Coolit®

User's Manual

September 2021

Daat Research Corp.
Hanover, New Hampshire
USA

Table of Contents

1 Welcome to Coolit

1.1 How to Use this Manual

The manual is built around tutorials and reference chapters. We recommend that all new Coolit users as a minimum complete tutorials 1 and 2 to get acquainted with the software. The chapters following the tutorials can be used to answer specific questions.

The manual uses the following convention, when we write, "select the **File|Open** menu" it means that you first select the File menu on the menu bar and then select the Open command within the File menu.

1.2 Getting Help

1.2.1 Online Help System

In addition to the User's Manual, you can use the online help to obtain context-sensitive assistance. Online help can be activated by any of the following methods:

1. Select the **Help|Help Topics** menu or click on , or

2. Press the Help button in a dialog box, or

3. Press F1 on the keyboard.

1.2.2 How to Contact Us; Software Updates

Before you contact us, please make sure to follow these steps.

First, verify that you are using the latest version of Coolit by clicking **Help|Check for Updates**. By default, Coolit checks for updates automatically and notifies you when an update becomes available. You can disable automatic checking from the **Tools|Options** dialog.

Second, we provide free support only with respect to the operation of Coolit. We assume that you are familiar with the information, calculations, and reports that serve as input and output of Coolit. Please review pertinent information in the User's Manual, on-line help and your notes from the training course (if you took one) before contacting us. Context sensitive help is available in almost every dialog and by pressing F1 anywhere inside Coolit.

If you do have the latest release, reviewed the relevant information and are still experiencing problems, send the *ProjectName-support.zip* file from your work folder to support@daat.com and describe the problem in the body of the email.

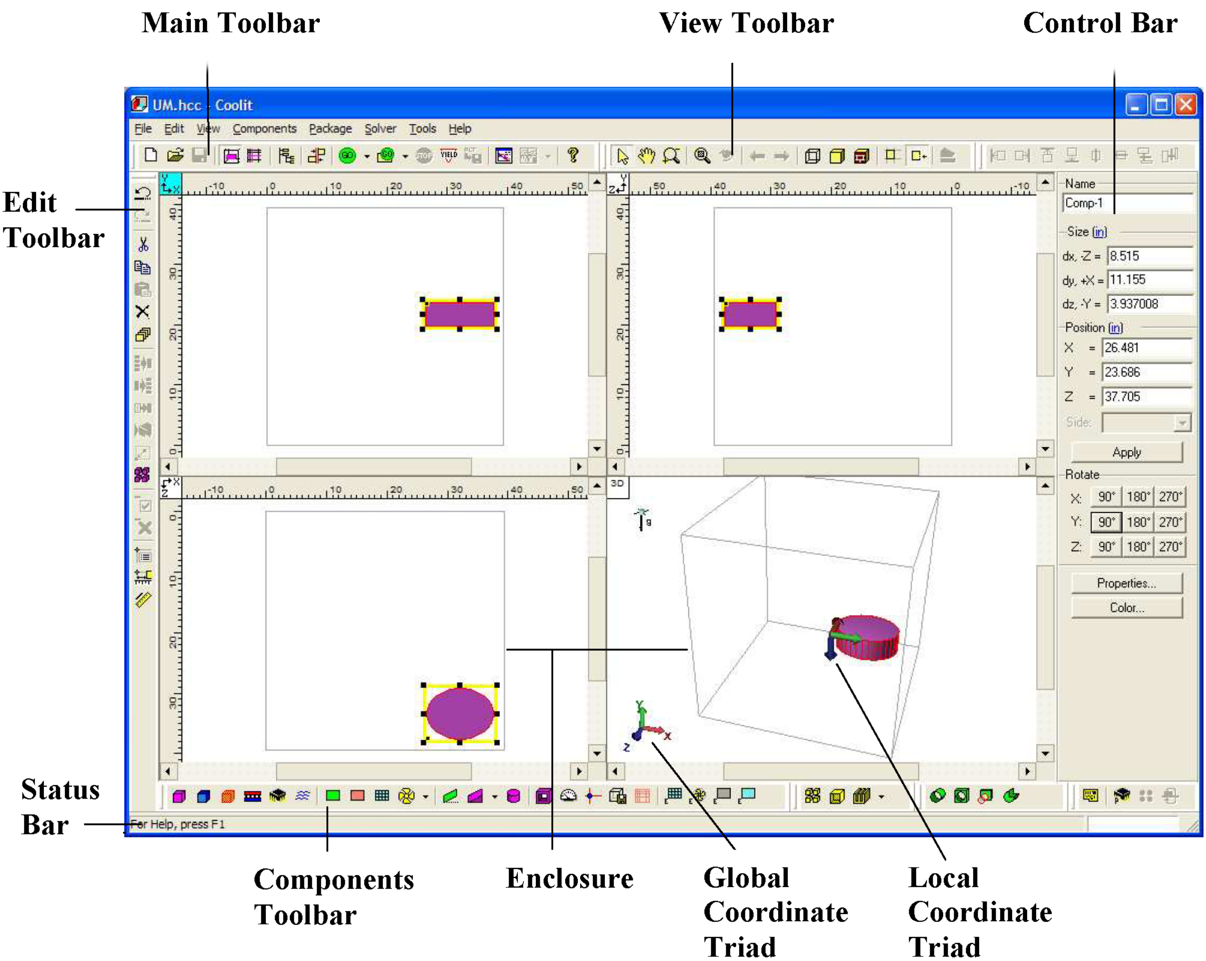
Here is how you can contact us:

- **Email:** support@daat.com
- **World Wide Web:** https://www.daat.com/

If we can't resolve your support inquiry via email, we will call you and, if required, login remotely to your computer to expedite a resolution.

1.3 Exploring the Working Area

By default, the main window opens with three 2D-view panels and one 3D-view panel. A blue left upper corner indicates the active panel. A double click on that corner maximizes the view. A double click on the corner of a maximized view will minimize it. You can also use the **View|Maximize View** menu or type **W** key to the same effect. Additional layouts, like the ones shown below, can be selected by accessing the **View|Configure Panels…** menu, or by right clicking the blue corner of the active window. If you have more than one monitor, it is often more productive to work with two windows, rather than one. Click **View|Add Second Window**.

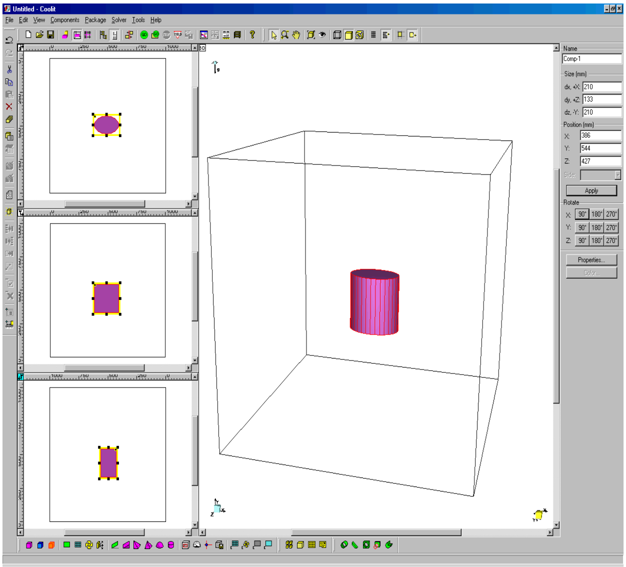


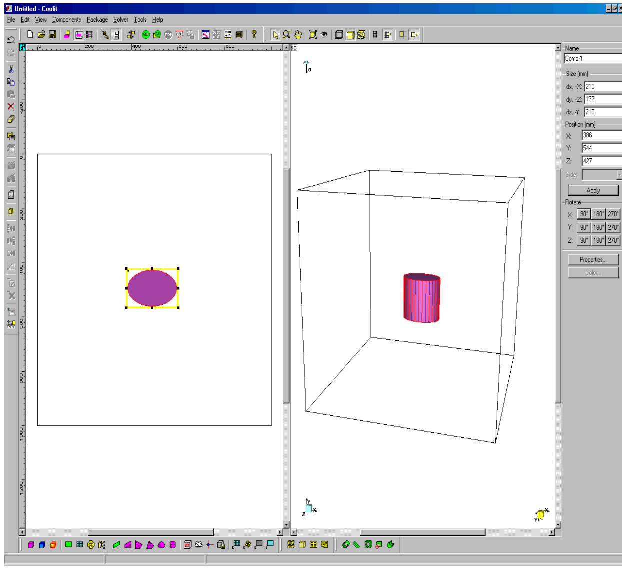
In Coolit, the **Enclosure** refers to the volume inside which computations will be done. It can be a cabinet or simply a volume in which you wish to carry out the simulation. When you start Coolit, the default size of Enclosure is 1 x 1 x 1 m.

In the 3D-view, there are two coordinate triads. The triad on the left defines the global coordinate system. The triad on the right defines the local coordinate system of the **active object**. The local triad is only shown when an object is active.

The **Control Bar** is used to define the Name, Nameplate, Size, Position, and orientation of an object. The lower case (dx, dy, dz) and the upper case (X, Y, Z) in the Control Bar indicate the local and global coordinate systems, respectively. To position an object, you can either drag the object with the mouse or type in the global coordinate location of its origin in the Control Bar. Note that the position cells in the Control Bar are calculator cells. The local coordinate system is always used when specifying the properties of a component such as dimensions and thermal conductivity.

The **Status Bar** provides dynamically updated context sensitive

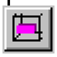
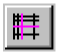




information about your working environment, such as the coordinates of active objects, grid size, and tips. The Status Bar can be hidden by unchecking **View|Status Bar**.

The Objects Manager is used to view and manipulate the hierarchy of Coolit objects within a project. You can open the Objects Manager by checking **View|Objects Manager,** by pressing , or by typing "b" on the keyboard.

There are six toolbars available in Coolit. You can select displayed toolbars via the **View|Toolbars…** menu. Click **Configure** to select which buttons to display and hide. All the toolbars are dockable. Just click them with the left mouse button and drag to a new location.

Coolit has three modes: Package, Component, and Grid. The modes can be activated using either the Package menu (**Package|Package Mode, Package|Components Mode**, **Package|Grid Mode**) or pressing the respective toolbar buttons:  for Package Mode,  for Components Mode, and  for Grid Mode.

Package Mode is used to create components, assemble components into complex objects, and assemble the entire system within the Enclosure. All objects inside the Enclosure are listed under the **Interior** folder in Objects Manager.

Components Mode is used to create components and assemble them into complex objects. Objects here are listed under Project Objects in the Objects Manager. They are available for the project but are not included inside the Enclosure. In order to add a Project Object to the Enclosure, drag the object into the Interior using Objects Manager.

Grid Mode is used to create and edit the computational grid. Click here for more information on Grid Mode.

2 Mouse and Keyboard Shortcuts

Mouse Functions

Input Device	Input Operation	System Response 2-D View Pane	System Response 3-D View Pane
2-Button Mouse	Left Mouse Button	Select active Object	Select active Object
	Right Mouse Button	Shows Object options	Shows Object options
	Ctrl+Left Mouse Button Double Click	Hide component pointed by the mouse cursor	Hide component pointed by the mouse cursor
	Shift+Left Mouse Button Click	Select the object pointed by the mouse cursor. The selected object will display a check mark next to its name in the Objects Manager	Select the object pointed by the mouse cursor. The selected object will display a check mark next to its name in the Objects Manager
3-Button Wheel Mouse	Left Mouse Button	Select active Object	Select active Object
	Right Mouse Button	Shows Object options	Shows Object options
	Ctrl+Left Mouse Button Double Click	Hide component pointed by the mouse cursor	Hide component pointed by the mouse cursor
	Shift+Left Mouse Button Click	Select the object pointed by the mouse cursor. The selected object will display a check mark next to its name in the Objects Manager	Select the object pointed by the mouse cursor. The selected object will display a check mark next to its name in the Objects Manager
	Mouse Wheel Turn	Zoom view	Zoom view
	Middle mouse button hold and drag	Pan	Rotate Object
	Ctrl + Middle mouse button hold and drag	Pan	Pan

Note: The mouse wheel direction for zoom can be reversed from the **Tools|Options** dialog, General tab.

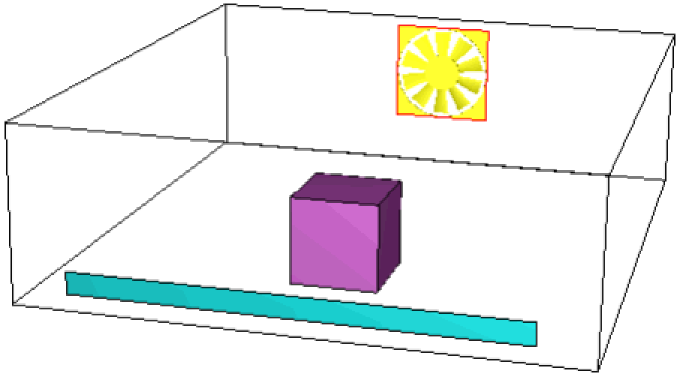
Keyboard Hot Key Commands

Shortcut	Description
Ctrl+D	Open Distance Toolkit
Ctrl+M	Open Distance Measure tool
Ctrl+N	Create a new project
Ctrl+O	Open an existing project
Ctrl+S	Save the current project
Ctrl+P	Print the active panel
Ctrl+Z	Undo
Ctrl+A	Redo
Ctrl+X	Cut
Ctrl+C	Copy
Ctrl+V	Paste
Ctrl+T	Transform local coordinates with component in-place
Del	Delete
Ctrl++	Clone
W	Maximize the currently active panel
H	Hold H and drag components horizontally in 2D panel views
V	Hold V and drag components vertically in 2D panel views
C	Hide/Display the Control Bar
B	Display the Objects Manager
M	Display the Convergence Monitor
O	Display the OptimizeIt dialog
Z	Switch to Zoom mode
S	Switch to Select mode
R	Switch to Pan/Rotate mode
P	Switch to component painting mode
Space	Toggle between Rotate and Select modes or between Zoom and Rotate modes
F2	Switch into Components mode
F3	Switch into Package mode
F4	Switch into Grid mode
F5	Reset all views to default
F7	Display components as wireframes
F8	Display components as solids of fixed colors
F9	Display components as solids of user-specified colors
F10	Toggle snap to objects mode
Ctrl+Tab or Ctrl+F6	Switch to next panel
Ctrl+Shift+Tab or Ctrl+Shift+F6	Switch to previous panel
Ctrl-Shift-H	Opens the *Minimum Allowed Object Size* field in Global Settings|Options dialog
Tab	Go to the next component
Shift+Tab	Go to the previous component
Esc	Deselect the currently selected component (if any)

3 Tutorial 1

3.1 Starting a New Project

In this tutorial, we will build and solve the model shown below. An assembled model is provided for your reference in the *tutorial_1.hcc* case file located in *Public Documents* folder *Coolit Tutorials\Tutorial_1*. Open Coolit to create a new project. Go to the **File|Save As** menu and save the project as *mytutorial_1.hcc*.



Use Project Properties dialog, opened from **File|Properties,** to write a short description of your project.

Coolit is set up to automatically save your project every minute in the *mytutorial_1.hcc* file. The automatic save time can be adjusted, or the option can be turned off in the **Tools|Options…** menu. Coolit will prompt you to save the project file upon exiting if any unsaved changes exist.

A backup file is placed in the same directory as the project file. Your backup, *mytutorial_1.Backup.hcc*, is created right after you load *mytutorial_1.hcc* for the first time.

3.2 Setting Defaults and Options

3.2.1 General Options: Startup, Saving, and Viewing

The first tab of the **Tools|Options…** dialog lets you set up start up, project saving, and viewing options. By default a backup file, filename.Backup.hcc, is created right after you load a file into Coolit. Autosave is also enabled, saving your project file every 10 minutes. One more layer of safety is provided by the **Save on Go** option. If checked (default), on pressing GO, Coolit saves a copy of the case file with the extension Solver.HCC. The name of this file and its time stamp are reported in CoolPlot's Objects Manager under Package.

The dialog also enables the display of the Reference Point and provides support for faster redrawing of the model (**Large Model Rendering**). The latter option somewhat degrades the quality of display, but the drawing is faster and it is advised for larger cases.

3.2.2 General Options: Undo and Redo

The Undo/Redo functions are used to recover previous operations in Coolit. The default number of Undo/Redo levels is 100. It can be changed using **Tools|Options…**. The Undo function is activated via the **Edit|Undo** menu, or  button or Ctrl+Z shortcut key. Similarly, you can access the Redo function via the **Edit|Redo** menu or  button or Ctrl+A shortcut key. Note that when you save manually, all Undo/Redo information is erased.

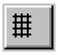
3.2.3 Warning Options

The Warning Options are specified in the **Tools|Options…** menu under the **Warnings** tab. Coolit currently provides three warning messages: on component delete, on move/resize, and on component disable. By default, the "Delete Object" and

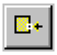
"Disable Object" warning messages are on while the "Move/Resize" warning is disabled.

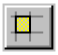
3.2.4 "Snap To" Options

The "Snap To" commands make component alignment easy by forcing objects to line up with the Snap to Grid, or with objects nearby.

To display the Snap to Grid, click on  in the toolbar.

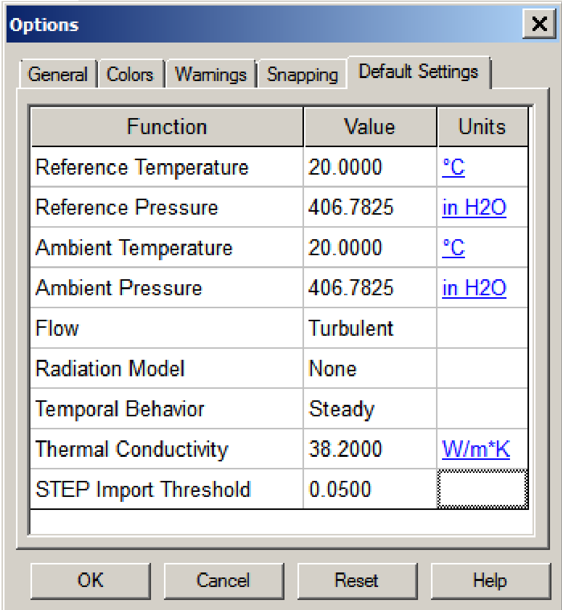
The grid is used to line up dragged objects vertically and horizontally with the nearest grid marker. The grid can be defined either uniform in all directions or independently for each coordinate direction in the **Tools|Options…** dialog, under the **Snapping** tab.

The **View|Snap to Anchor Lines** menu or  button aligns objects so that the anchor lines of a dragged object line up with those of objects nearby. The alignment occurs when a dragged object comes within the Snapping Radius of a stationary object. The Snapping Radius is set, by default, to 1% of the largest Enclosure side. To reset it, use the **Tools|Options…** dialog.

To display anchor grid lines, press  in the main toolbar, check the option in the **Tools|Options…** dialog, or select the **View|Show Anchor Lines** menu.

3.2.5 Default Settings Options

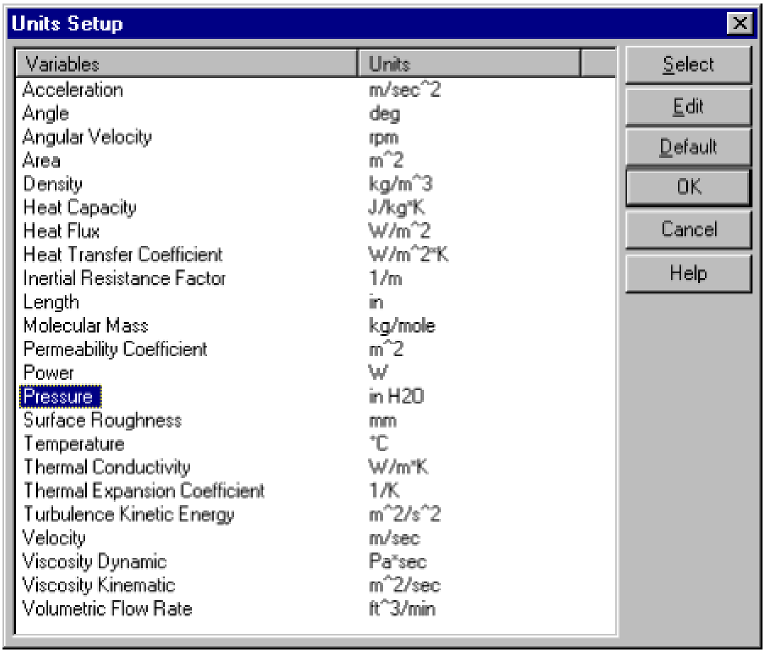
The default values for many Coolit variables can be set in the **Tools|Default Settings…** dialog. For new values to take effect, click OK to close the dialog and then close Coolit. Now if you start a new project, it will use new defaults from Default Settings.



3.3 Setting up the Units

Coolit provides you with the option to use different systems of units for different variables. For example, you can use mm for length, m/sec for velocity, ft^2/hr for kinematic viscosity, and BTU/min for power dissipation. The units are set up in the menu **Tools|Units Setup…** or they can be changed locally for individual variables by clicking the hyperlinked unit name next to the variable. To change units using the Units Setup dialog you can either press the Edit button, or double-click on the variable in the dialog. To return to the default setup, press the Default button.

The units shown in the Units Setup table are the units assumed by all the Coolit dialogs. For example, if you decide to change the length units from millimeters to inches, all the previously entered length values will be converted to inches.



3.4 Global Settings

This dialog consists of four pages.

On the **Environment** page, reference pressure, reference temperature, and the direction and magnitude of the gravity vector can be entered. Reference pressure can be calculated automatically using the altitude setting. Reference conditions are used differently depending on the type of problem. For steady state solutions with open Enclosures (i.e. Enclosures with Vents, and/or Opens, and/or Fans) the reference conditions are used for initialization and scaling only. The converged solution is independent of reference conditions. For steady state solutions with sealed Enclosures, the reference conditions are used to compute the mass of the gas in the Enclosure. The converged solution will depend on the reference condition values. For steady state problems with Heat Transfer Enabled unchecked, the reference temperature and pressure are assigned as the temperature and pressure of the selected Fluid. For transient solutions, reference conditions define initial conditions.

The **Fluid** page specifies the carrier fluid: gas, liquid, water, or vacuum. For *gas*, the default is air. Coolit computes the material properties of gases (thermal conductivity and viscosity) during the solution process as a function of local temperature using the kinetic theory of gases. The theory requires one to specify the molecular mass of the molecule, the Lennard-Jones parameters (the collision diameter and ε/K), heat capacity, and whether the gas is monatomic or not. The requested values for many frequently used gases are available by pressing the Materials button. If desired, you can specify the Relative Humidity of the gas at the Reference Conditions set on the Environment page. In most practical cases, the effect of humidity on computed temperature is negligible. If *water* is selected, physical properties are computed automatically from empirical formulas. If liquid is selected, there is a library of predefined liquids available by pressing the Materials button. If *vacuum* is selected there is no flow medium present.

The **Flow** page defines whether to solve for fluid flow and if the flow is laminar or turbulent. Selecting No Flow tells Coolit not to solve for fluid flow (i.e. no convective heat transfer will be accounted for). Selecting Laminar Flow will gray-out all component Property dialog boxes that have turbulence sensitive parameters. If Turbulent Flow is selected, then a Turbulence Model has to be specified. In this tutorial we will leave the selection at default (turbulent flow, Algebraic model).

On the **Heat Transfer** page you define whether to compute heat transfer, and if so, whether to account for radiation heat transport between objects and solar radiation. Selecting **None** for the Radiation Model field means that radiation heat transfer will not be computed. Coolit provides two Radiation Models. The **Fast** model computes view factors in a simplified manner and requires no user input besides the emissivity of component surfaces. The **Advanced** model accurately computes view factors and complete radiation exchange between all objects. Radiation exchange in the Advanced model is computed between patches on component and Enclosure surfaces. The smaller the designated patch size, the more accurate the model. A trade off is that smaller patches mean more patches, and therefore more computation time and memory. Initialization time can be significant when using the Advanced Model feature. Components smaller than the size specified in the **Min. Component Size** box will be neglected. The minimum size can also be expressed as a percent of the Enclosure size in the **% of Enclosure Size** box. When using the Advanced model you can disable radiation exchange for a specific component by unchecking the **Enable Emissivity** box in the component's Properties dialog.

If **Solar Radiation** is enabled, Coolit will compute the energy deposition from the sun. Specify the sun's position (Elevation, Azimuth, and North Azimuth direction) and Heat Flux on a plane perpendicular to the direction of the radiation. Azimuth angle is measured clockwise from the North and Elevation is the sun angle above the horizon. As the sun position is changed, it can be visually checked in the 3D view window.

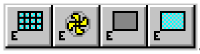
Using the data specified in this dialog, Coolit will compute the power deposition for individual objects based on their relative position, shadowing, and surface properties (Absorptivity). Solar Radiation and Radiation can be turned on/off independently.

3.5 Setting up an Enclosure

When Coolit starts up, the Enclosure object is active and has a yellow outline with black handles. To change the Enclosure size, either drag the handles, or type the new dimensions in Control Bar. In this example, define an Enclosure with dx = 500, dy = 160 and dz = 550 (all dimensions are in mm).

The sides of the Enclosure are defined in the **Package|** **Enclosure Properties…** menu. Alternatively, click the Enclosure with the mouse and press the Properties button in the Control Bar, or double-click on the Enclosure.

The Enclosure Properties dialog permits Enclosure sides of the following types: Wall, Open, Symmetry, Cyclic, Vent, or Fan. The Enclosure sides define the Boundary Conditions for the numerical solution. Boundary Conditions can be changed by pressing the Properties button for each Enclosure side, but for now leave all Enclosure sides at default values.

You can use the Components menu or the Enclosure Components toolbar buttons, , to combine these side types with each other. The only exception is the Symmetry boundary. A Symmetry boundary is a reflecting plane that tells the Solver to mirror conditions across the boundary. Therefore, it cannot be combined with any other side type.

3.6 Creating Enclosure Components

Tutorial 1 includes an Open and a Fan as the Enclosure components. Click the appropriate toolbar buttons to create these components and enter the information from the table below.

Name	Type	Side	Position (X, Y), mm	Size, mm	Q, m^3/sec
Front Open	Open	Zmax	(50, 15)	400x20	-
Rear Fan	Enclosure Fan	Zmin	(200, 50)	100x100	-0.01

Open the Fan properties dialog and enter the Q value into the Flow Rate box. Leave the ambient temperature and pressure conditions at their default values: T = 293.15 K and P_{gauge} = 0 Pa. Once you close the dialog, the Fan flow direction will be shown in all four views.

3.7 Creating Interior Components

Interior components are created similarly to Enclosure components. Create a Solid Block by clicking on the Solid Block button (or use the Components menu). Name it Power Supply Core, set all its dimensions to 80, and Position it at (X, Y, Z) = (210, 0, 350) (all dimensions are in mm). We will leave the **Nameplate** field blank in all tutorials. This field is used to specify a component label or a nameplate that is displayed in 3D view.

Press the Properties button (or double-click the component) and change Power to 5 W. Accept the default values for the rest of the properties.

Note that the (x, y, z) = (0, 0, 0) corner of the component is marked with a solid square. As you assemble components into complex objects or place them inside the Enclosure, the component's position as reported by Coolit will always be indicated in relation to that marked corner. The orientation of components is important because some component types have direction-specific properties.

Since this component will be used in Tutorial 2, we will save it in a Library for future use. To create a Library, open the Objects Manager by pressing in the toolbar or B on the keyboard. Highlight the Power Supply Core and drag it into the Library folder at the top of the Objects Manager. A copy of Power Supply Core will appear in the Library folder. (If you hold the Ctrl key while dragging components in the Objects Manager, components will be **moved** rather than **copied**.) Note that Coolit automatically switches into Components Mode when viewing Library objects. Click the right mouse button and in the popup menu select **Library|Save Library As.** Type in *Transformers* as the library name and keep the file type as **.hdb*. Press the **Save** button to save the Library and close the dialog.

3.8 Solid and Wireframe Views

Components appearance can be controlled from the main toolbar or the Objects Manager. To view objects as solid, painted by component-type colors, select **View|Appearance|Component-type colors** from the main menu or press on the main toolbar. If you prefer to specify colors of objects individually, click on the main toolbar or select **View|Appearance|Custom color/texture** from the menu. The button **Color…** in the Control Bar and the Edit toolbar button , then become available for selecting colors.

Solid View is visually pleasing, but there are instances where information may be hidden. Press on the toolbar or select **View|Appearance|Wireframe** to display objects as wireframes.

You can selectively "hide" components by clicking on the , , or icons in the Objects Manager; the icon will change to . You can also use the Object Manager's **Appearance** menu to the same effect. Yet one more way to hide components is to hold the Ctrl down, and double click the component in any view. To unhide the component you can use undo, or click or select **View|Unhide All**.

3.9 Select, Zoom and Rotate

In Select mode (is depressed), click the desired component to make it active. Alternatively, you can activate components by clicking on their names in Objects Manager. The active component will show in yellow outline with black handles for re-sizing and moving the component.

Double clicking on a component will display the component's properties dialog. If the component is part of a complex object (an object assembled from several other objects), before double clicking, make sure that the object's tree is open in the Objects Manager. When an object is active, clicking the right mouse button will produce a pop-up menu with functions applicable for the active object.

If you have a mouse with a scroll wheel, there are zoom, pan, and rotate shortcuts available. Otherwise, to zoom in, press the Zoom button  and drag the mouse up; to zoom out, drag it down. To pan in 2D-views, press the Pan/Rotate button  and drag the mouse. To rotate in 3D-view, press  and drag the mouse. If you press and hold the Ctrl key before pressing the left mouse button, the view will be rotated around the axis perpendicular to the screen and passing through the 3D window's center point. The following keyboard commands are supported:

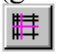
Command	Function
Left arrow	Rotate left
Right arrow	Rotate right
Up arrow	Rotate up
Down arrow	Rotate down
PgUp	Rotate clockwise
PgDn	Rotate counter-clockwise

To rotate the view more smoothly or coarsely, hold down the Shift key or Ctrl key, respectively. To restore all views to their defaults, press  button or the **View|Reset Views** menu.

3.10 Moving and Resizing Components

Once an object is active, it can be resized with the mouse or dragged to a new position. Alternatively, you can type the new position and size in the Control Bar. If you need to move an object strictly horizontally or vertically, hold the **H** (or **V**) key and drag with the mouse. If you have a large model and the move/resize operation is slow, click Tools|Options and under the General tab check the Large Model Rendering checkbox.

3.11 Viewing Grids

Coolit needs a grid of contiguous, non-overlapping control volumes (grid cells) to solve a problem. To generate the grid, press the  button or select the **Package|Grid Mode** menu. The default grid will appear in all 2D-views. You can edit the grid by manually adding, deleting or moving grid lines. Click here for more information about editing the Grid.

3.12 Running the Solver

To start the Solver, select **Solver|Start** or press . For additional solver options see Solver Settings. When you run the Solver, it is possible that Coolit will prompt you to examine your model using **Model Check**. Residuals will be displayed in the Convergence Monitor window as the iterative computations progress. The solution is considered converged when all the residual lines cross the horizontal dashed line - the specified convergence precision level (default - 10^{-3}). At that point, an information window displaying "Case Converged" will appear and the computed results can be viewed. To view results, select **Tools|Start CoolPlot** or click on  to start CoolPlot.

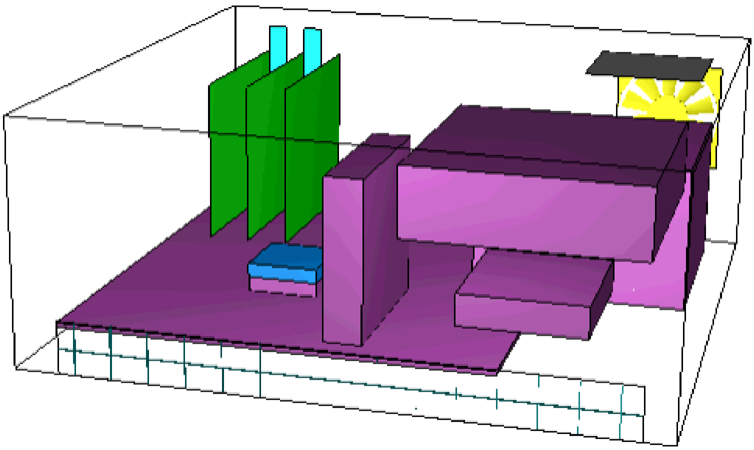
3.13 Restarting an Existing Case

If the solver is stopped before a converged solution is reached, you can restart it where it left off. To restart, select **Solver|Restart** or press , highlight the *mytutorial_1.fld* restart file, and click Open to restart.

4 Tutorial 2 - Building a Desktop Computer

4.1 Creating the Enclosure

In this tutorial, we will model the desktop computer shown below. The case file for the model is provided for your reference in the *tutorial_2.hcc* case file located in the *Public Documents* folder *Coolit Tutorials\Tutorial_2*. Start Coolit and select **File|Save As** to save the new project as *mytutorial_2.hcc*. Note that you can let Coolit automatically load the last project upon startup. If you wish to do so, check this option in **Tools|Options…**.



Set the Enclosure size either in the Control Bar, or by dragging the Enclosure handles with the mouse to dx = 420, dy = 165, and dz = 410 mm. Go to the **Package|Global Settings** dialog. Under 'Flow', select Turbulent. Specify the turbulence model as Algebraic Model. Under Heat Transfer check Enable Heat Transfer, and select, Fast, as the radiation model. Leave all the other options at their default settings. Click **OK** to exit the dialog.

Select **Package|Enclosure Properties…** to define the Enclosure. In this tutorial, several components touch the

Enclosure sides. Specifying constant temperature Walls, as was done in Tutorial 1, in effect implies an infinite heat sink at the boundary of those components. In reality, the temperature at the surface of contact is unknown and should be computed. Assuming that the computer is on a desk surface in a room with quiescent air, specify the bottom Wall, Y_{min}, as insulated. To do that, click Properties of the Y_{min} wall, select Specified Heat Flux, and leave it at the default value of zero. Leave Emissivity at the default value of 0.9. All the other sides of the Enclosure will be set up as Walls, with specified heat transfer coefficients. For each Enclosure side, Click Properties, select the Heat Transfer Coefficient option, set Ambient Temperature to 20 ºC, Heat Transfer Coefficient to 6.0 $W/m^2 \cdot K$, and Emissivity to 0.9.

4.2 Creating Enclosure Components

In Coolit, ventilation openings can be one of two types: Vent and Open. Use a Vent component if the opening has flow resistance, such as a grille, otherwise use Open.

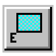
Click and rename it as Front Vent. Place the vent on the Z_{max} side. Make the width, dx = 370 mm, the height, dy = 35 mm, and the thickness 2 mm. Position the vent at (X, Y) = (30, 0) mm. Press Apply. Next, press the Properties button in the Control Bar. Choose Perforated Plate as the Modeled Object. Set the Approach Velocity to 1 m/s, the Hole Hydraulic Diameter to 5 mm, and the Thermal Conductivity to 155 W/m*K. Leave the rest at the default values and press OK to exit.

Next, create an exhaust fan by clicking on , Enclosure Fan. Name it Back Fan and place it on the Z_{min} side. Set the Enclosure Fan size to 80x80 mm and the position to (X, Y) = (320, 60) mm.

Press the **Properties** button in the Control Bar. Click the Fan Curve button and enter the pressure drop, dP, and volumetric flow rate, Q, from the table below. Use the Enter key on your keyboard when entering the data to move to the next cell or create a new line. When you have finished, click **OK** to exit the dialog. You can also copy the data from the table below or from any other Windows document, and paste it into the Fan Curve dialog using the  button.

ΔP, Pa	Q, ft^3/min
55	0
49	4
41	8
32	12
25	16
22	20
20	24
15	28
10	32
0	36

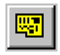
Positive (ΔP, Q) values will make the Fan blow in the positive z-axis direction in the local coordinate system. Leave the default properties for the Gauge Pressure and Ambient Temperature and click OK to exit. Check the Back Fan flow direction in either XZ or 3D view window. The fan arrow should point outward, if it is not, open the Fan Properties dialog and click the Reverse button to change flow direction.

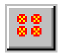
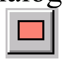
Click on  to add an Open, "Slot 1", on the Z_{min} side with a width of, dx = 15 mm, and a height of, dy = 140 mm, located at (X, Y) = (30, 10) mm. Use the default properties for Slot 1.

Create a second slot by clicking again on . The new Open will be created and placed exactly on top of Slot 1. Name the new Open as Slot 2, type in X = 60 mm in the Control bar, leave all other data unchanged, and click Apply.

Finally, add an insulating Wall Patch to the top Enclosure wall to simulate a router sitting on top of the case. Click on  to add a Wall Patch, "Router", on the Y_{max} side. Specify the size as 80 x 80 mm and locate it at (X, Z) = (320, 90) mm. Press the Properties button and select Specified Heat Flux from the pull-down window. Leave the default properties of 0 W/m^2 for the Heat Flux and 0.9 for the Emissivity. Click OK to exit.

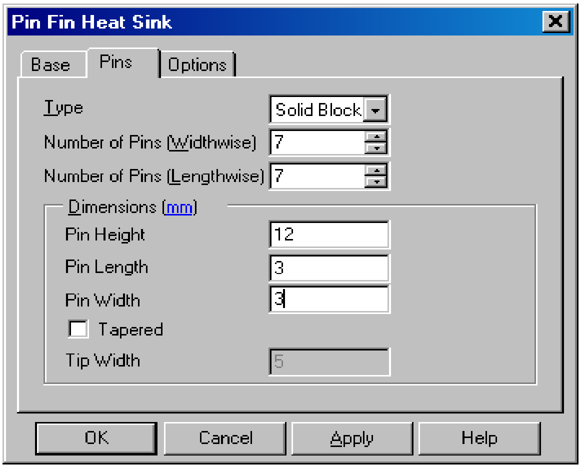
4.3 Creating Interior Components

First, we will create the System Board. Click  in the PCB toolbar. Rename the PCB "System Board". Open the Objects Manager and under the PCB object select the Board. In Control Bar, specify the Board size as (dx, dy, dz) = (350, 300, 3) mm and place it at (X, Y, Z) = (0, 7, 0) mm. In the Properties menu, under Thermal Conductivity choose Enter Manually and set the In-plane thermal conductivity to 17 W/mK and the Through-plane conductivity to 0.29 W/mK. Set the Power to 5W and click **OK** to close the dialog.

Click  in the PCB toolbar to create a Resistor Model object on top of Board. Name it CPU. Specify the size of the CPU as (dx, dy, dz) = (50, 50, 10) mm and position it at (X, Y, Z) = (100, 10, 200) mm. In the Properties menu, set it as the default 2-resistor model with a Power of 25W. Click  to add a Thermal Via to the CPU object. Open the Properties dialog of the Thermal Via and specify the Diameter as 0.1mm and the pitch as in x and y as 1mm. Click **OK** to close the dialog. Next click directly on the CPU component, and then click  to add a Contact Resistance to the top of the CPU object. Name the Contact Resistance "Thermal Pad" and specify it as an Interface Material with a Thermal Conductivity to 1 W/mK.

Now create a pin fin heat sink on the CPU component. First, in the Objects Manager click on the CPU component, and then use the Pin/Plate Fin Heat Sink button,  and select the Pin Fin option. A correctly sized pin fin sink will appear on top of CPU. Click the **Properties** button to open the Pin Fin Heat Sink dialog. On the Base page, name the heat sink “CPU Pin Fin Heat Sink”. Specify the Thickness of the base as 3 mm. Click the **Properties…** button and then the **Material…** button and select Aluminum 3003 from the pull-down menu. Click **OK** to close both the Materials and Properties dialogs. In this tutorial, we will use an approximate model of the heat sink, so check the box next to Compact Model. This will automatically construct an appropriate compact model of the heat sink.

On the Pins page, set the Number of pins in the Widthwise and Lengthwise directions to 7. Specify the Pin Height as 12 mm and the Pin Length and Width as 3 mm. Click Apply to view the created heat sink and then OK to save and exit the dialog.



Create the other components and specify their properties according to the table below. Leave the heat capacity and the density of the Solid Block components at their default values. They will not be used in this steady state computation.

Name	Type	Size (dx, dy, dz) mm	Position (X, Y, Z), mm	Power, W	Conductivity, W/m*K (kx, ky, kz) =	Emissivity
CD-ROM	Solid Block	160x150x45	(250, 110, 250)	0	209	0.7
Hard Disk	Solid Block	110x25x105	(200, 35, 300)	10	209	0.7
Floppy Disk	Solid Block	90x80x20	(285, 60, 320)	0	209	0.7
Power Supply Insulation	Solid Block	140x150x100	(250, 10, 60)	0	3.2	0.8

Click [toolbar button] in the toolbar to create a Plate. Size the PCI card as (dx, dy, dz) = (140, 120, 2) mm and place it at (X, Y, Z) = (30, 10, 0) mm. Rotate it around the Z-axis 90º so that its local z-axis is in the same direction as the global X-axis. Components may be rotated after being placed in Enclosure, but the direction-specific properties will remain fixed to the component's local coordinate system.

Access the **Properties** menu and set the Thermal Conductivity to 0.29 W/mK and the Roughness to 0 mm on both sides of the Plate. Click the Plate button again to create a second card. It will be placed exactly over the first card. Rename it and change the X-coordinate to 60 mm. Press Apply. Do the same with a third card except that X = 90 mm now. Note that we used both the PCB component (for the System Board) and Plate (for PCI Cards) to model PCBs. Read the How to Use Solid Block (Solid Block model is used for modeling PCBs) and Plate section for an in-depth discussion on how to decide which component to use.

4.4 Copying Objects between Projects

You need to copy the Power Supply Core object from the Library saved in Tutorial 1 to the current project. To open the Library, right-mouse-click inside the left pane of the Objects Manager and select **Library|Open Library.** Load the *transformers.hdb* library. The object Power Supply Core is now listed in the Library directory. Double click it to open the Properties dialog and change the power dissipation to 30 W.

Drag the Power Supply Core to the Interior folder and specify its position in the Enclosure: (X, Y, Z)=(285, 20, 90). The Core is now shown inside the Power Supply Insulation component.

Alternatively, you can use the standard Windows Copy/Paste to copy objects between open projects. Click an object and select **Edit|Copy** or Ctrl+C or [toolbar button]. Open another project where you wish to paste the object. Click **Edit|Paste** or click Ctrl+V or  to paste. The pasted object will appear in the Objects Manager.

4.5 Overlapping Components - Priority

If there are overlapping components, Coolit has to determine which component has precedence. By definition, components with higher priority define the intersection. In Objects Manager, components appearing lower on the list have a higher priority.

Check the relative positions of the components Power Supply Insulation and Power Supply Core in Objects Manager. If the Power Supply Core is listed first, it is incorrect, as it will be "swallowed" by the larger Insulation component. To change priority, simply rearrange components in the Objects Manager listing by drag and drop.

The only **exception** to the above rule is the Heat Source component. When Heat Source components intersect, the common area will have the power dissipation added from all the contributing Heat Sources.

4.6 Complex Objects

Complex objects are combinations of several components. You can assemble a complex object several ways:

- from individual components
- from components within other complex objects
- by adding new components to an already existing complex object
- by adding one complex object to another.

You can also create complex objects by using predefined Parametric Complex Objects or Object Constructors. This can be done in both the Components and the Package mode. In this tutorial, we will do it directly in the Package mode.

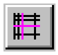
We will create a complex object called Power Supply, which will include Power Supply Insulation and Power Supply Core. Open Objects Manager. Check to make sure that Power Supply Core is listed after Power Supply Insulation. The selected objects will show with a blue outline. Now, click the Object Manager's Selected|Assemble popup menu to assemble. The complex object, Obj-1, will be created. Rename it Power Supply.

We will create a new complex object called Motherboard using a somewhat different approach. Open Objects Manager and place it to the side of the screen. (It is not needed when using this method. We will keep it only to show the mechanics of what is happening). In the view panels, press and hold down the Shift key and click once on all three PCI Cards, and the System Board (leave the Heat Sink for now, we will add it later). Notice that their outlines turned blue and in Objects Manager, the box next to their names is now checked. Click the Assemble button  (in the toolbar, not in the Objects Manager) to assemble. The default name Obj-2 and the position of the new complex object is shown in the Control Bar. Rename it Motherboard.

Sometimes you need to add one component to an existing complex object. For example, now you need to add the Heat Sink to the Motherboard complex object. Open the Objects Manager and click the "+" sign next to the Motherboard name to open the tree. Drag and drop the CPU Heat Sink component into the Motherboard folder.

We can also edit components within an existing complex object. Open Objects Manager to make sure that the Motherboard object's tree is open. (When it is closed, you cannot choose individual components.) Click on PCI-1 Card, press Properties and specify 1 mm roughness in the (+) direction and 5 W power dissipation. Repeat the same procedure for PCI Card-2 and Card-3 as well.

4.7 Solving the Problem

In this tutorial, we will use the default grid. It will be generated automatically, when you press  to run the Solver. (If you wish to view the default grid, press .) The solver parameters will also be left at default values. Read the Solver Settings section for more information. When you press GO to run the Solver, Coolit may prompt you to examine your model using Model Check.

The Solver will stop after any one of these three events occurs:

- All the equation's residuals cross the convergence precision line (i.e. you achieve convergence)
- The Solver reaches the specified maximum number of iterations
- The Solver is stopped by either selecting **Solver|Stop** or pressing . The Solver can also be stopped using the Solver|Abort menu (or icon ). If aborted, no files will be saved.

Note that only in the first case did you obtain a converged solution. If the Solver has a problem with something in the case (e.g. not enough memory), a message will appear.

After the Solver stops, a plot file will be created (*mytutorial_2.plt*). Select the **Tools|Start CoolPlot** menu or press  button to open CoolPlot and display the results.

5 Tutorial 2a – Using Grid Zones

In Tutorial 2 above, we used the Compact Model option to create an approximate model a pin fin heat sink. Using this option greatly reduced the number of grid cells required to model heat sinks. Alternatively, the Grid Zone feature allows you to model such components without simplifications by creating a localized, high-density grid around the heat sink. To demonstrate this feature, we will uncheck the Use Compact Model option in the CPU Heat Sink properties dialog and use the heat sink as is.

A completed model is provided for your reference in *Coolit Tutorials\Tutorial_2a\tutorial_2a.hcc*. Open *mytutorial_2.hcc*, save it as *mytutorial_2a.hcc*. Open the Properties menu for the "CPU Pin Fin Heatsink" object. On the Base page, uncheck the Use Compact Model box and click **OK** to close the dialog.

Now enter Grid Mode by pressing F4. Notice how the pin fin heat sink has created many grid lines that extend across the whole Enclosure. To check how many grid cells are in a case, open Objects Manager and click on Package in the model tree. At the bottom of Objects Manager window, you will see a value for Total Grid Cells. Right now, that value is greater than 80,000.

Get back into Package Mode by pressing F3. To create a Grid Zone, click  and select Grid Zone. Note that upon creating this object a warning message will appear. If radiation heat transfer is turned on, the Advanced Model must be used when there are Grid Zones in the model. Coolit will automatically prompt you to change this setting if you forget. Name the grid Zonal Grid and assign it a size of (dx, dy, dz) = (100, 56, 115) and a position of (X, Y, Z) = (75, 4, 170). Now switch back into Grid Mode and notice how the grid lines created by the heat sink are "captured" inside the Zonal Grid. A quick look in Objects Manager will show that Package now contains around 60,000 grid cells.

Grid Zones can be treated like other Coolit objects; they can be moved, named, and assembled into complex objects. To manipulate the grid inside a zone, merely select the grid either in Objects Manager or by clicking it with the mouse in the construction views. Changes within one Grid Zone will not affect the grid in other zones. Because we have refined the grid significantly around the heat sink, the Algebraic turbulence model is no longer appropriate. Under **Package|Global Settings|Flow** change Turbulence Model to Eddy Viscosity.

Finally, solve the case by clicking  and view the solution in CoolPlot.

6 Tutorial 2b - Using Textures

6.1 Applying Textures

In this tutorial, we will learn to apply textures and colors. A completed model is provided for your reference in *Coolit\Tutorials\Tutorial_2b\tutorial_2b.hcc*. To start, select **File|Open** *mytutorial_2a.hcc*. Save it as *mytutorial_2b.hcc*. Since textures are applied and viewed in 3-D, click on the 3-D view pane and select **View|Maximize View**. Next, press the  icon to enter User Specified Colors/Textures mode. Notice that all components have turned to the default gray color, as no colors or textures have been applied yet.

Click the  icon to open the Select Texture dialog. Select **Standard|Electronic Components**. Click on the **PCB 1** texture. The texture can be previewed by clicking the  icon or by right-mouse-clicking the texture and selecting Preview. Close the Preview window and double-click the **PCB 1** texture to select it. The name of the currently selected texture is shown in the pull-down menu in the painting toolbar. Verify that the selected texture is **PCB 1** and click the  icon. Notice that a paintbrush icon has appeared next to the pointer. This indicates that the selected texture is ready to be applied. Apply it to the three PCI cards by clicking on their surfaces. To apply a color or texture to an entire object at once, click the  button. To apply a color or texture to a single face, click the  button. Rotate the texture you applied to one of the PCI cards by clicking the  icon and then clicking the textured surface.

Open the Select Texture dialog again, and select the **sheet metal panel** texture. Apply it to the Power Supply object. Next, click the pull-down box in the Painting tools toolbar. A list of recently used colors and textures will appear. Click **PCB**

1 and apply it to the System Board object. (Previously used colors and textures can also be selected and then applied using the **Pick color/texture** tool, .)

Click the icon to open the Color dialog. Here you can select basic colors, or specify custom colors using the color map on the right. Colors selected here will also appear in the pull-down menu in the Painting toolbar. Select a color and apply it to the Floppy Disk object.

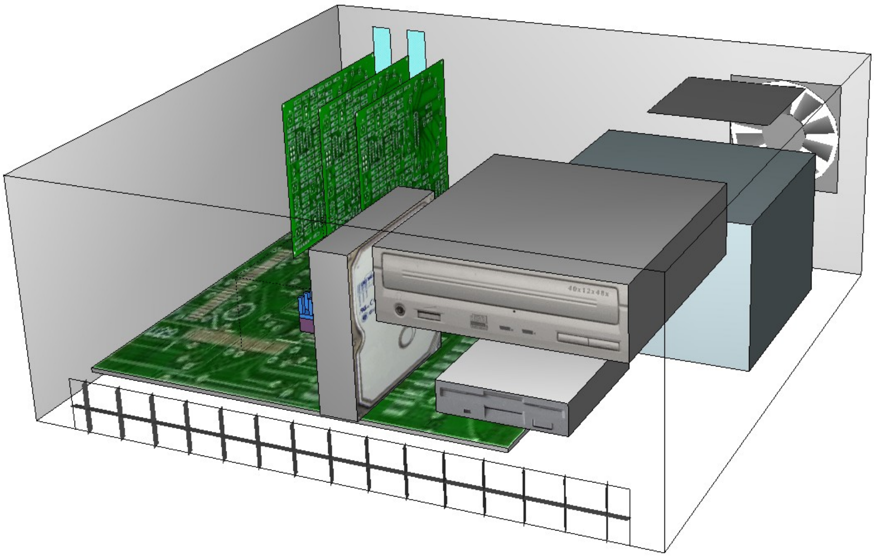
Click the icon to return to Select mode and click the Hard Drive Object. Click the icon to open the Visual Properties window. Here Objects can be viewed and textured/colored individually. All painting functions in the Visual Properties window work the same as in the main Coolit window. Select a color or texture and apply it to the Hard Drive. Click OK to close the window.

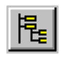
6.2 Importing Your Own Textures

You can apply your own textures to Coolit objects. Textures can be made from any raster format images including digital photos (jpeg, bmp, gif, tif, png). For this tutorial, a sample bitmap has been provided: *Coolit Tutorials\Tutorial_2b\sample.bmp*. To import a texture file, first press the icon to open the Select Texture dialog. Press the **Texture Library File >>** button and select New Library. This will create a folder called NewLibrary. Click the **Texture Library File >>** button and select Save As Library. Save the library as **My Textures**. Create a subfolder within the My Textures library by clicking . Click the NewFolder name and change it to **Tutorial 2b**. Click to import the provided sample.bmp texture. Once the sample texture is loaded into the Library, right click on it and select Properties. Check the **Stretch to face** box so that the bitmap is stretched to fit the surface it is applied to. Press OK to close the dialog and then double click the sample texture to select it. Apply this texture to the CD ROM Object.

6.3 Textures in CoolPlot

Using default Texture Libraries and colors you can generate a model that looks something like this:

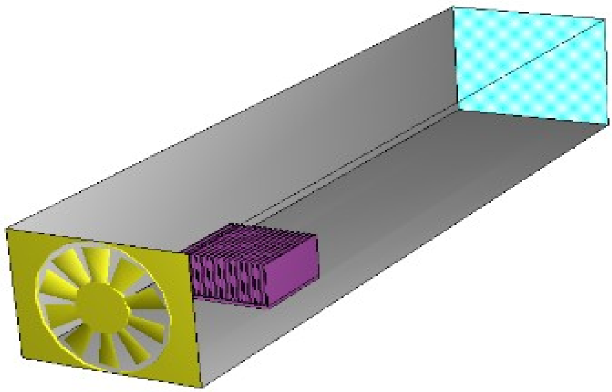


Click the  icon to rerun the model so that the textures get loaded into CoolPlot. CoolPlot may ask you to direct it to your custom texture library file location if you used custom textures in your model. Open the Objects Manager by clicking the  icon. First, hide or delete the Section object. Expand the Interior folder. Right click the  icon for the Hard Disk object and select Wireframe. To paint the Enclosure, PCI cards, and Power Supply in Textures again, right click their  icons in the Objects Manager and select Custom Paint. The ability to display individual components using different modes can be useful for preparing visually appealing presentations.

7 Tutorial 3 - Using OptimizeIt

7.1 Building the model

The purpose of this tutorial is to familiarize you with the features and functions of OptimizeIt. OptimizeIt is a powerful tool that can be used to study a system's sensitivity to design variations and optimize the system's design. The system we will be modeling is the Heat Sink wind tunnel shown below. An assembled model is provided for your reference in the *tutorial_3.hcc* case file located in the *Public Documents* folder *Coolit Tutorials\Tutorial_3.*



Start Coolit, select **File|Save As,** and save the new project as *mytutorial_4.hcc*. Press **OK** to close the dialog. Click on the Enclosure and set the size to (dX, dY, dZ) = (100, 75, 450) mm. Click Apply and then **Properties**. In the Boundary Conditions section of the dialog, set the Zmin side to Open and the Zmax side to Fan. Click the Zmax **Properties** button to open the Enclosure Fan dialog. Set the Velocity to 3 m/sec. For the remaining walls, click **Properties** and select Specified Heat Flux and press **OK**. Press **OK** to close the dialogs and then open the Objects Manager. Unhide the Enclosure Fan by clicking on the  icon and verify that the Fan is pointing into the Enclosure. If not, press the **Properties** button to open the Enclosure Fan properties dialog. Click **Reverse** to switch the Fan direction and click **OK** to close the dialog. Hide the Fan again by clicking the icon and then close the Objects Manager.

Click to create an Enclosure Wall Patch. Name it "Heat Source," place it on the Ymin side at (X, Z = 25, 300) mm, and size it 50x50 mm. Click Apply, then enter the Properties dialog for the Heat Source object. Select **Specified Power** from the pull down menu, enter 40W and close the dialog. Next, create a plate fin heat sink by pressing the button and selecting Plate Fin. On the Base page, name the heat sink "OI Heat Sink." Specify the Width and Length as 50 mm. Specify the Thickness as 2 mm. Click **Properties** and then **Material** in the dialog that appears. Select Aluminum 6063-T5 from the Select Material dialog, and press **OK** twice to close both dialogs. On the Plate Fins page specify the Number of Fins as 16. Enter the Fin Height as 25 mm and the Fin Width as 1 mm. Click **OK** to close the dialog. Rotate the Heat Sink about the X-axis by pressing the **X: 270°** button. Verify that the local coordinates are aligned as follows with z = Y and y = -Z and place the Heat Sink at (25, 0, 350) mm, so that it is on top of Heat Source.

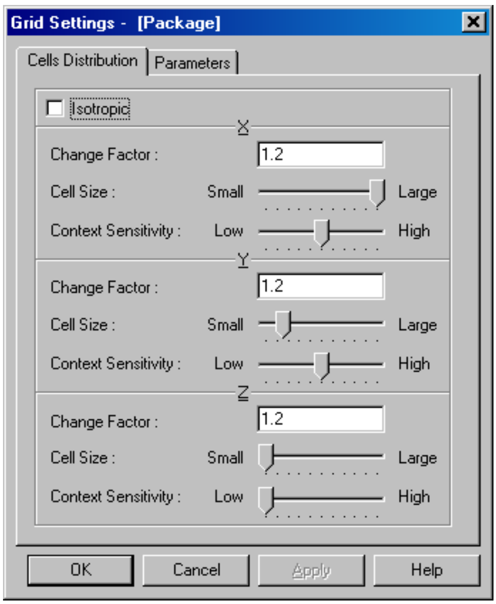
7.2 Working with Grid for OptimizeIt

Now you should examine the Grid. Click the icon or press F4 to enter **Grid Mode**. Notice that the default grid has poor resolution in the Y and Z directions. The Heat Sink object only has 2 grid cells along its height and 1 cell along its length. Normally, you could improve the grid by manually adding grid lines to the model, or by clicking the Refine button. However, OptimizeIt only uses the default grid – it will not recognize manual edits.

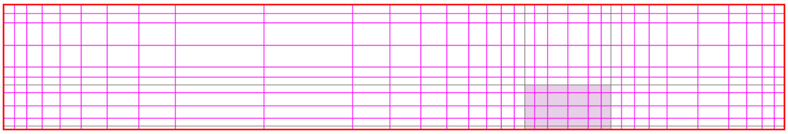
Here is why: OptimizeIt needs to create grids for different cases, each with different component sizes and positions. In order to ensure a meaningful comparison of computed results, the grid must be "consistent" in all cases. Consistent means that the grid must be generated according to the same rules in each case. This is ensured by using the default grid.

Now the task is to change the default. Once that is done, the new default will be used in all OptimizeIt cases for the current session.

To change the default grid settings, select **Package|Grid Settings**. In the Grid Settings dialog, uncheck the Isotropic box. In the 'X' box, move the Cell Size slider all the way right, to "Large". Click the Apply button to see the effect of your changes. In the 'Y' box, move the Cell Size slider to the second tick mark from the left. In the 'Z' box, move both sliders all the way to the left. When finished, the Grid Settings dialog should look like this:



Click OK to close the Grid Settings dialog. The grid in the Y/Z panel should now look something like this:



7.3 Performing a Parametric Study

Press the  button to open OptimizeIt. On the Study Type page, select Parametric Study. On the Parameterization page, click the **Add Object** button and double-click the OI Heat Sink. Click the cell under the Property heading and a pull-down menu will appear. Select Pins/Fins Number, Widthwise.

Next, click the cell under the Parameter heading and select New Parameter from the pull down menu. The Optimization Parameters dialog will open. Name the Parameter "number_of_fins" and select the Type as Dimensionless. For Dependencies, select Independent Parameter. Under Uniform Distribution Settings, set Min = 16, Step Size = 2, and Max = 22. Click Add Distribution to add the values to the table at left. Click **Add** to add this Parameter to the table and press **OK** to close the dialog.

Now we want to parameterize the plate fin thickness. To do this, add another Heat Sink to the Coolit Object list and select its Property as Pin/Fin Width. Using the pull down menu under Parameter, select New Parameter to open the Optimization Parameters dialog. Name this Parameter "fin_thickness." Set the Type to Length. Click the Values table and enter the following values: (0.75, 1, 1.25) mm. Click **Add** to add this Parameter to the table and press **OK** to close the dialog.

Next, open the Objective Function page. This function will be reported by Coolit for each set of Parameter Values. Click **Add Object** and select the Heat Sink. Click the Objective Function heading and select Max Temperature from the pull-down menu.

Leave the Weight set as 1.00. Weight is only required in Optimization Studies when two or more Objective Function are being used.

Skip over the Constraints page. Constraints are only used in Optimization Studies. Open the Case Map page and press Build Map. Twelve cases will be displayed at left; one for each combination of Parameter Values. Press **Start** to begin solving the cases. Note that individual Cases can be skipped or recalculated by clicking the **Status Icon** to the left of the Case File name. The maximum OptimizeIt Heat Sink temperature, our Objective Function, is shown in the right column as each case is solved. The Objective Function column can be sorted by clicking on the Objective Function column header. In this example, the case with 20 fins, 0.75mm thick provided the minimum temperature.

Each case can be viewed by clicking on the case and then pressing **View Case** for the model in Coolit, or **View Plot** for the results in CoolPlot. The values in the table can be copied to the clipboard by pressing the **>>Clipboard** button. The Clipboard contents can then pasted into an external text or spreadsheet document. Pressing **Show Monitor** will display Convergence Monitor.

7.4 Linking Parameters

In the previous example, we assumed all Parameters to be independent and hence we needed 12 cases to test all possible Parameter combinations. The Linking feature allows you to define some Parameters as dependent. An example of where this feature would be useful is a fan and shroud complex object. The shroud has to fit the fan, so their sizes must be linked. The linking feature will be used in this section of the tutorial.

Open the Parameterization tab in the OptimizeIt dialog and click the Parameter button at right. Click the **fin_thickness** Parameter in the table at left. Under Dependencies select

Linked with and in the pull down menu select **number_of_fins**. Click Update. OptimizeIt will now inform you that the number of Parameter Values for **fin_thickness** does not match the number of Parameter values for the Parameter it was linked with. This is because **fin_thickness** has fewer values than **number_of_fins**. In such cases parameter values that have no match are ignored. Click OK to close the warning and then click Close to exit the Optimization Parameters dialog. Go to the Case Map tab and click Build Map and then click **Proceed with build**. Notice that our number of cases has decreased to 3 and the number of fins now varies with the fin thickness.

7.5 Performing an Optimization Study

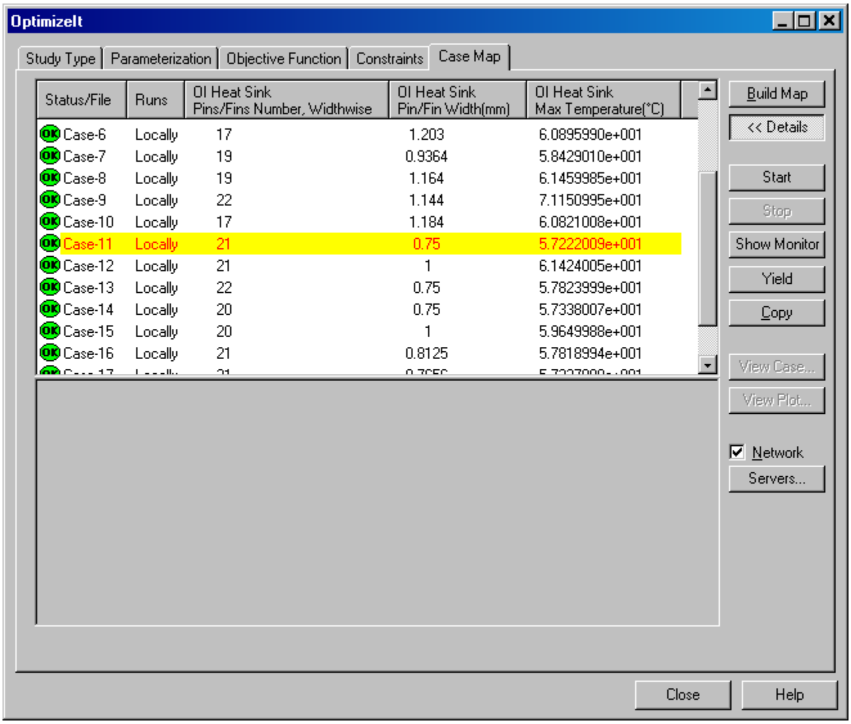
Return to the Study Type page and select Optimization Study. OptimizeIt will attempt to find values of your parameters that maximize or minimize the Objective Function.

When Optimization Study is selected the Optimization Options become available on the Study Type page. The Cases Per Parameter is the minimum number of cases that will be run for each parameter. A higher number will give you a better chance of finding your true optimum point, but may require more cases to be run. The recommended default is 5. The Optimization Precision is the threshold for stopping the optimization. If the percent change in the Objective Function value, from one case to the next, is less than then Optimization Precision, the case will be considered an extremum and Coolit will stop.

Once Optimization Study has been selected, go to the Parameterization page. Click **Parameter** to open the Optimization Parameters dialog. Click "number_of_fins" in the parameters table and delete the values of 18 and 20 from the Values table by clicking on them and pressing the Delete key. Click **Update** and then click on "fin_thickness" in the parameters table. Change the Dependencies to Independent Parameter and delete the value 1 from the Values table. Click

Update and then **Close** to exit the dialog. You now have only two bounding values for each Parameter. OptimizeIt will use to find the optimum.

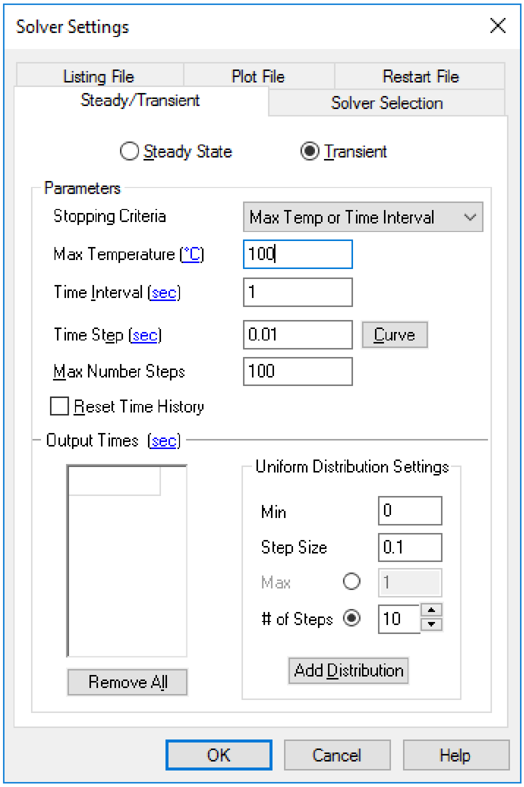
On the Objective Functions page verify that "OI Heat Sink" is the Coolit Object, "Max Temperature" is the Objective Function, and "Minimize" is the Type. Go to the Case Map page and click **Build Map** and then click **Proceed with build** on the window that pops up. Note that 10 cases have been created to begin with; 5 for each parameter. Additional cases will be added by OptimizeIt if needed. Press **Start** to run the cases.



After running the cases you will see that OptimizeIt has found a case that shows an improvement over our parametric study.

8 Tutorial 4: Obtaining Transient Solutions

Though most electronics cooling applications are steady state, transient analysis is required to investigate time-dependent scenarios such as warm-up and cool-down.



First, check the Transient radio button in the **Solver Settings| Steady/Transient** dialog. Select **Time Interval** in the Stopping Criteria box. Specify Time Step and either the Total Time or the Maximum Number of time steps. The solver will stop when the selected limit is reached. Variable Time Step can be specified by clicking the **Curve** button and entering time steps as a function of time. This can be particularly useful in problem with ramp up intervals during which a smaller time step can be used. In CoolPlot, you can view the results for time steps specified in Coolit's Output Times table, by selecting them in the Time pull down menu.

By default, the time count starts from the previous case's end time. If you want to restart from that case, but wish time starting from zero, you should check the **Reset Time History** box.

Finally, if your system contains Fans, fan will be disabled. If your steady case Fan was specified using fan curve, before solving a transient case, you should find the steady state solution and either use the computed operating point to specify constant flow rate for your transient simulation or use the **Get from Restart File** option instead.

To begin a transient analysis at the end of a steady state solution, for example, to model the cooling period following power-down of a board, follow these steps:

1. Solve the steady state problem.

2. Enable transient analysis.

3. Specify the duration (**Time Interval**), **Time Step**, and **Output Times**.

4. Change the component power dissipation settings to reflect power-down.

5. Restart the case using the steady state case restart file.

If the first leg of your case is also time-dependent rather than steady, e.g. you are computing power-up of a case for 5 min and then power-down. Solve the first case as transient for 5 minutes and then use the computed results from a restart file for the power down case as above.

Time Interval specifies time for a given leg of a case, not total time. That is if the power up phase took 5 min and the power down is 3 min, you will specify Time Interval for power-up phase of 5 min and for power-down 3 min. Coolit will automatically find out about all the previous phases of the computed process from restart files and will take it into account. If your first phase was a steady state case, in the restarted case you will specify Time Interval of 3 min in our example (and your starting time in this case will be set to zero).

As an exercise, we will solve a transient case based on the model built in Tutorial 2. An assembled model is provided for your reference in the *tutorial_4.hcc* case file located in the *Public Documents* folder *Coolit Tutorials\Tutorial_4.*

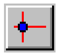
First, open *mytutorial_2.hcc* file and save it as *mytutorial_4.hcc.* Enable transient analysis in the **Solver Settings|Steady/Transient** dialog. Because in transient problems the fan flow has to be fixed, rather than specified via a fan curve, open the Fan Properties dialog of Enclosure Fan, and either type in the operating point value obtained in Tutorial 2 for steady state: 33.9 CFM or better still use the **Get from Restart File** option. The latter will not only provide correct flow rate, but will also use the flow distribution obtained in the steady state solution, instead of using uniform flow profile as done when constant Flow Rate is specified.

Now we will simulate a spike in CPU power. First, replace the CPU resistor model with a Solid Block. Click the CPU in Objects Manager and click the Properties… button. You will notice that next to the Power field the button labeled Transient is now available; click it. A dialog very similar to the Fan Curve dialog pops up. Here you can enter data points that describe a power dissipation curve for the component. Enter the values in the chart below and then click OK to clear the dialog.

Time (sec)	Power (W)
0	50
30	50
30.1	25
60	25

For our demonstration case, we will leave the values for Heat Capacity and Density at their defaults. Note, that a transient solution will depend on these values so it is important to enter correct data.

Since we are interested in a 1-minute period, set Time Interval to 60 sec. Enter 5 sec for Time Step. Now enter Output Times to write to the plot file. Add 0 and 30 sec to the table. You do not need to add an entry for the last time step as Coolit will always write a plot file at the end of the simulation. Be aware that the plot file can grow very large if your case is large and the Output Times table contains many entries. If that might be the case, uncheck the Save All Steps into a Single File option in the Solver Settings|Plot File dialog.

It can be useful to monitor the solution during a transient case. To do this, use Monitor Probe. Monitor Probes report data in the Convergence Monitor for temperature, pressure and velocity at the Probe point. Click  to create a Monitor Probe, name it CPU Probe and place it at the center of CPU; (X, Y, Z) = (125, 15, 225). As you are solving the case, you can view the Monitor Probe data in the Convergence Monitor.

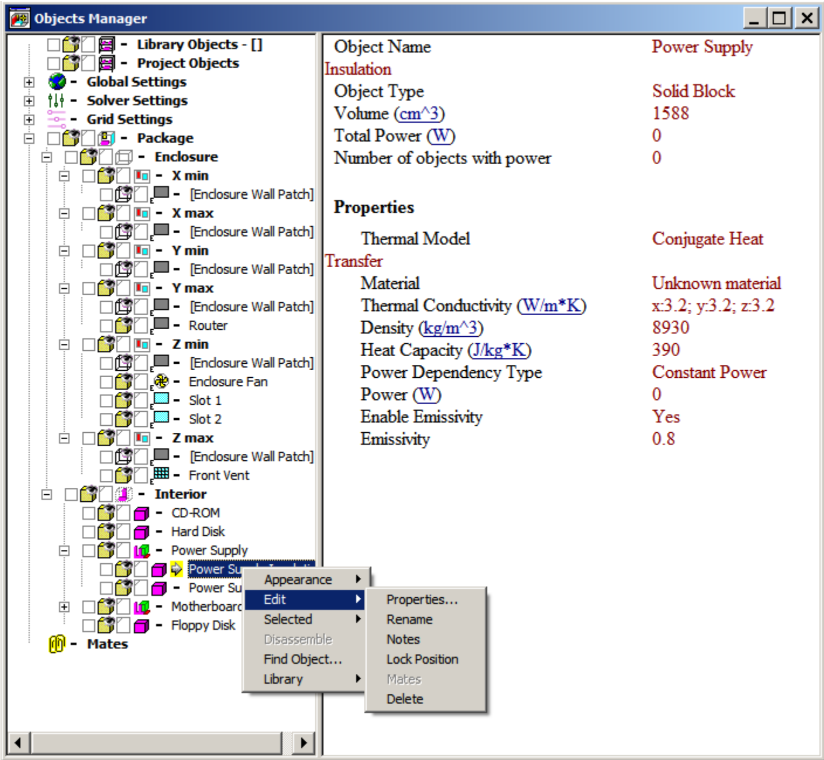
While the case runs, the Convergence Monitor displays a progress bar instead of residuals that are displayed in a steady state case. View the Monitor Probe data by using the pull-down menu in the upper right and selecting the variable you wish to see (temperature, pressure, or velocity). In this example case, pressure and velocity are not reported because the probe is inside a Solid Block.

When the case is finished, open CoolPlot. By default, all time steps are saved in a single file. Use the pull-down menu labeled "Time" in the Control Bar to display results from different time steps. If you prefer, or if it is necessary due to the case size and the number of saved time steps, you can save each time step into a separate file by unchecking the **Save All Time Steps in a Single File** box in the Solver Settings|Plot File dialog.

Coolit saves restart files every 10 time steps and keeps 4 last files FN.fld, FN-1.fld, FN-2.fld, and FN-3.fld, with FN.fld being the most recent. It also saves a restart file for every **Output Times** table value under the name: <CaseName>_<time in 13.5e format>_sec.fld. On stopping, whether on completion or user stop, FN.fld is saved.

9 Objects Manager

Objects Manager is the one-stop place for checking and editing the entire model and components settings. To launch the Objects Manager, select **View|Objects Manager** or click or press B on the keyboard. To quickly review the model settings, you can use the keyboard's arrow keys to slide up/down the objects tree in the left pane.



The menus shown in the picture and discussed in the following sections are activated via right mouse click anywhere in the populated portion of the left pane.

9.1 Drag and Drop Operations

Using the mouse, you can drag objects within each section of the Objects Manager, e.g. within Enclosure or Interior, as well as between sections such as Library, Projects Objects, and Interior. As the object is dragged, the cursor changes to indicate where you can drop the object. Once dropped, the object can be positioned either with the mouse or from the Control Bar.

There cannot be two objects with the same name in the same category, e.g. in Project Objects or in the Interior. If you attempt to add an object to, for example, Interior, with the same name as a component already there, Coolit will prompt you to rename it.

If you hold the Ctrl key while dragging objects between sections, such as Library, Interior etc, objects will be **moved** rather than **copied**.

9.2 Double Clicking

Double clicking on an item in the Objects Manager's sections: Global, Solver, or Grid Settings, Enclosure, or Interior will launch the properties dialog for the clicked item. Double clicking on a component name in the Mates section will launch the Model Check dialog's Mates page for the clicked component.

9.3 Keyboard Navigation

You can use the keyboard to navigate the Objects Manager even when it is closed. When it is open, use up ↑ and down ↓ arrow keys to move up and down the list. Pressing and holding the right arrow →, will open complex objects and move down the list, pressing and the left arrow ← will collapse them.

When the Objects Manager window is closed, first click an object with the mouse, then use the Tab key to move down and

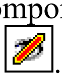
Shift Tab to move up within a level. Use Page Up to collapse an object's tree and Page Down to open it. See **Keyboard Shortcuts** in Coolit Help for the complete list of shortcuts.

In Windows 7 and later, clicking Alt key will display all the main menus, drop-down menus and entries in dialogs with underlined characters, e.g. Plate. Clicking the underlined character on the keyboard, will move the cursor to the selected menu or dialog box. (Note: you can make the underlined shortcuts always visible via the Windows Control Panel's "Make the Keyboard Easier to Use" section.)

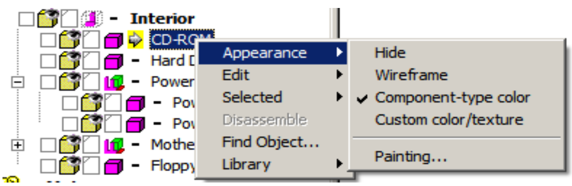
9.4 Priority

The order of components listed in the Objects Manager defines their respective priority. The component listed last on the Objects Manager list has the highest priority. If you wish to change priority of a component, drag it either up (to decrease priority) or down the list (to increase priority). When components intersect, the higher priority component defines the intersection. The only **exception** to this rule is the Heat Source component. When Heat Source components intersect, the common area will have the power dissipation added from all the contributing Heat Sources.

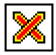
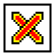
9.5 Objects Display Options

Using the Objects Manager, you can hide components and objects by clicking the , , or  icon next to the component's symbol. The icon of a hidden component changes to . To redisplay the component, click  again. To re-display all hidden objects, select the **View|Unhide All** menu.

You can also use the right mouse click **Appearance** menu to the same effect.



9.6 Disabling Objects

In the course of "what if" iterations, it is often convenient to temporarily remove objects from the system. Coolit provides a convenient way of doing it, by disabling an object. Disabled objects are not displayed and not used in the model. To disable an object, click its icon in the Objects Manager. The disabled object is displayed with the cross icon  over it. To re-enable the object, click on .

9.7 Editing Complex Objects

Complex objects have a + sign next their icon. Clicking on the "+" icon will display the object's constituent parts; the + icon is now changed to "-". Now you can select and edit any constituent component using either the mouse or the Objects Manager. When the object's tree is collapsed by pressing the "-" icon, the constituent components become inaccessible and the complex object can be moved as a whole only.

9.8 Selecting and Assembling Objects

To select objects using the Objects Manager, check the box to

the left of the object's name, or select a group consecutive of objects by clicking on the first object's name, holding the Shift key and clicking the last object's name. The selected objects will show with a blue outline. You can assemble selected objects into a complex object, provided they are on the same level of the model tree, by clicking the **Selected|Assemble** menu.

One more way of selecting objects is provided by the Find function. Click Find Object… menu and in the Find Object dialog check the Select Found Items box.

You can also change properties, size, color, position, disable/enable status, and delete multiple selected objects by clicking the Selected… menu and then **Properties, Size**, **Color, Move, Disable, or Delete** drop down menu. In Properties, you can either assign the Power value to each component individually or divide-up the Power value among the selected components proportional to their volume. To do the latter, check the "Distribute Power among selected components" box. Be careful when you use Properties as *all* of the material properties of the selected objects will be set the same. To unselect all selected objects, click **View|Unselect All** or click on the main toolbar.

9.9 Locking Position of Objects

This function is described here.

9.10 Mates Section

All the Mates groups are displayed in this section. When you select a Mate and click the Edit|Mates in the popup menu, Mates page in the Model Check dialog will open where you can check, delete and fix broken mates. The Edit|Mate menu is also available when Mated objects in the Interior section are clicked. If object does not have Mates, the Mates menu is grayed out.

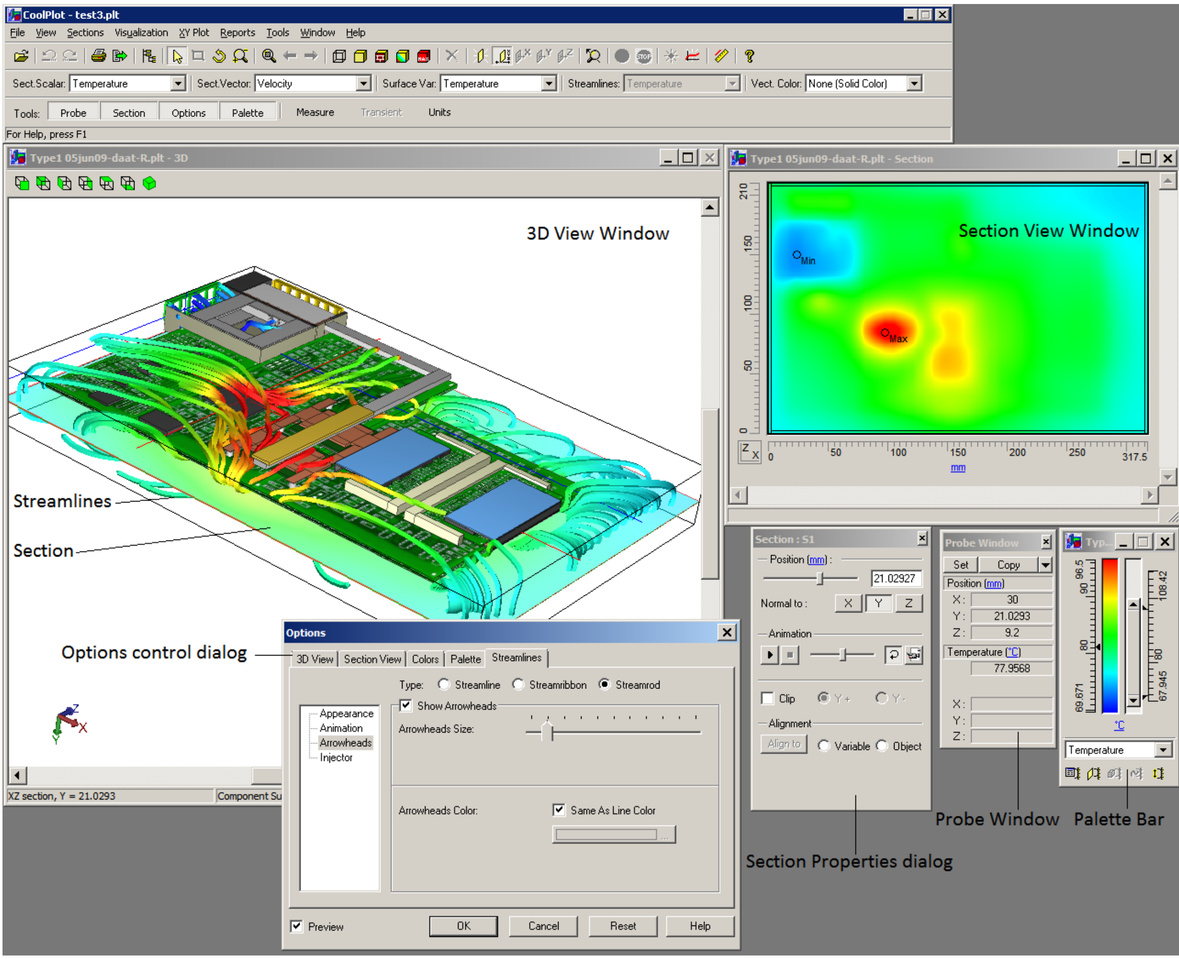
9.11 Adding Notes to Objects

To add notes to an object, click next to the object's icon. After writing the notes, the icon will change to .

10 CoolPlot

10.1 Overview

CoolPlot is used for visualization and animation of computed results. It can also generate images, both still and animated, for export to other software, such as PowerPoint and Word, and for making project reports. Select **Tools|Start CoolPlot** or click on  to start CoolPlot. Computed results will be displayed in two graphics windows shown in the picture below.



As you make changes to the graphics display, they are automatically tracked and saved. If you close CoolPlot and then later reopen the same file, the layout of windows and views should be the same as when you exited CoolPlot. If you wish to save multiple layouts of your CoolPlot session, use **File|Save Configuration File As**. You can activate saved layouts using **Open Configuration File**.

10.2 3D View Window

The 3D-view window shows a 3D-view of the entire system. Here, object surfaces are shown painted in the selected surface value. To select a different surface variable, use the pull-down box labeled Surface Var on the CoolPlot toolbar. Surfaces can also be displayed in the maximum selected variable value by pressing , Wireframe by pressing , Component colors by pressing , or Textures by pressing . To return to Surfaces painted by variable, press . The Object painting mode can be changed individually for each object using the Objects Manager.

In the 3D-view multiple Sections can be placed, and Streamlines can be positioned and animated. You can selectively hide any displayed object either by holding the Ctrl key and a double clicking on the object, or by using CoolPlot's Objects Manager. In order to re-display all components, select **View|Unhide All**.

You can mark up the image in 3D view window by inserting **text, lines, and ellipses. Click the** right mouse button and select **Mark Up Image** from the drop-down menu or you can click , , or on the toolbar. You can select created Mark Up items either in 3D view or in the Objects Manager. Double-clicking will open the Properties dialog. To move a Mark Up item, select and drag.

10.3 Sections and the Section View Windows

The right window in the image above is the Section-view window. It shows a 2D-view of the currently selected Section. To select a section, click on it in the 3D-view window. You can change the variables displayed in the Section view window by using the pull-down boxes on the CoolPlot toolbar.

In the Section-view window, if you double-click either of the plot axes, the image will be flipped along the that axis. Double-clicking will rotate the image. When the mouse cursor is moved over the image, it is shown as crosshairs with a unit vector indicating the vector variable's direction. The Probe Window will display the position of crosshairs and the values of currently selected scalar and vector variables. The variables can be selected using the toolbar pull-down boxes, Sect. Scalar and Sect. Vector. The displayed vectors attributes can be modified by pressing , to open the Options dialog.

In the 3D-view, any number of Sections can be created in any coordinate plane by pressing or selecting **Sections|Create Section**. To select a Section, press the Select button and click on a Section or use the Objects Manager. The selected Section can be dragged with the mouse or by using the arrow keys on the keyboard; it can be re-oriented by clicking , , and . The Status Bar will display the position and orientation of the currently active Section.

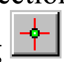
The position and orientation of a selected Section can be done using the **Section Properties** dialog, which is opened by double-clicking the section or clicking the right mouse button and selecting Section Properties. You can also use this dialog, to animate the selected Section, or to align it either with the Max/Min of a variable, or Min/Center/Max of a component. Finally, Sections can be used to clip the view, so that the image to one side of a Section is displayed as a wireframe. To do that, check the **Clip** box and select either + or – axis. Note, that a Section must be selected in order for Section functions to operate, otherwise they are grayed out.

The currently active Section can be hidden using the Objects Manager or deleted with either the Delete key or  in the toolbar.

To locate the positions of the maximum and the minimum value of a variable in the 3D-view window, open the Options dialog and on the 3D View page, Display Options section, check the **Show Min/Max** box. The spots where the variable has the maximum and minimum value will be displayed in the 3D window as crosshairs of appropriate color depending on the selected palette (e.g. blue for minimum and red for maximum).

10.3.1 Probe Window

When the cursor is moved onto the image in the Section View window, it will be displayed as a crosshairs. The **Probe Bar** will display the crosshairs coordinates and scalar and vector values at the crosshairs location. The white arrow from the center of the crosshair indicates the current vector's direction in the section's plane. Different scalars and vectors can be selected for display in the pull-down boxes on the CoolPlot toolbar.

A new **Probe** can be created by right-clicking in the Section View window and selecting **Add Probe** or by clicking  in the Probe Bar. The created Probe will appear both in 2D and 3D views. It will also be added to the Objects Manager. To change a Probe's position select the Probe either with the mouse or in the Objects Manager and type new coordinates in the Probe Bar.

The  button copies only numeric data from the Probe Bar to the Clipboard. The  button copies both the numeric data and headers to the Clipboard.

10.3.2 Information Window

Clicking ***I*** on keyboard or Sections|Create Information Area will allow you to create a box in the Section-view window. Upon creating the box the Information window will appear. The Information window will display the location and size of the created box, and approximate information about the currently selected Section Vector and Scalar. You can resize the information area by typing new numbers in the **Information Area** dialog or create a new area by Shift-drag in the Section view.

Note, that the displayed information is obtained from already interpolated data and as such should be used with caution and only for rough estimates. For accurate results you should use Gauge components.

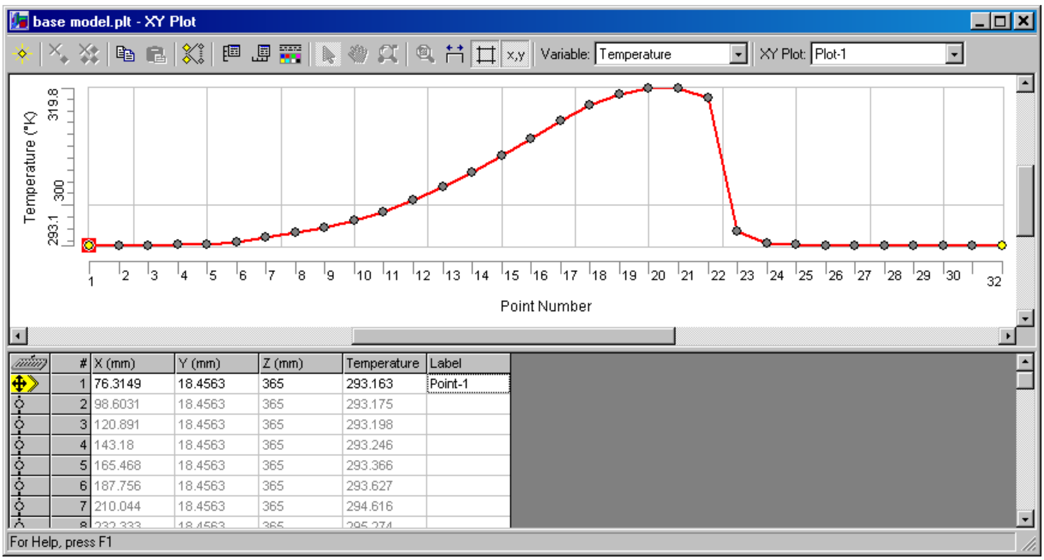
10.4 Palette Bar

The Palette Bar is used to control the colors used to represent the different variable values. The current variable is shown and can be selected in the pull-down menu at the bottom of the Palette bar. You can adjust the displayed range of the current variable by dragging the palette handles up or down. Modifying the range is particularly effective if the maximum or minimum significantly exceed mean values. You can also adjust the range to the maximum/minimum in an active cross section by pressing , or to the maximum/minimum in a selected component by pressing , or to the maximum/minimum in a selected streamline by pressing . The range can be reset back to default by pressing .

It is sometimes convenient to set the range wider than the domain range in a given problem. Pressing [icon], will open the Variable Range dialog that permits you to expand (but not shrink) the domain range of the displayed variable. To reset the displayed domain range minimum or maximum to the actual min/max in the domain, press [icon]. Press Reset button to reset both the minimum and maximum.

10.5 XY Data Plots

CoolPlot provides you with the ability to sample data at specific locations within the Enclosure and display that data in an XY (2D) chart. To start the utility in CoolPlot, click [icon] or select **XY Plot|Add XY Plot**. A window with two panels, as shown below will open. By default, CoolPlot will initially create a two-point plot. The points can now be moved with the mouse or keyboard arrows/up/down keys exactly as you would move streamline injectors in the 3D window. Alternatively, you can type in new points in the table under the graph or edit the existing points.



The created plot is now listed in Objects Manager's XY Plot folder as Plot-1. All created plots will be listed in this folder. The points on the plot can have labels, which can be specified either in the table or in the Objects Manager. By default, the labels are blank.

Similar to streamlines, when the header of a plot is selected in Objects Manager (Plot-1 in this example), CoolPlot shows a bounding box in the 3D view enclosing all points contained in the plot and the entire plot can be moved as a single object using the eight handles.

CoolPlot automatically opens the XY Plot window each time a new XY Plot object is created. To open an existing XY Plot, you can select **View|XY Plot Window**.

Often a profile of a variable along a straight line (in 3D space) is desired. An easy way to do this is to create the end points and then click  on the XY Plot toolbar and specify the Number of Intermediate Points to be uniformly distributed between end points. The intermediate points are displayed in the table with a light gray font and a small gray circle in the graph above the table. The intermediate points cannot be edited.

New points can be added by clicking  in the toolbar or by pressing Enter or Insert keys. To delete a point, click  in the toolbar or Delete key. To delete all points except the first two, click .

The Format Plot Area dialog is used to format axes and other elements of the plots. It can be accessed using any of the following buttons , , . Double click on either axis to shortcut to the dialog.

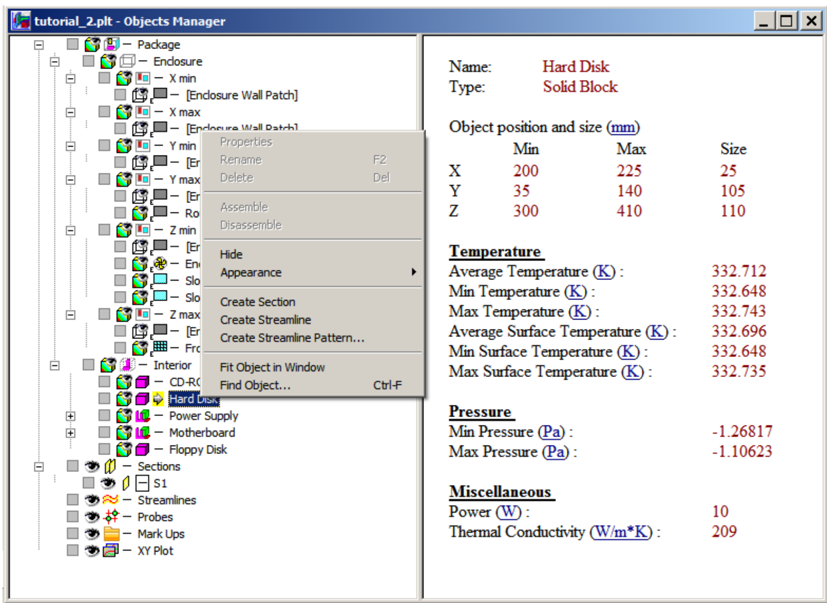
Columns in the table can be sorted by double-clicking any column header except Labels. Each double click switches the sorting order between unsorted, ascending and descending.

Sorting affects only the table; the graph panel remains the same no matter what sorting is specified in the table view.

The Copy command copies the selected table cells to the Clipboard in TAB delimited format. The Paste command copies the data from the clipboard (e.g. from Excel) into the CoolPlot table. The Paste command operates only when the Number of Intermediate Points is set to zero and the table is unsorted. You can paste data column by column or several columns at once. The table-selected area must have the same size and shape as the copied data.

10.6 CoolPlot Objects Manager

The CoolPlot Objects Manager consists of two panes. The left pane is similar to Coolit's Objects Manager, where you can select objects and define objects viewing options via "eye" icons.



The menus shown in the picture are activated via right mouse click anywhere in the populated portion of the left pane. The selected object can be fit to the 3D-view window by clicking the

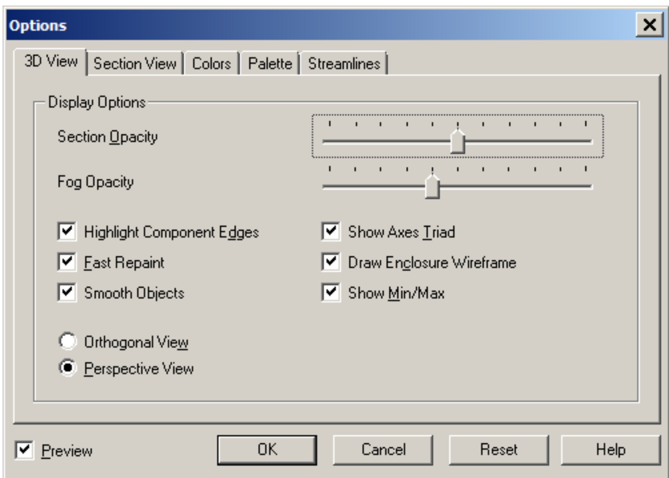
Fit Object menu. When a CoolPlot Object, such as a Section or Streamline is selected, the **Rename**, **Delete**, **Properties**, **Assemble**, and **Disassemble** menus become available.

As in Coolit, , , , and indicate components that are displayed in component type colors, as wireframe, with textures, and as hidden, respectively. The additional icons, and , indicate a component displayed in variable values and maximum variable value respectively. The displayed variable is selected in the pull-down menu Surface Var below the main CoolPlot toolbar.

The right pane of the Object Manager displays the name, position, and size of the selected object, as well as computed results. The displayed results depend on the object's type. The right pane information can be selected and copied to the Clipboard by highlighting it with the mouse, and pressing Ctrl-C. It can then be pasted into another Windows application, such as Excel, PowerPoint, or Word.

10.7 Options Dialog

Most viewing options can be set in the **Options** dialog, that can be opened from the Tools|Options menu.

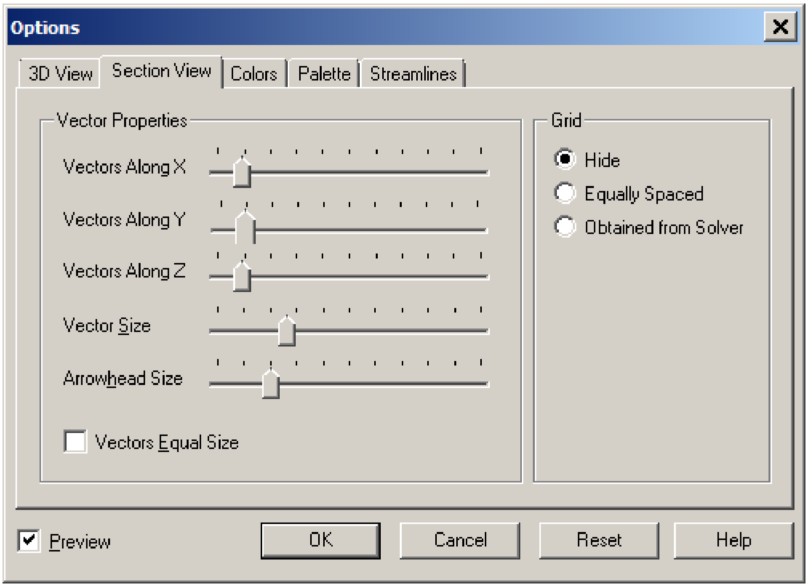


10.7.1 3D View

Display Options include the **Section Opacity** and **Fog Opacity** controls, **Fast Repaint**, and other viewing options. Checking **Fast Repaint** paints objects in maximum temperature only (or other surface variable) to speed up the repainting process during the animation. If your computer and graphics card allow it, unchecking Fast Repaint will enable the animation in full color.

When **Smooth Objects Geometry** is checked, the geometry of objects is shown with smooth surfaces, as in Coolit. However, depending on the grid, the geometry used in computing a solution can be different. Thus, for example, if you used a single grid cell inside a circular cylinder, the cylinder will be approximated with a rectangular parallelepiped. If Smooth Objects Geometry is checked, such a rectangular parallelepiped will be interpolated onto and displayed as a circular cylinder.

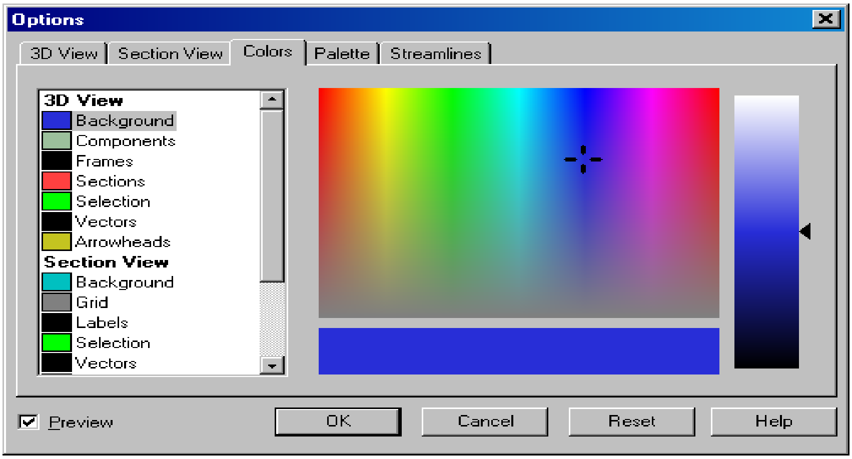
10.7.2 Section View



The Section View page controls Vectors in both Section and 3D Views and Grid in Section view. Under Vector Setup you can adjust the number of vectors displayed in each direction, the vector length, and the vector arrowhead size. To make all vectors the same length, check the **Vectors Equal Size** box. This option is particularly useful when velocity magnitude varies widely.

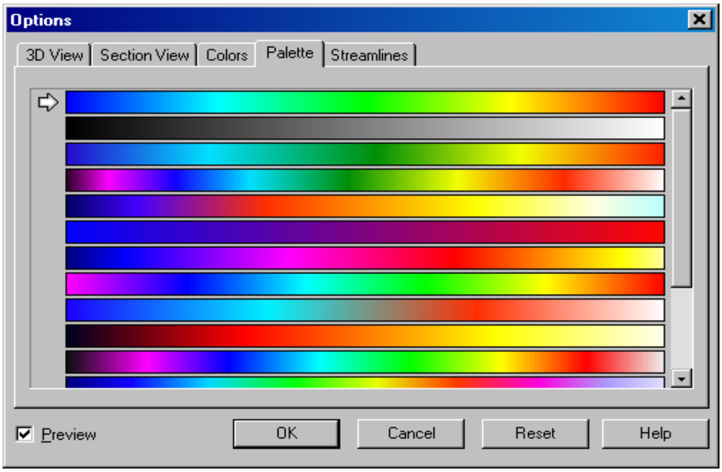
Under Grid, you can choose to hide the grid, show an equally spaced grid, or show the grid used to solve your problem.

10.7.3 Colors



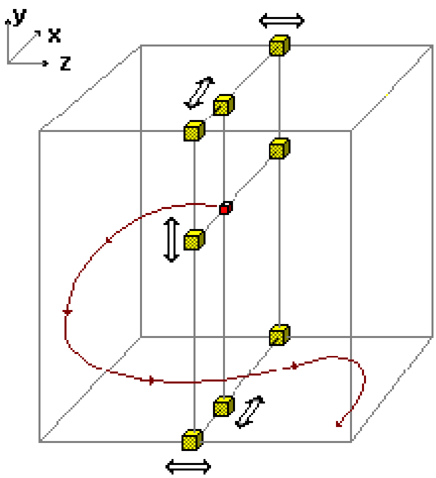
On the Colors page, you can set the colors of the features in all of the windows. Select the desired feature from the list at left. Then select the tone and value from the color field and value bar on the right. The presently selected color is displayed in the box at the bottom of the page.

10.7.4 Palette



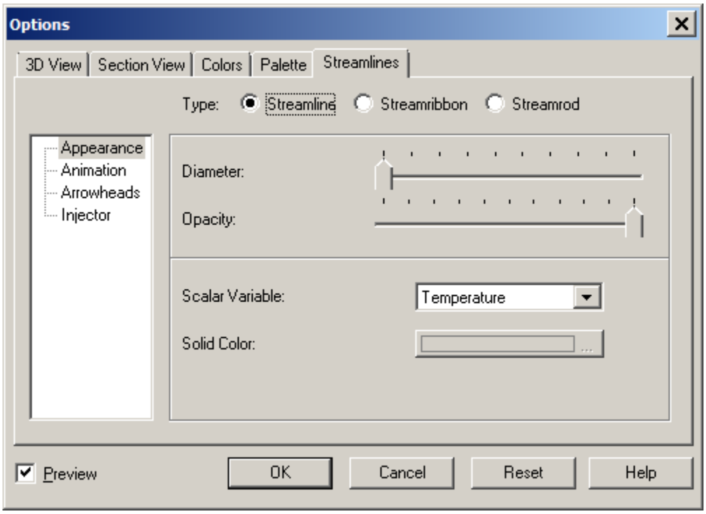
On the Palette page you can select the palette that your scalar values will be displayed in. Click on a colored bar to select it for your Palette.

10.7.5 Streamlines, Streamribbons, and Streamrods



To create a streamline, click the **Visualization|Streamlines** menu and select **Create One Streamline**. To delete a streamline, select the streamline and click  or press the Delete key on the keyboard. To create either a rectangular or circular streamline pattern click the **Visualization|Streamlines** menu and select **Create Streamline Pattern**. To create a pattern of streamlines on a Vent, Open, or Fan, double click on the component in the 3D view. The Streamline Pattern dialog will appear with the coordinates and dimensions of the component already entered. Click Apply to review the pattern before closing the dialog.

A streamline can be selected in Objects Manager or with a mouse. Once a streamline is selected, a crosshairs with eight handles will be displayed around the streamline injection point. Dragging these handles as shown in the picture above, will move the injector point.



Click the **Visualization|Streamlines** menu and select **Streamline Options** to open the **Options|Streamline** dialog. Here, you can switch the injector Type to Streamribbon or Streamrod, which provide information about both the flow direction and local swirl. Under **Appearance**, you can adjust the color, Opacity, and Width of your streamlines. The color of a streamline reflects the value of the scalar selected in the Scalar Variable pull-down box. Selecting None (Solid Color), will assign a color from the Solid Color box.

Under **Animation**, you can define the **Particle Size**, their quantity, and timing. Arrowhead size, color and spacing are set on the **Arrowheads** page. The **Injector** page lets you specify the Injector type. The default injector type is flow **Source**. The other two options are flow **Sink**, when the streamline ends at the injector, and flow **Source + Sink**, when streamlines both emanate from and end at the injector.

To animate Streamlines, Streamribbons or Streamrods, press . CoolPlot will generate a series of particles or ribbon/rod segments. As the particles traverse the region, you can rotate the domain to view the trajectory from different angles.

Similar to Coolit complex objects, you can assemble streamlines into groups using the Objects Manager. The group can be moved as a single object.

10.8 Additional Visualization and Animation Tools

In addition manually creating streamlines, CoolPlot can automatically distribute flow injectors over the Enclosure. Click the **Visualization|Animation** menu and then click either Using Ribbons or Using Rods. Flow animation will start. To stop the animation, click the Stop button. You can edit the properties of automatically generated streamlines, change their positions and manually add/delete new streamlines as desired.

The visualization of scalar fields, such as the temperature or pressure can be done in a variety of ways. You can select an existing Section or create a new one and then click the play button under **Animation** in the Section dialog. You can also visualize scalar fields by displaying streamlines and/or vectors and painting them in desired scalar color. **Fog** is yet another powerful visualization tool for scalars, activated by clicking

Visualization|Create Fog. You can adjust the opacity of Fog on the 3D View page of the Options dialog.

10.9 Reports, Still Images, and Animations

To print the 3D view or Section View, select the **File|Print** menu or click . If you wish to edit and annotate the picture before printing, or to create a digital image file or animation, or save computed results into a report file, click , or select **Reports|Project Report**, to open the **Report File** dialog. Project Report is an HTML document that by default contains only the project's summary: global settings, model statistics, solver settings, the grid and computed results as well as screenshots of 2D and 3D panes. Screenshots are taken when the Save As button is clicked.

In order to add project objects to the report click the **<<Add** button and select the desired objects. Standard Windows selection rules apply: for multiple objects selections hold down the Ctrl key and click objects, to select a range click the first object, hold down the Shift key and click the last object or drag the mouse around the selected objects. Selecting a complex object or a section, e.g. Interior, by default selects every object below the selected branch.

Use the **Mark Up Image** menu to insert text, lines, and ellipses into the 3D view.

Click **Make Screenshot** to capture the image inside the displayed frame of an active pane. Move and resize the frame as desired and then double-click inside it to save the screen capture to Clipboard to be pasted into other documents.

Click **Save Image As** to save a true color file of the active CoolPlot window in BMP, PNG, or JPEG format. After you specify the file name, you will be asked to specify image size before the file is created.

Load Image into MS Paint menu saves the active CoolPlot window into MS Paint. You will be asked to specify image size before MS Paint is opened. You can then edit the image and save it in any of MS Paint provided formats.

Streamline Animation to AVI File menu creates an AVI animation file for the currently active view with streamlines, ribbons, or rods.

Timestep Animation to AVI File creates an AVI animation file for the currently active view of a time-dependent simulation. The simulation must contain at least two time steps.

In animation cases, the movie image will be the same size as your active window. To reduce the movie file size, make the window smaller before creating an AVI file.

11 Model Check

Using this dialog, you can check your model for Conflicts and Warnings**.** When a Conflict is generated during model construction, the icon will appear on the Status Bar. The icon on the Status Bar indicates warnings. **Conflicts** must be fixed before the grid can be generated and the solver run. **Warnings** will not stop the solver from running.

You can access Model Check by clicking , selecting **Package|Model Check**, or double clicking the icons in the Status Bar, if present.

The left side of the dialog contains a tree displaying both Conflicts and Warnings. When the box next to a Warning type is checked, Coolit will prompt you to look at the Model Check dialog, if case contains Warnings. Sometimes Warnings are about issues you can’t do anything about, such as, for example, very thin fins, which produce closely spaced grid lines. You can choose to ignore Warnings and start the solver by selecting Cancel at the prompt.

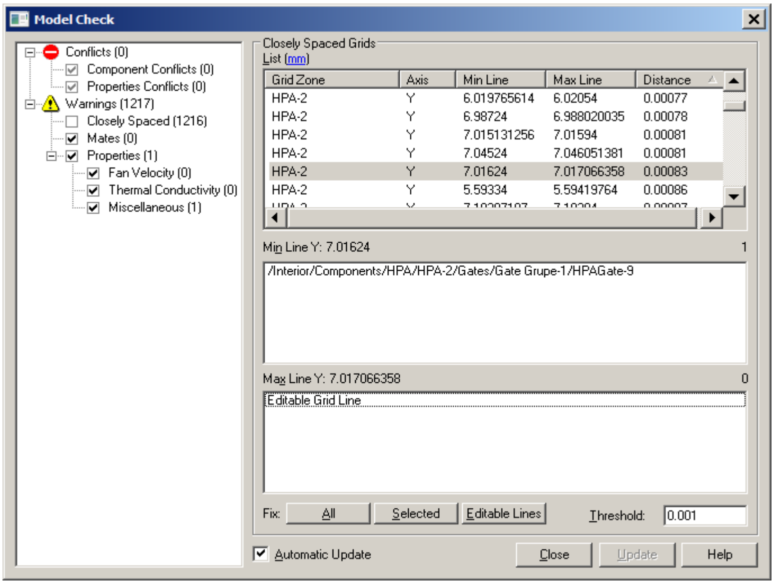
11.1 Conflicts

Component Conflicts – construction errors such as Plate on an Open or intersecting Cylinder components will cause a Components Conflict. When you click on Component Conflicts in the tree, a list of specific conflicts appears in the upper right pane. Click on a Conflict and the components involved in the Conflict will be displayed in two panes below. Double clicking on any component in these panes will highlight that component in the Coolit construction views.

11.2 Warnings

11.2.1 Closely Spaced

Two closely spaced grid lines create small grid cells that can cause round-off and convergence problems. Coolit uses non-dimensional **Threshold,** set in the lower right of the dialog, to evaluate the spacing of grid lines.



When you click on Closely Spaced in the left pane, a list of grid-line pairs appears in the upper right pane. Each entry is identified by its Grid Zone, the Global Axis on which the warnings occur, the locations of the Minimum and Maximum lines that cause the warning, and the Distance between the lines. Click on an entry in the list and the specific components that create the Minimum and Maximum lines will be displayed in the two panes below. Double clicking on any component in these panes will highlight that component in construction views. You can then move or resize components to remove closely spaced anchor lines or, if it is editable (magenta) grid lines, you can delete/move them as desired.

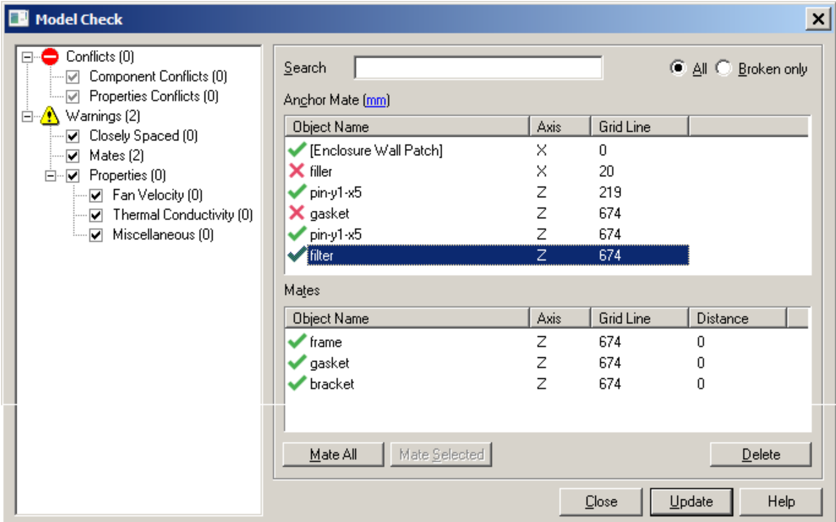
You have the option of automatically fixing closely spaced grids by ignoring differences in grid-line spacings defined by Threshold. You can always fix all Warnings that involve Editable Lines (magenta) either manually or by clicking the Fix **Editable Lines** button. You can try fixing all flagged components by clicking the Fix **All** button or only the selected ones, by clicking Fix **Selected**. A dialog appears with suggested anchor grid tolerances (these same settings are available in the **Package|Grid Settings…** menu under the Parameters tab). If you choose OK, the suggested settings will be applied and some components will be resized when the model is solved (this will not affect the way they are displayed or their dimensions in Coolit). Components will not be resized if the operation will result in the elimination of a gap between facing components. You can override this restriction by unchecking the **Do Not Join Components** box (not recommended).

If you wish to undo Fix operations on anchor grid lines, open the **Package|Grid Settings…** menu, select the Parameters tab, and set Anchor Grid Tolerances back to zero. The deleted editable grid lines are not restored by this operation. You can also re-enable the **Do Not Join Components** option here.

If the **Automatic Update** box is checked, the entire Closely Spaced Grids table is updated immediately upon every grid change. If you prefer to update it manually, uncheck the Automatic update box and use the Update button.

11.2.2 Mates

When you click an Anchor Mate in the top pane, its mates are shown in the lower pane.



If all mates of a group are mated, a green check mark is displayed next to the Anchor Mate.

To delete a Mates group select its Anchor Mate and click Delete. To delete a mate select it from the Mates pane and click Delete.

11.2.3 Properties

<u>Fan Velocity</u> – Coolit will check to see if the velocity specified for a fan is zero or if it exceeds the value in the Max Velocity field. Double clicking on a fan in the list will bring up the Fan Properties dialog. You can increase the Max Velocity value if fans in your problem have velocities higher than the default.

<u>Thermal Conductivity</u> –- Coolit will check to see if the thermal conductivity specified for a component is very low or exceeds the value specified in the Max Thermal Conductivity field. You can increase the Max Thermal Conductivity value if components in your problem have thermal conductivity higher than the default. Double clicking on a component in the list will bring up the properties dialog for that component. If the conductivity of your component is very low, you can set it to zero here and Coolit will not compute conduction through the component.

Miscellaneous – Coolit will check for potential errors such as components that have power and zero thermal conductivity, models with Enclosure Fans but no Enclosure Opens or Vents. It will also attempt to determine flow regime based on specified boundary conditions and flow and heat sources. If the software-determined regime is different from the specified, it will issue a warning. Such a warning is by now means definitive, but rather a suggestion to double check assumptions and calculations, which went into your flow regime definition.

12 Calculator Tools

12.1 CoolCalc

CoolCalc allows you to convert from one unit system to another and to compute mathematical expressions. To open CoolCalc select **Tools|Start CoolCalc** or click the icon. Scroll through the drop down menu to select the variable you want to work with. Next, select the unit in the second row of boxes you want to convert from, and fill in the value. In the third row, select the unit you want to convert to. The converted value will appear immediately in the value box. The above description is an example only; you can input values in any order.

Each window is also a calculator. Instead of a number as in the example above, you can input expressions of the form: 2.1*3/2-4.5*sin(pi)*e^2. Expressions up to 22 characters in length are permitted in the current version. The following operations and functions are supported:

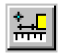
Command		Meaning
Operations	+	add
	-	subtract
	*	multiply
	/	divide
	^	raise to power
Constants	pi	3.14159265358979323846264 33
	e	2.71828182845904523536028 74
	deg	pi/180
Functions	sin(x)	sine of x
	cos(x)	cosine of x
	tg(x)	tangent of x
	arcsin(x)	inverse sine of x
	arccos(x)	inverse cosine of x
	arctg(x)	inverse tangent of x
	lg(x)	log base 10 of x
	ln(x)	natural log of x
	sqrt(x)	square root of x
	mod(x,y)	remainder from division of x by y

12.2 Calculator Cells

The Position entry boxes in Control Bar, and in Distance Toolkit, and the Lead/Pins/Balls Area for the new Contact Resistance component are calculator cells. Calculator cells are marked by the = sign. The calculator provides all the same arithmetic operations, trigonometric and algebraic functions, and constants as CoolCalc. Using calculator cells, you can enter positions of components as a formula, e.g. 200.3+7.1*3. This makes it convenient to specify positions of components with respect to other components. Thus, if a component has to be 23.5 mm away from another component that is at 182 mm, you can type in 182+23.5 and press Enter to compute the result.

13 Positioning and Sizing Tools

13.1 Distance Toolkit

The Distance Toolkit is opened by pressing the  icon. It is used for determining the distance between a **Reference Point** and a selected object, and moving and resizing components.

When first opened, the dialog will report Reference Point coordinates with respect to the global coordinate system. The position of the reference point is specified in the **Set Reference Point** dialog. This dialog can be opened by clicking  or by pressing the Edit button in the Distance Toolkit dialog. With the Distance Toolkit dialog open, click on an object to select it. The Distance Toolkit will then display the distances from the reference point, to various positions on the object.

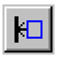
Using the Distance Toolkit, you can move and resize components, either with the mouse, or by using tabular inputs. To move or resize an object, press the **Move** or **Resize** button, respectively, and type in the new coordinates. Note that these coordinate cells are calculator cells. Click the Apply button or use the Enter key to do any calculator cell calculations and make any position or size changes.

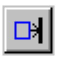
13.2 Measure Tool

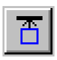
The distance Measure tool is used for positioning and measuring the distance between objects. Select **Edit|Measure** or click,  to start Measure tool. One point in the domain, by default it is (X, Y, Z) = (0, 0, 0), is shown in red. Click any black point in either 2D or 3D views and Coolit will display a dashed line between the red and the blue point. The Measure dialog will show the distance between the points. The points can be placed by entering a numeric position in the dialog, or by selecting a vertex of a specific object. The red point can be fixed to any position, or allowed to move to the blue dot's last position. If the red dot is fixed to a specific vertex, and a new position is entered for the point, the object will be moved so that the vertex the red point is at, is moved to the entered position.

13.3 Alignment Toolbar

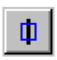
The Alignment Tools are used to align objects in any of the 2D view windows. The alignment function is always relative to the active 2D window. More than one object must be selected to activate the alignment tools. It is recommended that before selecting objects to align, you click  to unselect any previously selected objects. In order to select an object you can either check the box to the left of the object's name in the Objects Manager, or hold the Shift key and click (once) the object, or drag a “rubber band" around the object(s). The selected objects will be displayed with a blue outline. The first selected object is the "anchor" object; it is not moved when the alignment tools are used; the other selected objects will be moved relative to it.

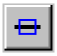
 will move the selected object(s) so that their left sides are aligned.

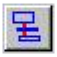
 will move the selected object(s) so that their right sides are aligned.

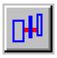
 will move the selected object(s) so that their top sides are aligned.

 will move the selected object(s) so that their bottom sides are aligned.

 will align the center(s) of the selected object(s) vertically

 will align the center(s) of the selected object(s) horizontally

 will place the selected objects an equal vertical distance between two bounding objects

 will place the selected object an equal horizontal distance between two bounding objects.

13.4 Lock Position command

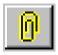
Lock Position command locks objects with respect to their parent in the Objects Manager tree. Thus, if locked object is on the top level, its position is locked with respect to Enclosure. If locked object is inside of a complex object, its position is fixed with respect to the complex object. But the complex object itself can be moved with respect to Enclosure and other objects.

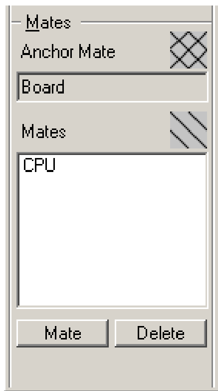
Right-mouse-click an object in the Objects Manager and select Edit|LockPosition. The locked object will now display with a golden lock:

and the Lock Position button in Objects Manager will be depressed. To unlock a locked object, click the button again.

13.5 Mates

Mates function creates a positional relationship between components. Components designated as Mates are supposed to be in contact along marked surfaces. If they are not, you get a warning on the Model Check dialog's Mates page.

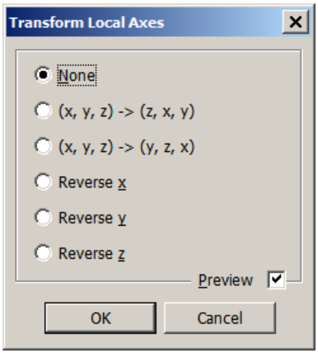
To mark mates, click  on the toolbar. The cursor will change to a paperclip. In 3D view, use the cursor to mark the side of the first component, which will serve as the anchor plane. The anchor component's surface is marked with a crosshatch. Using the cursor mark the other mates of the group, which will be indicated by regular hatching. All the marked mates are added to the table at the bottom of Control Bar. Once finished marking, click the Mate button on Control Bar. All the mate components will snap to become coplanar with the anchor mate plane.



To move a mate into position in the anchor mate's plane, it's best to use the pane view coplanar with the anchor mate plane. After components are mated, Coolit permits breaking the mate contact. If this happens, you get a warning in the Model Check dialog's Mates page. The broken mates are indicated by **X**. If you wish to fix a broken mate, select it and then click the Mate button at the bottom of the Model Check dialog's Mates page.

13.6 Local Coordinates Rotation Tool

Components with directional properties such as anisotropic Solid Blocks, TECs etc. often need to be rotated in place, so that the body of component does not move, while its local coordinate triad, which defines the direction of properties, is rotated. This can be accomplished by right-mouse-clicking the component that needs to be transformed and selecting the Transform Local Triad (or click the Ctrl+T hot key).



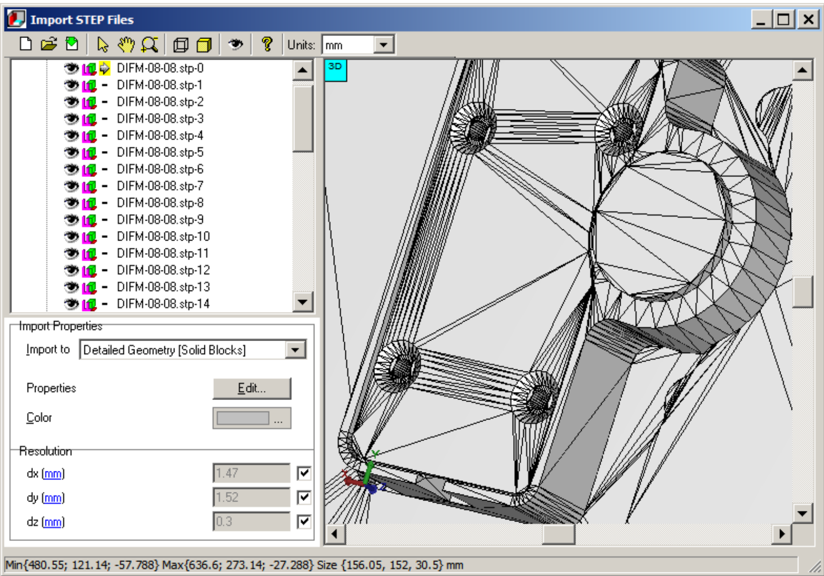
In the dialog select the desired transformation, visually verify it and click OK to exit.

14 Import/Export

14.1 Import/Export of STEP Files

To import STEP files into Coolit, select File|Import|STEP File. Click  to select the desired *.stp file to import.

Make sure that the units displayed in the Units box match your import file's units (i.e. if you exported from CAD software in millimeters, make sure you are importing in millimeters). In the left window, select parts you wish to import. You can select the entire assembly, a group of parts, or one part at a time. In the **Import to** box select the type of Coolit object you wish to import your STEP part as. You can assign a color and properties to the imported object by clicking the Color or Edit… buttons.



The import's accuracy is controlled by the Resolution section settings. Coolit displays the parts geometry as specified in STEP, not as approximated. The approximation is done with the user-specified accuracy and not shown. The displayed features smaller that the specified Resolution may or may not be there. If it is important to import those small geometry features, you should change Resolution settings accordingly. Note, that finer Resolution will require a finer grid and hence longer solution times and more RAM. You can see whether the geometry feature was accounted for by looking at the grid. If, for example, a 1 mm hole is shown inside of larger grid cell, it is definitely not accounted for.

Import the selected part(s) by clicking . The imported parts will now be hidden in the Import STEP Files dialog window, but will show up in Coolit and will be listed there under Project Objects in the Objects Manager.

To export Coolit projects in STEP format select File|Export|STEP File… and specify the exported file name and units.

14.2 Import/Export of STL Files

STL files can be exported from most CAD packages. If your project is an assembly, it has to be exported as such in order for import to work correctly. Here we will describe the export procedure from SolidWorks, it is done analogously from other packages.

Click SaveAs, and select STL as the Save as type.

Click Options, select Binary file format and make sure that the check box "Save all components of an assembly as a single file" is unchecked.

Click OK to export the files to the specified directory.

To import the resulting files in Coolit, select File|Import|STL File. Click to select the desired *.stl part files to import.

Make sure that the units displayed in the Units box match your import file's units (i.e. if you exported from CAD software in millimeters, make sure you are importing in millimeters).

In the left window, select parts you wish to import. You can select the entire assembly, a group of parts, or one part at a time. In the **Import to** box select the type of Coolit object you wish to import your STL part as. For example, instead of bringing in a cylinder as a construction of Solid Blocks, select Bounding Cylinder in the Convert to box. You can assign a color and properties to the imported object by clicking the Color or Edit… buttons.

Import Accuracy defines the ratio of volumes of the original CAD (STEP) object, V_{CAD}, to its volume after it is imported into Coolit, V_{Coolit}, as follows:

Import the selected part(s) by clicking . The imported parts will now be hidden in the Import STL Files dialog window, but will show up in Coolit and will be listed there under Project Objects in the Objects Manager.

To export Coolit projects in STL format select File|Export|STL File… and specify the exported file name, units, type (binary or ASCII), and whether to export each part separately or into a single file.

14.3 Import/Export of IDF Files

Go to **File|Import** and select **IDF File**. The IDF Import dialog will open, and on the first page you can specify a board file and component library files that you want to import. Press the Next button. If all the components were successfully imported, you will get to the Settings page, otherwise, the In Progress page will display components that Coolit could not find in the libraries in the Status window, e.g.

Undefined component: U26 - ignored.

Undefined component: U27 - ignored.

Undefined component: U29 - ignored.

Undefined component: X3 - ignored.

Files were successfully loaded.

On the Settings page, Coolit will list the imported board name and the imported components sorted by their Ref. Des. At this stage, you can decide which of the imported components to import into a Coolit model. By default, all of them will be imported, which is indicated by the check mark next to the component's name.

You have several tools for selecting components. First, you can check/uncheck boxes next to the components' names. For more options, see the pull-down box under the Imported Components window, which lets you unselect all components or unselect components smaller than a certain size, or components starting from a certain Ref.Des. or Part Number. For example, if you wish not to import components that are smaller than 0.5", select: Size smaller than (in), and type 0.5 in the field below. Click Unselect to finish. The imported board with components will appear under Project Components in the Coolit Objects Manager.

To avoid potential round-off problems, you can specify the precision for both the component size and position. The default is 0, i.e. components are imported exactly as specified in the IDF file. If you specify Precision to be, e.g. 0.5 mm, the components' sizes and positions will be rounded off to the nearest 0.5 mm. For example, a component with a 10.234 mm linear size and 173.546 mm position will be rounded off to 10.0 mm and its position to 173.5 mm.

By default, Coolit imports all components as Solid Blocks; it does not import cutouts in the board, and all components are imported as their bounding boxes.

If you want the imported board and components converted into Coolit PCB objects: PCB proper and resistor models of chips, check **the Convert to PCB Object** option. Check **Import**

Board Cutouts and **Precise Board Outline** if your board is non-rectangular and/or it has cutouts.

To export PCB objects in your project in IDF format select **File|Export|IDF File…** and specify the folder for the exported file.

14.4 Import of IC Package Thermal Models

This function is part of the optional IC Package Constructor and Library module. Coolit can import thermal models of IC packages specified in Icepak format. Such models are usually available for download from manufacturer's web sites. After you download models you need, email them to support@daat.com. The models will be converted to Coolit format (with the ICP file extension) and emailed back to you within 2 business days. To import ICP files, click **File|Import|IC Package File**.

14.5 Import DXF Traces

Click Import DXF… to open the Import DXF Layer dialog. Click  to select and load the *.dxf file to import. Specify units used in the dxf file and click  to import. The imported file will be listed as a layer on the Properties page in the Printed Circuit Board dialog.

14.6 Imported Object Properties Dialog

You can edit the visual, material, and geometry properties of STEP and STL imported object in the Imported Objects Properties dialog, which is activated by double clicking an imported object either in the Objects Manager or directly in one of Coolit panes. You can change the component's Name, Material Properties, and Color. You can also modify the imported geometry by removing small spacings (uncheck the box and specify minimum size in *Remove spacings smaller than*), change the Import Accuracy, and the Imported Object

Grid Tolerance. **Import Accuracy** defines the ratio of volumes of the original CAD (STEP) object, V_{CAD}, to its volume after it is imported into Coolit, V_{Coolit}, as follows:

Import Accuracy (%) = $| V_{CAD} - V_{Coolit}| / V_{CAD}$

The smaller the Import Accuracy value, the better the import, but more and finer grids are required. The default is 1%.

Imported Object Grid Tolerance defines the minimum spacing allowed between Imported Object's grid lines (light gray) in each coordinate direction. Spaces smaller than the specified Tolerance are automatically removed by resizing the offending grids/components.

14.7 Transform Local Triad

CAD files don't contain physics and hence components such as TECs, Resistor Models, Solid Blocks with anisotropic conductivity etc. can be imported with incorrect orientation. Coolit employs an algorithm to guess the orientation, but it is about 50% accurate, so that roughly half of your anisotropic components may need to be flipped. To do that you should right-mouse-click the component you want to flip, select the Transform Local Triad and then select the orientation. By default the Preview box is checked, so that you can immediately see the result of your selection.

14.8 Import/Export Power File

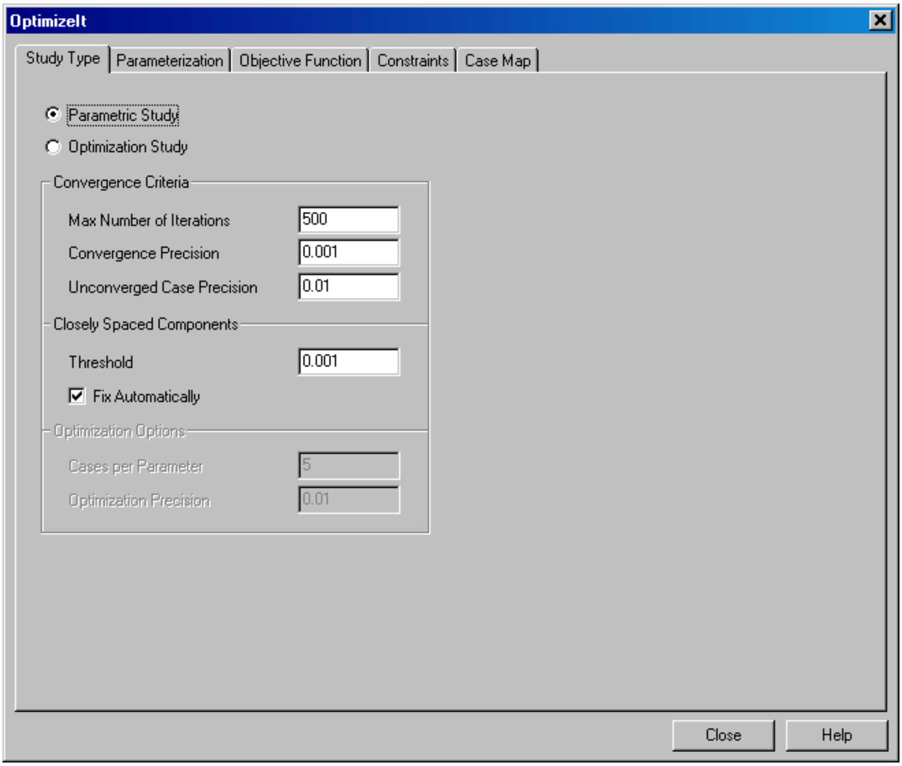
Power File is generated by the File|Export|Power File menu and selecting either All Objects... or Selected Objects… The resulting Excel-formatted spreadsheet can be edited and imported back into Coolit. There are two fields in the file: the objects name, column A, and the objects power, column B. If an object has variable power, column B will contain the word Table N, N=1, 2, …, and the referenced table can be found in columns D and E of the spreadsheet.

15 OptimizeIt Module

OptimizeIt is an optional module that can be used to study a system's sensitivity to design variations and optimize the system's design.

15.1 Study Type Page

The Study Type page is where you select the type of study you want to perform. In a **Parametric Study**, all possible combinations of entered parameters are tested. In an **Optimization Study**, OptimizeIt will automatically select parameter values within the specified ranges to test using an optimization algorithm.



The Max Number of Iterations and Convergence Precision settings are the same as in Solver Settings. Adjustments made here will be reflected in the **Solver Settings** dialog, and vice versa.

The Unconverged Case Precision is used in Optimization Studies for cases that have not fully converged. If a case in an Optimization Study has converged such that its residuals are less than the Unconverged Case Precision, the Objective Function value for that case will be used in the optimization process. Cases with residual values greater than the Unconverged Case Precision will be ignored by the optimization procedure.

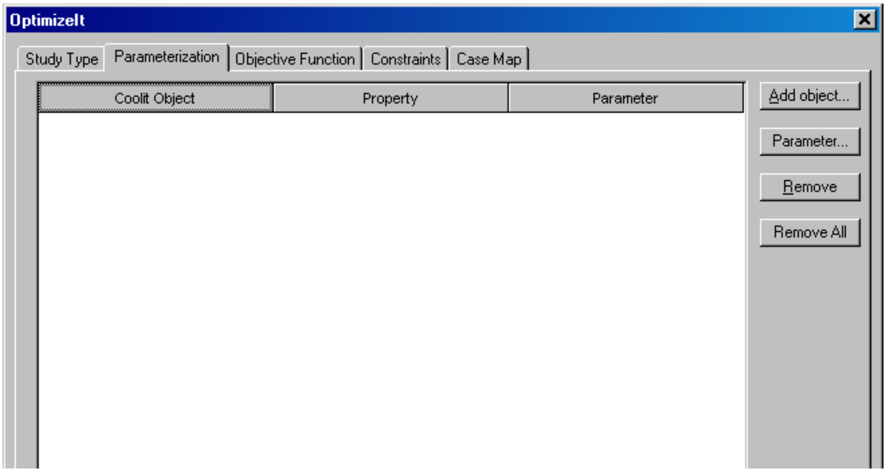
The Closely Spaced Components settings are the same as those in Model Check. Changes made here will be shown in Model Check and vice versa.

The minimum number of Cases per Parameter is available only if Optimization Study is checked. The higher the number of Cases per Parameter, the more likely that you will find your optimum design. However, this may require more cases to be run. The recommended (default) is 5 cases per parameter.

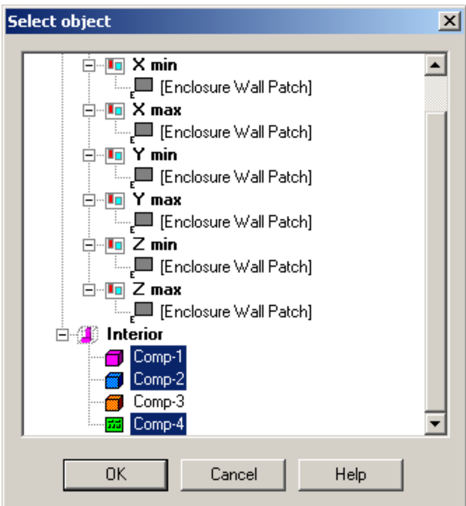
The Optimization Precision is the threshold for stopping the optimization. If the percent change in the Objective Function value, from one case to the next, is less than then Optimization Precision, the case will be considered an extremum.

15.2 Parameterization Page

The Parameterization page is where you define the Objects to be parameterized, the Properties of each Object that will be parameterized, and the numeric values of the parameters.

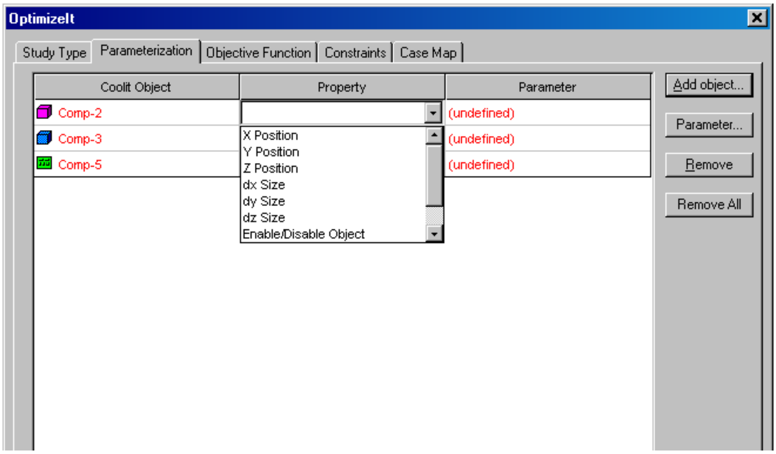


Objects are added by clicking the **Add object** button, which opens the Select Object dialog.



Multiple objects can be selected by dragging a box around them, holding the Shift key to select consecutive objects, or by holding the Ctrl key and clicking different objects.

Parameterizable properties for a selected Object appear in a pull-down menu when any cell in the **Property** column is clicked.



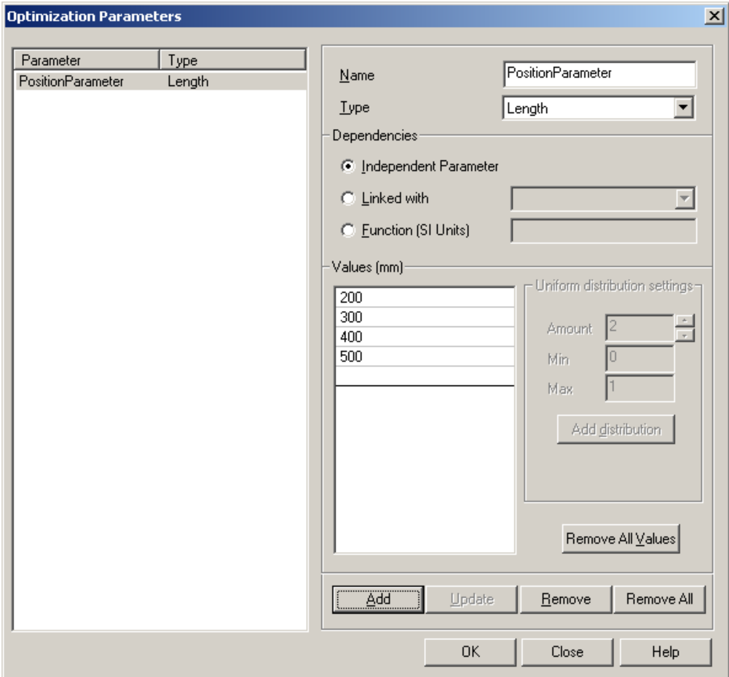
Parameter values for selected Property can be defined by clicking the **Parameter** column cell and selecting the name from the pull-down menu. Select **New Parameter** from the pull-down menu to define a new Parameter for an object's Property. Parameters can be edited by clicking the **Parameter…** button.

Incorrectly or incompletely defined table entries are indicated by red text. The status line below the table will provide descriptions of errors. You can leave table entries undefined, but the associated Parameter will be skipped when generating the Case Map.

15.3 Optimization Parameters Dialog

In this dialog, you can create new, or modify existing Optimization Parameters. To create a new Optimization Parameter, enter its Name, Type, Dependencies, and fill the Values table. Click the **Add** button to add the created

Parameter to the Parameter list at left.



Parameter Name and Type

Parameters are referenced by their Name and Type. The Name must start with a letter, and may contain only numbers and letters (no spaces allowed). Mathematical functions or constant definitions (i.e. Sin, Cos, Pi, e, etc.) cannot be used as Names. The chosen Parameter Type must be consistent with the associated Property. For example, if your Property is an object's size, you must specify Length as the Parameter Type. Dimensionless Properties such as a Heat Sink's Number of Pins, have the Dimensionless Parameter Type.

Parameter Dependencies

A Parametric Study can include one or more Optimization Parameters. **Independent Parameters** are changed individually. While an Independent Parameter is being changed, all other Independent Parameter Values are held constant. Case Maps will contain all possible combinations of Independent Parameter Values. For example, if you use two Independent Parameters that have, N and M Values, OptimizeIt will generate N*M Cases.

If there is a mutual dependence between Parameters, choose the **Linked with** Dependency to link one Parameter to another. The linked Parameters will change simultaneously. For two linked Parameters, with N and M Values, OptimizeIt will generate either N or M number of cases, whichever is smaller. An example of Linked parameters would be a fan and shroud complex object. The shroud has to fit the fan, so their sizes must change simultaneously (be linked). Three or more Parameters can be linked together by linking them all to the same Independent Parameter.

A **Function** Dependency allows you to specify a mathematical relationship between Parameters. Functions can include common mathematical operations, functions, constants, and Names of previously defined Parameters. The Function Parameters will be calculated automatically for each Case and their values will be shown in the Case Map.

Since Parameters are generally dimensioned quantities, the Function Dependency calculator converts all units to SI, performs the calculation, and then reconverts the units back to your specifications. This convention allows you to mix and change units. For example, suppose a manufacturer supplies you with a specification for a heating strip with a power dissipation, P, in BTU/min and a length, L, in inches:

$$P = 2.345\, L^2$$

The coefficient 2.345 has units of BTU/min/in^2. OptimizeIt assumes that all coefficients are entered in SI units. Hence the correct coefficient would be:

2.345 [BTU/min/in^2] = 63915 [W/m^2]

The function should be entered as 63915*L^2. From now on, OptimizeIt, will automatically make the correct unit conversions for function evaluation.

Parameter dependencies have no effect in Optimization Studies as parameter values are selected automatically by OptimizeIt. All parameters in an Optimization Study should be specified as Independent Parameters.

Parameter Values

Parameter values are defined in the **Values** table. Parameter values can be entered manually or by using the **Uniform Distribution** tool. You can create a range of values by specifying a minimum and maximum value, and step size, or a minimum value, step size, and number of steps. To modify an existing value, click on it and retype the value.

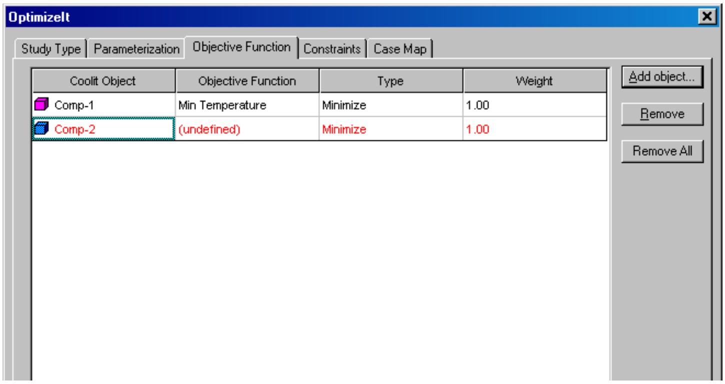
In Optimization Studies it is only necessary to specify the maximum and minimum values for parameters. OptimizeIt will automatically select which values to test within the specified range.

Modification of Existing Parameters

All Parameters defined in your project are shown in the parameter list on the left side of the Optimization Parameters dialog. If you click on a Parameter from the list, its information will be loaded into the dialog for editing. Note that if a Parameter has already been associated with an object and Property, you will not be able to edit the Parameter Type, or remove the Parameter. You will have to select another Parameter, and then delete the old one, or remove the entire row in the Parameterization page.

15.4 Objective Functions Page

In the Objective Function page you can set selected objects' attributes (i.e. maximum temperature, average surface temperature, pressure) to be solved for as the various design Parameters are applied. By analyzing the values of the Objective Functions reported on the Case Map page, you can choose your optimum design Parameters.



The total number of Objective Functions is not limited, but only certain object attributes can be specified. For now, supported Objective Functions include: minimum temperature, maximum temperature, average surface temperature, minimum surface temperature, maximum surface temperature, minimum pressure, and maximum pressure.

Applicable **Objective Functions** for a selected Object appear in a pull-down menu when any cell in the Objective Function column is clicked. The Objective Function is the Object's feature you want to optimize.

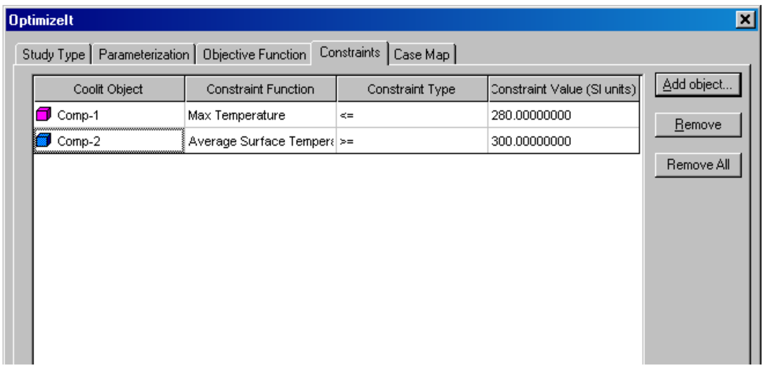
The **Type** of objective function can be selected from the pull-down menu that appears when any cell in the Type column is clicked. The Type is the kind of optimization you want to perform; either maximizing or minimizing.

The **Weight** of the Objective Function can be specified by clicking any cell in the Weight column. It is only used in Optimization Studies when a single objective function is formed as a linear superposition of all Objective Functions defined in the dialog. The Weight is the relative importance of the constituent Objective Functions.

Incorrectly or incompletely defined Objective Functions are indicated by red text in the table. The status line below the table will provide descriptions of errors. You can leave an Objective Function undefined, but it will be skipped when generating the Case Map.

15.5 Constraints Page

The Constraints page is where additional design constraints can be specified in an Optimization Study. OptimizeIt will use these design constraints while finding the specified objective function. Constraints can be used only in Optimization Studies.



Applicable Constraint Functions for a selected Object appear in a pull-down menu when any cell in the Constraint Function column is clicked. The Constraint Function is the Object's feature you wish to apply a numeric requirement to.

The Constraint Type for an Object can be selected from the pull-down menu that appears when any cell in the Constraint Type column is clicked. The Constraint Type is the inequality you wish to apply to your Object's numeric requirement. Possible constraint types are <= (less than or equal to), == (equal to), or >= (greater than or equal to).

The Constraint Value for an Object can be specified by clicking any cell in the Constraint Value column. Note that all Constraint values must be entered in SI units.

Red text in a table entry indicates that the entry is incomplete or is defined incorrectly. The status line below the table will provide a description of the error.

15.6 Case Map Page

The Case Map page is where the various Parameter combinations and their respective Objective Functions are listed. It is used for starting and stopping the optimization process and exploring individual Cases. Each Case represents a different combination of Parameter values. The Case models and solutions are stored in a separate folder. When optimization starts, Coolit will solve each Case, compute the Objective Function, and update the table information.

Each row in the Case Map table provides information about a specific Case. This information includes the Parameter Values, Case status icon, Case filename, and the computed Objective Function values. If you have a previously built Case Map, and press **Build Map**, OptimizeIt will warn you that all entries in the Case Map will be lost if you proceed with the build. If you are performing an Optimization Study, you will be given the option to use the already existing case map, or to build a new one. It is generally beneficial to use any previous results, as it will avoid re-running cases. It is also helpful if you have already run a parametric case and have bracketed your optimum configuration. The information from the table can be exported by pressing the **Copy** button and then using the paste command in an external spreadsheet editor.

Case status icons indicate the following:

Icon	Meaning
	The Case is waiting for processing and is not solved yet; Objective Functions have not been computed.
	The Case was solved successfully, Objective Functions were computed, and a CoolPlot file is available for viewing.
	The Case contains Conflicts and cannot be solved. You can view the Case to explore what caused the Conflicts by pressing the View Case button and then viewing the Model Check dialog.
	The Case is currently being solved.
	The Case was not built and solved due to an incorrect parameterization rule (e.g. specified heat sink's Number of Pins = 0).
	The Case did not converge within the specified Maximum Number of Iterations (default is 500). Cases indicated with this icon can be Restarted.

The Case Status can be changed by clicking on the status icon. You can exclude Cases from the optimization process by changing their Status icon to . If the  icon is shown, you can either restart from an already solved, but not fully converged case, or you can choose to change the Case Status to  to Start, to solve the case from scratch. Objective Function values can be sorted in ascending or descending order by clicking the Objective Function column header.

15.7 Messages and Warnings

Messages appearing in OptimizeIt Details Section. Data in braces {}, e.g. {100} depends on additional optimization information and are different for different messages.

Message Text	Message appears when...	Info
Processing Case Definitions...	OptimizeIt is initializing Parameterization Page	
{10} Case Definition(s) valid	OptimizeIt has finished initializing Parameterization page	Indicates number of valid Parameters
Processing Objective Functions...	OptimizeIt is initializing Objective Functions page	
{10} Objective	OptimizeIt has finished initializing	Indicates number of

Function(s) valid	Objective Functions page	valid Objective Functions
Processing Constraints...	OptimizeIt has finished initializing Constraints Page	
{10} Constraint(s) valid	OptimizeIt has finished initializing Constraints Page	Indicates number of valid Constraints
Objective Function #{line} invalid – {reason}	Line {line} invalid in Objective Functions Page for reason {reason}	Possible reasons: 1. Parameterized object not selected 2. Objective Function not selected
Removing Invalid Cases...	OptimizeIt is removing invalid cases from the Case Map	
{999} Case(s) valid	OptimizeIt has found {999} valid cases.	Indicates total number of cases in the Case Map after Build
For Parametric Studies Only		
Case Definition #{line} invalid – {reason}	Line {line} invalid in Parameterization Page for reason {reason}	Possible reasons: 1. Parameterized object not selected 2. Property not selected 3. Parameter not selected 4. Parameter type and property type incompatible 5. Parameterized object and property incompatible
Case Definition #{line} invalid - parameter {Param1} has no values	Parameter {Param1} referenced in line {line} in Parameterization Page has no values	This parameter was skipped when building the Case Map.
Case Definition #{line} - parameter {Param1} treated as independent	Parameter {Param1} referenced in line {line} in Parameterization Page is linked to another parameter that doesn't appear in Parameterization Page.	This parameter was treated as independent when building the Case Map.

Case Definition #{line} invalid - function {Term} in parameter {Param1} cannot be evaluated	Coolit cannot evaluate the function for {Param1}(e.g. Function has reference to other parameter that doesn't appear in the Page).	Example: F = Param2+500.0, but Param2 doesn't appear in the Parameterization Page or has no values. This case is skipped when building the Case Map.
Function {Func} undefined at {Arguments}; case removed	Parameter {Param1} referenced in line {line} is undefined for the specified set of values.	Example: F = 500/Param2, but Param2 = 0.0 This case is skipped when building the Case Map.
Linked parameter {Param1} undefined when parameter {Param2} = {100 mm}; case removed	Parameter {Param1} referenced in line {line} is Linked, but is undefined for certain value of {Param2}.	Example: Param 1 = {0, 10}, Param2 = {0, 10, 20} Param1 is undefined when Param2 = 20.
	For Constrained Studies Only	
Constraint #{line} invalid – {reason}	Line {line} invalid in Constraints Page for reason {reason}	Possible reasons: 1. Parameterized object not selected 2. Constraint not selected
Testing bounds...	Start testing parameter ranges	
Wrong Bound {case arguments}	Enclosure or component conflicts	Check for geometric or component conflicts
Setup complete, Press Start to begin Optimization Process	Constrained setup is ready	
Estimated number of cases – {20}	Estimated number of cases message	
Incomplete data specified	OptimizeIt cannot start, not enough information	Example: no Objective Functions defined
Incorrect bounds	OptimizeIt cannot start, boundary case invalid	Check Parameter range for geometric or other conflicts
Linked parameter {Param1} cannot be used in Constrained Optimization	Parameter {Param1} referenced in line {line} is Linked	Linked Parameters cannot be used in Constrained Optimizations

Messages appearing as separate Message Boxes:

Message Text	Message appears when...
Optimization completed	OptimizeIt is finished
Cannot start optimization. See Details section for more information.	OptimizeIt cannot start
Case map contains {999} cases. Proceed?	A warning, that you have more than 100 cases (Parametric Study)
Coolit cannot create Optimization directory	Cannot create optimization directory, Possibly disk full, or read/write permissions not set properly

16 Coolit Components

16.1 Interior Components

16.1.1 Solid Block

A Solid Block is a solid rectangular parallelepiped, which can be used to represent power supplies, chips, or any other rectangular objects. Two types of Solid Blocks are permitted, those with a specified temperature (**a**) and those with an unknown temperature (**b**). To use option (a) select Specify Temperature in the Properties dialog. To use option (b) select the Conjugate Heat Transfer option in the Solid Block properties dialog. Coolit will compute the temperature distribution in the Solid Block.

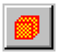
In conjugate heat transfer steady state problems, you have to specify the **Thermal conductivity**, **Power**, and **Maximum Temperature**. In transient conjugate heat problems, in addition to the above you need to specify the Density, Heat Capacity, and Initial Temperature. **Max Temperature** is used in CoolPlot to flag overheated components by painting them as "burnt" (orange-black pattern).

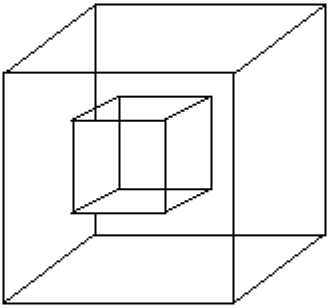
Using the drop down menu, the Power can be specified as uniformly distributed constant, temperature-dependent (for detailed description see here), Joule, or time-dependent (for transient models only). Joule heating is computed from $Q = \frac{I^2 \rho L}{SV}$. Here I is the current, ρ is the resistivity, L is the size in the **Current Direction** (in local, component's, coordinates, x, y, or z), S is the area normal to the Current Direction, and V is the volume of the Heat Source. **Resistivity**, ρ, is computed from $\rho = \rho(T_{ref})[1+C(T-T_{ref})]$, $T_{low} \le T \le T_{high}$, where $\rho(T_{ref})$ is resistivity at temperature T_{ref}, C is the temperature coefficient, the temperature, T, is assumed to vary within a range defined by T_{low} and T_{high}.

16.1.2 Porous Volume Resistance

A Porous Volume Resistance  is a rectangular parallelepiped that can be used to model small, densely spaced objects, such as pin fins. The Porous Volume Resistance component uses the Porous Media model. For more information, see the guidelines for using the Porous Volume Resistance model.

16.1.3 Heat Source

A Heat Source  specifies the amount of heat dissipation in a given volume. Heat Sources are usually inserted inside of other components. When the heat dissipation is evenly distributed throughout a component, it is more convenient to specify the power dissipation in the properties dialog for the component.



Note that when Heat Sources intersect, the power dissipation is added in the overlapping portion. For example, consider the object above consisting of two Heat Sources. Suppose that your task is to specify the power dissipation so that the total power, P, is distributed in 1/3 to 2/3 proportion between the smaller and the larger Heat Sources, respectively. Setting the power for the smaller Heat Source to P/3 will not be correct, as the total power from the volume occupied by the smaller Heat Source will be greater than P/3. The power, which has to be specified to satisfy the specified ratio, can be found from the following system of equations:

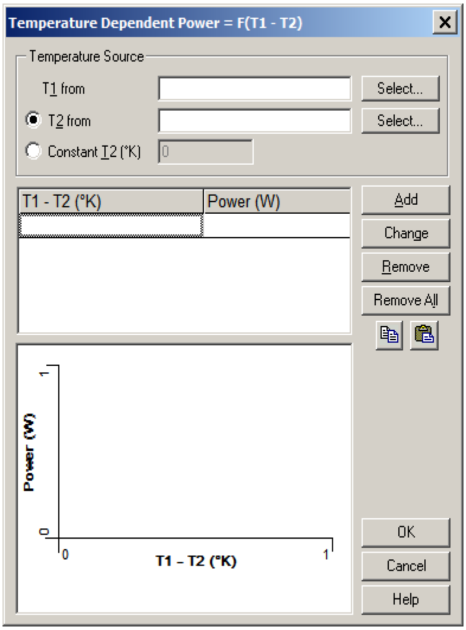
$$P_l \cdot \frac{V_l - V_s}{V_l} = \frac{2}{3} P$$
$$P_l + P_s = P$$

where P and V designate the power dissipation and volume, respectively, and subscripts s and l stand for small and large.

On the other hand, if the above sketch represented two Solid Blocks with specified power dissipation, the result would be very different. If the smaller Solid Block had a higher priority, P_s would be distributed over the entire volume of the smaller component, while P_l only over the non-intersected portion of the larger component.

Using the drop down menu, the Power can be specified as uniformly distributed constant, temperature-dependent, Joule, or time-dependent (for transient models only). Joule heating is computed from $Q = \frac{I^2 \rho L}{SV}$. Here I is the current, ρ is the resistivity, L is the size in the **Current Direction** (in local, component's, coordinates, x, y, or z), S is the area normal to the Current Direction, and V is the volume of the Heat Source. **Resistivity**, ρ, is computed from $\rho = \rho(T_{ref})[1+C(T-T_{ref})]$, $T_{low} \leq T \leq T_{high}$, where $\rho(T_{ref})$ is resistivity at temperature T_{ref}, C is the temperature coefficient, the temperature, T, is assumed to vary within a range defined by T_{low} and T_{high}.

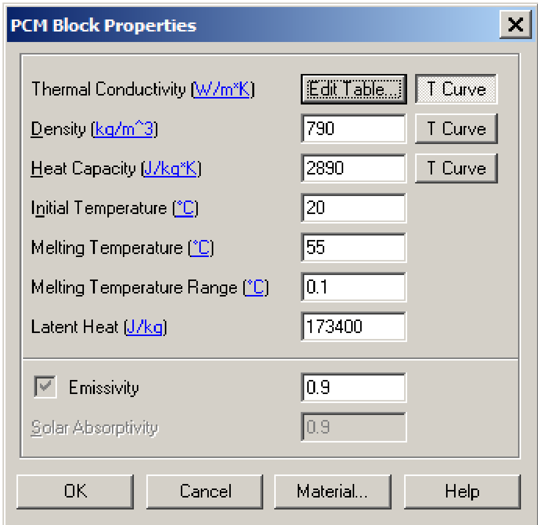
Clicking the **T-Dep** button will specify a **temperature-dependent** power curve for the Heat Source. This can be useful for modeling, for example, resistive heaters. At least one Monitor Probe or Gauge is required for this option. Specify the source of T1 temperature by clicking Select. T2 can be supplied by either another Monitor Probe or Gauge or can be fixed using the Constant T2 option. If your power curve is a function of only one temperature, use the Constant T2 option and set T2 equal to zero. Monitor Probes provide temperature reading at a single point in space. This is often sufficient when the temperature is sought in an area with small temperature gradients. When temperature gradients are high, it is better to use Gauge, which measures the average temperature at the Gauge's surface. A typical example of using Gauge would be an air outlet, where the temperature can range widely over the outlet's cross section.



The power curve can be specified by entering points using either the table or the plot. The table operates like an Excel spreadsheet, so that you can use either the Tab or Enter button to navigate it. You can also use the mouse and Change and Add buttons, if that is your preference. Clicking the right mouse button within the plot will add a new point. An existing point can be dragged to a new location with the mouse. The T1-T2 column in the table is automatically sorted to ensure monotone behavior.

16.1.4 Phase Change Materials Block

Encapsulated phase change materials (PCM) have a high fusion capacity and can absorb large amounts of heat when melting and release heat when solidifying. The phase change heat release (during melting) or absorption (when solidifying) is a time-dependent process. If your model has a PCM block and you model a steady state case, PCM blocks work identically to Solid Blocks. All PCM properties describing their transient behavior, such as latent heat, heat capacity etc. will be grayed out.



To create a PCM block click [icon] or Components|Interior Components|PCM Block. The PCM physical properties are entered via a dialog and can be general non-linear functions of the temperature.

The Melting Temperature Range characterizes the temperature interval over which melting/solidification occurs and is any positive number or zero.

16.1.5 TEC (Thermoelectric Cooler)

A Thermoelectric Cooler is a compact electric cooling device used to dissipate heat away from components. It has a hot side (red) and a cold side (blue).

On the **Modeled Object** page, you can specify a cold side temperature or a current for the TEC.

On the **TEC Settings** page you can specify the number of couples, the G-Factor, which is defined as the area divided by the length of a TE element, and the maximum current. The electrical resistivity and Seebeck coefficient can be specified as constants, or as temperature-dependent by clicking the T-Dep button. Emissivity and Solar Absorptivity can be specified if radiation heat transfer and solar radiation are enabled in the Global Settings.

On the **TEC Elements** page you can specify a density and heat capacity for the TEC. You can also specify thermal conductivity as either a constant, or as temperature dependent by pressing the T-Dep button. Click the **Library** button to select a TEC from library.

16.1.6 Cold Plate

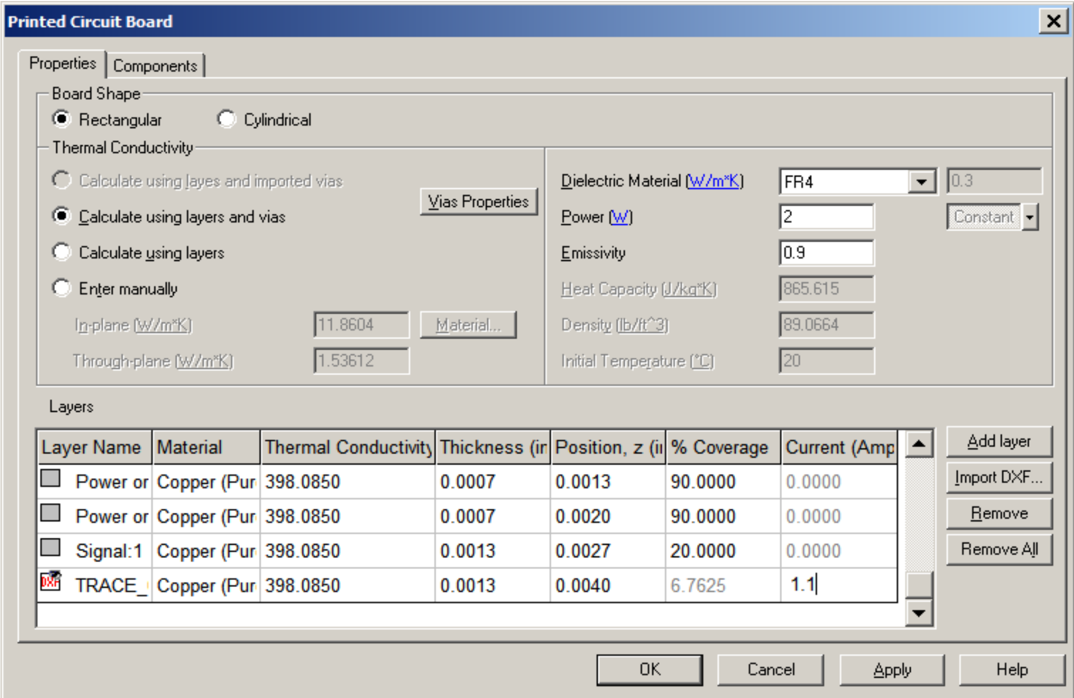
Cold Plate is a lumped-parameter model of a cold plate, which computes heat exchange between components placed on the cold plate, while omitting details of heat transfer within the cold plate. Specify the required properties, such as the Thermal Resistivity of the Cold Plate and coolant type and properties. These properties are available from manufacturer specification sheets. Typically curves of thermal resistivity versus flow rate of a particulate coolant are given, where you can pick out values for your specific system. If manufacturer provides only thermal resistance rather than the resistivity, you need to know at what flow rate it was measured and whether the total or only one side area of cold plate was used to compute it. Usually maximum flow rate and one side area are assumed, but sometimes other assumptions are made. If you need to make a quick hand-estimate of temperature rise in a component placed on Cold Plate, you first compute the thermal resistance of the component by multiplying the Cold Plate's Thermal Resistivity by the area of their contact. Multiplying the computed resistance by the component's heat dissipation rate, you will obtain a temperature rise estimate.

16.1.7 PCB Object

The PCB object is a complex object, which consists of a board and other components provided on the PCB toolbar (IC Package and Via) as well as non-PCB objects, such as heat sinks, added to the PCB object. PCB toolbar components are available (not grayed out) only when PCB object is active. Click  to create a PCB. The PCB is specified in its Properties dialog by double clicking on the created PCB, or by clicking it, and pressing the **Properties** button in the Control Bar.

On the Properties page, the PCB's thermal conductivity can be defined using the **Enter manually** option, or by specifying board layers and materials and selecting either the **Calculate Using layers and imported vias** (assuming that you imported an IDF file with via information)**, Calculate Using layers and vias**, or **Calculate using layers** option.

To add a layer click either **Add Layer** or **Import DXF...** buttons. The Add Layer button adds a layer with uniformly distributed amount of copper specified in the %Coverage column. Clicking Import DXF... opens the Import DXF Layer dialog for importing artwork created in board design software. The **%Coverage** for imported layers is computed based on the imported artwork and cannot be changed. Copper for such layers is distributed in accordance with artwork. To delete a selected layer click **Remove**.



If the layer position is important, make sure that the entry in the Position column is correct and the grid is sufficient to resolve the specified layers. Thus, if you were to specify only one cell in the through-plane direction, all the layers will be averaged out over that one cell. The thermal conductivity shown in the grayed out Through-plane and In-plane boxes are averaged. The values used in computation will approach that of actual layers depending on the grid resolution.

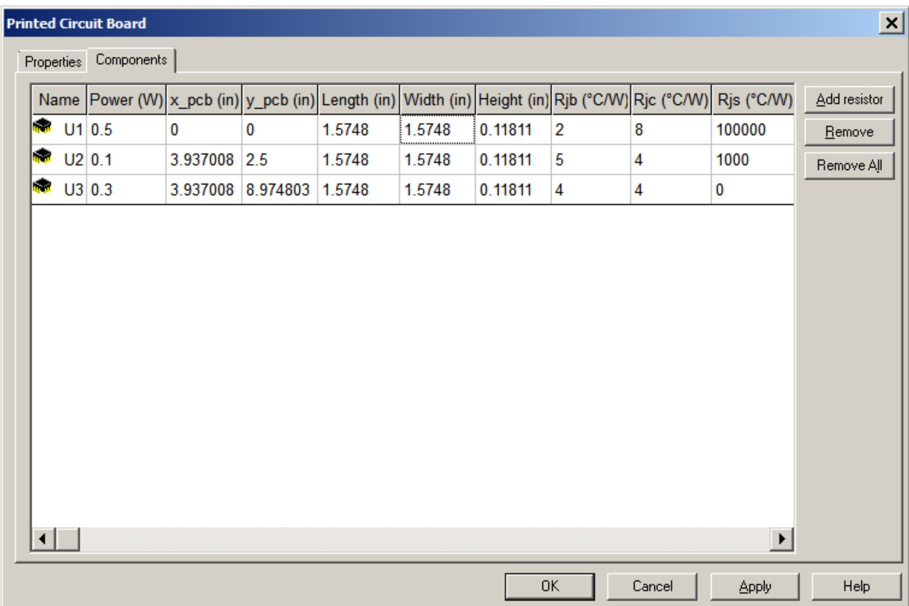
To change a specified value in the table, click on the desired box, and enter the new value. The total **Thickness** of all the layers must be smaller than the board thickness. The **% Coverage** is the area of the board covered by the Material in the layer. It is used to compute the board's conductivity. The thickness and position of dielectric material layers is computed automatically based on the board thickness and the total thickness of all copper layers.

The dielectric material, a uniformly distributed power dissipation, emissivity, and solar absorptivity can be entered in the respective boxes at the top of the dialog. The Density, Heat Capacity, and Initial Temperature are required for transient cases only. For steady state cases, input boxes for these variables are grayed out.

Using the drop down menu, the **Power** can be specified as uniformly distributed constant, temperature-dependent, Joule, or time-dependent (for transient models only). Joule heating is computed from $Q = \frac{I^2 \rho L}{SV}$. Here I is the current, ρ is the resistivity, L is the size in the **Current Direction** (in local, component's, coordinates, x, y, or z), S is the area normal to the Current Direction, and V is the volume of the Heat Source. **Resistivity**, ρ, is computed from $\rho = \rho\,(T_{ref})[1+C(T-T_{ref})]$, $T_{low} \leq T \leq T_{high}$, where $\rho(T_{ref})$ is resistivity at temperature T_{ref}, C is the temperature coefficient, the temperature, T, is assumed to vary within a range defined by T_{low} and T_{high}.

If you wish to specify Joule power in imported trace layers, type the trace's current value in the **Current** column of the Layers table. Note, that the Joule power in traces is not included in the uniformly distributed Power specified in the Power box.

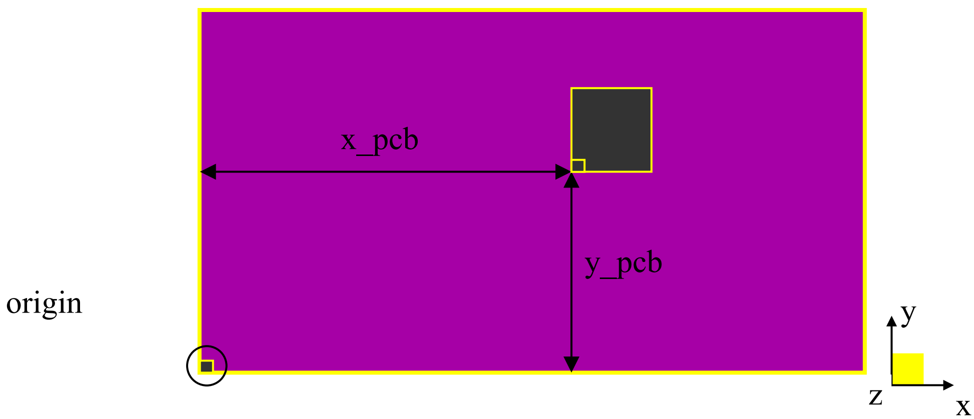
Components



Name	Power (W)	x_pcb (in)	y_pcb (in)	Length (in)	Width (in)	Height (in)	Rjb (°C/W)	Rjc (°C/W)	Rjs (°C/W)
U1	0.5	0	0	1.5748	1.5748	0.11811	2	8	100000
U2	0.1	3.937008	2.5	1.5748	1.5748	0.11811	5	4	1000
U3	0.3	3.937008	8.974803	1.5748	1.5748	0.11811	4	4	0

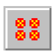
The Components page lists all components assembled within the PCB object and their properties. The components can be located on the PCB or anywhere in the domain, but as long as they are included in the given PCB assembly, they will be listed on the Components page of that PCB.

In this table, "x_pcb" and "y_pcb" refer to the board's coordinate system defined as follows:



Once a PCB has been created, the other PCB components become available. Clicking  will add an IC Package. The created component will be automatically placed on the active PCB. PCB components can be placed, moved, and resized like any other Coolit object.

Vias

To add a thermal via to a component, click on the component, and click the  button. Vias are specified with the following parameters:

Number of vias – specify the total number of vias.

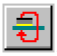
Diameter – specify the diameter of individual vias.

Plating Thickness – specify the wall thickness of the via.

Plating Material – select plating material.

Fill Material – select fill material.

Switch Sides

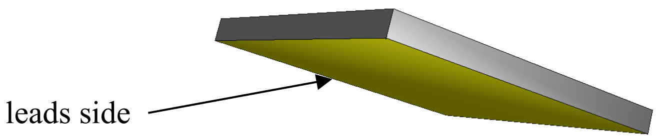
To switch a resistor model to the opposite side of a board click .

16.1.1 IC Packages: Detailed and Compact Models

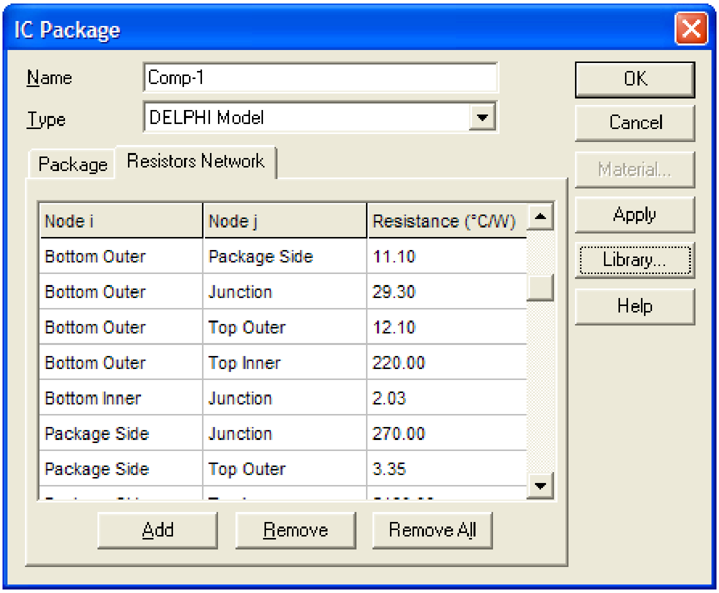
Electronic components and packages can be modeled as either compact (resistor) models or detailed models. IC Packages can be created as part of a PCB object using  from the PCB toolbar, or standalone using  from the Components toolbar.

To create a resistor model, select **Resistor Model** in the **Type** field of the IC Package dialog and under the **Package** tab specify the IC Package size and resistance. Check the **Temperature Monitor** box to create a monitor point in the middle of a Resistor Model. The temperatures at the monitor point can be viewed on the Temperature page of Convergence Monitor. The Temperature Monitor will have the same name as the Resistor Model and the color you specified. Under the Properties tab of the dialog, specify the **Power** dissipation and **Max Temperature**. Components exceeding the specified Max Temperature are painted (in CoolPlot) in "burnt" orange-black pattern. The **Density**, **Heat Capacity**, and **Initial Temperature** are grayed out for steady state cases. **Surface Properties** are grayed out if the Radiation model is set to None.

When using Resistor models, make sure that the yellow side of the component, the leads, is toward the board.

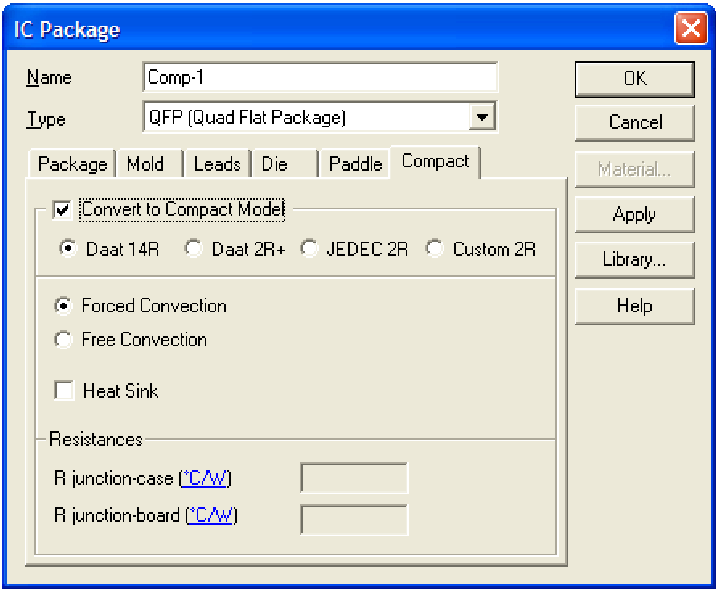


Selecting DELPHI Model in the Type pull-down menu opens the dialog that lets you specify the package size and the DELPHI resistor network. Typically this information is supplied by the manufacturer of the package. If the package thickness is specified as several layers, you should add their thickness and enter the total in the Thickness field (it's a calculator field, so you can add numbers right there).



Selecting BGA, DIP, QFP, or TO-220, -252, -263 in the Type pull-down menu, opens the respective IC Package constructor dialog for building detailed model of the package. After detailed model is specified, the last tab of the dialog, Compact, lets you convert the detailed model to a compact resistor model.

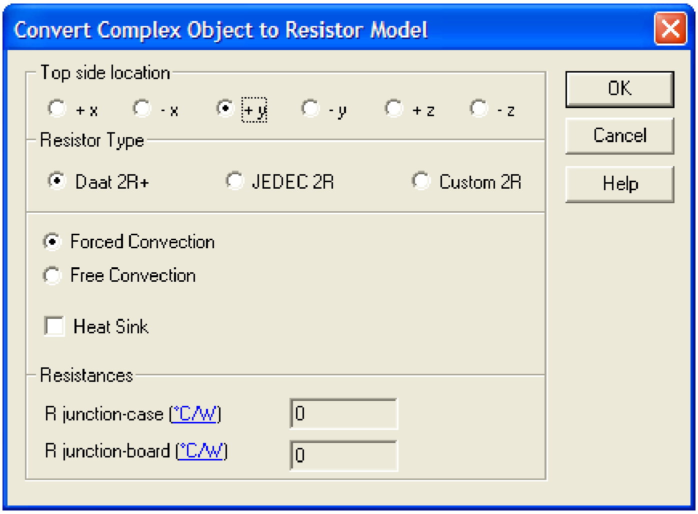
Under the Compact tab, check the Convert to Compact Model box and select the model type: Daat 14R, Daat 2R+, JEDEC 2R, or Custom 2R. Selecting either of Daat models builds a proprietary compact model with 2 or 14 resistors. Specify whether the IC package is in free or forced convection and whether a heat sink is used. Click Apply to compute resistor values or OK to compute the values and exit. After you click Apply, there is a delay while model is set up and solved.



Selecting the JEDEC 2R option requires no additional inputs, just click Apply to view the computed resistor values or OK to compute the values and exit. Selecting Custom, lets you specify the resistor values. You can get them from a manufacturer spec sheet or you can compute them yourself from detailed model. For example, you can place Gauges at the top and bottom surfaces of your detailed model, solve the case and obtain from the Gauges respective T_{max} and heat fluxes. The chip's junction temperature is also known (it's the maximum temperature for the model located somewhere in the die). So you now have all the ingredients to compute R_{jb} and R_{jc}.

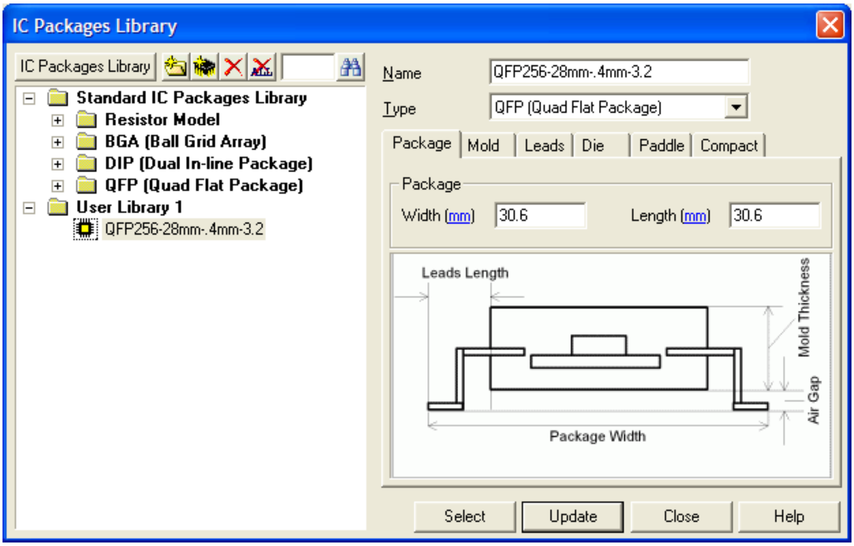
In our experiments, both Daat models have been more accurate than the JEDEC 2R model (i.e. the conventional two-resistor model) over a wide range of conditions (forced and free convection, with and without heat sink). The average error for Daat 2R+ model was 7% vs. 15% for JEDEC 2R and the maximum 14% vs 25%. The error was defined as the difference between junction temperatures of detailed and compact models.

Detailed models created by the IC Package constructor are lumped-parameter models, containing parts essential for computing heat transfer. Some elements of the model may not be exactly what you have. For example, BGA balls matrix is always filled from the outside in. If your BGA design calls for a different ball distribution and you believe that it is important for accurate heat transfer, you can convert your BGA model to Complex Object and then modify the ball distribution as appropriate. You can then create a compact model of such a Complex Object (or any Complex Object made of Plates and Solids Blocks and containing one Heat Source) by pressing  or **Edit|Convert to Resistor Model**.



The IC Package dialog comes with a Standard IC Packages library. The components from the Standard library cannot be edited and new components cannot be added to it. To copy a component from the library into the IC Package dialog, click the desired component and then click the Select button.

To create a new library, click the IC Packages Library button and select New. A new Library[] folder will appear under the Standard IC Package Library folder.
Components can be added to the created library either by using



drag-and-drop from the Standard library or by selecting a component in the Standard library, then highlighting the new Library and clicking the Add button. The selected component will be copied to the new library and can now be edited. To edit properties of a component, type its new properties in the dialog and press the Update button.

To save a new component library, click the IC Package Library button and select Save As.

16.1.2 LED package

The LED object is a compact thermal model of LED packages. It is defined by the following parameters:

- **Forward Current**, I, constant value, default 200 mA.

- **Forward Voltage**, U, constant or temperature dependent value, default 2 V.
- **Wall Plug Efficiency**, E, constant value, default 75%.
- **Thermal Resistance:** enter R_{jb}, R_{jc}, and R_{js} resistances; the default is $R_{jb} = R_{jc} = 20$ and $R_{js} = 0$.

The LED power dissipation is defined as $P = (1\text{-}E/100)*U*I$.

16.1.3 Heat Pipe

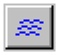
Heat pipes, devices that use two-phase phenomena for heat transfer, do not have a fixed thermal conductivity like solid materials modeled using Solid Block. A heat pipes' effective thermal conductivity improves with length and will also change with the amount of power being transferred. Effective thermal conductivities of a well-designed heat pipe range from 10 to 10,000 times the thermal conductivity of copper depending on the length of the heat pipe.

The performance of heat pipe changes with heat flux. Below the critical heat flux, the heat is transferred by boiling and condensation. The Coolit Heat Pipe model assumes normal operation below critical heat flux.

The complex physics of the heat transfer within a heat pipe are characterized by **Thermal Resistance**, which is usually available from manufacturers. If not, manufacturers usually provide data and equations from which thermal resistance can be estimated.

If your heat pipe has bends, you can assemble it from multiple segments. The segments' ends can either touch or intersect. You have to specify the Thermal Resistance for each segment, so that the total resistance matches that of the entire Heat Pipe assembly.

16.1.4 Fluid Region

The Fluid Region object  is used to define a 2nd fluid domain within an Enclosure. There can be as many Fluid Regions as desired, each with a different fluid. Fluid Regions may touch Enclosure Wall Patch sides and may touch and intersect most components with the exception of planar and volume porous resistances, Fans, and Grid Zones. Fluid Region sides are impermeable, no-slip surfaces. Thermally, they behave as 1 mm thick Plates with $k=10^3$ W/m*K. The rules of priority for intersecting/overlapping components apply.

If Fluid Regions with the same fluid are in contact with each other, the contact area is open and fluid from one region can flow into another. You can use this property to create complex ducts and general fluid domains with inlets and outlets modeled using **Flow Source**/**Sink** components. Fluid Regions in contact, but with different fluids are not allowed.

Fluid Regions can be specified as Liquid, Water, or Gas. Selecting the fluid as Liquid will require you to specify the Dynamic Viscosity, Conductivity, Heat Capacity, and Density. The Dynamic Viscosity and Density can be specified as functions of temperature by pressing the corresponding **T-Dep button**. A library of liquids is available by pressing the **Materials...** button.

If Water is selected, physical properties will be computed automatically from empirical formulas as functions of temperature.

For Gas selection, the default is air. Coolit computes the material properties of gases (thermal conductivity and viscosity) during the solution process as a function of temperature using the kinetic theory of gases. The theory requires one to specify the molecular mass of the molecule, the Lennard-Jones parameters (the collision diameter and ε/K), heat capacity, and whether the gas is monatomic or not. The requested values for frequently used gases are available by pressing the **Materials...** button. If desired, you can specify the Relative Humidity of the gas at the Reference Conditions set on the Environment page dialog.

To simulate fluid inflow/outflow from Fluid Regions, Flow Source components should be used.

16.1.5 Plate

A Plate is a solid thin rectangular object, which can be used to represent PCBs or similar components. The main difference between a Plate and a Solid Block is that the Plate's heat transfer is calculated exactly in the normal direction to the Plate's surface and only approximately in the tangent plane. Another important difference is that the Plate model is computed using fewer cells than a Solid Block. In the normal direction to the Plate's surface, no cells are required - a Plate is defined by a single grid line. On the other hand, a Solid Block requires a minimum of two grid lines, in each coordinate direction. Plate is a quasi-3D component: it has height, width, and thickness, but the latter is accounted for only in the numerical model, geometrically the thickness is zero. Click here for information on when to use Plates versus Solid Blocks.

Since the Plate model has no thickness, when a Solid Block is placed on top of a Plate it must be positioned at the same level as the Plate. For example, suppose a Plate is located at Z = 1.0 (i.e. in the XY plane). A Solid Block component must be placed exactly at Z = 1.0 to be on top of the Plate.

A Plate requires a thermal conductivity and if the flow is turbulent - the surface roughness. The surface roughness simulates the effect of fluid flow on various small components on the Plate surface, which otherwise are not feasible to model separately. The Plate sides can have different roughness. The (+) side has a positive normal, i.e. it is aligned with the positive direction of the coordinate axis normal to the Plate and the (-)

side has a negative normal. The positive side and the negative side of the Plate can be determined from the local coordinates triad displayed in the 3D-view.

An example of using the surface roughness parameter is a motherboard with many small components on top of it. If the side of the board with the small components on it is facing the positive direction, set the Surface Roughness (+) value to the average height of the small components (e.g. 2mm). If the other side of the board is smooth then you should set the Surface Roughness (-) value to 0. Similarly, the Emissivity and Solar Absorptivity can be specified for each side independently.

For more information on using Plates, click here.

16.1.6 Contact Resistance

A Contact Resistance is a component used to model thermal resistance between two surfaces. A Contact Resistance can be used to model Leads/Pins/Balls, an Air Gap between components, and an Interface Material. When modeling Leads/Pins/Balls you need to specify the surface area of the material (e.g. cross section area of a lead, times the number of leads) and the lead's thermal conductivity. For your convenience, the Leads/Pins/Balls Area box is a calculator. The Specified Resistance option is used to specify a contact resistance with known resistivity. The Interface Material option lets you specify an interface material with the known thermal conductivity. The Air Gap option is used to model a narrow air gap between components. No flow or radiation is modeled in an air gap; only conduction is considered.

16.1.7 Planar Porous Resistance

A Planar Porous Resistance component is a thin permeable rectangular object, which can be used to represent, for example, perforated plates or thin filters. The Planar Porous Resistance component uses the Porous Media model. See the Guidelines for specifying PPR parameters.

16.1.8 Axial Fan

Click the arrow  and select Axial Fan in the pull-down menu. Axial Fan is a 'lumped parameter' model the purpose of which is to model the flows within the system, while omitting the details of the flow within the fan itself. Thus, for example, one should not expect to get an accurate description of flow between fan blades, but rather to get correct amount of flow and heat transfer through the fan.

Axial Fans are infinitesimally thin (2D) objects, which have the surface area equal to that of the modeled fan. In steady flows, the flow rate through the fan can be computed from the user-specified fan curve, specified constant, specified as a function of temperature or specified fixed/constant based on a previously computed profile available from a restart file (the **Get from Restart File** option).

In transient cases, only specified flow Fans are allowed. Here one can specify either **Flow Rate** or **Get from Restart File** option (assuming that a restart file is available, e.g. from computing steady state case).

Layout Page

A failed fan can be simulated by checking the **Failed Fan** box in the upper left corner of the Axial Fan dialog. A Failed Fan behaves like an opening. Specify the Flow Rate or Velocity Magnitude or Fan Curve. The **Fan Curve** can be specified by pressing the Fan Curve button and entering the points in Fan Curve dialog using either the table or the plot. The table operates like an Excel spreadsheet, so that you can use either the Tab or Enter button to navigate it. You can also copy and paste tables directly, to and from Excel spreadsheets, using the and buttons in the Fan Curve dialog. You can also use the mouse and Change and Add buttons, or you can enter the points on the plot below the table. Clicking the right mouse button within the plot will add a new point. An existing point can be dragged to a new location with the mouse. The Q column in the table is automatically sorted to ensure monotone behavior. To allow for changes in air density due to altitude, click the **Altitude adjust** button. dP will be recalculated automatically to reflect the altitude specified in the **Global Settings** dialog.

Angular Velocity specifies the fan blades' angular velocity. It can be specified only for flows perpendicular to the fan, otherwise the Angular Velocity field is grayed out. The following convention is used for the sign of the angular velocity: for an observer looking at the Fan from the beginning of the normal to the Fan axis, clockwise rotation is positive, counterclockwise - negative. Both the flow direction and rotation are shown in all four panels.

The **Swirl Slip Factor** is the percentage of the Angular Velocity that contributes to flow Swirl.

For turbulent flows, the **Turbulence Level** should be specified in accordance with the selected turbulence model.

Any heat dissipated by the fan is specified by entering the value for **Power**.

Hub and Shroud Page

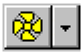
On this page you can specify the size and properties of a hub and shroud for the fan. Pressing the Apply button will show you the effect of selected settings. You can modify the settings until the OK button is pressed and the dialog is closed. The fan hub is shown as a green disk in the middle of the fan and the shroud is made out of Plates. If you do not wish to model a hub or shroud, leave their sizes as zero.

Flow Direction Page

On this page you can enter a direction for the fan velocity. Positive velocity always points in the positive z-direction in the Fan's local coordinate system. For example, for the positive flow normal to the Fan the Velocity Unit Vector will have (0, 0, 1) components. To assign other flow directions, follow the diagram in the dialog. Pressing the Reverse button, will reflect the flow through z-plane. For example, for flow in the negative z-direction the Velocity Unit Vector will change to (0, 0, -1). For more information and an example see FAQ: How to model fan at angle.

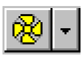
Click the **Library** button to select an axial fan from library.

16.1.9 Blower Fan

Click the down arrow  and select Blower Fan in the pull-down menu. Blower is the 'lumped parameter' model of centrifugal blowers the purpose of which is to model flows in the surrounding system, while omitting details of the flow within the fan itself. The flow rate through the fan and the pressure drop across the fan are computed from the user-specified fan curve. The blower model has an outlet and one or two inlets. The air is drawn into the blower through inlet(s) and then turned ninety degrees out of the outlet.

You can specify the Blower Fan geometry in its Properties dialog. A failed blower fan can be simulated by checking the **Failed Fan** box. Similar to **Axial Fan**, specify the Flow Rate or Velocity Magnitude or Fan Curve as well as the **Power**.

16.1.10 Crossflow Fan

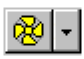
Click the down arrow  and select Crossflow Fan in the pull-down menu. Crossflow Fan is the 'lumped parameter' model the purpose of which is to model the flows within the system, while omitting the details of the flow within the crossflow fan itself. The flow rate through the fan and the pressure drop across the fan are computed from the user-specified fan curve.

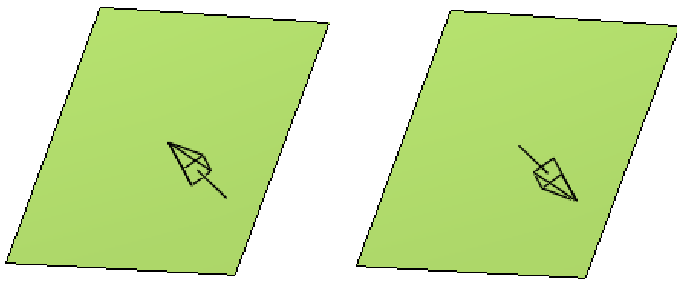
On the **Inlets/Outlets** page, you can specify the physical dimensions of your fan. If there is no inlet on a particular side, specify its size as zero. On the **Properties** page, you specify the flow parameters of the fan. You can specify a constant Flow Rate or Velocity with the corresponding radio buttons, or use the Fan Curve table.

The Fan Curve can be specified by pressing the Fan Curve button and entering the points in Fan Curve dialog using either the table or the plot. The table operates like an Excel spreadsheet, so that you can use either the Tab or Enter button to navigate it. You can also copy and paste tables directly, to and from Excel spreadsheets, using the  and  buttons in Fan Curve dialog. You can also use the mouse and Change and Add buttons, or you can enter the points on the plot below the table. Clicking the right mouse button within the plot will add a new point. An existing point can be dragged to a new location with the mouse. The Q column in the table is automatically sorted to ensure monotone behavior. To allow for changes in air density due to altitude, click the **Altitude adjust** button. dP will be recalculated automatically to reflect the altitude specified in the **Global Settings** dialog.

Turbulence Level can be specified for turbulent flow problems. Any heat dissipated by the fan can be entered into the **Power** box.

16.1.11 Flow Source (Sink)

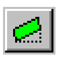
Click the down arrow  and select **Flow Source** in the pull-down menu. A Flow Source (Sink) is an infinitesimally thin, 2-sided object, which is used to simulate, for example,ventilation ducts where the flow entering or exiting the Enclosure is driven by a fan or blower outside of the Enclosure. The side of the Flow Source with the arrow is a flow source or sink, and the other side is an impermeable, no-slip surface. Specify the **Mass Flow Rate** (positive for Flow Source and negative for Flow Sink), the Velocity Unit Vector, and the Ambient Temperature. Positive velocity always points in the positive z-direction in the Flow Source's local coordinate system. For example, for the positive flow normal to the Flow Source the Velocity Unit Vector will have (0,0,1) components. To assign other flow directions, follow the diagram in the dialog. For turbulent flows, the Turbulence Level should be specified in accordance with the selected turbulence model.



Flow Sink Flow Source

In transient flows, Ambient Temperature can be specified as a function of time by pressing the Transient button and entering data into the table.

16.1.12 Baffle

Click the button  to create a Baffle component. The Control Bar will automatically change to display three additional parameters: Length, Thickness, and Angle of the Baffle. The Size and Position coordinates refer to the bounding box of the Baffle, which is visible during the Baffle's construction. The Baffle component is made of Plate components and uses the Plate Properties dialog. The Baffle component cannot be drilled nor can it intersect other components.

16.1.13 Prism, Tetrahedron, Pyramid, Cap, Cylinder, and Inclined Block

The Prism, Tetrahedron, Pyramid, Spherical Cap, Cylinder, , and Inclined Block components can be used to model non-rectangular objects. They are made of Solid Block components and use the Solid Block Properties dialog. When any of these components are created, the Control Bar will display size of a bounding box around the component. For Inclined Block, the Control Bar will display three additional parameters: Length, Thickness, and Angle.

These components, except for Cylinders and Prisms, cannot be drilled nor can they intersect other components. Coolit will approximate these components on a Cartesian grid using both rectangular and triangular cells. Cylinders and Prisms can be cut by both Cutouts and Solid Blocks, but not vice versa, in one direction only. For Cylinders the allowed direction is local z (axial) and for Prisms – local x.

16.1.14 Cutout

A Cutout is a component that drills holes in other components. Size is the only property required for Cutout components. Cutouts can be assembled in complex objects and component priority rules apply to them as well. For example, when a Cutout is intersected by a component with a higher priority, the component will not be drilled. Tetrahedrons, Pyramids, and Caps cannot be cut.

16.1.15 Gauge

A Gauge is a 2D component designed to measure the flow rate and heat flux through its surface. The Gauge does not interfere with flow or other components. Gauges can intersect any object, and can be placed directly on the Enclosure. After the computed results are displayed in CoolPlot, open Objects

Manager and select a Gauge component. The Object Manager window will display the following information:

Name: rfa_meter1
Type: Gauge (2D)

Object position and size (mm)

	Min	Max	Size
X	0	158.598	158.598
Y	87.122	87.122	0
Z	140.564	160.757	20.193

Temperature

Average Temperature (°C) :	61.4605
Min Temperature (°C) :	60.0453
Max Temperature (°C) :	64.3005
Average surface temperature (°C) :	61.4605
Min surface temperature (°C) :	60.0453
Max surface temperature (°C) :	64.3005
Bulk temperature (°C) :	61.4405

Heat Flux

Full Heat Flux (W) :	-4.01652
Convection Heat Flux (W) :	-4.01652
Conduction Heat Flux (W) :	0
Radiation Heat Flux (W) :	0

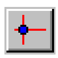
Flow Rate

Mass Flow Rate (kg/sec) :	0.022284
Vol. Flow Rate (m^3/sec) :	0.0210541

Pressure

Average Pressure (Pa) :	-126.691
Min Pressure (Pa) :	-126.876
Max Pressure (Pa) :	-63.1975
Pressure Drop (Pa) :	-11.2053
Pressure Gradient (Pa/m) :	-1030.73

16.1.16 Monitor Probe

The Monitor Probe  component is used to report temperature, pressure, or velocity at a specified point in Convergence Monitor. Monitor Probes require position (x, y, z) of the point, variables to monitor, and Line Style and Color, which are used to display the evolution of selected variables with iterations (for steady problems) or time (for transient problems). The default values for variables are temperature, pressure and velocity magnitude. They, as well as Line Style and Color, can be changed by selecting **Properties…**

To insert Monitor Probe into the center of a component, such as Solid Block, right-mouse-click the component and select **Insert Monitor Probe**.

Monitor Probe data can be saved and then pasted in MS-Excel or other applications. The Monitor data are obtained from the nearest grid cell without interpolation. Thus, if the grid cell is large, considerable discrepancy may result between what is reported at the same location in CoolPlot (done using 3D interpolation) and by Monitor Probe.

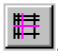
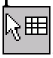
When selected in Coolit, Monitor Probes are displayed as a dot with crosshairs and without cross hairs when not selected. No grid lines are generated for Monitor Probes. In Convergence Monitor, a pull-down menu allows you to select whether to display convergence history (default) or Monitor Probes. At any time, you can hide/unhide any of probe lines by clicking , , or , or change color and line style by double-clicking either the name, or legend line.

Once solution is stopped, similar to all the other components Monitor Probes are transferred to CoolPlot. The computed solution at Monitor Points can be viewed in CoolPlot's Objects Manager.

16.1.17 Grid Zone

Click  to select Grid Zone. The Grid Zone component is used to define an area within the computational domain that has a different grid from the rest of the problem. A Grid Zones's grid does not propagate into the rest of the domain.

Grid Zones are used to reduce total grid cell count in cases where small and complex components are modeled within larger systems. Examples of Grid Zone applications include pin fin heat sinks, detailed components and packages etc.

Grid Settings are specified for each Grid Zone independently. First, select the zone either in Objects Manager or by clicking on it in the construction views. Open **Package|Grid Settings…** or click the Settings button in the Control Bar. The title bar at the top of the Grid Settings dialog indicates which Grid Zone is being worked on. To view the grid in each zone, switch into grid mode by clicking , or pressing F4 and click a Grid Zone in the Objects Manager. Alternatively, when the mouse passes over the location of a Grid Zone, the cursor changes to ; click to select that Zone.

Grid Zones can cut through components and can be snapped to component and Enclosure surfaces. You can also nest multiple Grid Zones one inside another, but without intersection. Two standalone Grid Zones, can touch and/or intersect one another. Such two touching/intersecting Grid Zones can be nested inside other Grid Zones, but they cannot touch or intersect them. The overlapped domain between two intersecting Grid Zones is not allowed to touch Enclosure.

When computing radiation heat transfer, only the Advanced Model can be used if the case contains a Grid Zone.

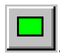
16.2 Data Center Components (optional module)

16.2.1 Computer Room Air Conditioner (CRAC)

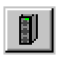
A CRAC  models both downflow and upflow CRAC units. By default, CRAC units are placed on the raised floor although they can be placed above or even below the floor as needed. A library of CRAC units from major manufacturers can be accessed by pressing the Library button. You can also build custom CRACs or modify a model from the library to fit your needs. Pressing the Apply button will show you the effect of selected settings. You can modify the settings until the OK button is pressed and the dialog is closed.

Check the Turn off/Failed box if you wish to simulate a failed CRAC. If the Turning Vane Installed box is checked, a vane is installed under the raised floor.

The cooling characteristics of a CRAC are specified by clicking the Thermal Model tab of the CRAC dialog. There you can specify either Constant Supply Temperature or Variable Supply Temperature with or without thermostatic control. The Variable Supply Temperature option implies the dependence of T_{supply} on T_{return}. This dependence can be specified in one of two ways: via the Operating Point or Two Points. If you enable the Thermostatic Control temperature control, the specified temperature is used as follows. If T_{return} is greater than the specified Thermostatic Control temperature, then T_{supply} is computed from Operating Point or Two Points from the dialog. Otherwise T_{supply} is adjusted until T_{return} equals the specified Thermostatic Control temperature.

In Rooms with drop ceilings, you can automatically attach return ducts to all downflow CRAC units in the Room Settings dialog. Alternatively you can build custom ducts using Hollow Box, , made of Plate, , components.

16.2.2 Cooling Devices

A Coolit Device  is a model of an air cooler other than CRAC. CoolitDC provides the ability to build above-the-cabinet cooling units (such as Liebert XDV), overhead cooling units (such as Liebert XDO), and in-row coolers (e.g. Liebert XDH, APC RC etc.).

A library of cooling units from major manufacturers can be accessed by pressing the Library button. You can modify to some extent models from the library to fit your needs. Cooling characteristics of Cooling Devices are specified on the Thermal Model page of the dialog.

Custom modifications allowed for Cooling Devices have to follow the convention and geometry of the original device being modified. If your custom cooling device does not fit the mold, you should use the CRAC dialog to build it.

16.2.3 Racks

A Rack is a model of a data center rack housing computing equipment. Under the General tab of the Rack dialog you can specify the Rack Name, Nameplate, size, inlet T_{max} and whether the rack has a rear door heat exchanger installed. Inlet T_{max} is used to identify overheated racks in reports and in CoolPlot. On the Flow Configuration page of the dialog you can specify rack inlets, outlets, exhaust ducts, and cable cutouts.

If you have a cutout, check the Cable Cutout box and specify the total area of the cutout opening and percent open area. For example, if the cutout is 6”x6”, with 70% of it blocked by cables, you would enter: Area = 36 in^2 and Percent Open = 30%.

Finally, the Air Flow and Heat Load page lets you specify the rack shelf-by-shelf. When you specify a Server shelf, you can enter either a value in either T_{Rise} or Flow Rate column. The other value is computed for you. E.g. if you specify T_{Rise}=20F, CoolitDC will fill out the Flow Rate column to ensure 20F rise for the specified Heat Load. A library of racks from major manufacturers can be accessed by pressing the Library button.

16.2.4 Tiles

The Tile dialog models both solid and permeable floor tiles. In data center applications this can be a regular solid or perforated tile, a perforated tile with dampers and cutouts, as well as more complex devices like diffusers and forced convection floor units. Tiles can be placed both on the Raised Floor and Drop Ceiling. A library of tiles from major manufacturers can be accessed by pressing the Library button.

16.3 Enclosure Properties and Components

16.3.1 Enclosure Properties dialog

In this dialog one can specify the Enclosure size and Boundary Conditions. Enclosure sides can be one of the five types: Wall, Open, Fan, Vent, Symmetry, or Cyclic. These initial definitions can be overridden by placing new Enclosure components over the ones specified in this dialog. For example, if you specify a side as Wall, you can later put on the whole or portion of it an Open or Fan. The portion of the Wall covered by components added later will redefine that portion of the Wall. Symmetry and Cyclic-type sides require no properties. Enclosure objects cannot be placed on Cyclic boundaries.

It is often convenient to resize the Enclosure from a specific side. For example, you set up a case with a custom grid, solved it, and now you wish to reduce the spacing between the X_{min} wall and the Interior Components. If you use the Control Bar to modify the size or drag the Enclosure side with the mouse, Enclosure will be resized from the X_{max} side. Moving components towards the left (X_{min}) wall will destroy your grid.

Coolit provides two solutions for resizing Enclosure from X_{min} side. If you want to use the mouse to resize the Enclosure, hold the Shift key while dragging the side. Enclosure will move X_{min} towards your Interior Components. The second method is to use the Enclosure Properties dialog. Enter the new Enclosure size and check the box next to it. When you click OK, Enclosure will be resized from X_{min} side.

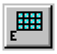
The bottom part of the dialog is available for users who purchased the optional **Data Center** module. If you model a

Data Center, Enclosure Size on this page refers to the room size. You can define whether the room has raised floor and drop ceiling. If **Raised Floor** is checked you will need to specify height of the raised floor and the tile size (default 2'x2').

If **Drop Ceiling** is present, check the appropriate box and specify the height of the ceiling above the room floor (not raised floor) and the **Leakage Area**. You can also have return ducts added automatically to your CRACs by checking the **Create Return Ducts** for CRACs box.

Leakage Area entry specifies the area of unknown and unintentional openings and air passages due to cracks, joints, seams and other such imperfections in the floor, which let the air leak through the floor. It is assumed that such leakage is randomly distributed over the entire floor area. Typically the value for this entry is between 0.1% and 1% depending on the quality and age of construction. The Leakage Area entry does not assume any unsealed tile cutouts, which should be accounted separately and individually.

16.3.2 Vent

A Vent  is a planar porous media model of a ventilation grille. If your vent does not have a grille, you should use an Open. In addition to the flow resistance parameters for Planar Porous Resistance model, a Vent requires the thermal conductivity of the Vent material, the Gauge Pressure and the Temperature. A Vent can be placed on either a Wall or an Open.

16.3.3 Enclosure Fan

Enclosure Fans  are infinitesimally thin (2D) objects, which have a surface area equal to that of the modeled fan. The flow rate through the fan and the pressure drop across the fan are computed from the user-specified fan curve. Enclosure Fans can be placed only on Enclosure sides. If you need to block a portion of the Fan, e.g. to represent a motor, use either the Wall Patch or Solid Block components. The entire volumetric flow will then be directed through the unobstructed area only.

In steady flows, the flow rate through the fan can be computed from the user-specified fan curve, specified constant, specified as a function of temperature or specified fixed/constant based on a previously computed profile available from a restart file (the **Get from Restart File** option).

In transient cases, only specified flow Fans are allowed. Here one can specify either Flow Rate or Get from Restart File option (assuming that a restart file is available, e.g. from computing steady state case).

You can also specify the Velocity Unit Vector, Angular Velocity, Swirl Slip Factor, and the ambient conditions. Positive velocity always points in the positive z-direction in the Fan's local coordinate system. For example, for the positive flow normal to the Fan the Velocity Unit Vector will have (0, 0, 1) components. To assign other flow directions, follow the diagram in the Enclosure Fan dialog. Pressing the Reverse button will reflect the flow through the z-normal plane. For example, for flow in the positive z-direction the Velocity Unit Vector will change to (0, 0, -1). For more information and an example see How to model fan at angle.

The Fan Curve is specified by pressing the Fan Curve button and entering the points in the Fan Curve dialog using either the table or the plot. The table operates like an Excel spreadsheet, so that you can use either the Tab or Enter button to navigate it. You can also copy and paste tables directly, to and from Excel spreadsheets, using the  and  buttons in fan curve dialog. Alternatively, you can use the mouse and Change and Add buttons, or enter the points on the plot below the table. Clicking the right mouse button within the plot will add a new point. An existing point can be dragged to a new location with the mouse.

The Q column in the table is automatically sorted to ensure monotone behavior. To allow for changes in air density due to altitude, click the **Altitude adjust** button. dP will be recalculated automatically to reflect the altitude specified in the Global Settings dialog.

Angular Velocity specifies the Fan's angular velocity in RPM. It can be specified only for flows perpendicular to the Fan, otherwise the Angular Velocity field is grayed out. The following convention is used for the sign of the angular velocity: for an observer looking at the Fan from the origin of the normal to the Fan axis, clockwise rotation is positive, counterclockwise – negative. Both the flow direction and rotation are shown in all four panels. The Swirl Slip Factor is the percentage of the Angular Velocity that contributes to flow Swirl.

If the flow is turbulent and you use either the Differential or an Eddy Viscosity model (see information on Turbulence Models) you need to specify the turbulence level:

- For the Differential model: it is the turbulence intensity, $\sqrt{\frac{u^2}{U^2}}$
- For the Eddy Viscosity models: it is Eddy viscosity, ν_t.

In transient flows, Gauge Pressure and Ambient Temperature can be specified as functions of time by pressing the Transient button next to the corresponding variable and entering data into the table. Only constant flow Fans can be used in transient flows.

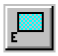
Several additional conventions are used with the Fan model. First, note that the fan curve uses the pressure difference, ΔP, rather than the absolute pressure. If the computed ΔP is outside of the specified fan curve, Coolit will use the limiting values in the table. Consider, for instance, the following fan curve:

ΔP	Q
44.0	0.0
37.0	2.6
28.0	5.0
15.0	7.0
2.0	8.7

Here if the computed ΔP is higher than 44.0, then Q will be set to 0.0; if ΔP is less than 2.0, Q will be set to 8.7 and no extrapolation will be made.

Positive Q means that the flow is in the positive direction of the local z-axis as shown below. The variable ΔP is always computed as follows: $\Delta P = P^{+} - P^{-}$, where P^{+} is the pressure on the Fan side facing in the positive axis direction.

16.3.4 Open

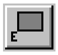
An Open  is an unobstructed vent. The properties of an Open are its Gauge Pressure and Ambient Temperature. If the flow is turbulent, an Open requires:

- Turbulence intensity, $\sqrt{\frac{u^2}{U^2}}$ (for the Differential model)
- Eddy viscosity, ν_t (for the Eddy Viscosity models)

If you have an outdoor application that radiates to the sky, you can account for this by unchecking the box next to T_{sky} and entering a temperature. T_{sky} is the effective sky temperature. Its value depends on atmospheric conditions changing from about -43C under a cold clear sky to about 12C under warm cloudy conditions.

An Open can be placed on either a Wall or an Open side of Enclosure. Click here for more information.

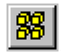
16.3.5 Wall Patch

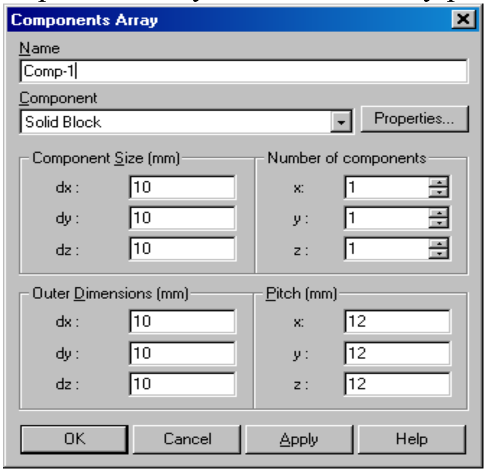
The Wall Patch  component is used to create Enclosure Walls with variable properties or to place patches of the Wall on other Enclosure components such as an Open. For example, consider a Wall at T = 300 K temperature with a patch where a heat flux is given. First, set this whole boundary to be a Wall side type with T = 300 K. Then, create a Wall Patch with the given heat flux and place it over the Wall. The patch will override the initial boundary condition of T = 300 K. A Wall Patch can be placed on a Wall, Open, or Fan side type of an Enclosure.

16.4 Parametric Objects

Parametric Objects are objects made from multiple components that are created and assembled using dialogs. To rebuild a Parametric Object, open the object's Properties dialog and change parameters. The entire object will be rebuilt automatically. If you wish to modify just one component of a Parametric Object, the Parametric Object must first be converted to a Complex Object using the Convert to Complex Object function .

16.4.1 Component Array

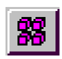
A Component Array can be created by pressing . The



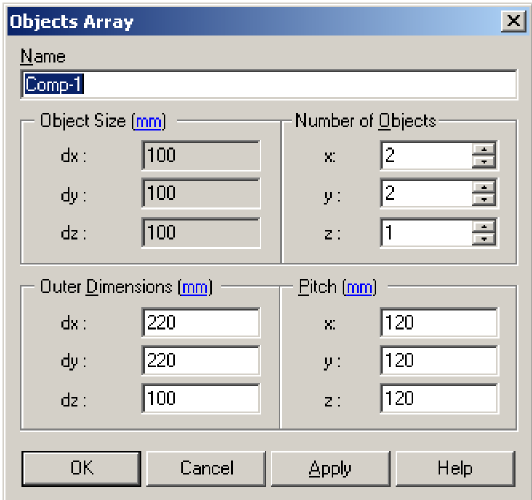
Name can be any combination of letters, symbols, numbers or spaces up to 63 characters long. Select the Component Type for the Array in the pull-down box and set the Component's Size and Properties. Finally, define the Number of Components in each direction of the array and the Pitch, i.e. the spacing between objects. Note that the x, y and z in the dialog box refer to the **local** coordinate system of component.

Press the Apply button to see the effect of selected settings. You can modify the settings until the OK button is pressed to close the dialog.

16.4.2 Objects Array

An Objects Array can be created by pressing . Objects Array is used to create a three dimensional array of the active object.

First specify the array's Name, which can be any combination of letters, symbols, numbers, or spaces up to 63 characters long. Then, define the Number of Objects in each direction of the array and the Pitch or Outer Dimensions. Notice that the last two fields are linked: when you change one the other one is automatically recomputed.



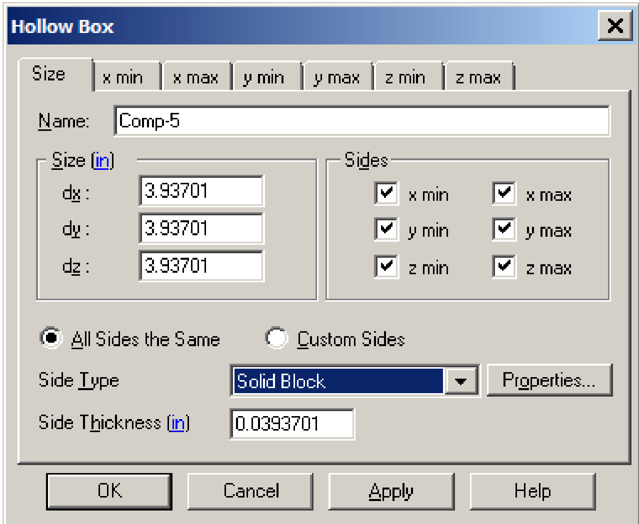
Pressing the Apply button will show you the effect of specified settings. You can modify the settings until the OK button is pressed and the dialog is closed.

16.4.3 Hollow Box

The Hollow Box  is a rectangular hollow box constructed of Solid Blocks, Plates, Planar Porous Resistances etc. The Name can be any combination of letters, symbols, numbers or spaces up to 63 characters long.

If all sides are the same (default) all the settings are on the first page of the dialog, where you specify exterior dimensions of the Hollow Box and the Sides' Type, Properties, and Thickness.

By default, all six sides are created. You can skip any one side by unchecking a corresponding box in the Sides section of the dialog. For example, to create a rectangular pipe (plenum), uncheck boxes on opposite sides of the box, say, Xmin and Xmax.



Press the Apply button to see the effect of selected settings. You can modify the settings until the OK button is pressed to close the dialog.

To specify sides individually, select **Custom Sides** and then specify each side on its page.

16.4.4 Heat Sinks

Click  to create a heat sink. A pull-down menu will prompt you to select a Plate Fin, Pin Fin, or Radial heat sink.

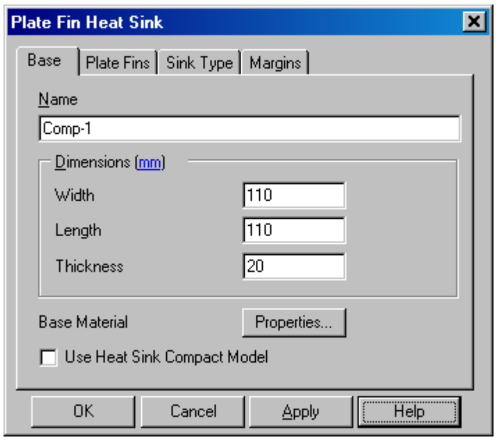
Plate Fin Heat Sinks

On the **Base** page, the component name, base dimensions, and base properties are defined. The Name can be any combination of letters, symbols, numbers, or spaces up to 63 characters long.

To model the heat sink approximately, check the box next to Use Heat Sink Compact Model. This will automatically generate the appropriate Porous Volume Resistance component to substitute for the specified heat sink. This modeling option offers an efficient alternative to detailed modeling of heat sinks, especially when assessing the overall cooling strategy in a system.

On the **Plate Fins** page, the number of fins, fin height, and fin width (thickness) are specified. Tapered fins can be specified by checking the Tapered box, and entering a Tip Width.

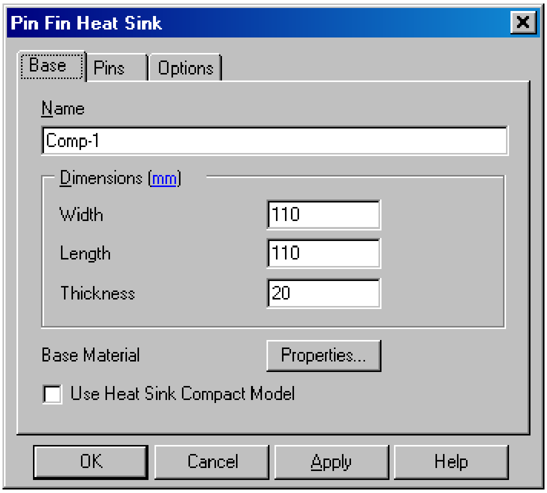
The type of heat sink is specified on the **Sink Type** page. If an Extruded/Cast heat sink is selected, no further specifications are needed on this page. Bonded fin heat sinks require a Groove Width and Groove Depth for the fin slots, a Thermal Conductivity for the adhesive, and a Fin Material. On the **Margins** page, margins can be added to any side of the heat sink base.



Pin Fin Heat Sinks

On the **Base** page, the component name, base dimensions, and base properties are defined. The Name can be any combination of letters, symbols, numbers, or spaces up to 63 characters long. To model the heat sink approximately, check the box next to Use Heat Sink Compact Model. This will automatically generate the appropriate Porous Volume Resistance component to substitute for the specified heat sink. This modeling option offers an efficient alternative to detailed modeling of heat sinks, especially when assessing the overall cooling strategy in a system.

On the **Pins** page, the Type of pin can be specified as either a Solid Block (rectangular pins), or as a Cylinder (round pins). The number of pins can be specified along the base's length and width. The Pin Length is the pin size in the lengthwise direction of the base. The Pin Width is the pin size in the widthwise direction of the base. Tapered pins can be specified by checking



the Tapered box and specifying a Tip Width. On the **Options** page, Margins and pin Offsets can be specified.

Radial Heat Sink

The component name is specified in the Name box. The Name can be any combination of letters, symbols, numbers, or spaces up to 63 characters long.

The overall heat sink height is specified in the Heat Sink Height box. The properties of the heat sink are selected by pressing the **Properties** button.

The Fin Diameter and Fin Thickness are specified in their respective boxes. The Center Post Diameter is for specifying the diameter of the post that joins the fins.

A contact post at the bottom of the heat sink is specified by checking the Contact Post box and entering its height and diameter.

16.5 Object Constructors

Object Constructors are used to build the following complex objects: Hollow Cylinder, Inclined Block, Solid Block with a Hole, Cylinder Cutout, and Cylinder Sector. Unlike Parametric Complex Objects discussed above, Object Constructors create regular complex objects that do not retain construction parameters.

16.5.1 Hollow Cylinder

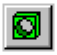
The Hollow Cylinder is a complex object made of Plates and Baffles. Click  to create the component. Specify Name, which can be any combination of letters, symbols, numbers or spaces up to 63 characters long.

In the Size section, specify the diameter(s) of the cylinder and height. The Wall Thickness and Properties specify the cylinder’s wall thickness and material properties.

Number of Segments defines the number of Plates/Baffles of which the cylinder is built. More segments will define a better cylinder, but require more grid lines. Normally 8 to 12 segments are sufficient for a well-defined cylinder. Check the Arc box, if you want less than a 360-degree cylinder.

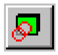
Press the Apply button to see the effect of selected settings. You can modify the settings until the OK button is pressed to close the dialog.

16.5.2 Solid Block with a Cylindrical Hole

Press  to create the object. Specify the object's Name, which can be any combination of letters, symbols, numbers or spaces up to 63 characters long. Specify the shape of the hole, its diameter(s), and height (z), the number of Quadrants and segments. More segments will define a better cylinder but will require more grids. Click Properties to specify the Solid Block's material properties.

Press the Apply button to see the effect of selected settings. You can modify the settings until the OK button is pressed to close the dialog.

16.5.3 Cylinder Cutout

The Cylinder Cutout  is a cylindrical object made out of Cutouts for drilling circular (elliptic) holes in the Solid Block, Plate, Planar and Volume Resistance components.

Name can be any combination of letters, symbols, numbers or spaces up to 63 characters long. In the Size section, specify the diameter(s) of the cylinder and the height. More segments will define a better cylinder, but require more grids. Normally 7 to 11 segments are sufficient to have a well-defined cylinder.

Press the Apply button to see the effect of selected settings. You can modify the settings until the OK button is pressed to close the dialog.

16.5.4 Cylinder Sector

The Cylinder Sector  is a circular or elliptical cylinder created from Solid Blocks and Prisms.

Name can be any combination of letters, symbols, numbers or spaces up to 63 characters long. In the Size section, specify the diameter(s) of the cylinder and the height. More segments will define a better cylinder, but require more grid lines. Normally 8 to 12 segments are sufficient to have a well-defined cylinder. Select the quadrants you want created by clicking on the appropriate icons in the dialog. Check the “Make it hollow” box and specify an interior diameter to create a thick walled tube. There is a lower limit on the thickness of the tube walls so Coolit will warn you if the specified interior diameter is too large. Select Properties to change the material properties of the Cylinder Sector.

Press the Apply button to see the effect of selected settings. You can modify the settings until the OK button is pressed to close the dialog.

17 Advanced Topics

17.1 Reference Conditions

Reference Conditions (Reference Temperature and Reference Pressure) in the **Package|Global Settings** dialog play different roles in different problems. For steady-state problems with heat transfer and with open Enclosures (i.e. Enclosures with Vents, and/or Opens, and/or Fans), Reference Conditions are used for initialization and scaling only. The converged solution is independent of Reference Conditions.

For steady-state problems with heat transfer and with sealed Enclosures, Reference Conditions are used to compute the gas mass in the Enclosure. Therefore, the converged solution will directly depend on Reference Conditions.

For transient problems, Reference Conditions define the initial conditions and hence determine the solution.

For steady state problems without heat transfer (i.e. the Enable Heat Transfer box in the Global Conditions dialog is unchecked), Reference Conditions are used to compute the density and viscosity of air.

17.2 Grids and Grid Settings Dialog

17.2.1 The Default Grid

Coolit needs a grid of discrete cells (finite volumes) to compute a solution. To generate the grid, press the icon, or select the Package|Grid Mode menu, or press the F4 key. The default grid will appear in the three 2D views. Note the checked **Automatic Grid** box on Control Bar. This indicates that you are currently using automatically generated grid and that the program is in automatic grid generation mode. As you modify the geometry, the entire grid will automatically change. If the box is unchecked, the grid is in the manual mode, and all the magenta grid lines will stay unchanged unless you edit them manually.

The number of grid lines in each direction is listed in the status bar window. Anchor (gray) grid lines will be drawn on the borders of all components and moveable (magenta) grid lines will be drawn to ensure a satisfactory grid. It is best to use the wireframe view mode to view the grid

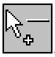
Note that in finite volume methods the solution is assumed to be constant within each cell, therefore you should place more cells where the solution varies rapidly.

17.2.2 Manually Editing the Grid

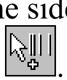
Uncheck the Automatic Grid box to enable the manual grid editing controls. To delete a grid line, press the Delete button on the Control Bar. The cursor will change to . Click near the desired line. The moveable (magenta) grid line closest to the cursor will be deleted.

When the mouse passes over a moveable grid line with the Select button depressed, the cursor will change to either  for vertical lines or to  for horizontal lines. When the cursor changes, the grid line can be moved by clicking and dragging. Coolit requires a certain minimum number of grid lines maintained between and within components. These minimums are specified in the Parameters page of the Grid Settings dialog. If this minimum is not met, grid lines will be added automatically during the grid check.

Two more restrictions are automatically (and dynamically) enforced when editing the grid: the **Maximum Cell Stretch Ratio** and **Maximum Cell Aspect Ratio**. The Maximum Cell Stretch Ratio monitors and enforces the ratio of neighboring grid cell sizes in any one direction, e.g. dX_i/dX_{i-1}, or dZ_j/dZ_{j-1}, where dX_i is the size of cell *i* in X-direction, $i = 1,2,3,...NX$. The Maximum Cell Aspect Ratio monitors and enforces the ratio of one cell's dimensions in two directions, e.g. dX_i/dY_i or dY_i/dZ_i. The default values for Maximum Stretch Ratio is 5 and for Aspect Ratio is 100. These values can be changed in the Grid Settings dialog. If you add/delete/move grid lines so that either restriction is violated, grid lines restoring the restrictions will be automatically added/deleted/moved.

Grid lines can be added by first pressing the Add | Line, Add --- Line, Multiple | Lines, or Multiple --- Lines button in the Grid Control Bar. The cursor will change to reflect the selection to , , , , respectively. Click the cursor at the desired grid line location. The specified selection will be added.

If the multiple grid lines option is selected, the grids will be added between the grid lines on either side of the cursor. For multiple grid lines three options have to be specified in the Add Parameters section: the type of grid variation (Symmetric, Expansion, and Contraction), the number of grid lines added (Amount) and the Change Factor.

Change Factor determines the rate of change of grid cell size. It can vary between 1 and 2. For example, the Change Factor 1.1 means that each cell is greater/smaller than its immediate neighbor by 10%. With the Change Factor greater than 1, there are 3 options with respect to the grid distribution. The Contraction/Expansion options will concentrate the grids either to one side or the other as indicated by their respective cursors: , . The Symmetric option, , will attract the grids to both bounding grid lines.

17.2.3 Grid Commands

Coolit provides six command options for grid control: Clear, Check, Smooth, Refine, Default, and Settings. These commands are available only if Coolit is in the manual grid mode (i.e. the

Automatic Grid box on Control Bar is unchecked). The Clear command will remove all non-essential grid lines. The grid Check ensures that the Solver will be able to run the problem; that is that the minimum grid requirements are satisfied. If this minimum is not met, grid lines will be added automatically during the Grid Check. The Smooth command will redistribute the already present grid lines more uniformly. This button can be pressed repeatedly for greater effect. The Refine command adds new grid lines. This command can be used repeatedly.

Pressing the Default button replaces the currently displayed grid with the default grid and enables the Automatic Grid mode.

17.2.4 Grid Settings Dialog

The default grid can be modified using the controls in the **Grid Settings dialog** (when in Grid mode, press the **Settings…** button on Control Bar). Select the desired grid zone (or the Package) in the combination box at the top of the dialog. Checking the Isotropic box under the Grid Size tab will make the X-direction entries apply to all directions.

The Number of Cells entry controls the number of cells in coordinate directions. Move the slider and click Apply button. The grid will change and the resulting grid values can be viewed at the top of the dialog. The Grid Expansion Factor variable determines the rate of grid expansion.

The **Parameters** page permits you to define the minimum spacing between anchor grid lines in each coordinate direction (**Grid Tolerance**). Spaces smaller than the specified Tolerance will be eliminated by resizing the offending components, subject to the following limitation. If anchor lines belong to components separated by an air gap, in order to snap the components (and eliminate the gap), you will also need to uncheck the **Do Not Join Components** box. By default, the box is checked and such components will not be snapped, irrespectively of the Anchor Grid Tolerance setting.

On the Parameters page, you can also specify the minimum number of grid cells in air channels between components (at least 1 cell) and in Solid Block components (at least 1). When there is only 1 cell in a channel, CoolPlot will display zero velocity in the channel. In this case, use Gauge to measure the flow.

Maximum Cell Stretch Ratio and **Maximum Cell Aspect Ratio** grid quality measures are discussed here.

17.2.5 Guidelines for a Well-Designed Grid

A good grid is essential for computing an accurate solution and for quick convergence. To accurately resolve important flow features you should maintain minimum grid resolution. Specifically at least 4-5 cells are recommended within a shear layer and 5-6 cells in a separated region, and 3-4 cells in a flow passage.

It is standard practice to first use a coarse grid during the initial design and use a finer grid in the final stages of the project. A coarse grid is useful for providing general understanding of the flow and heat transfer. For quantitative analysis, a finer grid should be used.

To test if the computed solution is numerically accurate, refine the grid and compare the new solution to the original results. Look for trends in the data that indicate a diminishing change with continued refinement. Remember, there is no value in computing a solution that is numerically 5% accurate when the input data into the model is only 20% accurate.

17.3 Solver Settings Dialog

Prior to running the Solver, you should review the settings in the **Solver Settings** dialog. These settings affect the solution convergence rate and time, the required computer RAM, and output information generated by Coolit.

17.3.1 Steady/Transient

Once the Solver is started, calculations proceed iteratively, gradually approaching the solution. The type of solution (time-dependent or steady) is specified on the **Steady/Transient** page of the **Solver Settings** dialog. In most cases, you will be looking for a steady state solution. Convergence Precision in this case is proportional to the number of significant digits in the solution. The default is 10^{-3}. The Solver will stop when either all the residuals drop below the **Convergence Precision** level or when the **Max Number of Iterations** is reached.

Time Stepping options: **Local Time Stepping** and **Courant Number** are available only for steady state solutions. Courant number (C) defines a dimensionless time step, which is used to integrate the governing equations until steady state is reached. If the Courant Number box is checked, the Courant number will be set automatically.

Generally, smaller Courant numbers increase the stability of the convergence process (convergence lines in the monitor will have fewer and smaller kinks), but may increase the convergence time. Increasing Courant number may decrease the stability (more kinks with larger amplitudes sometimes to the point of divergence), but may reduce the convergence time.

Selecting Local Time Stepping will tell the solver to automatically compute the time step for every grid cell depending on the size of that cell and on the local flow velocity. By default Local Time Stepping is turned on.

Underrelaxation for Fans is used only when at least one fan curve is specified (i.e. more than one (dP, Q) point for at least one fan is given). In cases with steep fan curves, oscillations may occur. Reducing the Underrelaxation factor, which ranges from 0 to 1, may reduce or eliminate these oscillations and improve convergence. The default value 0.2 works for most cases.

Underrelaxation for Heat Sources is used only with temperature-dependent heat sources. In cases with steep Power-Temperature curves, oscillations may occur. Reducing the Underrelaxation factor, which ranges from 0 to 1, may reduce or eliminate these oscillations and improve convergence.

The default state of the **Heat Always On** check box is unchecked. If the box is checked, the heat transfer equation is solved at every iteration together with the rest of the governing equations, instead of allowing the software to decide when to solve it depending on flow conditions and global convergence.

For **Transient** problems, you will need to specify any two of the three requested parameters: **Time Interval**, **Time Step**, and **Max Number of Steps**. The remaining parameter will be computed from the specified two. In order to ensure time accuracy, the specified Time Step should be commensurate with important time scales of the problem being solved.

17.3.2 Solver Selection

Coolit provides two solver options: **Economy** and **Fast**. Despite their names, Fast solver is not always faster than the Economy solver, although this is often true especially for larger cases (>5M grid cells). The convergence rate (time) for Fast solver sometimes reaches over 10 times that of the Economy solver for very large problem. However Fast solver requires significantly more RAM. Typically the Economy solver needs ~0.5 Gb RAM per 1 M grid cells in single precision (32-bit word format) and 1 Gb per 1 M cells in double precision (64-bit word format). Fast solver needs approximately ~1.5 Gb per 1M cells in either precision/format.

Floating Point Format determines whether the single (32-bit word format) or double precision solver is used (64-bit word format). Double precision is often required for larger problems and for problems with highly non-uniform grids with stretched and generally widely ranging cell sizes.

The default is the single precision solver, which is recommended for most applications. The double precision **Economy** solver will require almost twice as much RAM and about 50% more time per iteration to run compared to the single precision solver. **Fast** solver in double precision requires 10-20% extra RAM and time. Note that the solver precision setting has little bearing on accuracy of computed results. This option is used to mitigate the round-off error when solving cases with large and highly uneven grids, which have difficulty converging in single precision.

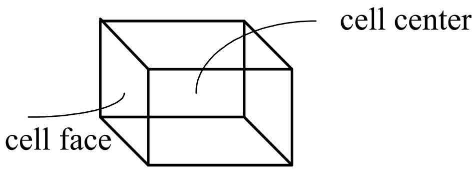
Parallel Processing section specifies the number of CPU cores used by the solver. (If you have not purchased this option, it will be grayed out.)

Momentum Equations Iterations section specifies the **Max Number of Iterations** of momentum equations per each global iteration reported by the solver and the corresponding convergence **Precision**. If convergence Precision is met before the Max Number of Iterations limit is reached, the solver would move on to computing the pressure etc. In steady-state cases, when momentum equations residuals lag other residuals, this option can reduce the required global iterations to convergence. In transient cases, it will increase the stability and allow larger time steps to be used. In both steady and transient cases, however, there is a trade-off between the solution time and this setting. Even though fewer iterations (or time steps) may be needed, the solution time may increase due to a longer solution time per iteration (time step). In most cases, the default setting is optimal.

17.3.3 Listing File

To save results for the whole domain select the List Volume radio button, otherwise choose List Planes (default). In most cases the Listing Planes option is the only practical choice due to the large amount of data that List Volume would produce.

The Listing File variables are organized into two groups: cell-centered and face-centered. (Face-centered data lie on grid planes and cell-centered at geometrical centers of grid cells.) At the grid cell centers you can output velocity vector components (V_x, V_y, V_z), the pressure, $P\text{-}P_{reference}$, the temperature T, and for turbulent flows, the turbulence kinetic energy, k (k is available only for the Differential turbulence model), and the relative turbulence kinematic viscosity, ν_t, (i.e. the turbulent viscosity divided by the molecular viscosity). At cell faces you can print the temperature, the heat transfer coefficient, h, and for turbulent flows, the y+ variable (see Wall Functions). Note that because y+ and h are defined only at solid surfaces (Enclosures Walls, Solid Blocks, Plates, Planar and Volume Resistances), Coolit will print zeros everywhere else. For Plate and Planar Resistance, Coolit prints values for both sides; the side facing the coordinate origin and the side facing away from it, designated by (-) and (+) respectively.



In addition to the selected variables, the Listing File will have a complete documentation of the physical properties data, grids, components listing, and the convergence history. The case documentation part of the Listing File is always printed, independently of whether the Produce Listing File box was checked.

To select which planes will be printed, first select the plane's direction (the number next to the radio button is the number of grid lines, or planes, available). Then enter the desired plane in the Plane text box and select Add. To remove planes from the output, highlight the plane to be removed in the list box, and click Remove.

17.3.4 Plot File

If the **Save Plot File** box is checked, the solver will store computed results for CoolPlot for graphical display. Available variables for the Plot File output include the temperature, T, the pressure, $P\text{-}P_{reference}$, the velocity vector, V, the heat flux vector, q, the heat transfer coefficient, h, and for turbulent flows the normalized turbulence viscosity, ν_t, (i.e. the turbulence viscosity divided by the molecular viscosity), and the turbulence kinetic energy, k (available for Differential model only). You can also set the reference temperature for calculating h.

The **Refine Triangular Grid of Imported Objects** option is used to improve the display of imported objects. This option requires considerable computer memory and has no effect on the model accuracy. In transient cases, you can save all output time step results into a single file (default) or in multiple files - one file per output time step. For large cases with many output steps, the last option is often a necessity.

17.3.5 Restart File

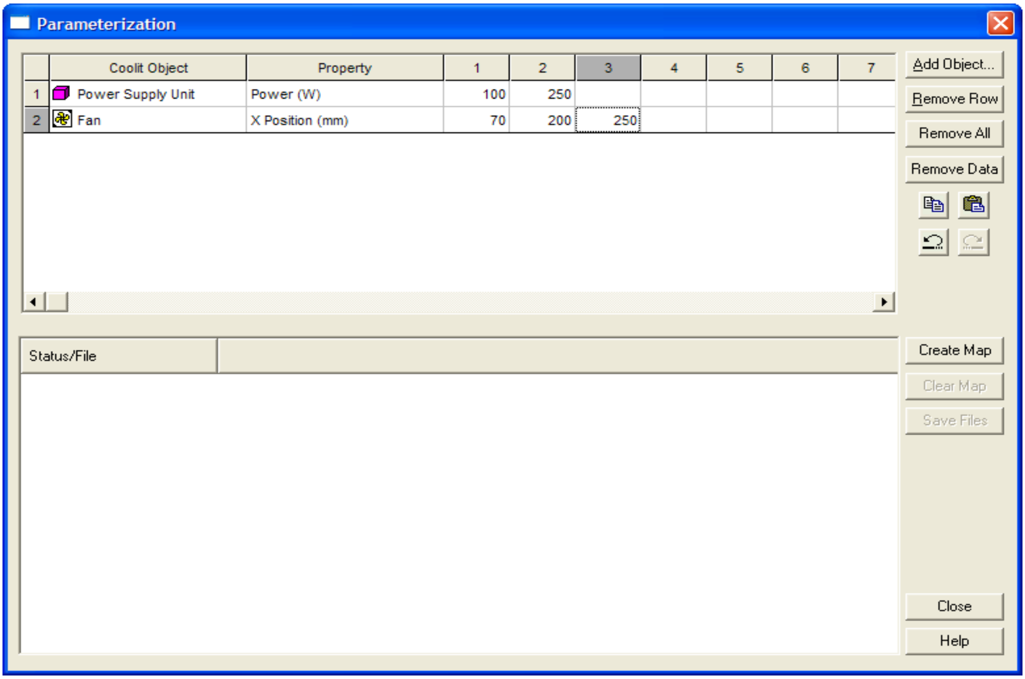
On the Restart File page, specify the Restart File name, the frequency of saving restart files, and the number of restart files to keep. The default for steady state flows is to save a file every 100 iterations and to keep only the last one file. For transient flows, the default is to save files every 10 time steps and to keep 4 last files. It also saves a restart file for every Output Times table value under the name: <CaseFileName>_<time in 13.5e format>_sec.fld.

Both steady and transient cases save the last iteration/time step restart files before stopping.

Click the button or Solver|Restart to load a restart file.

17.4 Parametric Models

The Parameterization dialog is opened by clicking Solver|



Parameterization… Click **Add Object…** to select objects to be parameterized. In the **Select Object** dialog, select objects you want to add to the project and click OK to close. For more than one object, all standard Windows object selection rules apply: Ctrl-click, Shift-click and the rubber band. In the Property column, click the pull down arrow to select the property to be parameterized.

Numerical values of parameters are entered sequentially starting from the first column in the table to the right of the Properties column. Tab, Shift-Tab, Enter, and Shift-Enter keys can be used to navigated the table. No empty cells in the middle of the table are allowed.

The **Remove Row** button removes the entire row: the Coolit Object, Property, and parameters; **Remove All**, removes the entire table; **Remove Data**, removes all parameter values. Copy/Paste buttons, permit the copying and pasting of parameter data to and from Coolit. If your Excel data contain empty cells, such cells will be automatically removed during the paste process. Undo/Redo buttons, will help restore accidentally deleted data.

Click the Create Map button, to create a map (list) of all possible files based on specified parameters. Each row in the Case Map table includes the Case file name and its parameter values. Click Save Files, to save the Case Map files with checked check boxes, save the batch file, *filename-batch.cbf*, and the map file, *filename-map.htm*. You can solve the saved cases either individually or using Batch Processing.

17.5 Guidelines for Planar Porous Resistance (PPR)

The Planar Porous Resistance model (PPR) is used to simulate perforated plates and vents. The main purpose of using PPRs is to reduce the number of grid cells required to solve a problem. Approximately integrating the PPR governing equations, the pressure drop across a PPR can be expressed as:

$$\Delta p \approx \frac{\mu H}{\kappa} V + \rho C H \frac{V|V|}{2} \qquad (1)$$

where Δp is the pressure drop across the PPR, μ is the dynamic viscosity of the fluid, H is the thickness of the PPR, κ is the permeability coefficient, **V** is the approach velocity, ρ is the density of fluid, and C is the inertial resistance factor. Note, that CH is the standard (dimensionless) loss coefficient. The unknowns in (1) are κ and C, which are required in Coolit's PPR Properties dialog.

To simplify the discussion, we will rewrite (1) using the following notation,

$$\Delta p = aV + bV^2 \qquad (2)$$

$$\text{where } a = \mu H/\kappa \text{ and } b = \rho CH / 2 \qquad (3)$$

Our aim is to obtain *a* and *b* coefficients in (2) as a function of percent open, *ϕ,* so that the resulting pressure drop is within 20% of the pressure drop computed by Coolit for Plates with perforations. Once you determine *a* and *b* for your range of velocities and percent open from the formulas below, you can compute κ and C from (3). In the event of $a=0$, κ is set to some large number, e.g. 10^{20}.

In what follows, we will provide ready to use formulas for generic staggered and inline perforated plates. If you prefer to obtain resistance coefficients for your specific perforated plates, we provide a detailed procedure for this in Numerical Experiments section.

17.5.1 Empirical Formulas

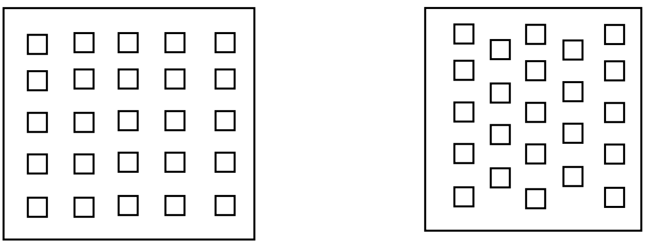


Figure 1. Inline (left) and staggered perforated plates considered in this section.

(a) Inline Perforated Plate Results

Empirical Formulas for Inline Perforated Plates				
	a		*b*	
V = [0.01, 5] m/s	$0.275\phi^2 - 0.696\phi + 0.473$	ϕ = [0.1, 0.9]	$0.5213\phi^{-2.443}$	ϕ = [0.1, 0.5)

			$-43.14\phi^3+108.06\phi^2-92.31\phi+27.16$	ϕ = [0.5, 0.9]
V = [0.5, 5] m/s	0	ϕ = [0.1, 0.75]	$0.610\ \phi^{-2.398}$	ϕ =[0.1, 0.35]
			69.38 exp(-6.350ϕ)	ϕ =[0.35, 0.75]

In this table, the ranges of velocities, V, and percent opens, ϕ, in which the formulas are valid, are shown in brackets after respective variables. A square bracket means that the value is included in the range, a parenthesis means that it is not.

Consider an example of a perforated plate with inline holes, an approach velocity around 1 m/s and percent open 0.5 (i.e. 50%). You have an option of using either the second formula in line 1 or the second formula in line 2:

Choice 1:

$$a = 0.275\phi^2 - 0.696\phi + 0.473$$

$$b = -43.14\phi^3+108.06\phi^2-92.31\phi+27.16.$$

Choice 2:

$$a = 0$$

$$b = 69.38\ \exp(-6.350\phi).$$

From the table for $\phi = 0.5$ below, note that the error at V=1 m/s for Choice 1 is 3.7%, while for Choice 2 it is 6.6%. Also note that the error for Choice 1 grows rapidly away from V=1, while for Choice 2 it is relatively stable. Thus, if you are confident that the approach velocity is close to 1 m/s, Choice 1 is preferable, while if there is a considerable uncertainty for V, the Choice 2 formula will provide a better prediction.

The following tables compare results computed using the above formulas to results computed by Coolit. The nomenclature is as follows: **Δp** is the pressure drop computed using Coolit, **Δp(1)** is the pressure drop computed using the formulas in the first row of the table above, **Δp(2)** is the pressure drop computed using the formulas in the second row of the table above, and **err, %** is the percent error between Δp and Δp(1), and between Δp and Δp(2), respectively. Note, that for illustration purposes, we sometimes used the formulas outside of their recommended range of applicability.

$\phi = 0.1$

V	Δp	Δp (1)	err, %	Δp (2)	err, %
0.01	1.79e-02	0.185E-01	3.5	0.153E-01	14.8
0.10	1.27e+00	0.149E+01	17.0	0.153E+01	20.1
0.50	3.42e+01	0.363E+02	6.3	0.381E+02	11.5
1.00	1.39e+02	0.145E+03	4.3	0.153E+03	9.7
3.00	1.28e+03	0.130E+04	1.7	0.137E+04	7.2
5.00	3.64e+03	0.362E+04	0.6	0.381E+04	4.8

$\phi = 0.25$

V	Δp	Δp (1)	err, %	Δp (2)	err, %
0.01	4.97e-03	0.470E-02	5.4	0.169E-02	65.9
0.03	2.15e-02	0.234E-01	8.6	0.153E-01	29.1
0.05	4.90e-02	0.543E-01	10.9	0.424E-01	13.5
0.10	1.70e-01	0.186E+00	9.3	0.169E+00	0.3
0.50	4.32e+00	0.401E+01	7.1	0.424E+01	1.9
1.00	1.79e+01	0.157E+02	12.1	0.169E+02	5.3
3.00	1.69e+02	0.140E+03	17.4	0.153E+03	9.8
5.00	4.91e+02	0.387E+03	21.2	0.424E+03	13.7

$\phi = 0.35$

V	Δp	Δp (1)	err, %	Δp (2)	err, %
0.01	3.40e-03	0.331E-02	2.7	0.752E-03	77.9
0.10	8.40e-02	0.941E-01	12.0	0.752E-01	10.5
0.50	1.74e+00	0.183E+01	4.9	0.188E+01	8.0
1.00	7.11e+00	0.704E+01	1.0	0.752E+01	5.7
3.00	6.86e+01	0.618E+02	10.0	0.676E+02	1.4
5.00	1.94e+02	0.171E+03	12.0	0.188E+03	3.1

$\phi = 0.5$

V	Δp	Δp (1)	err, %	Δp (2)	err, %
0.01	2.10e-03	0.220E-02	4.8	0.290E-03	86.2
0.03	7.30e-03	0.818E-02	12.0	0.261E-02	64.3
0.10	3.90e-02	0.457E-01	17.1	0.290E-01	25.6
0.50	6.60e-01	0.754E+00	14.2	0.725E+00	9.8
1.00	2.72e+00	0.282E+01	3.7	0.290E+01	6.6
3.00	2.80e+01	0.242E+02	13.5	0.261E+02	6.8

5.00	7.98e+01	0.667E+02	16.5	0.725E+02	9.2

$\phi = 0.65$

V	Δp	Δp (1)	err, %	Δp (2)	err, %
0.01	1.50e-03	0.146E-02	2.4	0.112E-03	92.5
0.10	2.20e-02	0.233E-01	6.1	0.112E-01	49.2
0.50	2.90e-01	0.310E+00	6.9	0.280E+00	3.6
1.00	1.07e+00	0.110E+01	3.1	0.112E+01	4.5
3.00	1.00e+01	0.911E+01	8.9	0.101E+02	0.7
5.00	2.94e+01	0.248E+02	15.5	0.280E+02	4.9

$\phi = 0.75$

V	Δp	Δp (1)	err, %	Δp (2)	err, %
0.01	1.10e-03	0.111E-02	0.7	0.593E-04	94.6
0.10	1.50e-02	0.157E-01	4.6	0.593E-02	60.5
0.50	1.70e-01	0.181E+00	6.3	0.148E+00	12.8
1.00	5.90e-01	0.617E+00	4.6	0.593E+00	0.5
3.00	5.26e+00	0.492E+01	6.4	0.534E+01	1.4
5.00	1.55e+01	0.133E+02	14.1	0.148E+02	4.4

$\phi = 0.9$

V	Δp	Δp (1)	err, %	Δp (2)	err, %
0.01	7.24e-04	0.710E-03	2.0	0.229E-04	96.8
0.10	8.84e-03	0.854E-02	3.4	0.229E-02	74.1
0.50	7.69e-02	0.748E-01	2.7	0.572E-01	25.7
1.00	2.26e-01	0.230E+00	1.7	0.229E+00	1.2
3.00	1.57e+00	0.165E+01	5.3	0.206E+01	31.1
5.00	4.33e+00	0.436E+01	0.7	0.572E+01	32.0

(b) Staggered Holes Plate Results

The nomenclature for staggered plate results is the same as for the inline plates. Note, that the upper limit of percent open for staggered plates is 0.5.

Empirical Formulas for Staggered Plate				
	a		*B*	
V = [0.01; 5] m/s	$0.4777+0.3367\phi - 4.22\phi^{2} + 4.833\phi^{3}$	ϕ = [0.1; 0.5]	$0.5051\phi^{-2.4507}$	ϕ = [0.1, 0.5]
V = [0.5; 5] m/s	0	ϕ = [0.1; 0.5]	$0.563\phi^{-2.430}$	ϕ = [0.1, 0.5]

The following tables compare results computed using the above formulas to results computed by Coolit. The nomenclature is the same as the one used for inline holes.

$\phi = 0.1$

V	Δp	Δp (1)	err, %	Δp (2)	err, %
0.01	1.84e-02	0.190E-01	3.3	0.152E-01	17.6
0.10	1.26e+00	0.147E+01	16.9	0.152E+01	20.3
0.50	3.36e+01	0.359E+02	6.8	0.379E+02	12.7
1.00	1.37e+02	0.143E+03	4.4	0.152E+03	10.6
3.00	1.28e+03	0.128E+04	0.4	0.136E+04	6.5
5.00	3.54e+03	0.357E+04	0.8	0.379E+04	7.0

$\phi = 0.2$

V	Δp	Δp (1)	err, %	Δp (2)	err, %
0.01	6.97e-03	0.676E-02	3.1	0.281E-02	59.7
0.10	2.82e-01	0.302E+00	7.2	0.281E+00	0.3
0.50	6.91e+00	0.673E+01	2.6	0.703E+01	1.7
1.00	2.87e+01	0.265E+02	7.7	0.281E+02	2.0
3.00	2.66e+02	0.236E+03	11.3	0.253E+03	4.9
5.00	7.50e+02	0.654E+03	12.8	0.703E+03	6.3

$\phi = 0.3$

V	Δp	Δp (1)	err, %	Δp (2)	err, %
0.01	4.40e-03	0.426E-02	3.2	0.105E-02	76.1
0.10	1.20e-01	0.129E+00	7.9	0.105E+00	12.5
0.50	2.77e+00	0.258E+01	6.9	0.262E+01	5.3
1.00	1.16e+01	0.999E+01	13.9	0.105E+02	9.5
3.00	1.09e+02	0.879E+02	19.4	0.945E+02	13.3
5.00	3.03e+02	0.243E+03	19.8	0.262E+03	13.4

$\phi = 0.4$

V	Δp	Δp (1)	err, %	Δp (2)	err, %
0.01	2.96e-03	0.294E-02	0.6	0.522E-03	82.4
0.10	6.31e-02	0.724E-01	14.7	0.522E-01	17.3
0.50	1.17e+00	0.132E+01	12.5	0.130E+01	11.5
1.00	4.71e+00	0.502E+01	6.5	0.522E+01	10.8
3.00	5.11e+01	0.437E+02	14.5	0.470E+02	8.1
5.00	1.45e+02	0.121E+03	16.9	0.130E+03	10.0

$\phi = 0.5$

V	Δp	Δp (1)	err, %	Δp (2)	err, %
0.01	2.20e-03	0.223E-02	1.3	0.303E-03	86.2
0.10	4.10e-02	0.471E-01	15.0	0.303E-01	26.0

0.50	6.50e-01	0.788E+00	21.2	0.758E+00	16.7
1.00	2.52e+00	0.296E+01	17.3	0.303E+01	20.4
3.00	2.49e+01	0.254E+02	2.2	0.273E+02	9.7
5.00	7.31e+01	0.700E+02	4.2	0.758E+02	3.8

17.5.2 Numerical Experiments

In this section, instead of relying on formulas from the previous section you will compute the resistance coefficients using numerical simulation. You will build two wind tunnels in Coolit: one for the PPR and the other for the perforated plate. In both cases, a wind tunnel is built as a rectangular tube made of Symmetry sides with an inlet (Fan) and outlet (Open). The PPR wind tunnel can be relatively short with coarse grids: the accuracy of predicted pressure drop in fact does not depend on either tunnel length or grid density. It is controlled exclusively by κ and C resistance coefficients that you specify in Coolit and the entire pressure drop occurs over the PPR.

The wind tunnel for the perforated plate has to be built differently. The pressure drop in real perforated plates occurs up- and downstream of the plate as the flow is slowed before the plate, it is forced through holes and then expands out of the holes forming vortices downstream. To correctly predict the pressure drop across the perforated plate, the flow up- and downstream of the plate has to be resolved. Therefore, you will need a longer wind tunnel with a finer grid. The length of the wind tunnel is determined by the flow pattern downstream of the perforated plate. It is desirable that the tunnel be long enough to prevent inflow through the outlet section. Since it is impossible to determine the tunnel length in advance, you may need to run several experiments with different tunnel lengths for the maximum anticipated approach velocity.

In the perforated plate case, you will also need to make a model of the plate. Normally, perforations are made with a symmetry pattern, examples of which are shown in Figure 1 above. If that is the case, you can save time by modeling only a symmetry element of the pattern and using symmetry walls for the wind tunnel. Examples of symmetry elements for Figure 1 are shown in Figure 2. If a perforated plate lacks a symmetry pattern, you can take a representative (sufficiently large) piece of the plate in order to determine the pressure drop.

The pressure difference required by the experiments is determined in both cases as the difference between the average pressure over the outlet, minus the average pressure over the inlet.

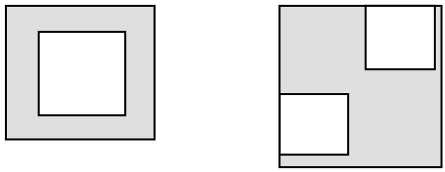


Figure 2. Examples of symmetry elements for inline and staggered perforated plates in Figure 1.

Once you have established the minimum length of the wind tunnel, you should run several computer experiments, using different approach velocities within the anticipated range in order to determine the resistance coefficients in eq. (2). If the range of anticipated velocities is relatively small and it fits in either the linear or the quadratic portion of the pressure curve, you can save time by assuming that one of the coefficients in (2) equals zero.

Typically, for device velocities (device velocity is equal to the approach velocity divided by percent open) less than 0.5 m/s the pressure is a linear function of velocity and you can assume the inertial resistance factor C to be zero. Above 0.5 m/s, the resistance is well described by the quadratic term alone, hence in eq. (2) $a = \mu H/\kappa = 0$, i.e. the permeability, κ, is set to some very large number. All the above cases are to be discussed in the following paragraphs individually.

Low Velocity Case:

In low velocity cases, the viscous (linear) forces are dominant. For example, assume a natural convection environment and the range of expected device velocities in the 0.01 -- 0.2 m/s range. You can safely assume the pressure drop to be a linear function of velocity, i.e. $b = C = 0$ and

$\Delta p = aV$, where $a = \mu H/\kappa$.

Only one data point is required to determine a. However, as in any experiment, whether a physical or numerical, it is better to obtain several data points and then average the result. Let us assume that you ran 3 experiments and computed three Δp's for 3 V's, that is

$\Delta p_1 = aV_1$

$\Delta p_2 = aV_2$

$\Delta p_3 = aV_3$.

Before computing the average, you should normalize the above equations to ensure that their contributions have a comparable weight. Dividing each equation by respective velocities is one way of doing this, e.g. $\Delta p_1 / V_1 = a$, etc. The average a is then simply

$a = (\Delta p_1 / V_1 + \Delta p_2 / V_2 + \Delta p_3 / V_3) / 3$.

The general case of N-experiments would result in:
$a = \frac{1}{N}\sum_{i=1}^{N}\frac{\Delta p_i}{V_i}$. Finally, you can determine the permeability coefficient from: $\kappa = \mu H / a$.

High Velocity Case:

In high velocity cases, inertial (quadratic) forces dominate. For example, imagine a forced convection problem with the expected range of device velocities from 1 m/s to 3 m/s. We assume that the viscous forces are negligible with $a = 0$ and hence $\kappa = \infty$. In practice, κ is set to some large number, e.g.

10^{20}. Following the procedure described above, and assuming N-experiments, we get

$$b = \frac{1}{N}\sum_{i=1}^{N}\frac{\Delta p_i}{V^2{}_i}, \quad \text{and} \quad C = 2b/(\rho H).$$

General Case:

Finally, we will consider the most general case in which the velocity range covers both linear and non-linear regions and, hence, neither coefficient in eq. (3) can be assumed to be zero. Applying the least squares method to find the *a* and *b* coefficients, we will minimize

$$\sum_{i=1}^{N}\left[\Delta p_i / V_i^2 - a/V_i \; - b\right]^2 = \min$$

where N is the number of numerical experiments, N>2. The resulting system would be:

$$\sum_{i=1}^{N}\left[\Delta p_i / V_i^2 - a/V_i \; - b\right] = 0 \qquad (3)$$

$$\sum_{i=1}^{N}\left[\Delta p_i / V_i^2 - a/V_i \; - b\right] / V_i \; = 0 \qquad (4)$$

The solution of system (3)-(4) is

$$a = \frac{\begin{vmatrix} \sum \frac{\Delta p_i}{V_i^2} & N \\ \sum \frac{\Delta p_i}{V_i^3} & \sum \frac{1}{V_i} \end{vmatrix}}{\begin{vmatrix} \sum \frac{1}{V_i} & N \\ \sum \frac{1}{V_i^2} & \sum \frac{1}{V_i} \end{vmatrix}}, \quad b = \frac{\begin{vmatrix} \sum \frac{1}{V_i} & \sum \frac{\Delta p_i}{V_i^2} \\ \sum \frac{1}{V_i^2} & \sum \frac{\Delta p_i}{V_i^3} \end{vmatrix}}{\begin{vmatrix} \sum \frac{1}{V_i} & N \\ \sum \frac{1}{V_i^2} & \sum \frac{1}{V_i} \end{vmatrix}}.$$

The permeability coefficient and inertial resistance factor can be found from: $\kappa = \mu H / a$ and $C = 2b / (\rho H)$.

17.6 Guidelines for Porous Volume Resistance (PVR)

PVR is a porous media based model, which is used to simulate 3D objects such as pin fin heat sinks. The main purpose of using a PVR is to reduce the number of grid cells required to solve a problem. For example, a typical pin fin heat sink would require at least 5x5 grid cells per pin and per space between pins. Assuming that the sink has 10x10 pins and that in a vertical direction you use 10 grid cells, it will require a total of 158,400 grids. A model utilizing an equivalent PVR can get away with as few as 125 grids.

The amount of air that will go through the heat sink depends on the relative resistance of the sink and the wind tunnel. The larger the ratio of the wind tunnel's cross-section to the cross section of heat sink, the less air will go through the sink. Ideally, you would need to guess the wind tunnel dimensions in such a way that its resistance is equal to that of the system that you are trying to model. The best option, if you can afford it, is to place the heat sink in an environment similar to the one in which it will be used.

When you don't have information about the target system, make a generic wind tunnel about six times the length of the sink's base, with width about two times the sink's base width, and with height about five times the height of the heat sink. If the heat sink has a plane of symmetry, you can use it to save on grids. In experiments described below, we used half an inline pin fin heat sink, and half a wind tunnel, as shown in the diagram below. You should use the same wind tunnel both for the pin fin sink and for the PVR sink to make a meaningful comparison. At the wind tunnel inlet specify a Fan with a velocity that you expect in the real case. At the tunnel's outlet specify the Open. Also, specify power deposition in a Wall Patch (or several Wall Patches if you have multiple heat sources) under the base of the heat sink.

As for the grid, at least 5x5 grid cells per pin, and per interpin space, is recommended.

To obtain the pressure drop, two Gauge components are required in the wind tunnel, one upstream and the other downstream of the heat sink. The Gauges should have the same cross section as the heat sink and they should be placed directly on the front and rear row of pins.

Once the wind tunnel is set up with the pin fin heat sink in it, run the case. Record the pressure difference $\Delta P_o = P_{upstream} - P_{downstream}$ and the maximum temperature T_{max}. $P_{upstream}$ and $P_{downstream}$ are the average pressures on the two Gauges. They will be displayed in the CoolPlot Objects Manager window when the Gauges are selected.

Now use the wind tunnel from the pin fin sink experiment and replace the pins there with the PVR. Parameters need to be determined for the PVR. First, estimate the Permeability Coefficient. A case is defined as "high velocity" if the Reynolds number based on approach velocity and the distance between pins is greater than 10. In such cases, inertial effects dominate and one can consider the linear resistance small. Therefore, you can set the Permeability Coefficient to a large number, e.g. the Coolit default is 10^{20} m^2. Next, specify the

Percent Open (see the definition in online Help file). Now the objective is to find the Inertial Resistance Factor, C, such that the pressure drop in the PVR is equal to the pressure drop computed for the pin fin heat sink. Use eq. (1) to obtain the initial C:

$$C = \frac{2\Delta P_o}{H\rho V^2} \qquad (1)$$

where ΔP_o is the pressure drop obtained from the pin fin experiment, ρ is the air density, V is the approach velocity, and H is the linear size of the heat sink in the direction of flow.

Uncheck the Enable Heat Transfer option in the Heat Transfer page of the Global Settings dialog. After setting the PVR parameters as discussed above, run the case. Compute ΔP_{PVR} from Gauges and compare with ΔP_o from the pin fin sink computation. If $\Delta P_{PVR} > \Delta P_o$, reduce C, otherwise increase it and rerun the case until $\Delta P_{PVR} = \Delta P$. Usually only a few runs are required to get satisfactory accuracy. To save time, you can restart new cases using previous case's restart file.

To reduce the number of iterations, you can use the following extrapolation formula. Compute C_1 from eq. (1) and obtain a corresponding ΔP_1. Increase or reduce C_1 depending on whether ΔP_1 is greater or smaller than the pin fin sink's ΔP_o. Compute ΔP_2 for the new C_2. Now estimate C_3 from:

$$C_3 = \frac{C_2(\Delta P_1 - \Delta P_o) - C_1(\Delta P_2 - \Delta P_o)}{\Delta P_1 - \Delta P_2}\,. \qquad (2)$$

Use C_3 from eq. (2) to compute a pressure drop, ΔP_3. If the pressure drop is not satisfactory, use it together with whichever is better, ΔP_1 or ΔP_2, to compute C_4 from eq. (2). Repeat, if necessary.

Once the Inertial Resistance Factor is determined, you can determine the effective thermal conductivity of the PVR, such that the maximum temperature of the PVR is equal to that of the heat sink. Turn on heat transfer by checking the Enable Heat Transfer option in the Heat Transfer page of the Global Settings dialog. Set the PVR's conductivity in directions parallel to the base equal to that of air, $k_\tau = k_{air}$ and in the direction normal to the base set it to the conductivity of the heat sink material, $k_n = k_{sink}$. Run the case and compare the computed PVR maximum temperature, $T_{\max}^{PVR}$, to that of the heat sink, $T_{\max}$. Normally the $T_{\max}^{PVR}$ will be higher than $T_{\max}$, so you should increase k_τ (keep k_n the same). If $T_{\max}^{PVR}$ is lower than $T_{\max}$, increase k_n and keep k_τ the same instead. Since the area that the PVR is modeling has only air and the heat sink, the conductivity of the PVR should fall between that of air and that of the heat sink.

If the second value is still not satisfactory, use a variant of eq. (2) to compute the third k_τ

$$k_3 = \frac{k_2(T_1 - T) - k_1(T_2 - T)}{T_1 - T_2} \qquad (3)$$

where we omitted *max* and τ subscripts, the subscripted variables refer to the PVR model and the unsubscripted variables refer to the heat sink.

18 Frequently Asked Questions

18.1 Why should I use a turbulence model?

Turbulence is associated with the presence of both spatial and temporal random fluctuation in the flow field. It develops gradually from laminar flows that display regular variations in both space and time, through a sequence of instabilities, which develop as the Reynolds number (Re), which measures the ratio of the inertia to viscous forces of flow, is increased. Turbulent fluctuations significantly increase the intensity of the momentum and heat transport. Typical values in electronics are between 10 to 100 times the rate of the corresponding molecular transport rates. Increasing the rate of heat transfer is normally a positive development for electronics cooling, while increasing the momentum transport is not, as it results in higher wall shear stress and hence higher pumping requirements. Under these conditions neglecting turbulence is not an option.

By now it has been definitively established that the Navier-Stokes equations solved by CFD programs, do describe turbulence. That is in theory, you don't need any additional equations to model turbulent flows. In practice, the turbulence is associated with very small-scale fluid motions (eddies). Thus the ratio of the smallest length scale to the size of domain in turbulent flow is proportional to $Re^{-3/4}$. That is, even for very slow turbulent flows with the Reynolds number of a few thousands, grids on the order of $1{,}000^3$ are required. Such direct simulations, therefore, are of purely academic interest.

The only practical way to compute turbulent flows that exists today is to use turbulence models. Most popular models use the eddy viscosity concept proposed by Boussinesq. This concept is based on the analogy between molecular transport of momentum and heat and their turbulent transport by the eddies. Using this approach one solves the ensemble-averaged Navier-Stokes equations and a model that computes the eddy transport.

Unlike the molecular viscosity and thermal conductivity, which are strictly material properties of fluid, the eddy properties depend on the nature of the flow and vary significantly over the solution domain.

As of now, there is no single turbulence model that works equally well across all flow types and geometries. However, there are models that work better than others for specific flow conditions. Coolit provides a choice of 4 turbulence models, which have been shown to work well in low Reynolds number, buoyant flows encountered in electronics cooling applications.

18.2 When should I use a turbulence model?

A turbulence model can be used in the following three cases; when the flow is

- turbulent,
- mixed (laminar and turbulent zones), and
- time-dependent.

If the flow is fully developed turbulent (which is uncommon in electronics cooling applications), the decision is simple: you should use one of the Coolit provided models.

When the flow is mixed, i.e. laminar, transitional, and turbulent regions coexist within the solution domain, the decision whether to use a turbulence model will depend on locations of important components. If all critical components are in turbulent regions, for example, next to the fan, the decision to use a turbulent model is clear. When they are in both turbulent and laminar regions, eddy viscosity models should be used. These models have been shown to predict mixed flows with good accuracy.

In many problems the temperature field in components is steady, while flow in at least some areas of the modeled domain is time-dependent. For example, in many natural convection problems, the rising plume often becomes unstable (time-dependent) or in forced convection problems time-dependent vortex shedding occurs behind blunt objects. In such cases no steady state solution exists and hence there is no convergence. Unless your interest is in obtaining accurate resolution of time behavior of the unstable portion of the flow, solving such problems as time-dependent is not a good idea. Normally the temperature inside components is steady and if your objective is to find that temperature you have two options.

Solve the problem as laminar until solution residuals come to a stably oscillating pattern. You can define the solution at that stage as "converged". Although the flow pattern in the oscillating portion of flow is apparently periodically changing, the temperature in components does not. The second option is to use the eddy viscosity turbulence models to dampen the time-dependent oscillations and force a steady state solution.

18.3 How to Tell if Flow is Turbulent?

The parameter indicating whether transition to turbulence may occur is the so-called critical Reynolds number, Re_{cr}. That is if $Re < Re_{cr}$, then flow is always laminar independently of turbulence level at the inlet. The inlet turbulence will decay to zero. If $Re > Re_{cr}$, the flow may or may not be laminar, depending on inlet conditions. For example, while in circular tubes $Re_{cr} = 2{,}300$, laminar flow has been observed with Re as high as 50,000.

For a flow between parallel flat plates, two different definitions of Re are used. The first definition, $Re_L=UL/\nu$, where U is the inlet flow velocity, and ν is the air viscosity, is based on the plate length, L. It is appropriate for the so-called “short” channels, whose ratio of the thickness of the boundary layer, δ, formed at the channel exit and the width of the channel, d, is less than ½, i.e. $\delta /d < ½$. The boundary layer thickness can be estimated from: $\delta = L \cdot Re_L^{-1/2}$. For smooth short plates, $Re_{cr} = 10^5 - 10^6$.

For "long" channels, those with $\delta /d > ½$ and with smooth walls, the Reynolds number is based on the channel width, d, $Re_d = Ud/\nu$. For such channels the flow is laminar if $Re_d < Re_{cr} \approx 2{,}500$.

There are several turbulence models capable of predicting a transition to turbulence due to the high inlet turbulence (the so-called bypass transition mechanism). Two such eddy viscosity models are available in Coolit. With complex flows when it is not obvious whether the flow is turbulent, these models should be considered.

The above discussion concerned forced convection flows. For free convection, the commonly used parameter to determine transition is the Rayleigh number, Ra=Gr*Pr, which is a product of Grashof and Prandtl numbers. Assuming $Pr \approx 1$, for "long" channels $Ra_{cr} \approx 9{,}000$ and for "short" channels $Ra_{cr} \approx 10^9$, where "long" and "short" are defined above. The boundary layer thickness for natural convection along a vertical flat plate is $\delta \approx L \cdot Gr_L^{-1/4}$.

The practical problem is how to define Reynolds numbers in a complex geometry, which may have tens of length and velocity scales with the correspondingly large number of Reynolds numbers. We can suggest several practical guidelines to help determine the flow regime.

- Flow is always turbulent near a Fan outlet. On the suction side, though, the turbulence zone can be small.
- Flows between closely spaced components and surfaces are almost always laminar.
- Expanding flow streams become turbulent at lower Reynolds numbers, while converging streams require higher Re to become turbulent.

18.4 Which turbulence model to use?

Coolit provides algebraic, differential, and two eddy viscosity models.

The algebraic model is simple and it is faster to compute than eddy viscosity models. It provides reasonable accuracy for making quick estimates in many electronics cooling applications. For important cases, compare its predictions with either of the eddy viscosity models. Comparing the computed solution in key areas, will tell you whether the algebraic model is accurate enough for your purposes. The important disadvantage of the Algebraic model is that the user specified background turbulence (Turbulence Level) serves as the basis of computations. Background turbulence is not easy to estimate and hence you may want to use instead eddy viscosity models, which compute background turbulence. In eddy viscosity models, the user-specified background turbulence is used only to start computations.

The two eddy viscosity models - one by Secundov and the other by Spalart and Allmaras are similar. Both work well in mixed flows relaminarizing in regions where turbulence vanishes, such as the interfin space in heat sinks. In general, our experience has shown the Secundov model as somewhat more accurate for electronics cooling applications. The difference between these models is usually small especially for slower flows. When the difference is detectable, the Spalart-Allmaras model produces more conservative (warmer) results. On the other hand, Spalart-Allmaras model equations were optimized for good convergence, which often contributes to better convergence of all governing equations resulting in faster solution times. As always, the ultimate judge of accuracy is experiment. If you have experimental data, compare it with predictions of different models and select the one that works best for your system.

18.5 How to determine if the grid resolution is sufficient?

Solve a problem on a given grid. Refine the grid and solve the problem again. If the solution has not significantly changed the first grid was sufficient. Otherwise continue the grid refinement.

18.6 What are grid requirements for accurate pressure?

The pressure losses in plate fin heat sink type of geometry are caused primarily by the shear stress along the plate fins. Secondary contributions are the form drag created by the blunt ends of the fins and the flow contraction and expansion into and out of the heat sink.

Before you grid the problem, you must select a flow model. There are at least two possibilities:

- The flow is laminar both inside and outside the heat sink. In this case, use the laminar model.
- The flow is laminar or transitional inside the heat sink and transitional or turbulent outside. In this case, use an eddy viscosity model.

When computing the Reynolds number for flow in the duct before and after the heat sink use the hydraulic diameter of the duct as the length scale.

Once you have selected your flow model, grid the problem manually according to the guidelines below for each direction.

Perpendicular to the Fin Surfaces

- In Grid mode, use the Multiple Lines button with an expansion factor of 1.2 or less to grid between the fins. Begin with a grid that has at least 1 grid cell in the boundary layer computed from $H_{bl} = 10w / \sqrt{Re_{10w}}$. Here w is the distance between fins, $Re_{10w} = U_{ave} \cdot 10w / \nu$, where U_{ave} is the average velocity in the channel, and ν is the kinematic viscosity of the air. Refine the grid at least twice and monitor the pressure drop solution for change.

- For mixed flows that contain laminar, transitional, and turbulent flow regions, eddy viscosity models provide the best accuracy. As in the laminar case above, the grid must be fine enough to resolve the gradients near the walls in the channels. The non-dimensional parameter, y+, reported by CoolPlot provides a tool for evaluating the fineness of the grid near the wall. Typically, y+ < 2 on channel walls should compute the pressure drop to within 10%. Compute a solution with the default grid. View the results in CoolPlot and select the y+ option in the Surfaces pull-down menu. View the surface of the interior plate fins to evaluate the y+ value over the bulk of the fin surface (it may be useful to hide some fins and adjust the range of the Color Palette). While monitoring the pressure drop solution for change, use the Multiple Lines grid editing tools with an expansion factor of 1.2 or less to refine the grid in the channels until the y+ value drops below 2.

- It is computationally prohibitive to resolve the velocity gradients in high Reynolds number flows. In these cases, Coolit's wall functions kick in automatically to provide good accuracy on coarse grids.

In the Flow Direction

For all cases: in order to resolve the form drag and the developing boundary layer in the channel, you need to have several cells stacked up near the entrances and exits of the channels. Use the Multiple Lines grid editing tools with an expansion factor of 1.2 to create a grid with at least three cells in the first channel width before and after the entrance to the channels and after the exit of the channels. As always, refine the grid at least twice and monitor the pressure solution for change.

In the Fin Height Direction

For all cases: Grid density in this direction has a weak effect on the solution inside channels.

18.7 What are grid requirements for accurate temperature?

The grid requirements for computing accurate temperature solutions are significantly lower than that for the pressure despite the fact that the boundary layer thickness of velocity and temperature are very close. Fortunately, resolving the temperature boundary layer is not necessary in order to predict accurate temperature in both the solid and the air. Here's why.

Lets assume that a solid object dissipates Q Watts of power. A good CFD program should conserve momentum, mass, and energy EXACTLY. Such a program would then compute that the entire Q is dissipated in the air, irrespectively of the grid. This is an integral property, i.e. the property which is obtained by integrating around the solid's surface of differential heat fluxes for each surface grid cell. And even though the heat fluxes through individual surface grid cells may not be very accurate and dependent on the grid an integral over all surface cells would exactly match Q.

The temperature of solid object depends not only on Q but also on the temperature of the air around it. Specifically, in a CFD model, it depends on the temperature of the air in the first cell outside the solid. In most cases, the default grid is sufficiently fine to compute the bulk airflow velocity and temperature around a solid body.

This is in contrast to computing the pressure drop which needs an accurate derivative, or rate of change, of the velocity. This requires a much finer grid. For example, in a short channel with Re = 1600 (based on channel width), 8 cells were required in the channel to get within 10% error for the pressure drop; only 3 cell were required to get within 10% error for the average temperature of the walls.

Finally, earlier we assumed that the bulk airflow around the solid body is computed correctly. However, when you have a fan curve, the flow rate depends on the system resistance (pressure drop). In most cases, the pressure drop through a system is largely determined by form drag of solid bodies. In such cases, a coarse grid is often sufficient to correctly predict the pressure drop and hence the flow rate.

In all cases, you should refine the grid at least twice and compare the results. If the solution is still changing significantly, continue to refine the grid.

18.8 What does Convergence Monitor show?

As the problem is being solved, Coolit shows the Convergence Monitor - a 2D plot of solution residuals vs. iteration number. In Coolit, the solution residual defines how far the computed solution for a given variable (velocity components, pressure, and temperature) deviates from steady state. Normally the solution is called "converged" when the residuals for all the variables are less than 10^{-3}.

In addition to residuals, Convergence Monitor displays plots of the temperature, pressure, and velocity magnitude at Monitor Probe locations as well as maximum and minimum values over the entire field.

18.9 What can be done when case does not converge?

In most problems, which have a steady state solution, the convergence history consists of relatively smooth curves going down until they drop below the specified convergence precision line. In some problems, it never happens; and we say that the problem does not converge. Two most often encountered reasons for non-convergence are closely spaced grids and problems which have no steady state solution.

The convergence is affected by the closely spaced grids because computers use finite precision for computations and if the grid spacing is near the limit of computer precision, severe round off problems will occur. The important notion here is not the absolute distance between grids, but rather the normalized distance, Threshold = $|x_i - x_{i+1}| / x_{i+1}$, where x_i and x_{i+1} are coordinates of neighboring grid lines. Grid lines closer than Threshold, by default 0.001, are reported in Model Check dialog's warnings for Closely Spaced Components.

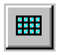
If a case with Closely Spaced warnings does not converge, you should eliminate the reported problems. This can be done either manually by moving and resizing components, so that they line up or by specifying the Anchor Grid Tolerance in the Grid Settings|Parameters dialog. The Tolerance should be greater than the largest of the offending distances listed by Model Check. This can also be done automatically in the Model Check dialog by pressing the Fix button.

Another often encountered reason for non-convergence is when a flow develops time-dependent flow patterns due to the instability. A common example of this phenomenon is an oscillating wake behind a blunt object. Such a periodic vortex shedding produces corresponding oscillations in residuals, that measure the departure of solution from steady state. You won't be able to find a steady state solution to such flow problems, because it does not exist.

Users have several options after recognizing that the wake's oscillations have little effect on the temperature of the cylinder: accept the solution with oscillating residuals as "converged"; use a turbulence model to dampen the oscillations and obtain a steady state solution, or solve case as time-dependent, obtain results over a period, and average them.

You can also sometimes reduce the magnitude of oscillating residuals by unchecking the Local Time Stepping option in the Solver Settings|Expert Parameters dialog and reducing the Courant number.

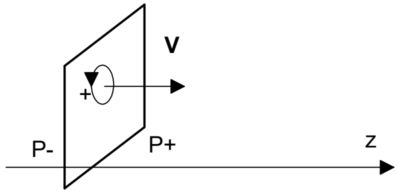
18.10 How to Create a Filter or Flow Resistance

To create a filter, click  to create a Planar Porous Resistance (if your filter is very thick, you may want to model it with a Porous Volume Resistance). After you name and position the PPR, select Properties and choose UAF Filter as the Modeled Object. Coolit comes with a library of UAF filters available in the pull-down menu. If, however, the filter you wish to model is not in the library, you can create a custom filter to suit your needs. Under the Modeled Object tab select Custom Filter/Flow Resistance. Under the Flow Properties tab enter an Approach Velocity, corresponding Pressure Drop, and Percent Open. Fill in the material properties and you are done. If your resistance curve is nonlinear, instead of Approach Velocity select Flow Rate and click the Curve button, to specify the filter resistance curve.

A third option exists for those who want to model a specialized filter or who are primarily interested in filter selection. Choose Custom as the Modeled Object and enter the requested data determined by following the guidelines for **PPR**.

18.11 How to Model Fan at Angle

To simulate fans at angles not normal to main coordinate directions, you can specify the Unit Vector at the angle your fan is positioned. You will then need to adjust the Coolit Fan’s area to ensure the correct velocity through the Fan. If either x-component or y-component is non-zero, then divide the Fan size in that direction by the z-component value. If both x- and y-components are non-zero, then divide the size in both directions by the square root of the z-component value. Note, that it is not important how you achieve the area change, as you can see from the example below.



Assume that you need to model a 100 x 100 mm fan with 200 CFM. The fan is turned 30° with respect to Z-axis. To model this fan, create a Coolit Fan perpendicular to the Z-axis. Open the Properties dialog and change the Fan's flow direction to 30°, by entering Alpha = 30° on the Flow Direction page. The effective area of such a model Fan is reduced proportionally to cos(Alpha), which is given in the z-component field of the Velocity Unit Vector as 0.866025. You can achieve the required area change by stretching the Coolit Fan in the x-direction, so that the new dx size is 100/0.866025 = 115.47 mm, while keeping the dy size unchanged. If you wish to maintain the square shape of the fan, you could increase both dx and dy sizes of the Fan by $1/\sqrt{0.866025}$, so that dx = dy = 107.457 mm.

18.12 How to Use the Clone Function

The Clone function is used to create a one-dimensional array of objects out of an active object. Click a component or a complex object and then press  or select **Edit|Clone** to open the Clone Object dialog. Specify the number of copies in the array and the pitch in the X, Y, and Z directions (the distance between the beginning of the object to the beginning of the following object). Press the Apply button to see the effect of selected settings. You can modify the settings until the OK button is pressed to close the dialog. The cloned objects are independent and do not constitute a new complex object.

18.13 How to Display Component Information

If you use the Solid View in Coolit (button), the components will be color-coded, e.g. Plates - green, Solid Blocks - violet, etc. In addition, as you pass the mouse cursor over the component, the Status Bar will display the component's name and location. You can double-click on any component in any of the 2D views or in the 3D view to open the component's properties dialog. Note that the double-click feature is active only when the Select button is depressed. Component information can also be viewed in the Objects Manager.

18.14 How to Change Defaults

To change default values for frequently used variables, select **Tools|Options...** and go to the Default Settings page. The new defaults will become effective when you restart Coolit.

18.15 How to Change Properties of Selected Objects

To change the properties of a group of Objects, first you need to select the desired objects. This can be done using the Objects Manager, or by using the rubber band selection. The rubber band selection works by clicking and dragging the mouse either in 2D or 3D view windows around the desired objects. As you drag the mouse, it will display a dashed rectangle. Upon releasing the mouse button, all the objects inside the rectangle will be selected.

To edit properties of these selected objects, open the Objects Manager and in the pop-up menu click Selected|Properties....

18.16 How to Create a Plenum

The easiest way to create a rectangular plenum (pipe) is to use the Hollow Box parametric complex object. Create a Hollow

Box with dimensions of the pipe and open opposite sides. If the pipe is not straight, create its segments using this approach and then assemble the pieces.

18.17 When to Use Solid Block vs Plate

The Solid Block model provides a complete description of heat transfer in a solid body. Given correct physical properties this model can predict the temperature distribution accurately. Solid Blocks require at least two grid lines (one grid cell) in every coordinate direction.

The Plate component uses an approximate model of heat transfer in solid objects. It requires a single grid line in the Plate's plane. Plate model uses an analytical solution to predict heat transfer normal to the Plate's surface. In tangential directions, when Plate is used in combination with Solid Blocks the heat transfer is accounted for using an approximate, but accurate model.

As a rule, if either of the following conditions is not satisfied, the Solid Block model should be used:

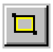
- Heat conduction from the object's sides is small.
- The object has one dimension that is much smaller than the other two.

An example of the first condition not satisfied is when an object with a heat source embedded into a good conductor. Unless the object is very thin, i.e. its thickness is less than 0.1 or better than its other dimensions, Plate model should not be used. Another example is of a heat source bearing object which is not thin, i.e. the second condition is not satisfied, but its sides don't conduct heat due to ambient conditions, for example, a chip on a PCB, which has either bottom only in contact with solid or both top and bottom (PCB below and heat sink on top). In such a case, Plate model would be very accurate.

Plate can be used to model thin copper plating of PCBs, thin copper layers inside of PCBs. Plates should not be used for modeling heat sink fins, as its in-plane conduction model works only when a Plate is either “sandwiched” between Solid Blocks or is on top of a Solid Block. Plates can also be used as baffles to direct air flow.

18.18 How to Find Pressure and Flow Through Cross Section

You can approximately determine the mean heat and air flow rate through any part of an active cross section in CoolPlot as well as the average pressure and temperature.

First, position the cross section by dragging it to the desired position or double click it and then type in the new position. Now click the  button and then draw a rectangle in the Section View window by dragging the mouse. As soon as you release the mouse, an information screen will appear which will show the size, the position, the average of the currently selected scalar, the average normal velocity, and the volumetric flow rate. To display information about another variable, select it as usual in the toolbar - the information screen will be updated automatically. If you wish to draw another rectangle, simply click the mouse in another spot and repeat.

Note that this method can be used only for rough estimates and can result in significant errors on coarse grids. For accurate analysis use Gauge components. Fans, Opens, and Vents do not require a Gauge. In these components, the fluxes are computed directly in the solver and can be viewed by selecting the object in CoolPlot's Objects Manager.

18.19 How to Solve Multiple Projects in Batch

The Batch Queue dialog permits you to schedule sequential solving of multiple cases. To access the dialog, select **Solver|**

Batch Processing… To create a new batch job, click the Add button, then select a Coolit project file from the dialog. The selected file will now be listed in the batch queue. Additional projects are added to the queue the same way. To change the order of projects in queue, simply drag and drop them as required.

Several icons are used to indicate the status of the batch job. In the leftmost column, an arrow shows the currently processed file. In the next column, the icon can take several forms. The icon indicates that the project is waiting to be processed. The icon shows the currently executing project. The icon tells that the project was successfully processed, and finally the icon indicates that the project run failed. Selecting the failed project will give you the corresponding error message in the status line of the dialog. The third column can have the two modes: - start and - restart, which can be toggled with the mouse. If is selected, the project starts from scratch. When is specified, you have two choices. Restart from the default, *projectname.fld* file, or press the Restart File button to select a different restart file.

Once the queue is ready, pressing the Start button starts the batch. The function of the dialog's buttons is summarized below:

New – start a new batch queue

Open – open an existing batch queue file

Save – save the current batch queue

Save As – save the current batch queue under a different name

Start – execute the current batch queue starting with the first project

Continue – execute the current batch queue starting with the project immediately following the last executed project

Stop – offers three choices: a) continue with the current project, b) stop the current project and start the next in line project, c) stop the entire batch queue

Output Plot File – saves a CoolPlot file of the currently running case

Yield - Slow down the solver to permit other applications run faster

Show Monitor - displays the Convergence monitor for the presently running case

Network - Checking the Network box allows for solving a batch of problems on several computers at once

Servers - Opens a dialog for selecting which computers to use while using the Network Solver

Close – close the batch queue dialog

Add – add a new project to the queue

Rename – the selected project

Restart File– select a restart file for the selected project

Remove – remove the selected project

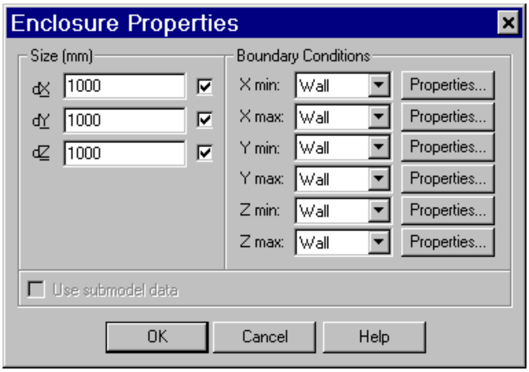
Remove All – remove the entire queue

18.20 How to Resize the Enclosure from Any Side

It is often convenient to resize the Enclosure from a specific side. For example, you set up a case with a custom grid, solved it, and now you wish to reduce the spacing between the X_{min}

wall and the Interior Components. If you use the Control Bar to modify the size or drag the Enclosure side with the mouse, Enclosure will be resized from the X_{max} side. Moving components towards the left (X_{min}) wall will destroy your grid.

Coolit provides two solutions for resizing the Enclosure from X_{min} side. If you want to use the mouse to resize the Enclosure, hold the Shift key while dragging the side. Enclosure will move X_{min} towards your Interior Components. The second method is to use the Enclosure Properties dialog. Enter the new Enclosure size and check the box next to it. When you click OK, Enclosure will be resized from X_{min} side.



18.21 How to Obtain Snapshot of Solution before Convergence

A large model may take considerable time to reach a fully converged solution. A mistake in the setup wastes that time. A good practice is to look at a snapshot of solution after just a few iterations to see if there are any apparent setup errors.

In order to do that, click the Output Plotting File button . The partially converged solution will be exported to the plotting file. Launch CoolPlot to look at the solution. You may, for example, discover a component that is unexpectedly hot, indicating a possible incorrect power dissipation or thermal conductivity value. Or, when a flow seems too fast, check fan curves.

18.22 How to Export to CAD Files?

Coolit models can be exported in STEP, STL, and IDF file formats. To export your model, choose **File|Export**.

18.23 How to Non-dimensionalize Error in Temperature?

Normally, you take the difference between the predicted value and the measured value and divide it by the measured value. Here is an example: Ambient = 40C, predicted junction temperature for a chip = 120C, measured = 110C. Error = (110-120) / 110 = 9% However, if you use the Kelvin scale to report temperature, you get: Ambient = 313K, predicted junction = 393K, measured = 383K Error = (383-393) / 383 = 3%. Obviously, the same data shouldn't give different errors.

To determine the error between Coolit predicted temperatures and measured values, you need to look at the temperature rise above ambient:

in degrees Celsius:
Error = [(110-40)-(120-40)] / (110-40) = [70-80] / 70 = 14%

or in degrees Kelvin:
Error = [(383-313)-(393-313)] / (383-313) = [70-80] / 70 = 14%

The same rule applies when comparing computed predictions to each other (for instance, when investigating the sensitivity of the computed temperature rise to the grid density). In this case, the reference temperature rise is not a measured value. So you should use the average temperature rise for all of the solutions to normalize the data. For example, two Coolit simulations give a component temperature of 101C and 98C respectively. The ambient is 20C. The sensitivity is:

sensitivity = (101-98) / [(101+98)/2-20] = 3 / (99.5-20) = 4%

The standard non-dimensionalization does not work well with temperatures, because the Celsius and Fahrenheit scales have "offsets" built into them that make zero Celsius or zero Fahrenheit not equal to absolute (Kelvin) zero. When you take differences, both in the numerator and denominator you subtract out the offset. The Kelvin scale does not have an offset (0 K is the absolute zero) and therefore does not have this problem.

Even in cases where the temperature is measured in Kelvin, using the standard non-dimensionalization is usually inappropriate. Typically it is used only when the value of the variable relative to zero is important. For example, if you are using Coolit to predict the pressure drop, dP, across a heat sink at a fixed flow rate. In this case, dP across your heat sink is measured relative to zero dP, so the standard non-dimensionalization equation is used to calculate the error.

In electronics cooling applications there is normally an ambient temperature, $T_{ambient}$, which dictates the lowest possible temperature for the problem. In such cases, that ambient temperature is the reference and any errors in Coolit predictions for temperatures should be calculated relative to $T_{ambient}$:

$$\text{Error} = (T_{measured} - T_{predicted}) / (T_{measured} - T_{ambient})$$

18.24 What is a lumped-parameter/compact model?

In this section, we will use the words “lumped-parameter” and “compact” interchangeably, with the word “compact” usually applied specifically to lumped-parameter models of IC packages. Devices within electronic systems are modeled using lumped-parameter approach when we are not concerned with or unable to obtain detailed description of the flow and heat transfer within the device and are satisfied with modeling the device’s effect on the rest of the system. Lumped parameter models are usually created by volume averaging and replacing detailed structure with a system of thermal and/or flow resistances, such that the effect of the averaged model on the rest of the system is similar to that of the actual device.

For example, model Fans use the actual fan curve, so the flow rate produced by the model is the same as the flow produced by the actual fan. But certainly no detail inside the fan is available, so there is no sense in refining the grid to obtain the flow structure inside the fan - it’s not there. Another example is the porous media model. It provides the flow resistance and impacts the ambient flow similar to what the actual porous media would, but as with the Fan model, there are no pores in the model. They are all averaged out and no grid refinement is going to recover flow within individual pores/passages.

Finally, consider compact models of IC packages. Considering nanostructure of IC packages, even the so-called detailed models are crude approximations. Compact, resistor-based models, are even cruder approximations, but they are often accurate enough to account for heat transfer and to predict with acceptable accuracy what matters – the case and junction temperatures of the IC device. As with all compact models, placing more grid cells inside a resistor model is not going to give any more information than having just a cell or two.

19 Theory

19.1 Gas Phase

Coolit uses the following Navier-Stokes model, which is solved to predict gas flow and temperature distribution.

$$\frac{\partial \rho}{\partial t}+\nabla\cdot(\rho V)=0 \qquad (1)$$

$$\rho\left[\frac{\partial V}{\partial t}+V\cdot\nabla V\right]=-\nabla P+\nabla\cdot 2\mu D+\rho g \qquad (2)$$

$$\rho c_p\left[\frac{\partial T}{\partial t}+V\cdot\nabla T\right]=-\nabla\cdot k\nabla T+Q \qquad (3)$$

$$\rho=\frac{mP}{RT} \qquad (4)$$

The variables are defined in the table below.

Variable	Definition
ρ	Gas density
V	Gas velocity vector
P	Pressure
D	Deformation tensor, $1/2(\nabla\mathbf{V}+(\nabla\mathbf{V})^T)$
g	Acceleration of gravity vector
μ	Viscosity
T	Temperature
K	Thermal conductivity
Q	Heat sources
m	Molecular mass of gas
c_p	Specific heat capacity
R	Gas constant (R=8.314 J/mol*K)

For laminar flows, the gas phase viscosity and thermal conductivity are assumed equal to their molecular values. For turbulent flows, they are computed as a sum of their molecular and turbulent components. The turbulent viscosity and thermal conductivity are computed in accordance with the selected turbulence model.

19.2 Interior Components

19.2.1 Solid Block Model

The Solid Block model is activated only when the user selects the conjugate heat transfer option.

$$\rho_0 c_p \frac{\partial T}{\partial t} = \nabla \cdot k \nabla T + Q \qquad (5)$$

If the Specified Temperature option is selected, no equations are solved in the Solid Block.

19.2.2 Porous Volume Resistance Model

The momentum, energy and mass transport inside Porous Volume Resistances (PVR) are computed using the following model:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho V) = 0 \qquad (6)$$

$$\rho \left[\frac{\partial V}{\partial t} + V \cdot \nabla V \right] = -\nabla p + \nabla \cdot 2 \mu D + \left[\frac{\mu}{\kappa} V - \rho C \frac{V|V|}{2} \right] + \rho g \qquad (7)$$

$$(\rho c_p)_{pvr} \frac{\partial T}{\partial t} + \rho c_p V \cdot \nabla T = -\nabla \cdot k_{pvr} \nabla T + Q \qquad (8)$$

where C is the inertial resistance factor, κ is the permeability coefficient,

$(\rho c_p)_{pvr} = \phi \rho_0 c_p + (1-\phi) \rho_s c_s$,

$k_{pvr} = \phi k + (1-\phi)\, k_s$,

ϕ is the porosity, $\varphi = \frac{v_{gas}}{v_{gas} + v_s}$, the subscript *s* refers to the solid matrix of the PVR. This model implicitly assumes that the temperature of gas and solid matrices is the same.

19.2.3 Plate

A Plate is a thin object on the surface of which there could be various small components not specified explicitly.

The influence of these components on flow is considered only when flow is turbulent through the surface roughness. The Plate thickness is always taken into account when computing conductive heat transfer between a Plate and 3D objects, such as Solid Block(s) in direct contact with the Plate.

Because the Plate thickness is much smaller than its width and height, the temperature distribution in a Plate is always assumed stationary. In the direction perpendicular to the Plate's surface the heat transfer is assumed locally one-dimensional:

$$\frac{d}{dx} k_{plate} \frac{dT}{dx} + \frac{Q}{h \cdot A} = 0$$

where Q is the total heat source capacity and h and A are the Plate thickness and area, respectively. This equation can be solved analytically for the temperatures of both sides of the Plate. One can also readily obtain heat fluxes for each side of the Plate.

Plate's surface in contact with Solid Blocks conducts heat both in the normal direction as described above and in tangential (in-plane) direction.

For more information on using plates, click here.

19.2.4 Planar Porous Resistance

The model used to describe Porous Planar Resistance is similar to that of the Plate except that the porous media governing equations (6)-(8), appropriately simplified for the 1D momentum, mass, and energy transport across the Porous Planar Resistance, are used.

19.2.5 Cold Plate

Cold Plate is a volumetric Coolit component that is specified by its thermal resistivity and coolant properties (density, heat capacity, volumetric flow rate, inlet temperature).

On every iteration (or time step) we update the outlet temperature of Cold Plate:

$$T_{outlet} = T_{inlet} + \frac{Q}{\rho \dot{V} C_p}$$

where T_{outlet} and T_{inlet} are outlet and inlet temperatures, Q is the total heat load over the entire cold plate area (heat flux integral over the Cold Plate surface area), $\dot{V}$ is the coolant volumetric flowrate, ρ is the coolant density, and C_p is the coolant heat capacity. Cold Plate is assumed to have constant temperature $T_{ColdPlate} = 0.5 * (T_{inlet} + T_{outlet})$ and the user-specified thermal resistivity.

In CoolPlot we output Cold Plate T_{outlet} and T_{inlet}. We paint the surface of the Cold Plate in T_{surf} color, where $T_{surf} = T_{ColdPlate} + q R$, where $q = q(x, y, z)$ is the heat flux through a grid cell surface face, and R is the resistivity of the Cold Plate. Maximal temperature of Cold Plate is defined as $T_{max} = \max(T_{surf}, T_{inlet}, T_{outlet})$.

19.2.6 Fan

The interior Fans are infinitesimally thin (2D) objects, which have the surface area equal to that of the modeled Fan. The flow rate through the Fan and the pressure drop across the Fan are computed from the user specified fan curves, see Enclosure Fans for more detail.

19.2.7 Heat Source

Depending on its placement, a Heat Source specifies the term Q in eq. (3), (5), or (8).

19.3 Enclosure Boundaries and Components

The Enclosure sides and Enclosure components are used by the code to specify boundary conditions for the governing equations. These boundary conditions are listed below.

19.3.1 Wall Patch

Wall Patch components imply the no-slip and impermeability boundary conditions for the momentum equations. For the heat transfer equation the four options are available:

- Specified temperature,
- Specified heat flux,
- Specified power,
- Specified heat transfer coefficient.

These boundary conditions are specified to be constant for the entire Wall Patch. If a distribution or different heat transfer boundary conditions are desired for different parts of the Enclosure side, multiple Wall Patch components should be used.

19.3.2 Open

An Open defines boundaries of the Enclosure at which ambient conditions (pressure and temperature) are specified. At the

Open, the flow can be either in or out of the Enclosure depending on the computed flow inside the domain and the specified ambient conditions. To determine the normal velocity vector component, V_n, Coolit uses the following boundary condition:

$$\rho_0 \frac{\partial V_n}{\partial t} + \frac{\partial p}{\partial n} = 0$$

where n is the local normal. The tangential velocity component is assumed to be subject to the no-slip condition. Depending on the sign of V_n, the heat transport equation is computed using either the given $T_{ambient}$ (when there is inflow) or using "soft" boundary conditions, $\frac{\partial^2 T}{\partial n^2} = 0$, for the outflow portions of an Open.

19.3.3 Symmetry

Symmetry sides are used to reduce your domain size when a plane of symmetry exists. For example, for a case where the left and right sides of a system are mirror images from a geometry and physics standpoint, the Enclosure size could be halved, and a symmetry boundary applied.

Symmetry boundaries of the Enclosure use the standard symmetry boundary conditions:

$V_n = 0$, $\frac{\partial V_\tau}{\partial n} = 0$, and $\frac{\partial T}{\partial n} = 0$.

19.3.4 Cyclic

Cyclic boundaries are used for systems containing repeated components, but the components are not mirror images of each other. For example, if you had several identical boards stacked together in a rack system, you could apply cyclic boundaries to solve a single slot problem instead of the whole stack.

Cyclic boundaries are specified in pairs. All conditions (thermal, flow) at the matching boundary are identical.

19.3.5 Vent

Vents represent permeable areas of Enclosure sides that present resistance to flow (e.g. a grille) and which are modeled using a Porous Media Model. In addition to the flow resistance parameters discussed in Porous Volume Resistance Model, a Vent requires the thermal conductivity of the Vent material, the gauge pressure and the temperature. The normal velocity component and heat flux across the Vent are computed from the 1D porous media equations.

19.3.6 Enclosure Fan

Fans, like Opens and Vents, are permeable sections of Enclosure sides and therefore require the gauge pressure and temperature. The velocity component through an Enclosure Fan is computed as in the case of Interior Fans using fan curves and swirl, while the heat flux through the Fan is computed as for Opens.

19.4 Turbulence Models

Coolit provides algebraic, differential, and two eddy viscosity models. The models are documented below and a discussion of their relative merits can be found in FAQ: Which Turbulence Model to Use.

All Coolit models compute the turbulence viscosity, μ_t, and thermal conductivity, k_t, which are used to augment their molecular components, i.e. $\mu = \mu_t + \mu_l$ and $k = k_t + k_l$. The governing system of equations remains the same, eqs. (1)-(3), except that now these equations are considered as ensemble averaged with the use of the Boussinesq hypothesis:

$$\rho \overline{u_i' u_j'} - \frac{1}{3} \rho \overline{\delta_{ij} u_i' u_j'} = \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \tag{9}$$

$$-\rho c_p \overline{u_i' T'} = \frac{c_p \mu_t}{\sigma_t} \frac{\partial \overline{T}}{\partial x_i} \tag{10}$$

where ' indicates the fluctuating components of velocity and temperature, $\overline{\ }$ and U denote ensemble averaged values, and σ_t is the turbulence Prandtl number.

FAQ contains several topics relevant to this section such as to why and when to use turbulence models, how to tell if flow is turbulent, and, finally, which turbulence model to use.

19.4.1 Algebraic Subgrid Scale Model

The Algebraic subgrid scale model of Smagorinsky (1965) and Deardorff (1971) computes the turbulent viscosity from an algebraic formula:

$$\mu_t = \rho c^2 \Delta^2 \sqrt{G} + \rho \nu_{t0} \tag{11}$$

where ν_{to} is the background turbulence kinematic viscosity and Δ is the subgrid turbulence scale, $\Delta = \sqrt[3]{\Delta x \Delta y \Delta z}$, Δx, Δy, Δz, are average grid spacings in x, y, and z directions, and G is the turbulence generation: G=2**D:D**, where **:** denotes Gibbs double dot product of deformation tensors.

The model requires three coefficients: c, the turbulent Prandlt number, σ_t, and the turbulence level, ν_{to}. The value of c varies in the [0.1, 0.26] interval, with the lower bound suitable for pure forced convection flows and the higher for pure natural convection. For mixed convection problems, when Fans are present, we recommend using $c = 0.18$. Coolit lists $\sigma_t = 0.9$ for either 2D or 3D as default. Finally, the background Turbulence Level in the Turbulence Model dialog is specified based on experimental data when available. Otherwise, set it to $10\cdot\nu$ for developed turbulence (default), where ν is the molecular viscosity of air ($\nu \approx 1.5\cdot 10^{-5}$ m^2/sec for air at room temperature).

Coarse Grid Version is the default option. It uses a proprietary formula for computing the turbulence length scale, Δ, such that the solution's sensitivity to the grid cell size is minimized.

This model has been successfully used for a variety of natural and mixed convection flows on relatively coarse grids. The model uses wall functions and the first grid cell should be in the log-layer, i.e. y+ >30. Because the algebraic subgrid turbulence model is the fastest to compute and is reasonably accurate, it provides a good option for many electronic cooling applications.

19.4.2 Differential Subgrid Scale Model

The differential (k-equation) subgrid model computes turbulent viscosity from a formula, which involves the turbulence kinetic energy, *K*, which in turn is computed from a conventional partial differential transport equation:

$$\mu_t = \frac{1}{20}\rho \Delta \sqrt{K}$$

$$\rho\left[\frac{\partial K}{\partial t} + V\cdot\nabla K\right] = \nabla\cdot\frac{\mu}{\sigma_k}\nabla K + \mu_t G - \frac{\rho K^{3/2}}{\Delta} + g\frac{\mu\nabla T}{\sigma_t T} \qquad (12)$$

where K is the turbulence kinetic energy, $\Delta=\sqrt{\Delta x^2+\Delta y^2+\Delta z^2}$, $\sigma_k=1$, and G is the turbulence generation: G=2**D**:**D**, where **:** denotes Gibbs double dot product of deformation tensors.

The model requires a turbulent analog of Prandtl number. Coolit lists σ_t= 0.9 for either 2D or 3D as default. In addition, the turbulence level is required in the Turbulence Model dialog, Enclosure Fan and Open components, and for interior Fan components.

Turbulence Level in the case of the differential model denotes the turbulence kinetic energy defined as $\sqrt{\frac{u'^2}{U^2}}$. The Turbulence Model dialog asks for the initial Turbulence Level, which is used to start the computations, but should have no effect on the converged solution (although it may have an effect on the rate of convergence). In the Fan and Open component dialogs, Turbulence Level entry is used as the boundary condition for eq. (12). Typically the value of the Turbulence Level varies between 0.01 and 0.1 with the lower value used for quiescent flow and the higher for mixing zones such as the ones generated by Fans.

19.4.3 Eddy Viscosity Models

Coolit implements two eddy viscosity models (Secundov et al., 1992) and (Spalart and Allmaras, 1992). The models are documented below and a discussion of their relative merits can be found in FAQ: Which Turbulence Model to Use.

For both forced and natural convection thermal flows, Eddy Viscosity models demonstrated better predictive ability than the standard k-ε model and with less computational work. Unlike the Algebraic and Differential subgrid turbulence models offered by Coolit, which always use wall functions, the Eddy Viscosity models use blended wall functions which automatically adjust depending on the value of local y+ for optimal accuracy.

Similar to the other Coolit's turbulence models, Eddy Viscosity models require a turbulent analog of **Prandtl number,** σ_t. Coolit suggests $\sigma_t = 0.9$ as default. The **Underrelaxation Factor** in the Turbulence Model dialog varies in the (0, 1] range. The default value is 0.3.

In addition, Coolit requires the turbulence level in the Turbulence Model dialog and for the Enclosure Fan, Open, and interior Fan components. **Turbulence Level**, denotes the turbulence kinematic viscosity, ν_t. The Turbulence Model dialog requires the initial Turbulence Level, which is used to start the computations, but should have no effect on the converged solution (although it may have effect on the rate of convergence). Typical values for the initial turbulence viscosity are ν_t=(10-50)ν, where ν is the molecular kinematic viscosity. In the Fan and Open component dialogs, Turbulence Level entry is used to specify the boundary ν_t for the governing equations below. Unless you have experimental data for ν_t,, it should be set to (10-50)ν with the lower value, 10ν, used for quiescent flow and the higher, 50ν, for highly turbulent mixing zones such as the ones generated by fans.

Secundov Eddy Viscosity Model

$$\rho \frac{\partial \nu_t}{\partial t} + \rho V \cdot \nabla \nu_t =$$

$$\nabla \cdot \left[\rho \left(\nu_l + C_0 \nu_t \right) \nabla \nu_t \right] + \nabla \left\{ \rho \left[\left(C_1 - C_0 \right) \nu_t - \nu_l \right] \right\} \cdot \nabla \nu_t + S_{\nu_t} \qquad (13)$$

where

$$S_{\nu_t} = \rho C_2 F_2 \left(\nu_t \Gamma_1 + A_1 \nu_{t^{4/3}} \Gamma_2^{2/3} + A_2 N_1 \sqrt{(\nu_l + \nu_t)\Gamma_1} \right) +$$
$$+ \rho C_3 \nu_t \left(\nabla^2 \nu_t + N_2 \right) - \rho C_4 \nu_t \left(\nabla \cdot \langle V \rangle + |\nabla \cdot \langle V \rangle| \right) -$$
$$- \rho C_6 \nu_t \frac{N_1}{d} - \rho C_7 \nu_l \nu_t \frac{F_1}{d^2}$$

$$F_1 = \frac{N_1 d + 0.1 \nu}{\nu_t + 0.1 \nu}, \quad \Gamma_1 = \sqrt{G}, \quad \Gamma_2 = \sqrt{\left(\nabla^2 u\right)^2 + \left(\nabla^2 v\right)^2 + \left(\nabla^2 w\right)^2},$$

$$F_2 = \frac{\chi^2 + 1.3\chi + 0.2}{\chi^2 - 1.3\chi + 1.0}, \quad \chi = \frac{\nu_t}{7\nu_l}, \quad N_1 = \sqrt{\nabla \nu_t \cdot \nabla \nu_t},$$

$N_2 = \sqrt{\nabla N_1 \cdot \nabla N_1}$, *d* is the distance to the nearest wall, A_1=-0.5, A_2=4.0, C_o=0.8, C_1=1.6, C_2=0.1, C_3=4.0, C_4=0.35, C_6=2.9, C_7=31.5.

Spalart-Allmaras Eddy Viscosity Model

Coolit implements the Spalart-Allmaras model with rotation correction, SA-R, (Dacles-Mariani et al, 1999). which corrects the strain rate of the original model to reduce the eddy viscosity in regions with vorticity exceeding strain rate:

$$\frac{\partial \tilde{\nu}}{\partial t} + u_i \frac{\partial \tilde{\nu}}{\partial x_i} = C_{b1} \tilde{S} \tilde{\nu} - C_{w1} f_w \left(\frac{\tilde{\nu}}{d} \right)^2 + \frac{1}{\sigma} \left[\frac{\partial}{\partial x_i} \left((\nu + \tilde{\nu}) \frac{\partial \tilde{\nu}}{\partial x_i} \right) + C_{b2} \frac{\partial \tilde{\nu}}{\partial x_j} \frac{\partial \tilde{\nu}}{\partial x_j} \right]$$

where $\mu_t = \rho \tilde{\nu} f_{v1}$, $f_{v1} = \frac{\chi^3}{\chi^3 + C_{v1}^3}$, $\chi = \frac{\tilde{\nu}}{\nu}$, ρ is the density, ν is the molecular kinematic viscosity, and μ is the molecular dynamic viscosity, $\hat{S} = \Omega + 2\min(0, S - \Omega) + \frac{\hat{\nu}}{\kappa^2 d^2} f_{v2}$ where $\Omega = \sqrt{2 W_{ij} W_{ij}}$ is the magnitude of the vorticity, *d* is the distance to the nearest wall, and

$$f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}}, \quad f_w = g \left[\frac{1 + C_{w3}^6}{g^6 + C_{w3}^6} \right]^{\frac{1}{6}}, \quad g = r + C_{w2} \left(r^6 - r \right)$$

$$r = min\left[\frac{\hat{v}}{\hat{S}\kappa^2 d^2}, 10\right],\ W_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i}\right),\ S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$$

$$C_{b1} = 0.1355,\ \sigma = \frac{2}{3},\ C_{b2} = 0.622,\ \kappa = 0.41$$

$$C_{w2} = 0.3,\ C_{w3} = 2,\ C_{v1} = 7.1,\ C_{w1} = \frac{C_{b1}}{\kappa^2} + \frac{1 + C_{b2}}{\sigma}$$

19.4.4 Wall Functions

When a turbulence model with wall functions is selected, (Algebraic and Differential Subgrid models always use wall functions) the near wall tangential velocity is assumed to satisfy the Law of the Wall, i.e.:

$$\frac{u_\tau}{u^*} = \frac{1}{\kappa}\ln\left(\frac{E}{1 + 0.3\frac{\rho_0 h u^*}{\mu}}\frac{\rho_0 y u^*}{\mu}\right) \qquad (14)$$

where h is the height of surface roughness, u_τ is the tangent to the wall velocity component in the nearest to the wall grid cell y away from the Wall, such that, $y^+ = \frac{yu^*}{\nu}$, κ=0.42, E=9.8, and $u^* = \sqrt{\frac{\tau_w}{\rho_o}}$, τ_w is the wall shear stress. Equation (14) is used as a boundary condition for the momentum transport equation (2).

Heat transfer equation (3) uses the heat wall functions: $q_w = \alpha_w(T_p - T_w)$. The α_w coefficient is to be determined using the Reynolds analogy for the heat and momentum transport: $\alpha_w = \left|\frac{\tau_w}{u}\right| c_p$. The kinetic energy transport equation (12) is solved with $\frac{\partial K}{\partial n} = 0$ boundary condition.

19.5 Radiation Models and Solar Radiation

Radiation can be an important heat transfer mechanism in electronics. In Coolit, radiation accounts for heat transfer exchange between objects inside the Enclosure. If Solar Radiation is enabled, Coolit will also compute the energy deposition from the sun, based on the specified sun position and heat flux, and the objects surface absorptivity. Solar Radiation and Radiation can be turned on/off independently.

Coolit offers two radiation models described below.

19.5.1 Fast Radiation Model

Electronics packages generally consist of a set of tightly packed components such as circuit boards, flow baffles, Fans, power supplies, disk drives, etc. Owing to the tight spacing, the "view" of components is usually restricted to those directly opposite them. The Fast radiation model takes advantage of this fact by accounting only for heat transfer between parallel surfaces. Furthermore, each cell on the surface of a given component only communicates with its direct line of sight neighbor. The effect of this is that a hot spot will be "felt" by components directly opposite the hot spot. Those components, which are off center from the hot spot, will not be affected except through convection and conduction.

The heat flux between two opposing surfaces is given by:

$$q=\sigma \frac{T_2^4-T_1^4}{\frac{1}{\varepsilon_2}+\frac{1}{\varepsilon_1}-1}$$

where: q is the heat flux, σ is the Stefan-Boltzman constant, $\sigma = 5.67 \cdot 10^{8} \frac{W}{m^2K^4}$, T_1 and T_2 are the surface temperatures; and ε_1, ε_2 are the surface emissivities. This model is restricted to gray body radiation (no temperature, wavelength or angular dependence of emissivity) with a non-absorbing, non-scattering media (a good assumption for air). Also, in this model, the radiating surfaces cannot be perfect reflectors, i.e. ε_1 and ε_2 must be greater than zero.

19.5.2 Advanced Radiation Model

This model provides a very accurate representation of radiation. Specifically it computes complete radiation exchange between all the components and enclosure surfaces. Each component and enclosure surface is covered by "patches" between which Coolit computes view factors as follows:

$$d^2(F_{jk})_{12} = \cos(\beta_1)\cos(\beta_2)*dS_1*dS_2/(r^2\pi).$$

The radiation exchange is then computed from:

$$\sum_{j=1}^{N}\left(\delta_{jk}-\left(1-\varepsilon_k\right)F_{jk}\right)q_j/\varepsilon_j=\sigma\sum_{j=1}^{N}\left(\delta_{jk}-F_{jk}\right)T_j^4$$

The patch size is specified in the Heat Transfer page of the Global Settings dialog. The smaller the patch, the more accurate (but longer) the radiation heat transfer simulation. Note, that the main computation overhead is up front, before the iterative calculation is started. If you are willing to wait for that initial computation, the rest of iterations will go almost as fast as they would with a smaller number of patches.

To reduce the start up time, you can exclude some components from radiation exchange by unchecking the Enable Emissivity box in their Properties dialog.

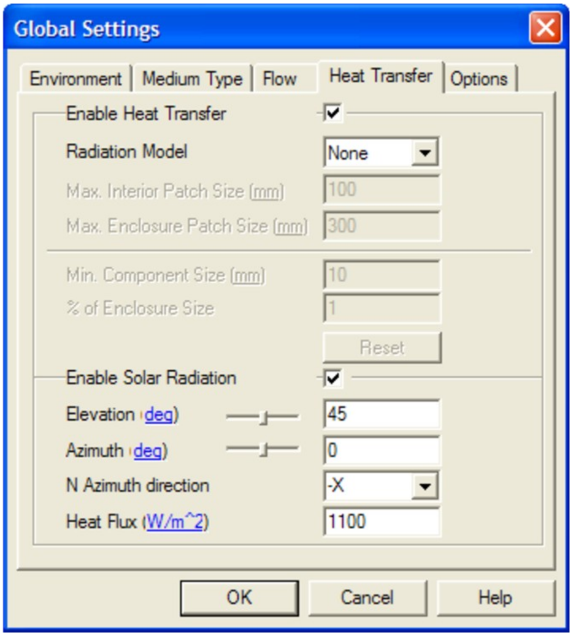
The Advanced model is restricted to gray body radiation (no temperature, wavelength or angular dependence of emissivity) with a non-absorbing, non-scattering media (a good assumption for air). The surfaces cannot be perfect reflectors, i.e. their emissivities must be greater than zero.

19.5.3 Comparison of the Models

The Fast model provides a conservative estimate of the temperature. That is the hot component's temperature will be predicted not lower than it actually is. Depending on the problem, this difference can vary from negligible (e.g. forced convection and/or small temperature differences) to significant (e.g. free convection with highly disparate temperatures).

19.5.4 Solar Radiation

The computation of Solar Radiation effects is enabled from the Global Settings|Heat Transfer dialog.



In this dialog you need to specify the sun's position (Elevation, Azimuth, and North Azimuth direction) and Heat Flux on a plane perpendicular to the direction of the radiation. As the sun position is changed in the dialog, it can be visually checked in the 3D view window. Azimuth angle is measured clockwise from the North and Elevation is the sun angle above the horizon.

Using the data specified in this dialog, Coolit will compute the power deposition for individual objects based on their relative position, shadowing, and surface properties (Absorptivity).

20 Solution Algorithms

Coolit uses the projection method (e.g. Dukowitz and Dvinsky, 1992) to solve the discretized governing equations.

20.1 Finite Volume Discretization

Finite volume methods used for discretization of partial differential equations are described in most textbooks dealing with CFD modeling and therefore will be only touched upon in this section. To make the continuum governing equations (1)-(14) amenable to solution by digital computers, one has to perform the following three steps:

- Transform continuum equations to discrete (difference) ones
- Linearize the resulting system of difference equations
- Iteratively solve the problem

The finite volume discretization is performed by integrating the governing equations over each control volume (grid cell). Assuming that u is a continuous function of (x, y, z), it is now replaced by a discrete function u_i as follows

$$u_i = \frac{1}{V_i} \int_{V_i} u dV$$

where V_i is the i-th grid cell volume. The resulting system of difference equations, **A**u=**b**, has to be now linearized to enable its iterative solution and solved using some suitable algorithm.

20.2 Projection Method

The projection method consists using the following fractional steps to iterate the governing equations:

$$\rho^n c_p \left[\frac{T^{n+1} - T^n}{\Delta t} + V^n \cdot \nabla T^{n+1} \right] = -\nabla \cdot \lambda \nabla T^{n+1} + Q \tag{16}$$

$$\rho^n \left[\frac{V^* - V^n}{\Delta t} + V^n \cdot \nabla V^* \right] = -\nabla p^n + \nabla \cdot 2\mu D^* + \rho^n g \tag{17}$$

$$\nabla \cdot \Delta t \nabla \delta p - \nabla \cdot \rho^n V^* = 0 \tag{18}$$

$$V^{n+1} = V^* + \frac{\Delta t}{\rho^n} \nabla \delta p \tag{19}$$

The superscript denotes the time level, $t = n\Delta t$, and the asterisk denotes an intermediate value, $\delta p = p^{n+1} - p^n$. The system (16) - (18) is solved using approximate factorization, conjugate-gradient, and multigrid methods.

20.3 Convergence and Convergence Control

20.3.1 Definitions

Most of the time, we are interested in obtaining a steady state solution. In such a case, one is concerned not with time-accuracy during the integration process of (16)-(19), but rather with converging to a steady state solution as fast as possible. In practice, when the solution residual becomes smaller than some pre-assigned convergence tolerance value, the solution is converged.

In Coolit, you can monitor the convergence process using the graphic solution monitor, which is displayed every time you run the solver. The solution is considered converged when all the residuals cross the horizontal axis.

In most cases, the convergence history will appear as a smooth hyperbola going down very quickly and then gradually changing to a slower rate of convergence. In some cases, the residuals will oscillate as they go down. In other cases, the residuals will oscillate and not converge, i.e. their mean will either stay constant or increase (diverge).

Such oscillating behavior occurs most often because a steady state solution does not exist for the problem or because of flow instability. In the next section, you will learn what can be done to eliminate instability and generally how to control the rate of convergence.

20.3.2 Convergence Control

Coolit provides a number of tools to control the rate of convergence. As a starting point, you should use all the default parameters. If the solution residuals do not converge, first check your grid to make sure that it is adequate to resolve the expected flow field. Especially troublesome are very narrow grid cells next to wide cells. If you notice such cells, you should remove them either by eliminating the narrow cells or by adding grids to the wide cells.

Next, you should review the following controls in the **Solver Settings|Expert Parameters** dialog:

Parameter	Default	Comments
Local Time Stepping	Checked (on)	Improves convergence on non-uniform grids. If residuals first go down and then oscillate around a nearly horizontal asymptote, uncheck. The problem most likely has time-dependent behavior (e.g. vortex shredding).
Automatic Courant Number (C)	Checked (on)	Lower C values will increase the stability and reduce oscillations.
Fan Underrelaxation	0.2	Reduce this value if you notice oscillation in the pressure and velocity residuals.

21 Bibliography

Smagorinsky J. et al., Monthly Weather Rev., 93(12), 727-733, 1965.

Deardorff J.W., J. Comput. Phys., 7, 120-133, 1971.

Gulyaev A.N., Kozlov V.Ye., and Secundov A.N., ECOLEN ,Scientific Research Center, Preprint #3, 1993.

Spalart P.R. and Allmaras S.R. AIAA-92-0439.

Dacles-Mariani, J., Kwak, D., and Zilliac, G. G., Int. J. for Numerical Methods in Fluids, Vol. 30, 1999, pp. 65-82.

Dukowitz J.K. and Dvinsky A.S., J. Comput. Phys. 102, 336, 1992.

Travis J.R., Nat. Heat Transfer Conf., Denver, Colorado, Aug. 5-9, 1985.

Index