. Select the relevant table tab: Increasing Curve or Decreasing Curve. 3. Click **Import CSV File**. 4. Navigate to, and select, a valid CSV file and click **Open**. 5. Repeat for the other table tab.

5. Check that the hysteresis loop is correct.

6. Click **OK** to save the data and close the Transient Temperature Chart.

Related Topics

Hysteresis Loops

Transient Temperature Chart

Transient Temperature Chart CSV File Format

Defining a Transient Attribute Function

Functions can be simple, single subfunctions or complex combinations of subfunctions.

Prerequisites

- Functions can only be applied to time-varying transients. The **Multiplier vs. Time** tab of the Transient Attribute Property Sheet is open and the Function Type is selected, see Defining a Transient Attribute.

Procedure

1. Ignore the Overlap field unless you are going to combine sub-functions over the same time period. If you are then you must decide how these are combined: Add or Multiply.
2. By default the Sub-Function table is empty. Click the + button at the right-hand side of the first row of the table to add a sub-function.

   The property sheet expands to show more fields.
3. Enter a Start Time and an End Time that defines the time period that the Sub-Function is active.
4. Select a Sub-Function Type from the dropdown list and enter data in the fields. Refer to "Transient Sub-Functions" on page 135 for details.
5. Use the Sub-Function Time Chart window to view the currently-selected sub-function's profile.
6. Add further Sub-Functions as necessary.
7. To repeat the function, see "Defining a Periodic Transient Attribute" on page 148.
8. Use the Function Time Chart window to view the resultant transient attribute derived from the combination of all the transient sub-functions.

   This window shows the effect of either adding or multiplying overlapping functions. The time range is from the earliest start time to the latest end time of all the sub-functions unless the combined function is periodic, then the first five cycles are shown.

   Note

   To zoom-in on an area of a chart, mouse-drag over the area and release. Right-click to restore the chart to the default size.

Related Topics

Transient Attribute Property Sheet

Transient Variation With Time

Defining a Transient Attribute

Using Multipliers

Defining a Periodic Transient Attribute

Time-varying transients, can be set up to be periodic, that is, at the end of the profile or function, the profile/function is repeated.

Prerequisites

- A Profile or Function transient has been defined.

Procedure

Check Periodic.

Results

The T start and T end values show the earliest start time and the latest end time of all the sub-functions.

The Transient Time Chart in View mode (Profiles) or the Function Time Chart (Functions) shows the first five cycles.

Related Topics

Defining a Transient Attribute Profile That Varies With Time

Defining a Transient Attribute Function

Defining a Linear Function Time-Varying Transient Attribute

In this example a transient attribute is defined to vary a heat source between 2 and 5 Watts linearly from 0 and 15 seconds. The required multiplication factor of the transient is then calculated to vary from 0.4 (2/5) to 1 (5/5).

Procedure

1. Create a new Thermal attribute and set the following in the property sheet:
    - Name: for example, “2 to 5 W Linear”
    - Thermal Model: Conduction
    - Total Power: 5 W — the maximum power that will be applied
2. For the Transient Attribute, select **Create New**.

    A new Transient attribute is opened at the **Multiplier vs. Time** tab.
3. Set the following in the property sheet:
    - Name: for example, “Linear Gradient 0.4”
    - Type: Function
    - Overlap: leave as is.
    - Add a Sub-Function to the list by clicking the + button and then set the End Time to 15 s.

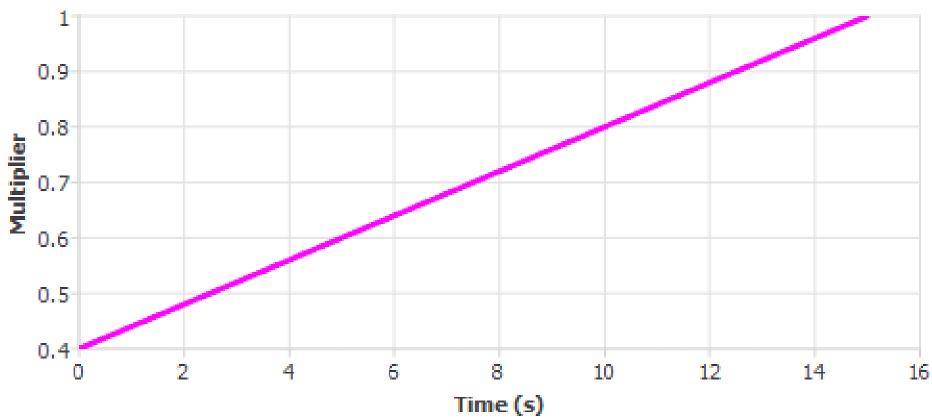
    a. For the Sub-Function, set the following:
        - Sub-Function Type: Linear

- Baseline Value: 0.4 because the required multiplication factor of the transient should then vary from 0.4 (2/5) to 1 (5/5).
- Baseline Time: 0 s
- Coefficient: 0.04 that is, the gradient of the function which, in this example = (1 − 0.4)/15

Results

- The sub-function is shown in Figure 13-15.

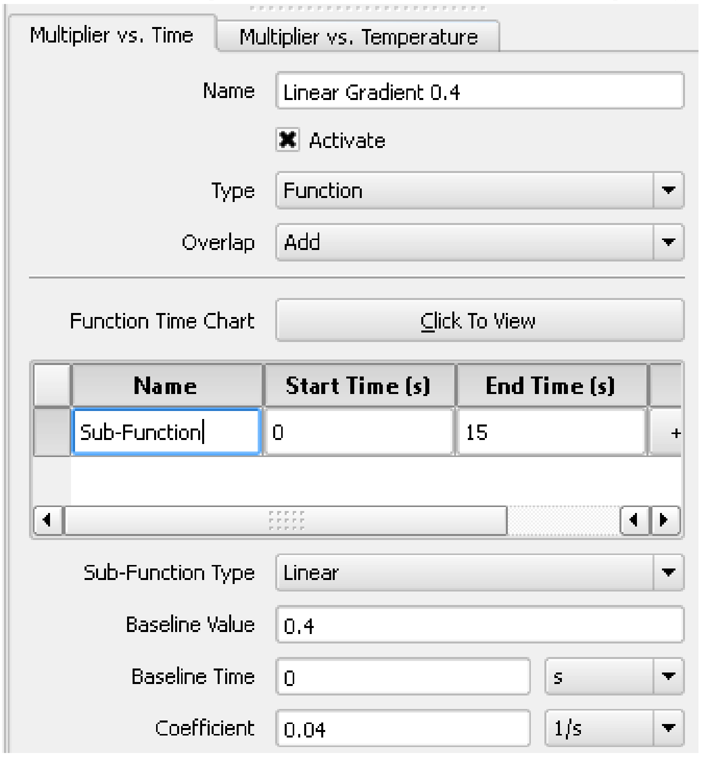
Figure 13-15. Transient Attribute Curve



When this is applied to a 5 W thermal attribute, the transient will vary linearly from 0.4 * 5 (that is, 2) W at 0 s to 1 * 5 (that is, 5) W at 15 s.

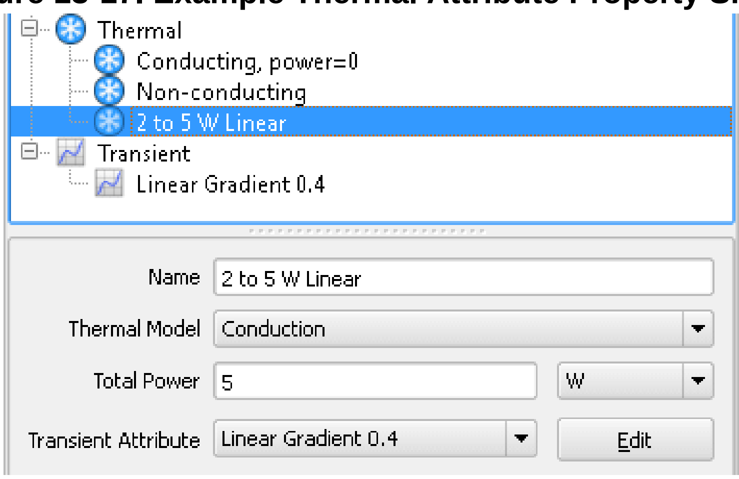
- The transient attribute property sheet is shown in Figure 13-16.

Figure 13-16. Example Transient Attribute Property Sheet



- The thermal attribute property sheet is shown in Figure 13-17.

Figure 13-17. Example Thermal Attribute Property Sheet



Related Topics

Transient Sub-Functions

Defining a Transient Attribute Function

Transient Attribute CSV Files

You have the option of defining a transient profile by specifying plot coordinates in a CSV file.

Transient Time Chart CSV File Format

A comma-separated text data file.

Use this file to set up a transient attribute profile which varies with time.

Format

A Transient Time Chart CSV File must conform to the following formatting and syntax rules:

- The units are those currently set by the chart.
- 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.

```
<time>,<multiplier>
<time>,<multiplier>
...
```

Parameters

- time

  A time at which you want to define a multiplier.
- multiplier

  The multiplier value at that time.

Related Topics

Transient Time Chart

Using Multipliers

Transient Temperature Chart CSV File Format

A comma-separated text data file.

Use this file to set up a transient attribute profile which varies with temperature.

Format

A Transient Temperature Chart CSV File must conform to the following formatting and syntax rules:

- The units are those currently set by the chart.
- 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.

```
<temperature>,<multiplier>
<temperature>,<multiplier>
...
```

Parameters

- temperature

  A temperature at which you want to define a multiplier.
- multiplier

  The multiplier value at that temperature.

Examples

- An Increasing Curve of a hysteresis loop (temperatures in degC):

```
35,1
90,1
90.2,0.2
125,0.2
```

- A corresponding Decreasing Curve for the same loop:

```
35,1
80,1
80.2,0.2
125,0.2
```

Related Topics

Transient Temperature Chart

Using Multipliers

Transient Attribute Property Sheets

This section describes the property sheet and time-dependent and temperature-dependent charts used when defining transient attributes.

Transient Attribute Property Sheet

To access: Select or edit a Transient attribute.

Use this property sheet to define a transient variation with time, temperature, or both.

Objects

Field	Description
Multiplier vs. Time tab	See Transient Attribute Property Sheet Multiplier vs. Time Tab.
Multiplier vs. Temperature tab	See Transient Attribute Property Sheet Multiplier vs. Temperature Tab.

Transient Attribute Property Sheet Multiplier vs. Time Tab

To access: Select a Transient attribute then the **Multiplier vs. Time** tab.

Use this property sheet to define a function over time, either by specifying a profile or a mathematical function. A function can be a combination of sub-functions.

Objects

Field	Description
Name	Identifies the transient attribute.
Activate	Check to activate this transient attribute.
Type	Profile — define the transient by a plot interpolated from points on the plot. Function — define the transient by mathematical function comprising one or more subfunctions in sequence or combination.
Transient Time Chart	(Profile) Click **Click to Edit** to open the Transient Time Chart in Edit mode. Click **Click to View** to open the Transient Time Chart in View mode.
Overlap	(Function) Add or Multiply subfunctions that overlap, that is, share the same time line. Applies to the complete function.
Function Time Chart	(Function, and one entry must have been added to the table of sub-functions) Click **Click To View** to open the Function Time Chart.
Table of Sub-Functions	(Function) Sub-Functions; their Names and their Start and End Times. A sub-function is active at its Start Time, but not at its End Time. This prevents sub-function overlap (and, therefore, addition or multiplication) when one sub-function is set to start at the same time as the end of a previous sub-function.
Sub-Function Type	(Function) Select a type from the dropdown list.
Linear Sub-Function Fields	
Baseline Value, Baseline Time, and Coefficient	See Linear Sub-Function for the equation and an example.
Power Law Sub-Function Fields	
Baseline Value, Baseline Time, Coefficient, and Power Index	See Power Law Sub-Function for the equation and an example.
Exponential Sub-Function Fields	

Field	Description
Baseline Value, Baseline Time, and A and B Coefficients	See Exponential Sub-Function for the equation and an example.
Sinusoidal Sub-Function Fields	
Baseline Value, Baseline Time, Coefficient, and Period	See Sinusoidal Sub-Function for the equation and an example.
Gaussian Sub-Function Fields	
A, B, and C Coefficients	See Gaussian Sub-Functionfor the equation and an example.
Pulse Sub-Function Fields	
Amplitude, Rise Time, High Time, and Fall Time	See Pulse Sub-Function for the equation and an example.
Double Exponential Sub-Function Fields	
Amplitude, Decay, and Rise	See Double Exponential Sub-Function for the equation and an example.
Sub-Function Time Chart	
Sub-Function Time Chart	Click **Click to View** to open the Sub-Function Time Chart.
Periodic Transients	
Periodic	Check to activate a periodic profile. Applicable to both profile and function transient types. For functions this applies to the complete function, that is, the combination of all sub-functions.
T start	The start time of the period. This is the earliest time that is defined for the function or period. Read Only
T end	The end time of the period. This is the latest time that is defined for the function or period. Read Only.

Related Topics

Transient Variation With Time

Transient Time Chart

Function Time Chart

Sub-Function Time Chart

Transient Attribute Property Sheet Multiplier vs. Temperature Tab

To access: Select a Transient attribute then the **Multiplier vs. Temperature** tab.

Use this property sheet to select a monitor point whose temperature will be used to define a transient multiplier function.

Objects

Field	Description
Name	Identifies the transient attribute.
Activate	Check to activate this attribute.
Associate Monitor Point	Select from the list of monitor points that are in the model. If there are no monitor points in the model then the list is empty.
Edit button	Click to select the associated monitor point in the data tree and open the monitor point property sheet.
Transient Temperature Chart	Click **Click to Edit** to open the Transient Temperature Chart.

Related Topics

Transient Variation With Temperature

Defining a Transient Attribute Profile That Varies With Time

Transient Temperature Chart

Function Time Chart

To access: Open a Transient Attribute property sheet **Multiplier vs. Time** tab then click the Function Time Chart **Click To View** button.

Use this chart to confirm the profile of the multiplier variation with time derived by combining sub-functions.

Description

This chart shows the profile of the complete transient function resulting from the combination of all the sub-functions.

- To zoom-in on an area of the chart, mouse-drag over the area and release.
- To restore the chart to the default size, right-click in the chart.

Objects

None.

Related Topics

Defining a Transient Attribute Function

Transient Attribute Property Sheet Multiplier vs. Time Tab

Sub-Function Time Chart

To access: Open a Transient Attribute property sheet **Multiplier vs. Time** tab then click the Sub-Function Time Chart **Click To View** button.

Use this chart to confirm the profile of the multiplier variation with time created by a sub-function.

Description

This chart shows the profile of the currently selected sub-function.

- To zoom-in on an area of the chart, mouse-drag over the area and release.
- To restore the chart to the default size, right-click in the chart.

Objects

None.

Related Topics

Transient Sub-Functions

Transient Attribute Property Sheet Multiplier vs. Time Tab

Transient Time Chart

To access: Open a Transient Attribute property sheet **Multiplier vs. Time** tab then click the Transient Time Chart **Click to Edit** or **Click to View** button.

Use this chart to create or confirm the profile of the multiplier variation with time.

Objects

Field	Description
Time	The units of time values 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 multiplier	
Time	A time at which you want to define a multiplier.
Multiplier	The multiplier value at that time.

Related Topics

Defining a Transient Attribute Profile That Varies With Time

Transient Attribute Property Sheet Multiplier vs. Time Tab

Transient Time Chart CSV File Format

Transient Temperature Chart

To access: Open a Transient Attribute property sheet **Multiplier vs. Temperature** tab then click the Transient Temperature Chart **Click to Edit** button.

Use this chart to set a profile, or two profiles to model hysteresis.

Objects

Field	Description
Temperature	The units of temperature values entered into the table.
Enable Hysteresis	Check this box to define separate increasing-temperature and decreasing-temperature curves.
Import CSV File	Click to open a file browser to select a valid CSV file.
A single table containing a row for each multiplier, or two tables, named Increasing Curve and Decreasing Curve if the Enable Hysteresis check box is checked.	
Temperature	A temperature at which you want to define a multiplier.
Multiplier	The multiplier value at that temperature.

Usage Notes

If you have two profiles defined and you uncheck Enable Hysteresis, then the Increasing Curve table values are used to define a single profile.

If Enable Hysteresis is checked, then **Import CSV File** populates the currently selected table.

Related Topics

Defining a Transient Attribute Profile That Varies With Temperature

Transient Attribute Property Sheet Multiplier vs. Temperature Tab

Transient Temperature Chart CSV File Format

Hysteresis Loops

Index