# CoolitPCB™

# User's Manual

September 2021

Daat Research Corp.
Hanover, New Hampshire
USA

# Table of Contents

# 1 Welcome to CoolitPCB

## 1.1 How to Use this Manual

The manual is built around three tutorials and several reference chapters. We recommend that all new CoolitPCB users complete the tutorials 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 pull-down 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

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

First, verify that you are using the latest version of CoolitPCB 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 CoolitPCB. We assume that you are familiar with the information, calculations, and reports that serve as input and output. 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 CoolitPCB.

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:

- **World Wide Web:** https://www.CoolitPCB.com/

- **Email:** support@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.

# 2 Tutorial 1: Card Slot

## 2.1 Starting a New Project

In this tutorial, we will build and solve a model of a card slot. An assembled model is provided for your reference in the *tutorial_1.hcb* case file located in the *Public Documents* folder *CoolitPCB Tutorials\Tutorial_1.* A movie showing the construction of this model is also available as explained in Download instructions you should have received.

Open CoolitPCB to create a new project. Go to the **File|Save As** menu, and save the project as *mytutorial_1.hcb*. From **File|Properties** menu, open the Project Properties dialog to write a short description of your project.

CoolitPCB is set up to automatically save your project every minute in the *mytutorial_1.hcb* file. The automatic save interval can be changed or the option can be turned off in the **Tools|Options…** menu. CoolitPCB will prompt you to save the project file upon exiting if any unsaved changes exist. Once you make a first modification in an open case, CoolitPCB will save the original case as *mytutorial_1.Backup.hcb*.

## 2.2 Setting Defaults and Options

### 2.2.1 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.hcb, is created right after you load a file into CoolitPCB. 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, CoolitPCB saves a copy of the case file with the extension Solver.HCB. 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 fast redrawing of the 3D view (**Large Model Rendering**). The latter option somewhat degrades the quality of display, but the drawing is faster and it is advised for larger cases.

### 2.2.2 General Options: Undo and Redo

The Undo/Redo functions are used to recover previous operations in CoolitPCB. 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.

### 2.2.3 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.

### 2.2.4 “Snap To” Options

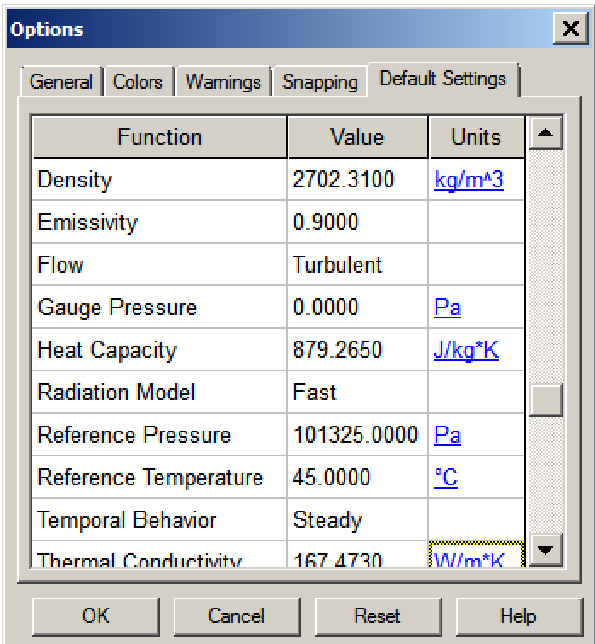
The “Snap To” commands make component alignment easy by forcing objects to line up with the Snap to Grid, or with objects nearby.

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.

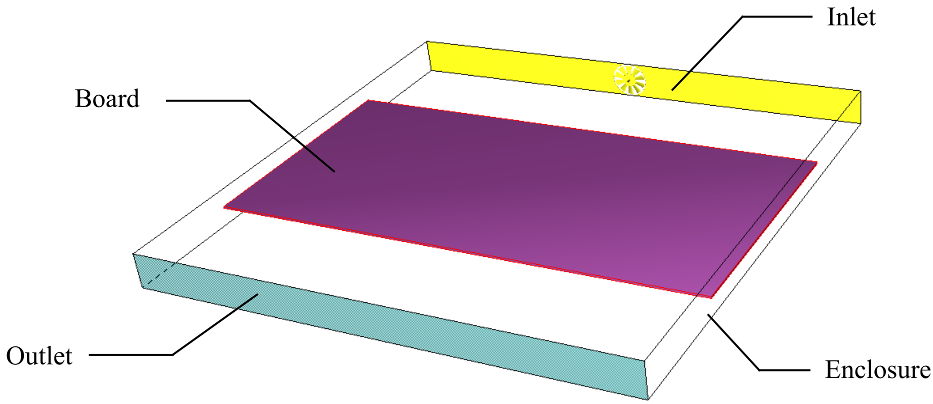
### 2.2.5 Default Settings Options

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.

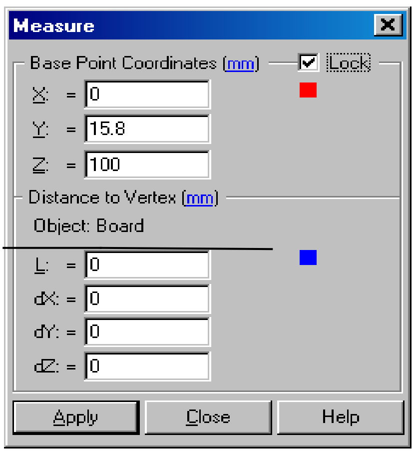


## 2.3 Adding Components to the Board

For this first tutorial, we will use the default "Card Slot" setup that will appear when you open CoolitPCB:



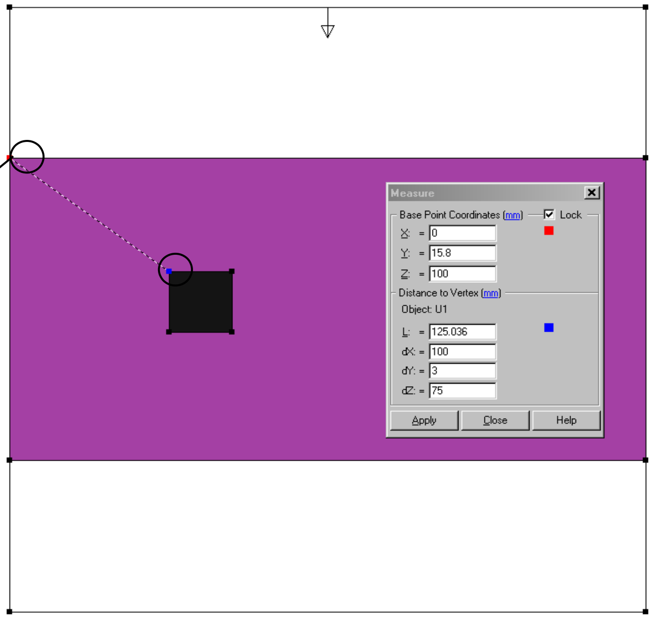

First we will set up the Measure tool to facilitate the placement of components on the board. Click  to open the Measure dialog; a small red dot called the Base Point will appear at the origin, (X,Y, Z) = (0,0,0) of the Enclosure. Un-check the **Lock** box in the dialog, move the mouse to the upper left vertex of the board and when the mouse pointer changes to a ruler icon click on the vertex twice. The red dot will move to the vertex. Verify that the Base Points Coordinates are (X,Y,Z) = (0, 15.8, 100) mm and check the **Lock** box to fix it as a Base Point. Now, we can reference all component positions from the Base Point instead of from the global coordinates origin.



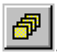
Keep the Measure dialog open and select (click) the board.

Click  on the PCB Toolbar or use the **Build|PCB|Add Resistor Model** menu command. A resistor component U1 will appear at the board's origin. Click somewhere in empty space, so that no objects are selected. Click on the component's upper left corner. Notice that a blue point will appear there. Enter dX = 100 mm, and dZ = 75 mm, in the Measure dialog. Click Apply and close the Measure dialog.

**Base Point**

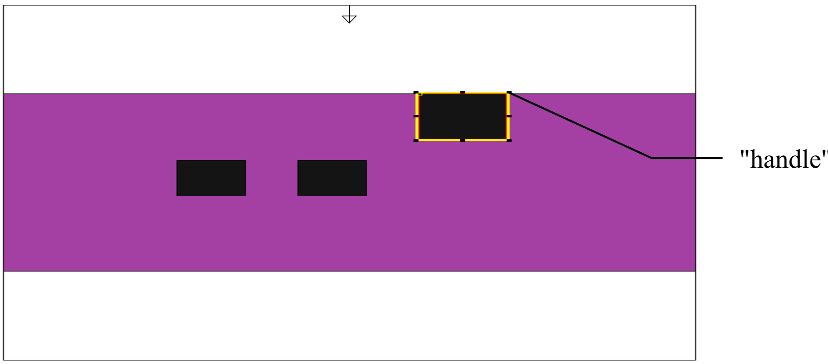



Next, double-click on the component, or click the **Properties** button on the Control Bar to open the Resistor Model properties dialog. Set the Component Type as 2-Resistor. Set the Power to 1 W, and the $R_{junction\text{-}board}$, and $R_{junction\text{-}case}$ to 10 °C/W. Leave the Emissivity as the default and click **OK** to close the dialog.

Next, create two copies of the U1 object, using the Clone command. Click on the U1 object and then click the Clone icon, , on the Edit toolbar. Set the Number of copies to 2 and the Pitch to (X, Y, Z) = (70, 0, 0) mm and close the dialog. In the Objects Manager double click the newly created U1-2 component and in the Resistor Model dialog change the Power to 3 W. Click **OK** to close this dialog.

Rename this component by clicking in the Name text box on the Control Bar and typing in U3. Press Enter on the keyboard or click **Apply** to effect the change. We will leave the **Nameplate** field blank in all tutorials. This field is used a component label or a nameplate that is displayed in 3D view.

Drag U3 to the back edge of the board until it snaps into position at the board's edge. Verify that the new position is roughly (X, Y, Z) = (240, 15.8, 100) mm in the Control Bar. Next grab onto the bottom right corner handle of U3 and drag it until the size changes to 50x50 mm. Verify this by watching the Size box in the Control Bar.




Finally, add a heat sink to U3 by clicking the  button and selecting the "Plate Fin" option. The default aluminum plate fin heat sink will be automatically sized and positioned on top of U3.

## 2.4 Solving the Problem and Viewing Results

Click or select **Solver|Start** to begin solving your problem. 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.

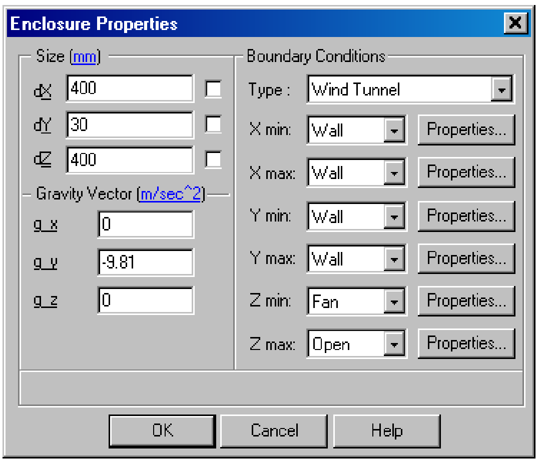
# 3 Tutorial 2: Setting up the Environment

In this tutorial we will test the board built in Tutorial 1 in wind tunnel and free convection environments. An assembled model for this tutorial is provided in the *tutorial_2.hcb* case file located in the *Public Documents* folder *CoolitPCB Tutorials\ Tutorial_2*.

## 3.1 Wind Tunnel Environment

Start CoolitPCB, open *mytutorial_1.hcb* and then save it using **File|Save As** *mytutorial_2.hcb*. Next we will change the Card Slot Enclosure used in Tutorial 1 to a 400x50x400 mm Wind Tunnel with a constant 100 CFM flow rate, 50°C ambient temperature, and zero gauge pressure.

To make these changes, open the Enclosure Properties dialog by double clicking on the Enclosure or using the **Build|Enclosure...** menu.




| Size Enclosure | |
|---|---|
| dX | 400 |
| dY | 50 |
| dZ | 400 |

In the Type pull down box of the Boundary Conditions section of the dialog, select Wind Tunnel. Set up the wind tunnel inlet (Z min) and outlet (Z max) according to the table below.

| Boundary Conditions | | |
|---|---|---|
| **Side** | **Type** | **Properties** |
| X min | Wall | Specified Heat Flux, 0 W/m$^2$K |
| X max | Wall | Specified Heat Flux, 0 W/m$^2$K |
| Y min | Wall | Specified Heat Flux, 0 W/m$^2$K |
| Y max | Wall | Specified Heat Flux, 0 W/m$^2$K |
| Z min | Fan | Constant Flow Rate, 100 ft$^3$/min, Ambient 50C |
| Z max | Open | Ambient 50C |

To set the Z min side, click the corresponding Properties button to open the Enclosure Properties dialog. Click the radio button next to Flow Rate to set the Enclosure Fan to a constant flow rate type and enter 100 CFM. Also, reset the Ambient Temperature to 50C.

Similarly, reset Ambient Temperature to 50C for the Z max Open side. Click GO to begin solving the problem. A progress bar will appear indicating the status of your solution. CoolPlot will automatically open after the solution has converged and will allow you to view your results.

## 3.2 Free Convection Environment

Open *mytutorial_2.hcb* and save it as *mytutorial_2free.hcb*. Open Enclosure Properties dialog and change the Enclosure type to Free Convection. All Enclosure sides change to Open. Click the Properties button next to each Open to specify Ambient Temperature at 50°C.

We now want the gravity vector parallel to the board. Instead of changing the Enclosure dimensions and rotating the board, we will change the gravity vector's direction as follows: ($g_x$, $g_y$, $g_z$) = (9.81, 0, 0) m/sec$^2$.

You can verify this change in the 3D view window, as the gravity vector should now be pointing in the same direction as the global X direction.

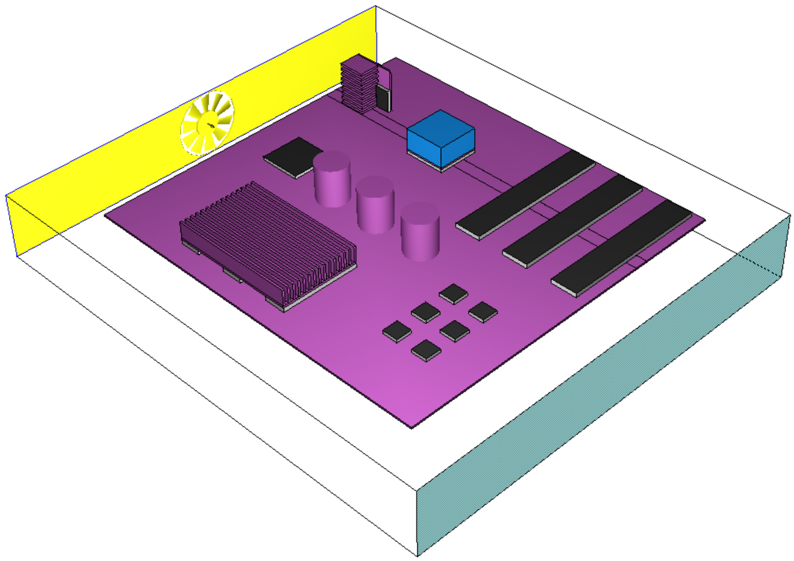
Notice that Heat Sink's fins are now perpendicular to the gravity and hence to the flow, which is inefficient. To rotate Heat Sink, first, click on it to select it and then, in the Control Bar, under Rotate, click the **Y: 90°** button once. Notice that the Heat Sink's orientation will then be correct, but it will have moved off the component. This is because the components' positions are referenced from the corner indicated by the small black box. The rotation axis also passes through this reference point.

Use the mouse to drag the Heat Sink back to the correct position. Click  to begin solving the problem. A progress bar will appear indicating the status of your solution. CoolPlot will automatically open after the solution has converged and will allow you to view your results.

# 4 Tutorial 3: Advanced Functions

## 4.1 Starting a New Project

In this tutorial, we will build and solve the model shown below. An assembled model for this tutorial is provided in the *tutorial_3.hcb* case file located in the *Public Documents* folder *CoolitPCB Tutorials\Tutorial_3*. Open CoolitPCB to create a new project. Go to the **File|Save As** menu and save the project as *mytutorial_3.hcb*.



## 4.2 Setting up the Environment

Use the default, Card Slot type Enclosure, and set the Enclosure size and boundaries as follows:

| Size Enclosure | |
|---|---|
| dX | 400 |
| dY | 60 |
| dZ | 450 |

| Boundary Conditions | | |
|---|---|---|
| **Side** | **Type** | **Properties** |
| X min | Wall | Specified Heat Flux, 0 W/m$^2$K |
| X max | Wall | Specified Heat Flux, 0 W/m$^2$K |
| Y min | Cyclic | none |
| Y max | Cyclic | none |
| Z min | Fan | Constant Flow Rate, 150 ft$^3$/min, Ambient 50C |
| Z max | Open | Ambient 50C |

Set the gravity vector parallel to the card: $(g_x, g_y, g_z) = (9.81, 0, 0)$ m/sec$^2$

## 4.3 Building the Board

First, adjust the board size and position by selecting it and by entering the following into the Control Bar, and clicking **Apply**:

| Board Position and Size | |
|---|---|
| dx | 350 mm |
| dy | 330 mm |
| dz | 1.6 mm |
| X | 70 mm |
| Y | 15 mm |
| Z | 25 mm |

The board's position in the Control Bar is reported with respect to the board's reference point marked with a small solid black square. Next open the Printed Circuit Board Properties dialog by either clicking the Properties button while the board is selected, or by double clicking directly on the board and specify the following conditions:

Layers

| Layer Name | Material | Thickness (mm) | % Coverage |
|---|---|---|---|
| Signal | Copper (Pure) | 0.0340 | 20.0000 |
| Power or Ground | Copper (Pure) | 0.0170 | 90.0000 |
| Signal:1 | Copper (Pure) | 0.0340 | 20.0000 |
| Signal 2 | Copper (Aluminized) | 0.0300 | 30.0000 |
| Signal 3 | Copper (Pure) | 0.0400 | 10.0000 |
| Signal 4 | Copper (Pure) | 0.0400 | 40.0000 |

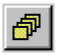
To add a layer, click the **Add layer** button. CoolitPCB will generate a new line in the table with some default values. To change any of the values, click it, and either enter a new value, or select a new option from the pull down menu. To delete a layer, click on any value in its row, and click the **Remove** button. Clicking **Remove All** will remove all the layers.

Notice that as you add/change layers, the grayed-out thermal conductivity values in the In-plane and Through-plane boxes above the table will change to reflect the board's composition.

## 4.4 Adding Components

Components can be added to a selected board by clicking  on the PCB Toolbar or from the **Build|PCB|Add Resistor Model** menu command. Add and specify the board components as shown in the table below. Leave the Emissivity at the 0.9 default.

| Name | Size(mm) (dx,dy,dz) | Position (mm) (X,Y,Z) | Component Type | Material | Power (W) | R-jb (C/W) | R-jc (C/W) |
|---|---|---|---|---|---|---|---|
| U1 | 40, 40, 3 | 300, 16.6, 60 | 2-Resistor | N/A | 5 | 7 | 7 |
| U2 | 40, 40, 3 | 300, 16.6, 140 | 0-Resistor | Ceramic | 2 | N/A | N/A |
| U3 | 40, 40, 3 | 210, 16.6, 60 | 1-Resistor | N/A | 2 | 3 | N/A |
| Pow amp | 20, 75, 4 | 90, 16.6, 200 | 2-Resistor | N/A | 10 | 5 | 5 |
| Op amp | 15, 15, 3 | 125, 16.6, 290 | 0-Resistor | Plastic | 0.25 | N/A | N/A |
| M. Plex | 25,140, 4 | 255, 16.6, 340 | 0-Resistor | Plastic | 0.25 | N/A | N/A |

Now select the **M. Plex** component, and clone it by clicking  on the Edit toolbar, setting the Number of Copies to 2 and the Pitch to (X, Y, Z) = (0, 0, -50) mm. Next, use Ctrl-C to copy the **Pow amp** component and then use Ctrl-V to paste it at (X, Y, Z) = (90, 16.6, 150) mm and at (X, Y, Z) = (90, 16.6, 100) mm.

A third technique for creating copies of objects is to use the Object Array function. Select the **Op amp** component and click  in the Edit toolbar to open the Objects Array dialog. Set the Number of Objects to 2 in the x-direction, 3 in the y-direction, and 1 in the z-direction, where x, y, and z are the object's local coordinates. Set the pitch to 30 mm in the x- and y-directions and 0 mm in the z-direction. Set the position of the

Op Amp Object Array to (X, Y, Z) = (128.5, 16.6, 290) mm
Click **OK** to close the dialog.

Now we will modify U1 and U2 components from the table above. Click on the U1 component and then click the Change Sides, , button in the PCB toolbar to move U1 to the other side of the board.

Add a thermal via to the U2 component, by first selecting U2 and then clicking the Attach Thermal Via, , button. This will automatically insert a via under U2. To edit the properties of this via, either click the **Properties** button, or select the Thermal Via object in Objects Manager and double click directly on its name. Specify the Diameter as 0.4 mm and the Plating thickness as 0.03 mm. Click **OK** to close the dialog.

We will now put a heat sink on U2, but before first, we will add a thermal pad between U2 and the heat sink. Select U2 and then click the Contact Resistance button, . The Thermal Resistance component will be created directly on the U2 component. Click the Properties button and specify an Interface Material with a thermal conductivity of 1W/mK.

Again, select U2 and then click the  button. Select Pin Fin from the popup menu. A heat sink will be placed directly on top of U2 and the thermal pad above it. Open the Pin Fin Heat Sink properties dialog and on the Base page specify the Thickness as 1.75mm; on Pins page, specify the number of pins in both directions as 8, the Pin Height as 20 mm, and the Pin Length and Width as 3 mm. Click Apply to view the settings. Assuming that component U2 is not of critical interest, we will use a Compact Model of the heat sink to save computational time. On the Base page of the dialog, check the box next to Use Compact Model and click **OK** to close the dialog.

Next, add a heat sink to the **Pow amp** components. Select one of them and then click  and select Plate Fin from the pull down menu. A heat sink will appear on top of the selected component. Open the Plate Fin Heat Sink properties dialog and on the Base page, set the length to 120mm, on the Plate Fins page, set the Fin Width to 2 mm, the Fin Height to 15mm, and the Number of Fins to 15. Click **OK** to close the dialog. Position the heat sink at (X, Y, Z) = (90, 20.6, 220).

Next, add three Cylinder components. Click the button to create a cylinder. Size it to (dx, dy, dz) = (30, 30, 35.25) mm and place it at (X, Y, Z) = (200, 16.6, 125) mm. Open the Cylinder properties dialog and specify Power of 2 W. Use the Clone function to make two copies of Cylinder at (X, Y, Z) = (200, 16.6, 175) and (200, 16.6, 225) mm.

Now we will create a power transistor with a heat sink. Click , name it Power Transistor, specify Power as 5W, size it to (dx, dy, dz) = (14.5, 21, 3) mm, and place it at (X, Y, Z) = (335, 16.6, 70) mm. Rotate Power Transistor 90° around Z-axis, so that it is perpendicular to the board. Next, click and create a Plate Fin heat sink. Specify the base Width and Length as 40 mm, the Number of Fins as 9, the Fin Height as 20 mm, and the Fin Width as 2 mm. On the Margins page, specify a Front margin of 14.5mm. Click **OK** to close the dialog, and move the sink such that Power Transistor is on the heat sink's margin (X, Y, Z) = (336.5, 16.6, 84.5) mm.

## 4.5 Applying and Importing Textures

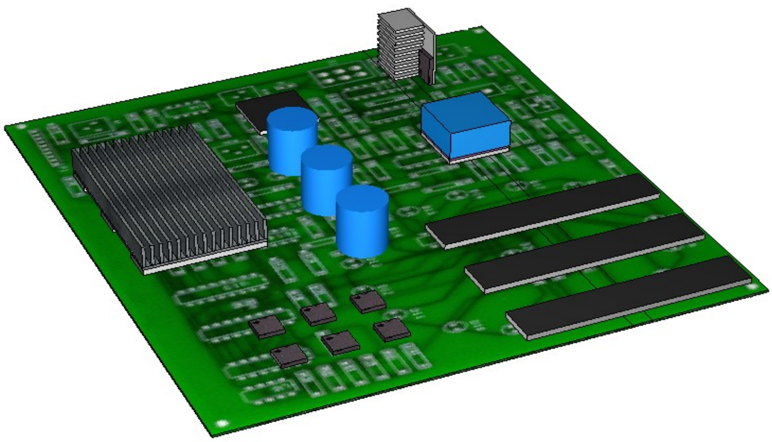
Press the icon to enter the User Specified Colors/Textures mode. All components turned to the default gray color.

Click the icon to open the Select Texture dialog. Select **Standard|Electronic Components**. The texture can be previewed by clicking the icon or by right-mouse-clicking the texture and selecting Preview. The name of the currently selected texture is shown in the pull-down menu in the painting toolbar. Click the button to enter painting mode. Notice that a paintbrush icon has appeared next to the pointer. This indicates that the selected texture is ready to be applied. 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. Press the icon to enter the texture rotation mode. Now click the textured surfaces that need to be rotated. Press again to exit the texture rotation mode.

From the pull-down box in the Painting tools toolbar you can select a color or texture from a list of recently used colors and textures. 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.

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



You can make your own textures from any bitmap (*.bmp) image including digital photos. For this tutorial, a sample bitmap *CoolitPCB Tutorials\Tutorial_2\sample.bmp* has been provided. 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 3**. 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.

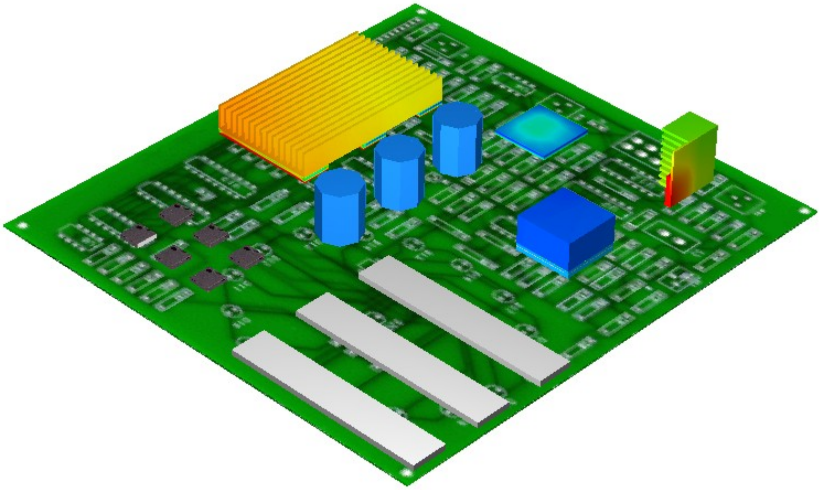
## 4.6 Running the Solver

Click or select **Solver|Start** to begin solving your problem. When you run the Solver, it is possible that Coolit will prompt you to examine your model using **Model Check**. CoolPlot will automatically open after the solution has converged.

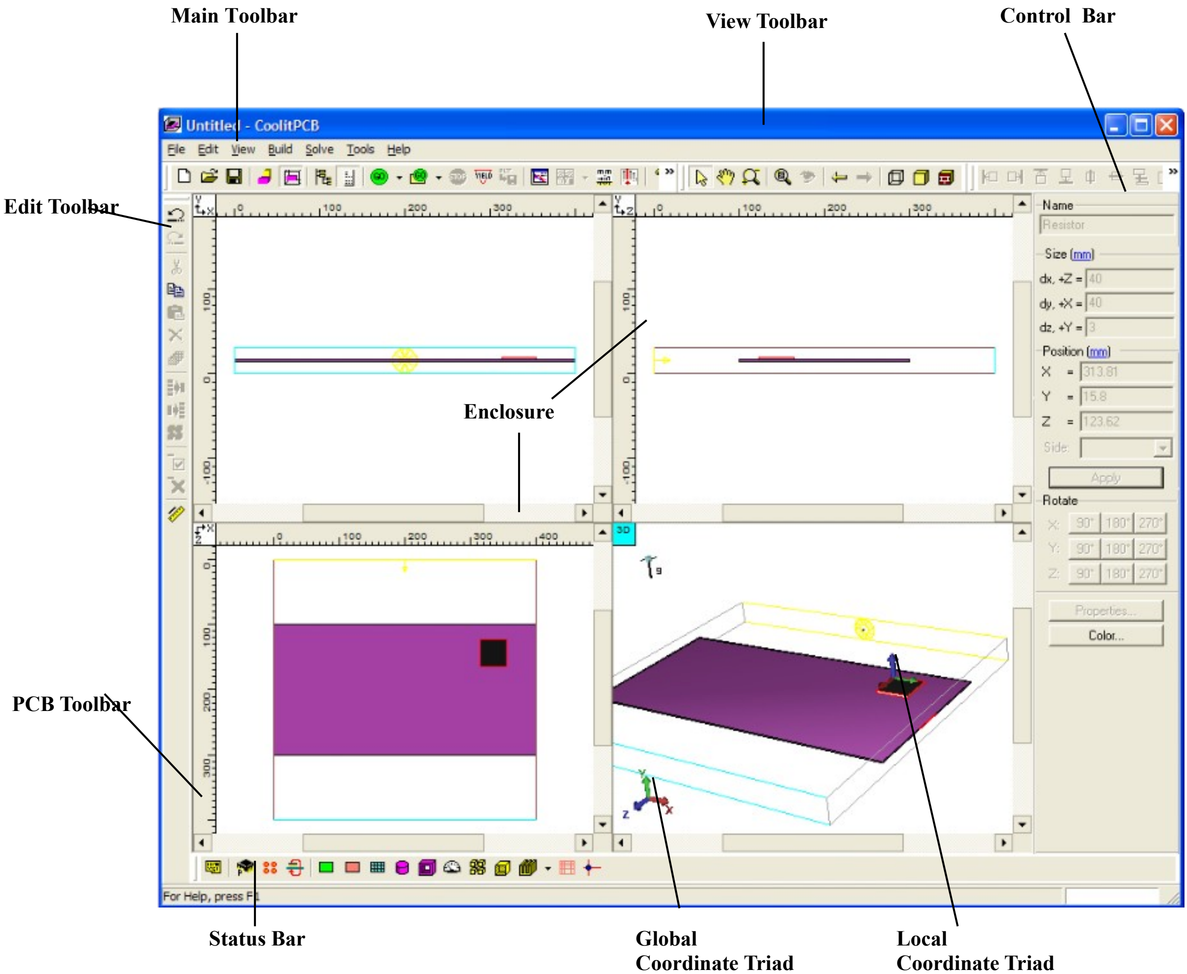
If you wish to review results before the solution has finished converging, click to output a plot file, and then click to start CoolPlot.

## 4.7 Textures in 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. Expand the Interior folder. You can right click on the icons for different objects and select a variety of display types: Wireframe, Component-type Colors, Custom Colors (textures), Variable, and Variable Max. The ability to display individual components using different modes can be useful for preparing visually appealing presentations.

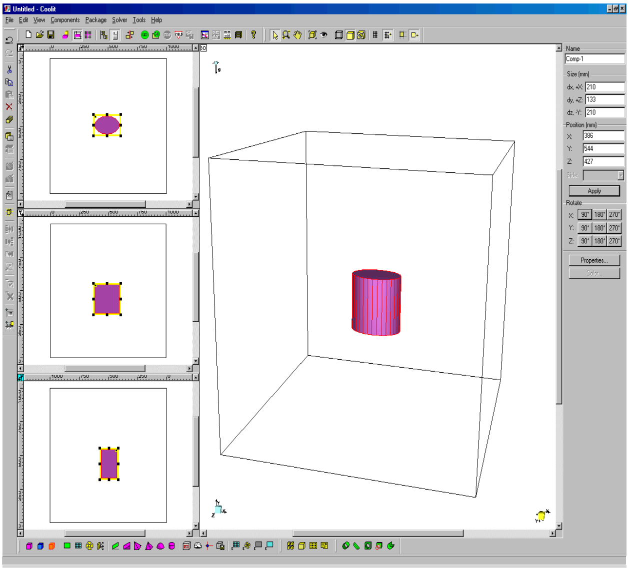


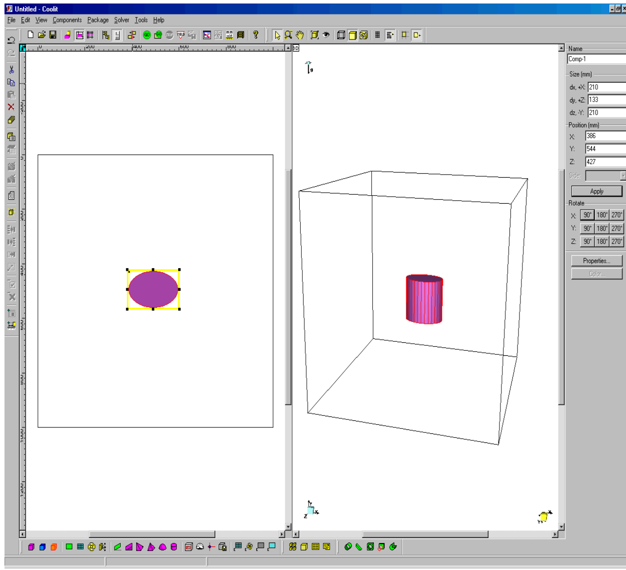
# 5 Exploring the Working Area




The picture shows the default layout of the main CoolitPCB interface. There are three 2D-view panes and one 3D-view pane. A blue left upper corner indicates the active window. 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 use the hotkey "W" 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 convenient to work with two windows, rather than one. Click





**View|Add Second Window**.

In CoolitPCB, the **Enclosure** refers to the volume inside which computations will be done.  It can be a card slot, a wind tunnel, or simply a volume in which you wish to carry out the simulation.

In the 3D-view, there are two **coordinate triads**.  The triad on the left defines the **global coordinate system**, which in this manual is designated with capital (X, Y, Z).  The triad on the right defines the **local coordinate system** of the selected object. The local triad is only shown when an object is selected. Local coordinates in this manual are designated using lower case (x, y, z).

The **Control Bar** is used to define the name, size, location, 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 selected 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 CoolitPCB objects within a project. You can open the Objects Manager by checking **View|Objects Manager,** by pressing , or by typing “b” (for “browser”) on the keyboard.

There are five toolbars available in CoolitPCB. You can select displayed toolbars via the **View|Toolbars…** menu. All the toolbars are dockable. Just click them with the left mouse button and drag to a new location.

CoolitPCB has two modes: Build and Components mode. Either mode can be activated using the Build menu (**Build|Build Mode, Build|Components Mode**).

Build 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 selected object into the Interior using Objects Manager.

## 5.1 Menus

### 5.1.1 File Menu

The file menu contains the same options typical of most Windows applications such as New, Open, Save, Save As, Print, Print Setup, and Exit. In the **Properites** dialog you can enter text information about the current model for future reference. The Import and Export functions are described later in the manual.

### 5.1.2 Edit Menu

The Edit menu contains standard editing functions, such as undo, redo, cut, copy, paste, and delete. The **Clone** function can be used to make multiple copies of the current Object. **Object Properties** opens the properties dialog for the current object.

You can select multiple objects, and assign them all the same properties by using **Selected Properties**. **Object Color** will open the Visual Properties dialog where you can paint and texture components. **Selected Color** assigns the same color to a group of selected objects. **Object Notes** can be used to enter text notes about the current object. **Assemble** will assemble a group of selected objects into a complex object. **Dissassemble** will disassemble a selected complex object.

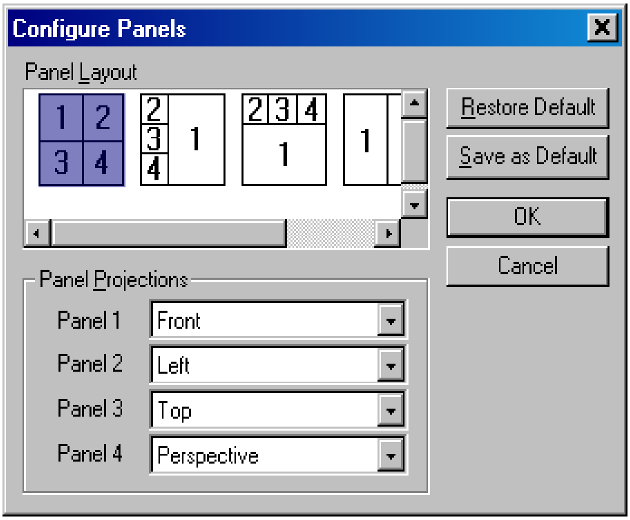
**Measure** opens up the Measure tool dialog.

### 5.1.3 View Menu

From the View menu you can control the appearance of objects and how the CoolitPCB window layout.

**Appearance|Wireframe** will display all objects as outlines. **Appearance|Component-type colors** will display all objects in the default color for each object type. **Appearance|Custom**

**colors/textures** will display all objects in their user-defined colors or textures.




**Maximize view** will maximize the size of the currently selected window. **Reset views** will reset all views to default zoom and rotation. **Configure Panels** opens the Configure Panels dialog where you can arrange the layout of the various views.

**Objects Manager** will open the Objects Manager window. In this window you can set individual display options for different objects, specify and view object properties, assign object priority, assemble and disassemble complex objects, and perform many other useful actions.

Clicking **Convergence Monitor** displays or hides the Convergence Monitor. The Convergence Monitor displays the progress of a solution.

**Toolbars...** opens the Toolbars dialog where you can select which toolbars to display, and which to hide. Click **Configure** to select which buttons to display and hide. All toolbars are "dockable" and can be moved to any place in the window by clicking and dragging.

Clicking **Status Bar** displays or hides the status bar. The status bar displays information about whatever object or button the mouse cursor is currently held over. Clicking **Control Bar** displays or hides the Control Bar. Control Bar is an integral part of the interface and is used to size and position components as well as access their properties dialogs. Clicking **Rulers** or **Scrollbars** displays or hides the rulers or scroll bars on the sides of each display panel.

**Unselect All Objects** will unselect all currently selected objects. **Unhide All Objects** will make all hidden objects visible. See the section on Object Display Options for more information on hiding and un-hiding objects. **Enable All Objects** will enable all disabled objects. Objects can be disabled in the Objects Manager.

### 5.1.4 Build Menu

**Enclosure...** opens up the **Enclosure Properties** dialog where the case environment is specified. The Size of the Enclosure and direction of the Gravity Vector can be entered as numeric values in the boxes provided. The Boundary Conditions for Wind Tunnel, Card Slot, Free Convection, as well as Custom environments can be specified from the Type pull down menu. The properties for each side can be specified by clicking the Properties button next to each side.

The **Medium Type** page specifies the carrier fluid: air, water, or vacuum. If *water* is selected, physical properties are computed automatically from empirical formulas. If *vacuum* is selected there is no flow medium present.

The **Flow** regime defines whether to solve for fluid flow and if the flow is laminar or turbulent. Selecting No Flow tells CoolitPCB 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.

Radiation Model defines radiation heat transport between components. Selecting **None** means that radiation heat transfer will not be computed. CoolitPCB 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 all view factors and complete radiation exchange between all objects. Radiation exchange in the Advanced model is computed between patches on component and Enclosure surfaces. Patch sizes are set automatically by the program. 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.

Under **PCB** there is a sub menu containing all the object types that can be added to your PCB.

From the Build menu you can also switch between Build and Package mode by selecting either the **Build Mode** or **Component Mode** options.

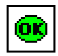
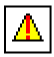
**Expert Options** allow you to solve time-dependent problems (**Transient Flow** option) and conduction/radiation problems without flow (**No Flow** option). Time step size and other parameters required by transient problems are set in the Solver Settings dialog.

### 5.1.5 Solve Menu

The Solve menu contains controls for using the Solver. **Start** begins the Solver and starts the processing of your model. **Output Plotting File** outputs a CoolPlot file of unconverged solution, while the solver is still running. **Stop** stops the solver and outputs a plot file. If the solver is stopped due to convergence or reaching the iterations limit it will create a plot file.

The **Coarse Grid**, **Medium Grid**, and **Fine Grid** options determine the number of grid cells used in your model. While a finer grid may produce a more accurate solution, it will take longer to solve. You can customize the grid further and manually manipulate grid lines from Grid mode, **Tools|Settings|Grid Mode**.

Selecting **Batch Processing...** will open the Batch Queue dialog. 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 CoolitPCB 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.

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

Remove – remove the selected project

Remove All – remove the entire queue

Selecting OptimizeIt will open the OptimizeIt dialog. OptimizeIt is an optional module.

### 5.1.6 Tools Menu

**Start** CoolPlot will open the CoolitPCB post processor. **Start CoolCalc** will open the CoolCalc calculator and unit conversion tool.

**Materials...** opens the Materials Library dialog. In this dialog you can view the materials in the Standard Materials library and add your own custom materials library. In the **Units Setup...** dialog you can set up the units you want to use for each variable edit existing units, and add new units. Units can also be selected in most dialogs by clicking the blue unit hyperlink.

The dialog enables the display and setting of the **Reference Point,** opens **Distance Toolkit**, and **Measure** tools.

**Write Report File...** will produce an *.html document containing a detailed description of your models setup. **Find Object...** will open the Find Object dialog. In this dialog you can search for a specific object by name, type, or power dissipation.

In the **Options...** dialog you can find controls for many of the general settings in CoolitPCB. It configures several parameters for CoolitPCB start up operation, auto-save, back up, undo depth, display colors, warnings, snapping, and provides support for fast redrawing of the 3D view (**Fast 3D Rendering** option). The latter option somewhat degrades the quality of display, but the drawing is much faster and it is recommended for large cases with numerous components and intersections.

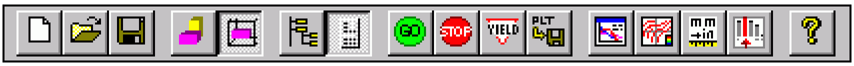
Under **Expert,** you will find a sub menu that controls additional user options.

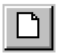
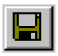
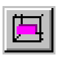
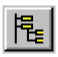
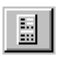
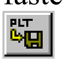
### 5.1.7 Help Menu

**Help Topics** will open the CoolitPCB help menu. Here you can fine context sensitive help about all elements of the user interface. It is fully indexed and searchable. **Registration...** is where you handle the license for your software. In the registration dialog, you create an ID file for your computer by pressing Save ID File. You can then e-mail this file to support@daat.com in order to receive a license for your software and license type you have. The license information is entered in this dialog by pressing the Update button and selecting the file you received from support.

**Tip of the Day** displays useful hints for working with CoolitPCB. These tips are generally shown at startup but can be toggled on and off by using the checkbox at the bottom left of the Tip of the Day window. **About CoolitPCB...** will display all the version and license information about your installed software.

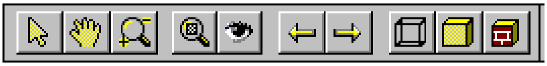
## 5.2 Main Toolbar



-  Creates a new project
-  Opens an existing project
-  Saves current project
-  Switches to Components mode
-  Switches to Build mode
-  Displays or hides the Objects Manager
-  Displays or hides the Control Bar
-  Starts the Solver
-  Stops the Solver
-  Slow down the solver to permit other applications run faster
-  Outputs a CoolPlot file. The function is used to save a snapshot of solution before convergence.

- Starts CoolPlot
- Starts CoolCalc
- Starts OptimizeIt (optional module)
- Starts the About CoolitPCB dialog

## 5.3 View Toolbar



- Switches into select mode
- Enter the Pan (in 2D pane) and Rotate (in 3D) mode. To operate: hold the left mouse button and drag the mouse.
- Enter Zoom mode. To zoom in drag the mouse up and to zoom out drag it down.
- Reset all panel views to default.
- Unhide all objects
- Returns panel view to a previous view.
- Switches panel view forward.
- Display in wire-frame mode.
- Shade displayed Objects in component-type colors.
- Shade objects in user-selected solid colors or textures.

### 5.3.1 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 activate 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 editing functions applicable for the active object.

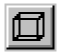
If you have a mouse with a scroll wheel, there are zoom, pan, and rotate shortcuts available. Otherwise, to zoom in, press the Zoom button and drag the mouse up; to zoom out, drag it down. To pan in 2D-views, press the Pan/Rotate button and drag the mouse. To rotate in 3D-view, press and drag the mouse. 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:

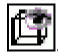
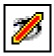
| Command | Function |
|---|---|
| Left arrow | Rotate left |
| Right arrow | Rotate right |
| Up arrow | Rotate up |
| Down arrow | Rotate down |
| PgUp | Rotate clockwise |
| PgDn | Rotate counter-clockwise |

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.

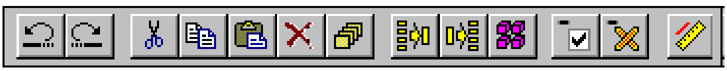
### 5.3.2 Solid and Wireframe Views

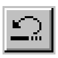
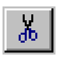
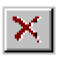
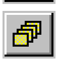
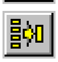
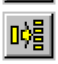
Components appearance can be controlled from the main toolbar or the Objects Manager. To view objects as solid, painted by component-type colors, select **View|Appearance|Component-type colors** from the main menu or press . If you prefer to specify colors of objects individually, click on the main toolbar or select **View|Appearance|Custom colors/textures** 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 main toolbar or select **View|Appearance|Wireframe** to display objects as wireframes. You can also use the Object Manager's **Appearance** menu to the same effect.

You can selectively "hide" components by clicking on the , , or  icons in the Objects Manager; the icon will change to . Alternatively, hold the Ctrl down, and double click the component in any view. To unhide the component you can either click  or select **View|Unhide All**.

## 5.4 Edit Toolbar



-  Undo
-  Redo
-  Cut
-  Copy
-  Paste
-  Delete
-  Clone
-  Assemble
-  Disassemble

- Create Objects Array
- Unselect all objects
- Enable all previously disables objects
- Activate the Measure Tool

### 5.4.1 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.

### 5.4.2 Clone

The Clone function is used to create a one-dimensional array of objects out of a selected object. Select 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. Check the Assemble box if you want the cloned objects assembled into a complex object.

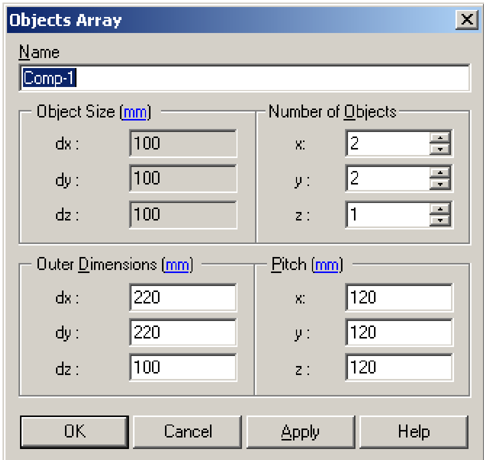
### 5.4.3 Objects Array

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

### 5.4.4 Assembling and Disassembling Complex Objects

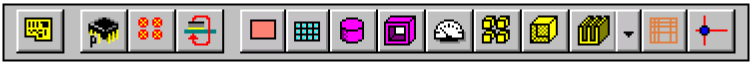
Complex objects are combinations of several components. You can assemble a complex object several ways:




- 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.

To assemble a complex object, select several objects and click the button. To disassemble a complex object, select the desired complex object and click the button. Complex objects can also be assembled and disassembled in the Objects Manager. Object Arrays are treated as Complex Objects.

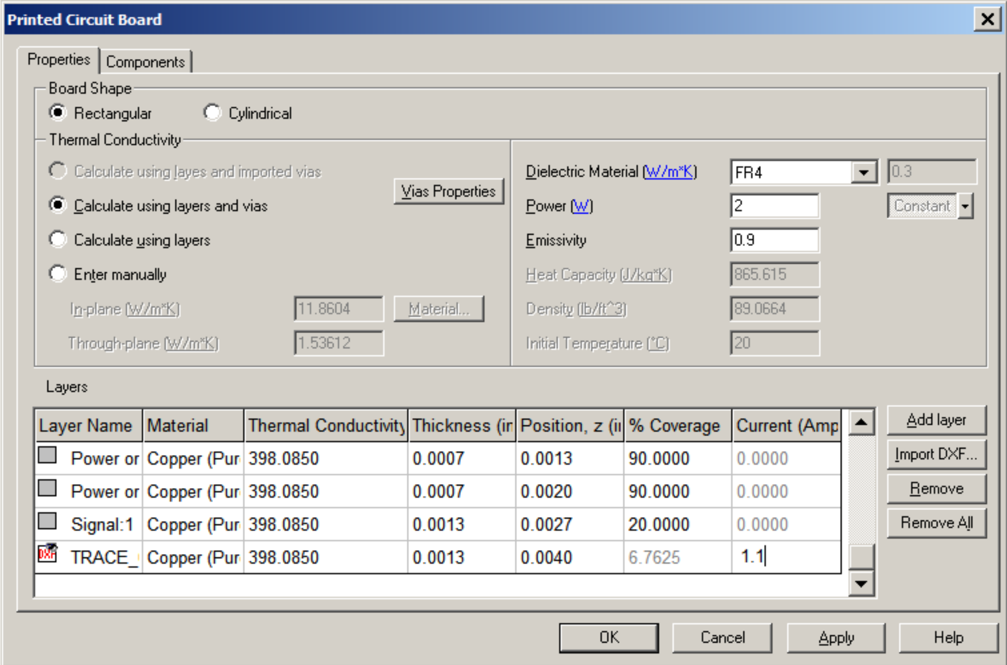
## 5.5 PCB Toolbar



- Create a PCB
- Create an IC Package object
- Create a Thermal Via object
- Switch IC Package placement to opposite side of PCB
- Create a Contact Resistance component
- Create a Planar Porous Resistance component
- Create a Cylinder component
- Create a Cutout component
- Create a Gauge component
- Create a Component Array object
- Create a Hollow Box object
- Create a Heat Sink component
- Create a Grid Zone object

### 5.5.1 PCB Object

Click to create a PCB. The PCB is specified in its Properties dialog by double clicking on the created PCB, or by selecting it, and clicking 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.

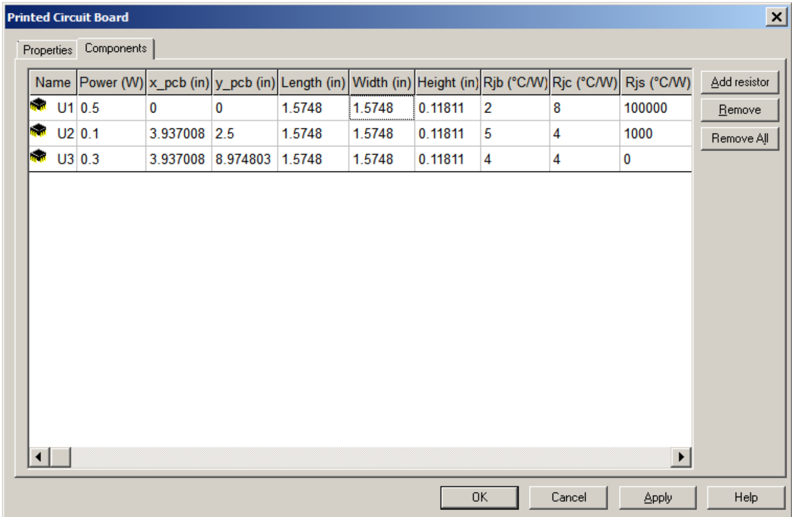
To change a specified value in the table, click on the desired box, and enter the new value. The total **Thickness** of all the layers must be smaller than the board thickness. The **% Coverage** is the area of the board covered by the Material in the layer. This is used to compute the board's conductivity. The total thickness 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 flows only. For steady state flows, 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, $\rho$ 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**, $\rho$, 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 $T_{high}$.

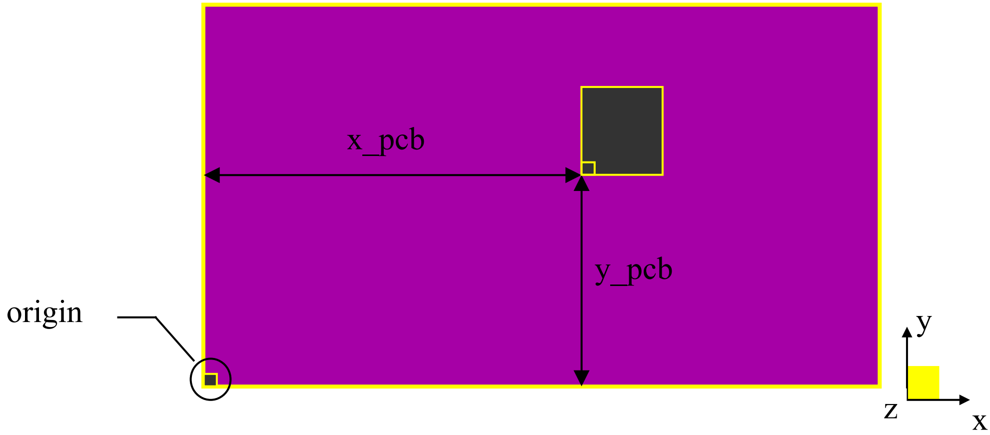
If you wish to specify Joule power in imported trace layers, type the trace's current value in the **Current** column of the Layers table. Note, that the Joule power in traces is not included in the uniformly distributed Power specified in the Power box.

## Components




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 and 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:

Once a PCB has been created, the other PCB components



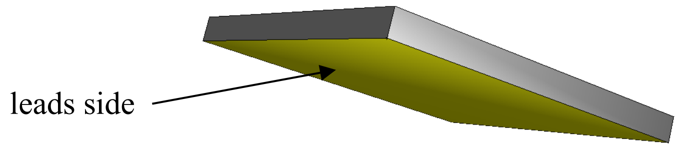

become available. Clicking  will add a Resistor Model component. The created component will be automatically placed on the selected PCB. They can be placed, moved, and resized like any other CoolitPCB object.

### 5.5.2 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.

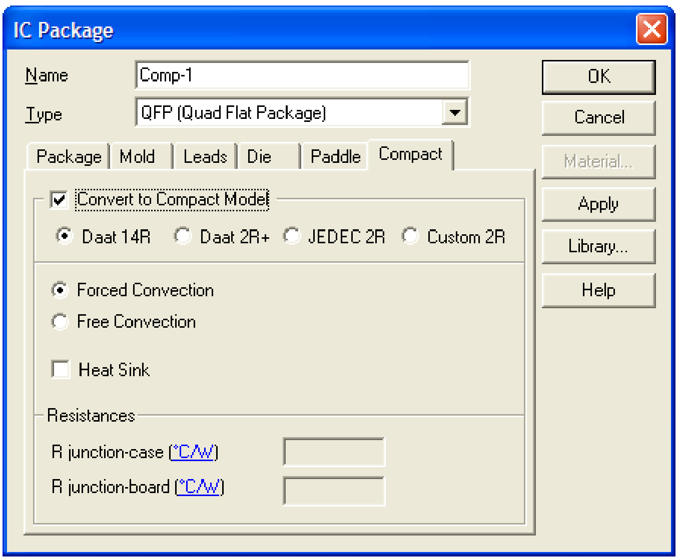
To create a resistor model, select **Resistor Model** in the **Type** field of the IC Package dialog and under the **Package** tab specify the IC Package size and resistance. Under the Properties tab of the dialog, specify the **Power** dissipation and **Max Temperature**. Components exceeding the specified 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 1 or 2-Resistor models, make sure that the yellow side of the component, the leads, is toward the board.




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 $R_{jc}$.

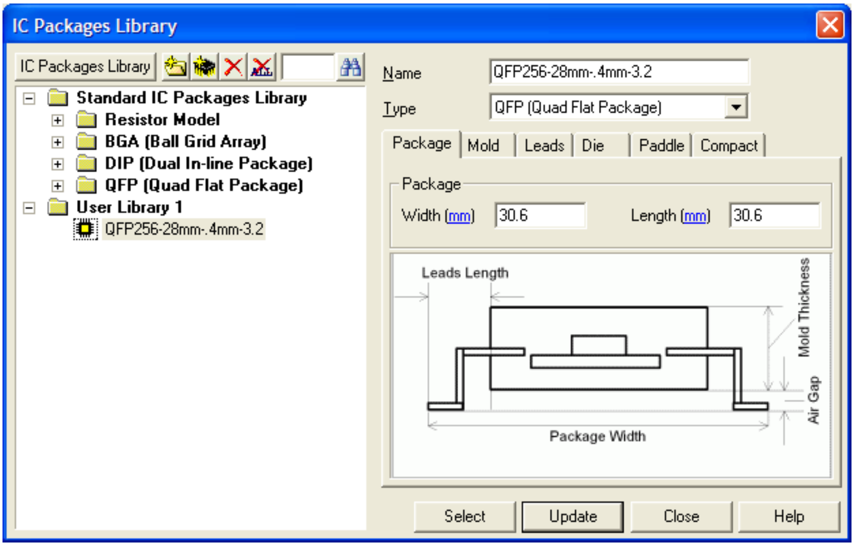
In our experiments, both Daat models have been significantly more accurate than the JEDEC 2R model (i.e. the conventional two-resistor model) over a wide range of conditions (forced and free convection, with and without heat sink). The average error for Daat 2R+ model was 7% vs. 15% for JEDEC 2R and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

The IC Package dialog comes with a Standard IC Packages library (an optional module). 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.

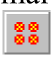
Components can be added to the created library either using drag-and-drop from the Standard library or by selecting a component in the Standard library, then highlighting the new

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.




Max Temperature specified on the Properties page is used in CoolPlot to flag overheated components by painting them as “burnt” (orange-black pattern).

## 5.5.3 Vias

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

Diameter - This is the diameter of individual vias.

Plating Thickness - This is the wall thickness of the via.

Pitch x_pcb - This is the spacing of the vias in the x_pcb direction. This spacing must be larger than the diameter of the vias.

Pitch y_pcb - This is the spacing of the vias in the y_pcb direction. This spacing must also be larger than the diameter of the vias.

Number in x_pcb - This is the number of vias in the x_pcb direction.

Number in y_pcb - This is the number of vias in the y_pcb direction.

Plating Material and Fill Material can also be specified.

### 5.5.4 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-E/100)*U*I.

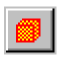
### 5.5.5 Solid Block (part of optional CAD module)

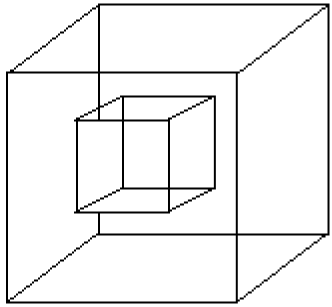
A Solid Block  is a solid rectangular parallelepiped, which can be used to represent power supplies, chips, or any other rectangular objects. Three types of Solid Blocks are permitted, those with a specified temperature (a) those with an unknown temperature (b) and those that simulate a Cold Plate (c). 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. Option (c) is used to simulate a liquid-cooled Cold Plate.

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. Maximum 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, $\rho$ 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**, $\rho$, 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 $T_{high}$.

### 5.5.6 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:

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

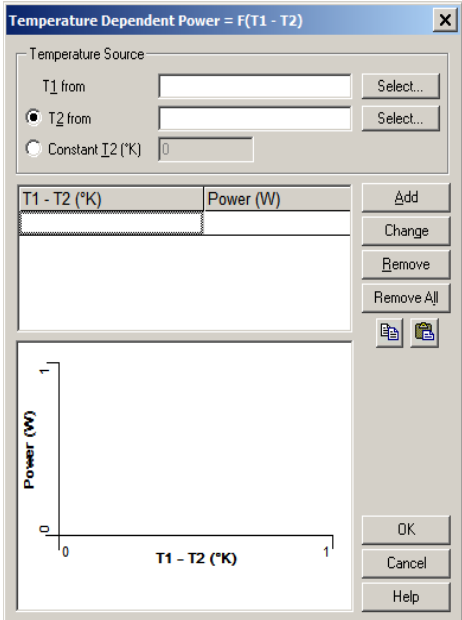
$$P_l + P_s = P$$

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.

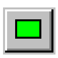
Using the drop down menu, the Power can be specified as uniformly distributed constant, temperature-dependent, Joule, or time-dependent (for transient models only). Joule heating is computed from $Q = \frac{I^2 \rho L}{SV}$. Here $I$ is the current, $\rho$ is the resistivity, $L$ is the size in the **Current 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**, $\rho$, 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.

### 5.5.7 Plate

A Plate  is a solid thin rectangular object, which can be used to represent PCBs or similar components. 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 very accurate model.

Since the Plate model has no thickness, when 3D components, such as Cylinder, are placed on top of a Plate they 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.

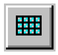
A Plate requires a thermal conductivity and if the flow is turbulent - the surface roughness. The surface roughness simulates the effect of fluid flow on various small components on the Plate surface, which otherwise are not feasible to model separately. The Plate sides can have different roughness. 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. Similarly, the Emissivity and Solar Absorptivity can be specified for each side independently.

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.

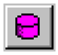
### 5.5.8 Contact Resistance

Contact Resistance is used to model thermal resistance between two surfaces. It 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 is modeled in an air gap; only conduction is considered.

### 5.5.9 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.

### 5.5.10 Cylinder

A Cylinder component  can be used to model capacitors or other cylinder-shaped objects. They can be specified with a power dissipation, a thermal conductivity, and an emissivity.

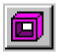
### 5.5.11 Baffle

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.

### 5.5.12 Prism

The triangular Prism component can be used to model non-rectangular objects. Prisms are made of Solid Block components and use the Solid Block Properties dialog. When a Prism is created, the Control Bar will display size of a bounding box around the component. Prisms cannot be drilled nor can they be intersected by other components.

### 5.5.13 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.

### 5.5.14 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 other objects, 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 |

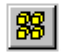
**Heat Flux**

| | |
|---|---|
| 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 |

### 5.5.15 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 the 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.

### 5.5.16 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.

### 5.5.17 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.

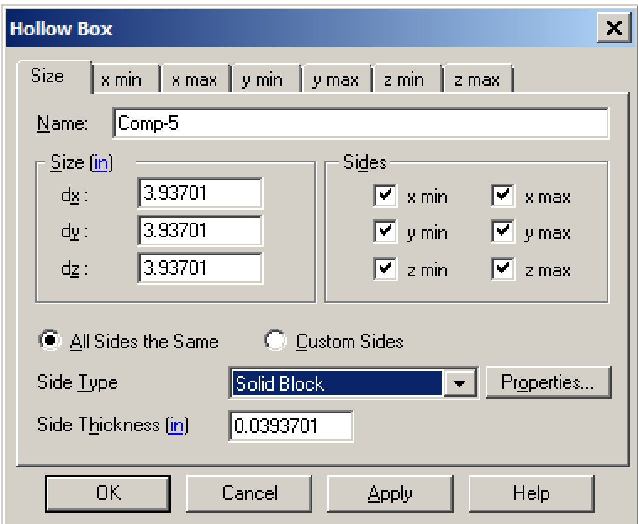
### 5.5.18 Hollow Box

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.

## 5.5.19 Heat Sink

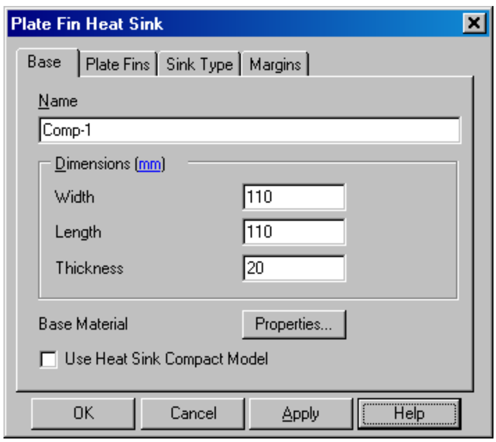
Click  to create a heat sink. A pull-down menu will prompt you to select a Plate Fin, Pin Fin, or Radial Fin heat sink.

### Plate Fin Heat Sinks

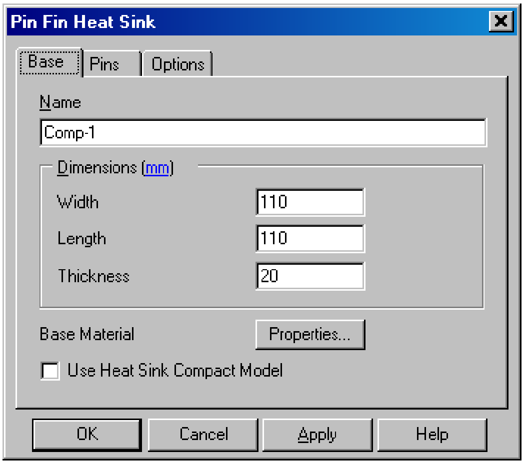
On the **Base** page, the component name, base dimensions, and base properties are defined. The Name can be any combination of letters, symbols, numbers, or spaces up to 63 characters long.

To model the heat sink approximately, check the box next to **Use Heat Sink Compact Model**. This will automatically generate the appropriate compact model 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



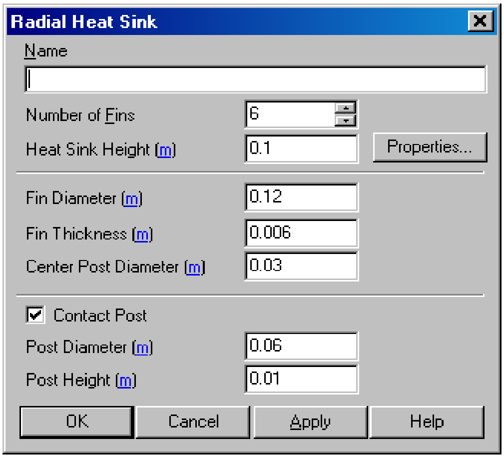

Conductivity for the adhesive, and a Fin Material. **Margins** can be added to any side of the heat sink base.

### 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 compact model 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.

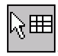
The Fin Diameter and Fin Thickness are specified in their respective boxes. The Center Post Diameter is for specifying the diameter of the post that joins the fins.

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

## 5.5.20 Grid Zone

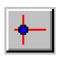
Click  to select Grid Zone. The Grid Zone component is used to define an area within the computational domain that has a different grid from the rest of the problem. A Grid Zones's grid does not propagate into the rest of the domain.

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 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.

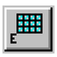
### 5.5.21 Monitor Probe

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…**

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.

### 5.5.22 Vent

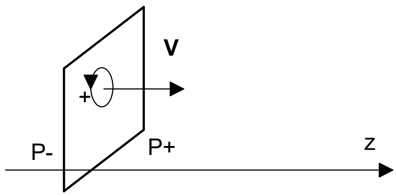
A Vent  is a planar porous media model of a ventilation grille. If your vent does not have a grille, you should use an Open. In addition to the flow resistance parameters for Planar Porous Resistance model, a Vent requires the thermal conductivity of the Vent material, the Gauge Pressure and the Temperature.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

### 5.5.23 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.

Specify the Flow Rate or Velocity (if you want a constant flow rate or velocity through the Fan) or a Fan Curve. You can also specify the Flow Direction 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 Flow Direction 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 Flow Direction Unit Vector will change to (0, 0, -1).




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$^{o}$ 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 $30^o$, by entering Alpha = $30^o$ 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.

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.

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.

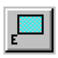
Several additional conventions are used with the Fan model. First, note that the fan curve uses the pressure difference, $\Delta P$, rather than the absolute pressure. If the computed $\Delta P$ is outside of the specified fan curve, Coolit will use the limiting values in the table. Consider, for instance, 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 $\Delta P$ is higher than 44.0, then Q will be set to 0.0; if $\Delta P$ 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 $\Delta P$ is always computed as follows: $\Delta P = P^+ - P^-$, where $P^+$ is the pressure on the Fan side facing in the positive axis direction.

## 5.5.24 Open

An Open  is an unobstructed vent. The properties of an Open are its Gauge Pressure and Temperature.

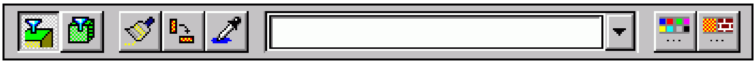
If you have an outdoor application that radiates to the sky, you can account for this by unchecking the box next to $T_{sky}$ and entering a temperature. $T_{sky}$ is the effective sky temperature. Its value depends on atmospheric conditions changing from about -43C under a cold clear sky to about 12C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

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

### 5.5.25 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.

## 5.6 Painting Tools Toolbar



- Applies selected color/texture to component's face. Faces are selected in 3D view panel.
- Applies selected color/texture to whole component. Components are selected in 3D view panel
- Switches to painting mode
- Rotates texture on component's face
- Picks color/texture
- Shows a list of most recently used colors/textures
- Displays select color dialog. Selected color is inserted into the list of most recently used colors/textures
- Displays the Select Texture dialog. Selected texture is inserted into the list of most recently used colors/textures

# 6 Mouse and Keyboard Shortcuts

### Mouse Functions

| Input Device | Input Operation | System Response<br>2-D View Pane | <br>3-D View Pane |
|---|---|---|---|
| 2 Button Mouse | Left Mouse Button | Select Object | Select 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 Object | Select 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 General tab of the **Tools|Options** dialog.

## 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 |
| Tab | Go to the next component |
| Shift+Tab | Go to the previous component |
| Esc | Deselect the currently selected component (if any) |

# 7 Importing and Exporting CAD Files

## 7.1 Import and 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 CoolitPCB 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, CoolitPCB 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 CoolitPCB 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 CoolitPCB 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, CoolitPCB 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 from your project in IDF format select **File|Export|IDF File…** and specify the folder for the exported file.

## 7.2 Import of IC Package Thermal Models

This function is available only if you purchased 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 and emailed back to you within 2 business days.

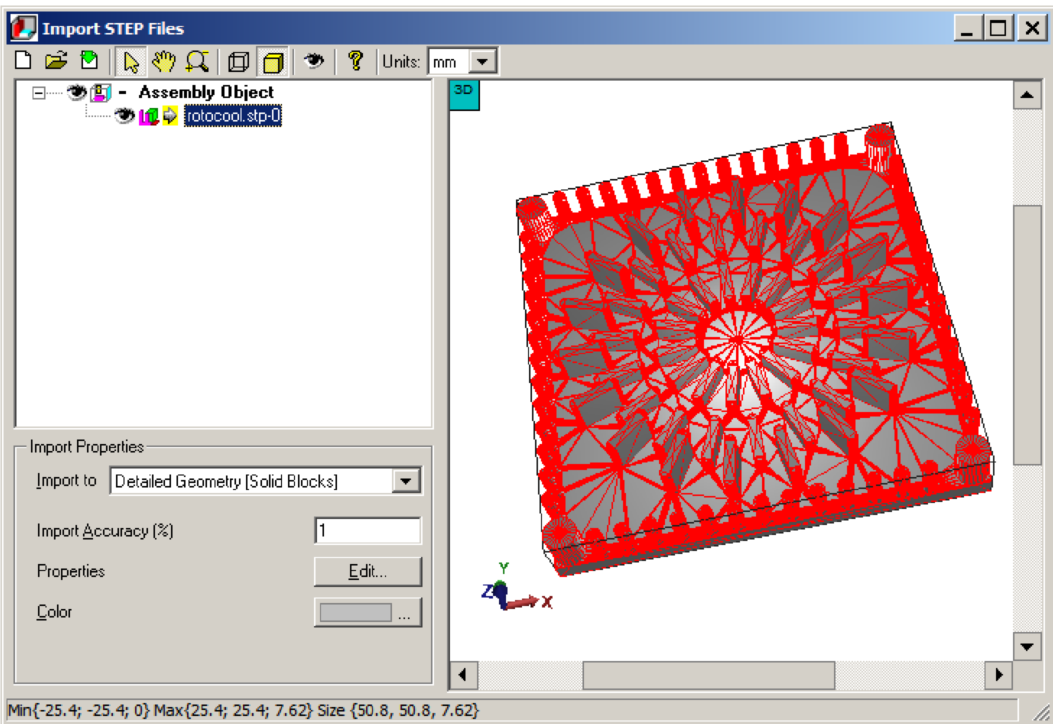
## 7.3 Import DXF Traces

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.

## 7.4 Import/Export of STEP Files

To import STEP files into CoolitPCB, 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 CoolitPCB 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.

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.

## 7.5 Export of STL Files

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.

## 7.6 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.

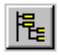
Import the selected part(s) by clicking . The imported parts will now be hidden in the Import STEP Files dialog window, but will show up in Coolit and will be listed there under Project Objects in the Objects Manager.

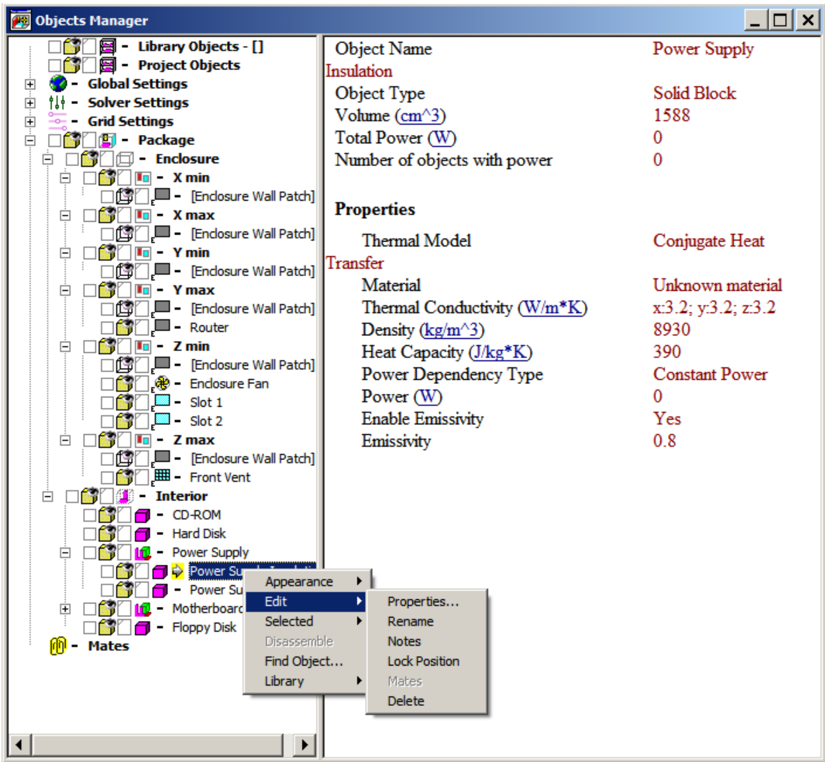
To export Coolit projects in STEP format select File|Export|STEP File… and specify the exported file name and units.

## 7.7 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. CoolitPCB 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.

# 8 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 object 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.

## 8.1 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**.

## 8.2 Double Clicking

Double clicking on a component name or parametric object name in the Objects Manager will launch the properties dialog for the clicked item.

## 8.3 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.)

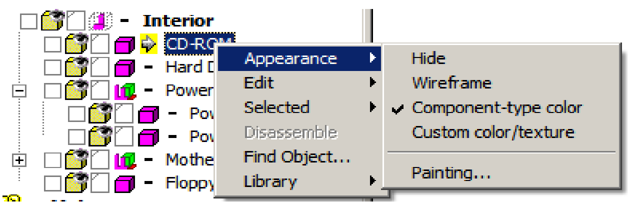
## 8.4 Overlapping Components - Priority

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. When components intersect, the higher priority component defines the intersection. If you wish to change priority of a component, drag it either up (to decrease priority) or down the list (to increase priority).

## 8.5 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.




## 8.6 Disabling Objects

In the course of "what if" iterations, it is often convenient to temporarily remove components and objects from the system.

CoolitPCB 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 .

## 8.7 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.

## 8.8 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.

## 8.9 Locking Position of Objects

This function is described here.

## 8.1 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 the object does not have Mates, the Mates menu is grayed out.

## 8.2 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 .

## 8.3 Using Libraries

The "Library Objects []" folder, is always at the top of the Objects Manager folder list. Objects can be dragged directly into this library from other Objects Manager folders, such as Interior. When viewing Library objects the program automatically switches into Components Mode. Click the right mouse button and in the popup menu select **Save Library As** to save the library.

# 9 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 the **Build|Model Check** menu, 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.

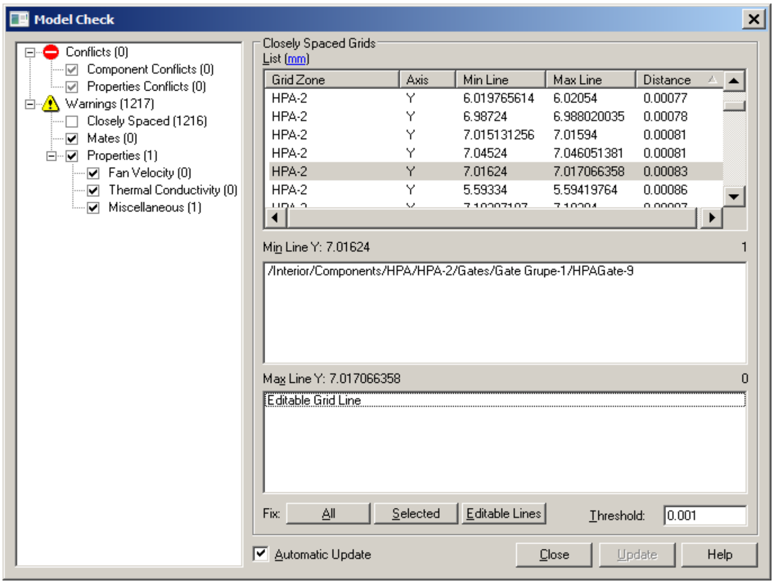
## 9.1 Conflicts

**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.

## 9.2 Warnings

### 9.2.1 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.

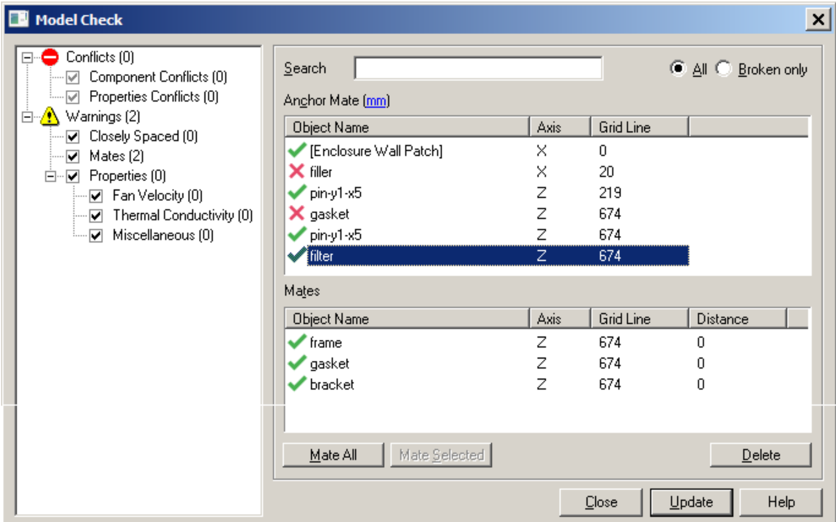
You have the option of automatically fixing closely spaced grids by ignoring differences in grid-line spacings defined by Threshold.  You can always fix all Warnings that involve Editable Lines (magenta) either manually or by clicking the Fix **Editable Lines** button. You can try fixing all flagged 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.

### 9.2.2 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.

### 9.2.3 Properties

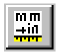
**<u>Fan Velocity</u>** – Coolit will check to see if the velocity specified for a fan is zero or if it exceeds the value in the Max Velocity field. Double clicking on a fan in the list will bring up the Fan Properties dialog. You can increase the Max Velocity value if fans in your problem have velocities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

# 10 Calculator Tools

## 10.1 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 | 3.14159265358979323846264433 |
| | e | 2.71828182845904523536028747 |
| | 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 |

## 10.2 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.

# 11 Positioning and Sizing Tools

## 11.1 Distance Toolkit

The Distance Toolkit is opened by selecting **Tools|Expert|Distance Toolkit**. It is used for determining the distance between a set reference point and a selected object, and moving and resizing components.

When first opened, the dialog will report the 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 selecting **Tools|Expert|Set Reference Point** 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.

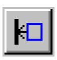
Using the Distance Toolkit, you can move and resize components, either with the mouse, or by using tabular inputs. To move or resize an object, depress the **Move** or **Resize** button, respectively, and type in the new coordinates. Note that these coordinate cells are calculator cells. Click the **Apply**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

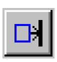
## 11.2 Measure Tool

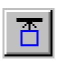
The distance Measure tool is used for positioning and measuring the distance between objects. Select **Edit|Measure** or click,  to start Measure tool. All objects are now displayed with black corner points and one point in the domain, by default it is (X, Y, Z) = (0, 0, 0), is shown in red. Click any black point in either 2D or 3D views and CoolitPCB 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 such that the vertex the red point is at, is moved to the entered position. Note that the all cells are calculator cells.

## 11.3 Alignment Toolbar

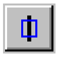
The Alignment Tools are used to align objects in any of the 2D view windows. The alignment function is always relative to the active 2D window. More than one object must be selected to activate the alignment tools. It is recommended that before selecting objects to align, you click  to unselect any previously 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.

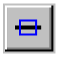
 will move the selected object(s) so that their left sides are aligned.

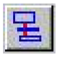
 will move the selected object(s) so that their right sides are aligned.

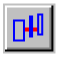
 will move the selected object(s) so that their top sides are aligned.

 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

## 11.4 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.

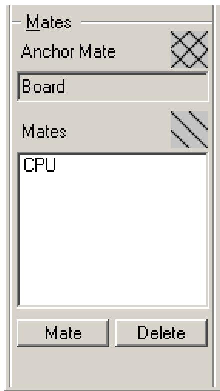
Right-mouse-click an object in the Objects Manager and select Edit|LockPosition. The locked object will now display with a golden lock:


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

## 11.5 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.

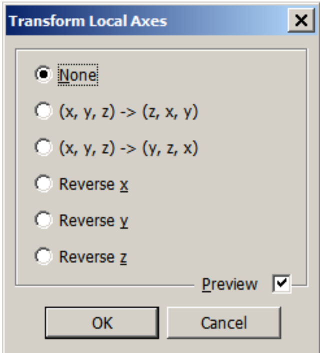



To move a mate into position in the anchor mate's plane, it's best to use the pane view coplanar with the anchor mate plane. After components are mated, Coolit permits breaking the mate contact. If this happens, you get a warning in the Model Check dialog's Mates page. The broken mates are indicated 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.

## 11.6 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).

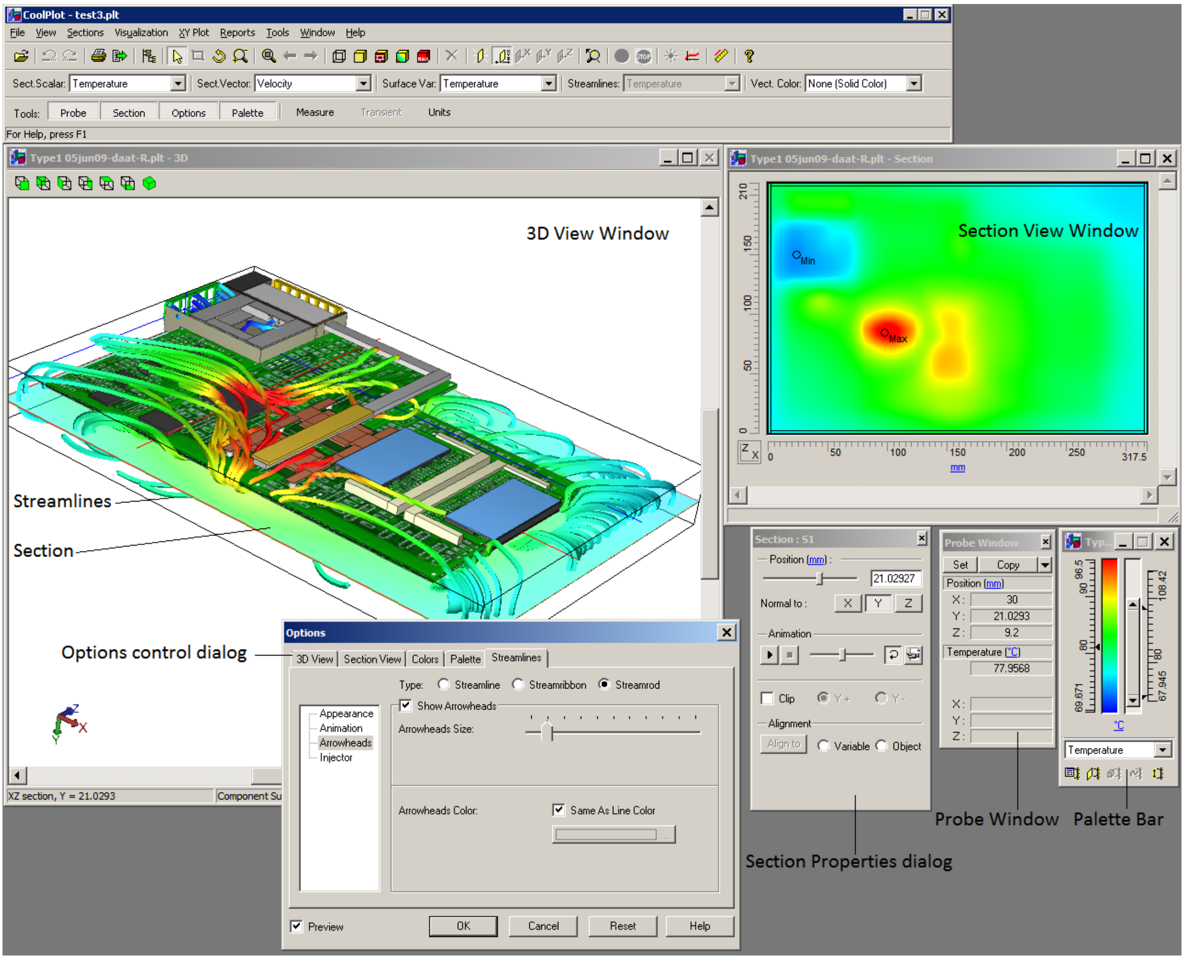
In the dialog select the desired transformation, visually verify it and click OK to exit.



# 12 CoolPlot

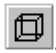
## 12.1 Overview

CoolPlot is 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**.

## 12.2 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 double clicking on the object, or by using Objects Manager. In order to re-display all components, select **View|Unhide All**.

You can add **Labels** in 3D view window by right mouse clicking where you want the label and selecting **Add Label** from the drop down menu or you can click  on the toolbar. Specify the Label's **Name** to be used in the Objects Manager and the **Text** that will appear in 3D view. You can select created Labels either in 3D view or in the Objects Manager. Double-clicking a Label will open the Label Properties dialog. To move a Label, select and drag.

## 12.3 Sections and the Section View Window

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 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.

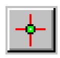
The position and orientation of a selected Section can be done using the **Section Properties** dialog, which is opened by double-clicking the section or clicking the right mouse button and selecting Section Properties. You can also use this dialog, to animate the selected Section, or to align it either 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).

### 12.3.1 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.

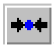
The button copies only numeric data from the Probe Bar to the Clipboard. The button copies both the numeric data and headers to the Clipboard.

### 12.3.2 Information Window

Clicking will allow you to create a box by clicking-and-dragging 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. Note, that the displayed information is obtained from already interpolated data and as such should be used only for rough estimates. If accurate results are desired, you should use Gauge components.

## 12.4 Palette Bar

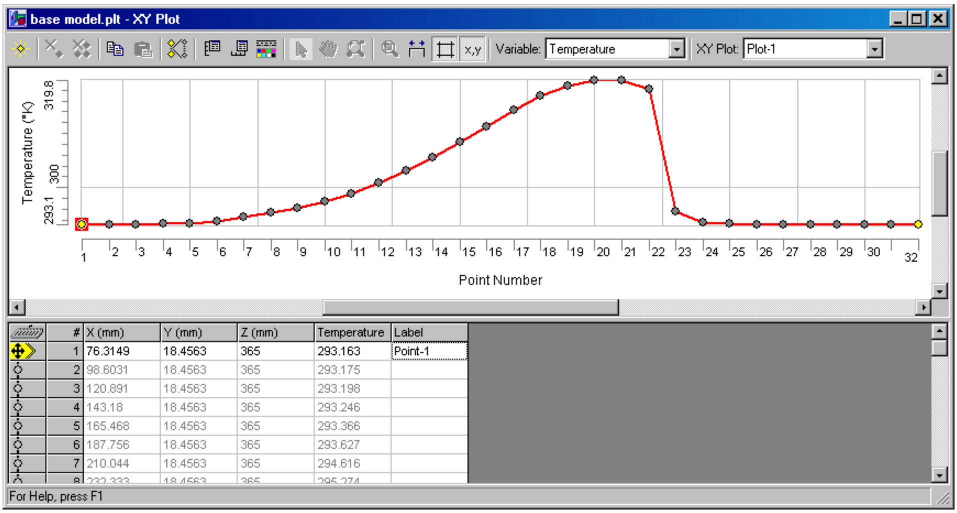
The Palette Bar is used to control the colors used to represent the different variable values. The current variable is shown and can be selected in the pull-down menu at the bottom of the Palette bar. You can adjust the displayed range of the current variable by dragging the palette handles up or down. 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 , 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 . Press Reset button to reset both the minimum and maximum.

## 12.5 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  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 .

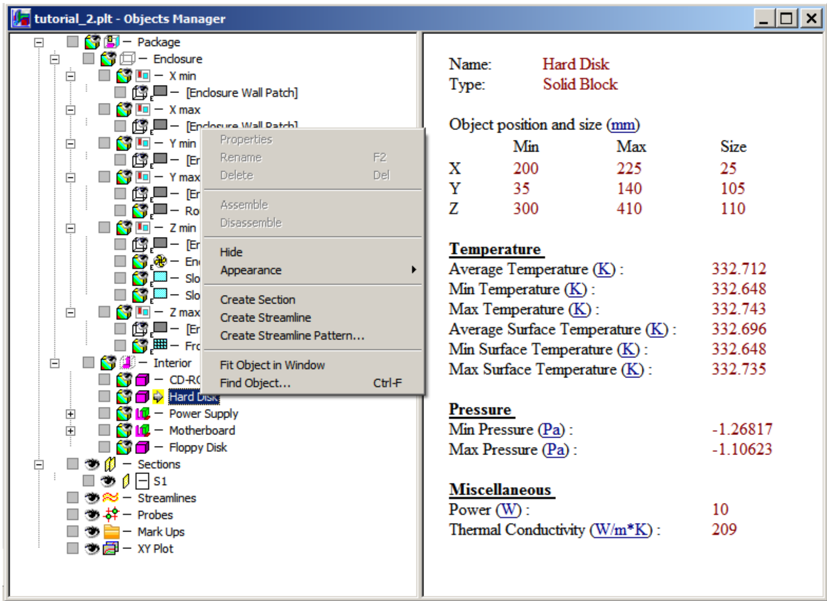
The Format Plot Area dialog is used to format axes and other elements of the plots. It can be accessed using any of the following buttons , , . Double click on either axis to shortcut to the dialog.

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.

## 12.6 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.

Double-clicking on a PCB object or selecting it and clicking the **Properties** button will open a table displaying all components associated with the selected PCB and their assigned properties and computed temperature values.

Printed Circuit Board

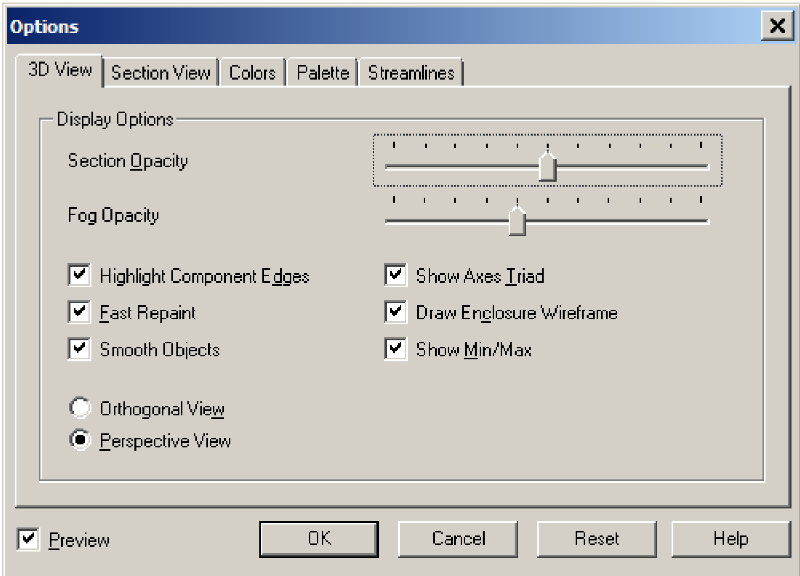
| Name | Pow... | x_... | y_pc... | Len... | Wi... | Hei... | Resistor... | R... | R... | T min (°C) | T max (°C) | T avg (°C) | T case (°C) | T junction (°C |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| U1 | 5 | 35 | 270 | 40 | 40 | 3 | 2-Resistor | 7 | 7 | 35.544 | 50.362 | 37.089 | 42.0836 | 38.6545 |
| U3 | 2 | 35 | 140 | 40 | 40 | 3 | 1-Resistor | 3 | | 25.9089 | 76.3134 | 26.2837 | 27.2244 | |
| Pow a... | 25 | 175 | 20 | 20 | 75 | 4 | 2-Resistor | 5 | 5 | 47.4523 | 90.0273 | 89.6062 | 54.1552 | 89.7227 |
| z1-y1-x1 | 0.25 | 265 | 58.5 | 15 | 15 | 3 | 0-Resistor | | | 54.8703 | 56.4102 | 56.105 | | |
| z1-y1-x2 | 0.25 | 295 | 58.5 | 15 | 15 | 3 | 0-Resistor | | | 54.4984 | 56.0964 | 55.7676 | | |
| z1-y2-x1 | 0.25 | 265 | 88.5 | 15 | 15 | 3 | 0-Resistor | | | 55.4218 | 57.1339 | 56.8079 | | |
| z1-y2-x2 | 0.25 | 295 | 88.5 | 15 | 15 | 3 | 0-Resistor | | | 54.987 | 56.7653 | 56.4037 | | |
| z1-y3-x1 | 0.25 | 265 | 118.5 | 15 | 15 | 3 | 0-Resistor | | | 55.3348 | 56.9419 | 56.7085 | | |
| z1-y3-x2 | 0.25 | 295 | 118.5 | 15 | 15 | 3 | 0-Resistor | | | 54.8727 | 56.6472 | 56.3389 | | |
| M. Plex | 0.25 | 315 | 185 | 25 | 140 | 4 | 0-Resistor | | | 48.8516 | 50.7204 | 50.3469 | | |
| M. Ple... | 0.25 | 265 | 185 | 25 | 140 | 4 | 0-Resistor | | | 48.7664 | 50.6876 | 50.2031 | | |
| M. Ple... | 0.25 | 215 | 185 | 25 | 140 | 4 | 0-Resistor | | | 47.902 | 50.5529 | 49.9045 | | |
| Pow a... | 25 | 125 | 20 | 20 | 75 | 4 | 2-Resistor | 5 | 5 | 47.6887 | 90.0303 | 89.6123 | 53.8907 | 89.6784 |
| Pow a... | 25 | 75 | 20 | 20 | 75 | 4 | 2-Resistor | 5 | 5 | 47.6322 | 90.211 | 89.8992 | 54.5173 | 89.9329 |
| U2 | 2 | 115 | 230 | 40 | 40 | 3 | 0-Resistor | | | 49.1803 | 49.477 | 49.3623 | | |

Close Copy

The selected object can be fit to the 3D-view window by clicking the **Fit Object** button. When a CoolPlot Object, such as a Section or Streamline is selected, the **Rename**, **Delete**, **Properties**, **Assemble**, and **Dissasemble** buttons become available.

## 12.7 Options Dialog

Most viewing options can be set in the **Options** dialog accessible from **Tools|Options** or using right mouse click in either 3D or Section view.

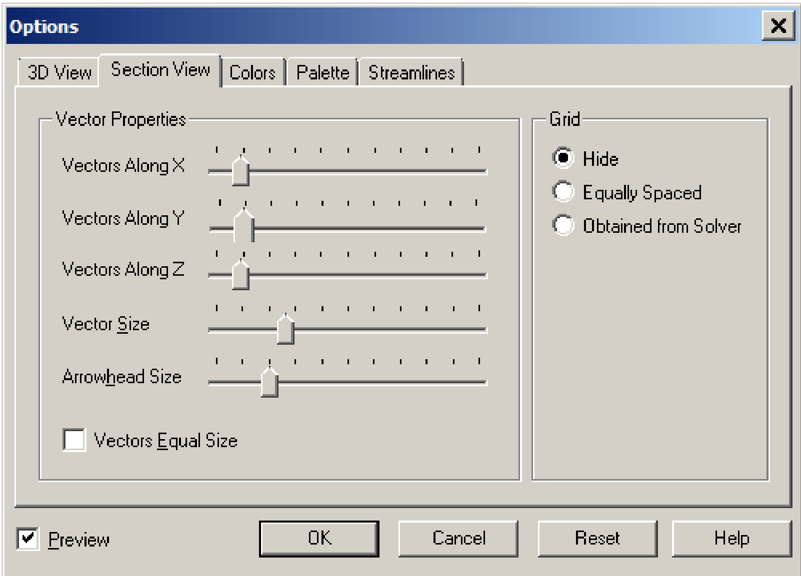



### 12.7.1 3D View

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 CoolitPCB. 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.

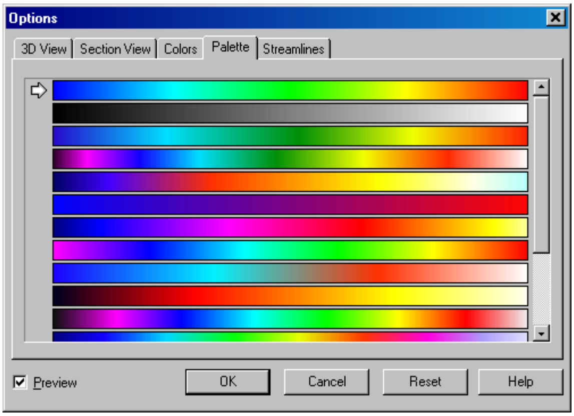
### 12.7.2 Section View




The Section View page controls the Section view window. 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.

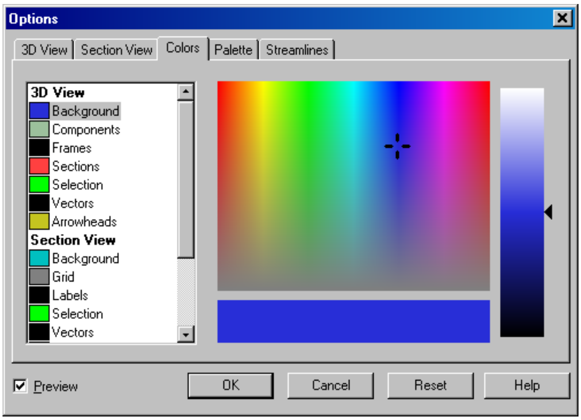
Under Grid, you can choose to hide the grid, show an equally spaced grid, or show the grid used to solve your problem.

### 12.7.3 Palette



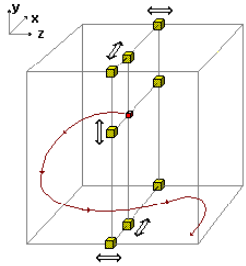

On the Palette page you can select the palette that your scalar values will be displayed in. Click on a colored bar to select.

### 12.7.4 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.

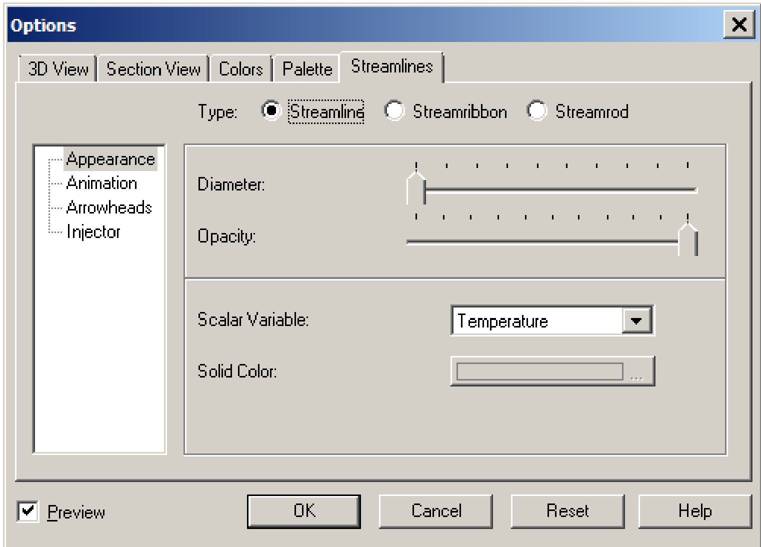
### 12.7.5 Streamlines, Streamribbons, and Streamrods




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.

## 12.8 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.

## 12.9 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.

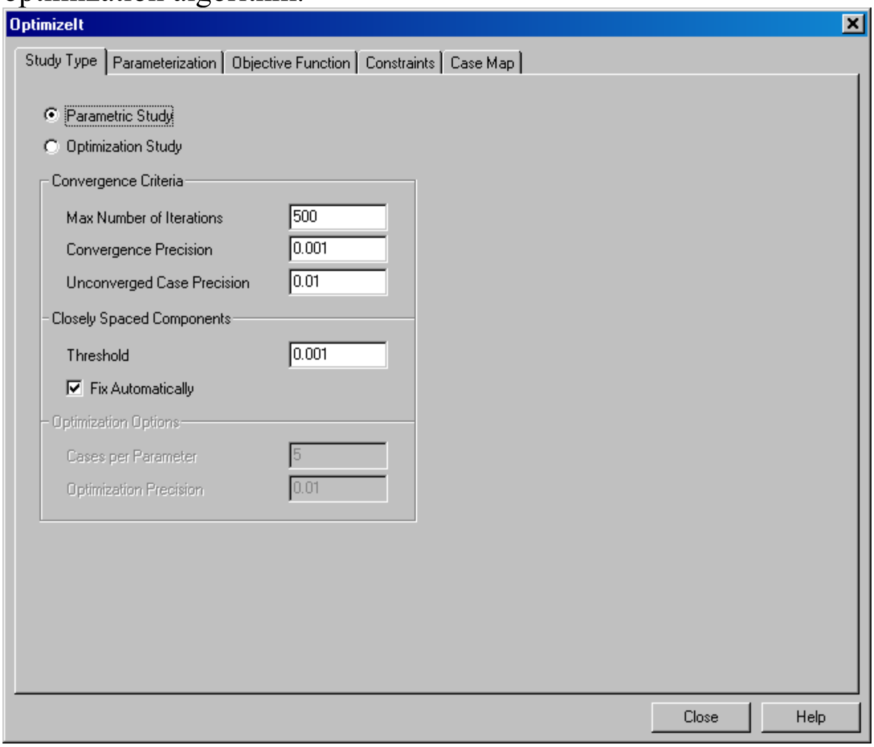
In animation cases, the movie image will be the same size as your active window. To reduce the movie file size, make the window smaller before creating an AVI file.

# 13 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. (OptimizeIt is an optional module.)

## 13.1 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|Parameters** 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.

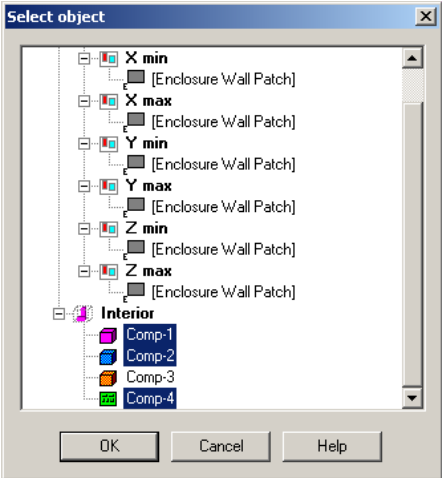
The minimum number of Cases per Parameter is available only if Optimization Study is checked. The higher the number of Cases per Parameter, the more likely that you will find your optimum design. However, this may require more cases to be run. The recommended (default) is 5 cases per parameter.

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.

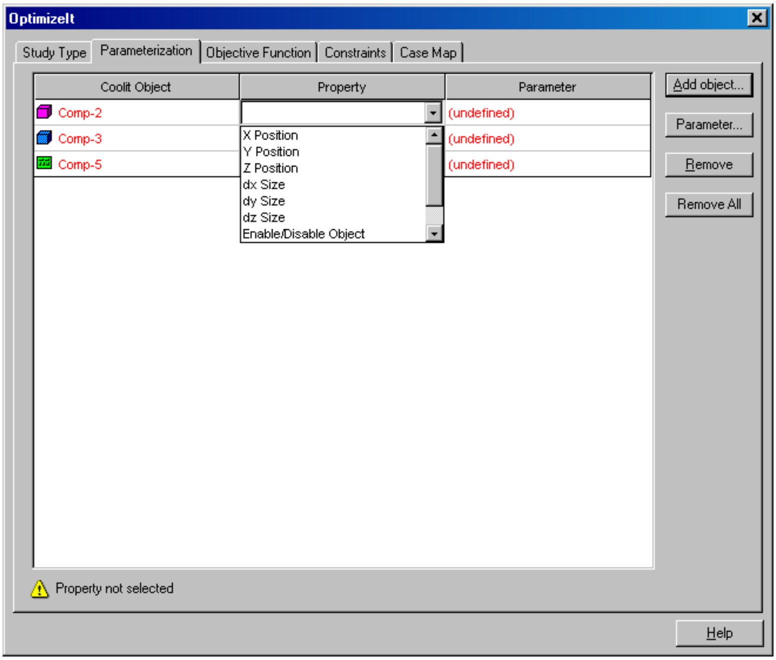
## 13.2 Parameterization Page

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.

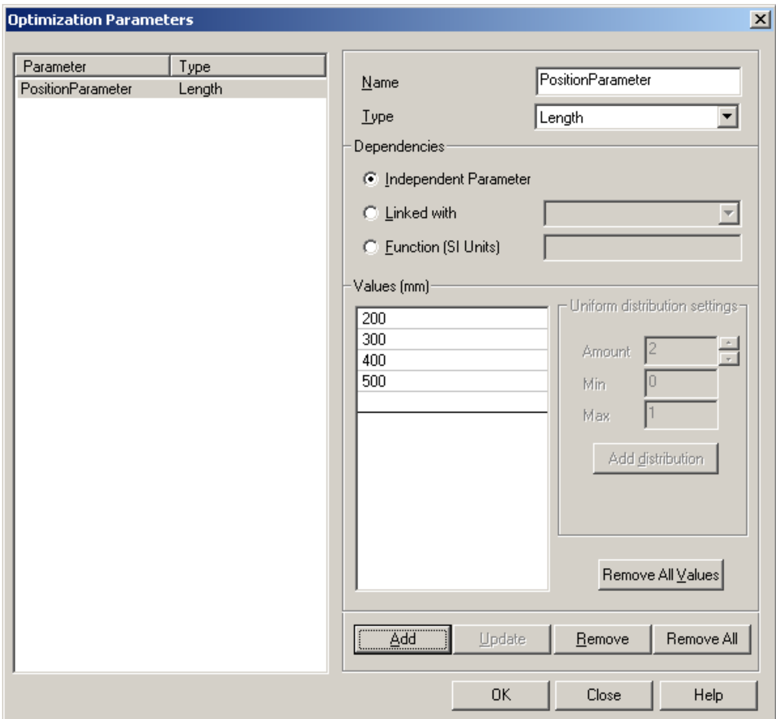



Parameter values for selected Property can be defined by clicking the **Parameter** column cell and selecting the name from the pull-down menu. Select **New Parameter** from the pull-down menu to define a new Parameter for an object's Property. Parameters can be edited by clicking the **Parameter…**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

## 13.3 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 number of 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:

$$2.345\ [\mathrm{BTU/min/in}^2] = 63915\ [\mathrm{W/m}^2]$$

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.

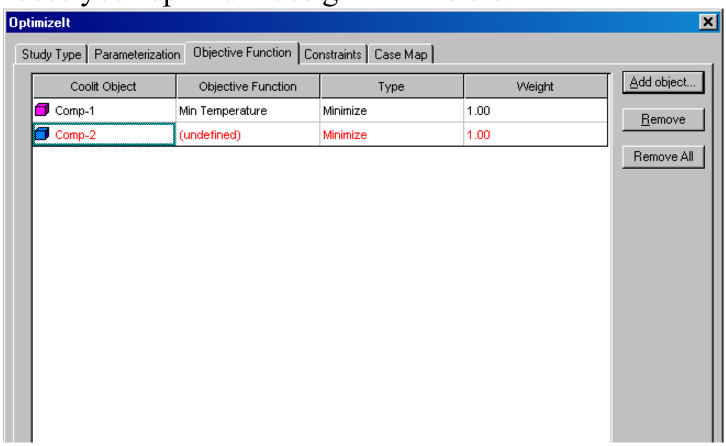
In Optimization Studies it is only necessary to specify the maximum and minimum values for parameters. OptimizeIt will automatically select which values to test within the specified range.

### 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.

## 13.4 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.

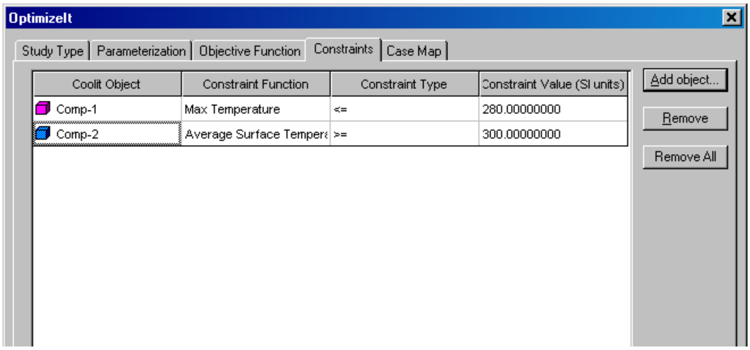
The **Type** of objective function can be selected from the pull-down menu that appears when any cell in the Type collumn is clicked. The Type is the kind of optimization you want to perform; either maximizing or minimizing.

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.

## 13.5 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.

## 13.6 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, CoolitPCB 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 |
|---|---|
| W | The Case is waiting for processing and is not solved yet; Objective Functions have not been computed. |
| OK | 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 Status can be changed by clicking on the status icon. You can exclude Cases from the optimization process by changing their Status icon to . Objective Function values can be sorted in ascending or descending order by clicking the Objective Function column header.

## 13.7 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 Function(s) valid | OptimizeIt has finished initializing Objective Functions page | Indicates number of 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:<br>1. Parameterized object not selected<br>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:<br>1. Parameterized object not selected<br>2. Property not selected<br>3. Parameter not |

| | | |
|---|---|---|
| | | selected<br>4. Parameter type and property type incompatible<br>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 | CoolitPCB 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}<br>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:<br>1. Parameterized object not selected<br>2. Constraint not selected |
| Testing bounds... | Start testing parameter ranges | |

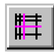
| | | |
|---|---|---|
| Wrong Bound {case arguments} | Enclosure or component conflicts | Check for geometric or component conflicts |
| Setup complete, Press Start to begin Optimization Process | Constrained setup is ready | |
| Estimated number of cases – {20} | Estimated number of cases message | |
| 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) |
| CoolitPCB cannot create Optimization directory | Cannot create optimization directory, Possibly disk full, or read/write permissions not set properly |

# 14 Grid Mode and Grid Settings

## 14.1 The Default Grid

CoolitPCB needs a grid of discrete cells (finite volumes) to compute a solution. To generate the grid, press the icon, 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.

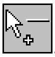
The number of grid lines in each direction is listed in the status bar window. Anchor (gray) grid lines will be drawn on the borders of all components and moveable (magenta) grid lines will be drawn to ensure a satisfactory grid. It is best to use the wireframe view mode to view the grid

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.

## 14.2 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. CoolitPCB 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.

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.

## 14.3 Grid Commands

CoolitPCB provides six command options for grid control: Clear, Check, Smooth, Refine, Default, and Settings. 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.

## 14.4 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.

## 14.5 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 a 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.

# 15 Solver Settings

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 CoolitPCB.

## 15.1 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. The value of C should be approximately equal to half of the maximum number of grid cells in the main flow direction. If, for example, your grid has (12, 15, 25) grid cells, and the main flow is expected in the x-direction, then you should set C to about 6. Physically, Courant number indicates the number of grid cells a signal will travel in one time (iteration) step. Thus, if the number of cells in x-direction is 20 and C = 10, it means that it will take 2 iterations to take the signal from one Wall to the other. The best convergence rates are obtained when the distance a signal travels over one iteration is commensurate with the important length scales of the problem.

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 decide when to solve it depending on flow conditions and global convergence.

For **Transient** problems, you will need to decide what stopping criteria you will use (**Time Interval** only or **Max Temperature** or **Time Interval**) and specify the requested parameters. For time accuracy, the specified Time Step should be commensurate with important time scales of the problem being solved.

## 15.2 Solver Selection

CoolitPCB provides two solver options: **Economy** and **Fast**. Despite their names, Fast solver is not always faster than the Economy solver, although this is true in most cases and especially for larger ones (>3M 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 of RAM per 1 M grid cells in single precision (32-bit word format) and 1 Gb of RAM per 1 M cells in double precision (64-bit word format). Fast solver needs approximately ~1.5 Gb of RAM 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 difficult grids with stretched cells 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 the accuracy of computed results. This option is used to mitigate the round-off error when solving difficult cases with very large and unevenly distributed grids, which have difficulty converging in single precision.

**Parallel Processing** section specifies the number of CPUs used by the solver. (If you have not purchased this option, it will be grayed out.) By default Coolit will attempt to use all CPUs.

**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 time per iteration (time step). In most cases, the default setting is optimal.

## 15.3 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 turbulent viscosity, $\nu_t$, (the turbulent viscosity divided by the molecular viscosity), and $y^+$. 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 often 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.

## 15.4 Restart File

Specify the Restart File name and the frequency of saving restart files in the Restart File page. The default is to save the file every 100 iterations. Click the button to load the restart file.

## 15.5 Report File

The Report File contains a documentation of the project. It is an HTML file with JavaScript, so that you could expand the report's section headers for better viewing. Depending on the settings of your internet browser, you may get a warning that the file is trying to run scripts or ActiveX controls. If you want to see the interactive contents of the Report File, select the Allow Blocked Content… option in your browser.

The Report File is written automatically every time Solver is started. By default, it contains only the project's summary: global settings, model statistics, solver settings, and the grid. 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.

To include computed results into a Report File, use the **Reports|Project Report** menu in CoolPlot. If you desire to save a report file without starting Solver, select **Tools|Write Report File…**

# 16 Theory

## 16.1 Gas Phase

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

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

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

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

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

The variables are defined in the table below.

| Variable | Definition |
|---|---|
| $\rho$ | Gas density |
| $\mathbf{V}$ | Gas velocity vector |
| P | Pressure |
| $\mathbf{D}$ | Deformation tensor, $1/2(\nabla\mathbf{V}+(\nabla\mathbf{V})^T)$ |
| $\mathbf{g}$ | Acceleration of gravity vector |
| $\mu$ | 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.

## 16.2 Porous Media Model

The momentum, energy and mass transport inside porous resistances are computed using the following model:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) = 0 \qquad (5)$$

$$\rho\left[\frac{\partial \mathbf{V}}{\partial t} + \mathbf{V} \cdot \nabla \mathbf{V}\right] = -\nabla p + \nabla \cdot 2\mu \mathbf{D} + \left[\frac{\mu}{\kappa}\mathbf{V} - \rho C \frac{\mathbf{V}|\mathbf{V}|}{2}\right] + \rho \mathbf{g} \qquad (6)$$

$$(\rho c_p)_{pvr} \frac{\partial T}{\partial t} + \rho c_p \mathbf{V} \cdot \nabla T = -\nabla \cdot k_{pvr} \nabla T + Q \qquad (7)$$

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

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

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

$\phi$ is the porosity, $\phi = \dfrac{v_{gas}}{v_{gas} + v_s}$, 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.

## 16.3 Enclosure Boundaries

In CoolitPCB, the **Enclosure** refers to the volume inside which computations will be done. It can be a card slot, a wind tunnel, or simply a volume in which you wish to carry out the simulation.

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.

It is often convenient to resize the Enclosure from a specific side. 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. If you want to use the mouse to resize the Enclosure from the $X_{min}$ side, hold the Shift key while dragging the side. Enclosure will move $X_{min}$ towards your Interior Components. If you want to resize the Enclosure from the $X_{min}$ side using the Enclosure Properties dialog, enter the new Enclosure size in the entry box, and check the check box next to it. When you click OK, Enclosure will be resized from $X_{min}$ side instead of the $X_{max}$ side.

### 16.3.1 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.

### 16.3.2 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, $V_n$, CoolitPCB 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.

### 16.3.3 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 $\frac{\partial T}{\partial n} = 0$ .

### 16.3.4 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.

### 16.3.5 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 a Vent requires the thermal conductivity of the Vent material, the Gauge Pressure and Temperature. The normal velocity component and heat flux across the Vent are computed from the 1-D porous media equations.

### 16.3.6 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.

## 16.4 Turbulence Modeling

CoolitPCB provides the Spalart-Allmaras eddy viscosity turbulence model documented below (Spalart and Allmaras, 1992).

The model computes the turbulence viscosity, $\mu_t$, 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:

$$\rho \overline{u_i' u_j'} - \frac{1}{3} \rho \overline{\delta_{ij} u_i' u_j'} = \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \qquad (9)$$

$$-\rho c_p \overline{u_i' T'} = \frac{c_p \mu_t}{\sigma_t} \frac{\partial \overline{T}}{\partial x_i} \qquad (10)$$

where ' indicates the fluctuating components of velocity and temperature, $\overline{\quad}$ and $U$ denote ensemble averaged (mean) values, and $\sigma_t$ is the turbulence Prandtl number.

### 16.4.1 Spalart-Allmaras Eddy Viscosity Model

For both forced and natural convection thermal flows, eddy viscosity models show better predictive ability than k-ε models and with less computational work. CoolitPCB eddy viscosity model use blended wall functions which automatically adjust depending on value of local y+ to provide the best accuracy.

CoolitPCB 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:

$$\frac{\partial v}{\partial t}+u_i\frac{\partial v}{\partial x_i}=C_{b1}Sv-C_{w1}f_w\left(\frac{v}{d}\right)^2+\frac{1}{\sigma}\left[\frac{\partial}{\partial x_i}\left((v+v)\frac{\partial v}{\partial x_i}\right)+C_{b2}\frac{\partial v}{\partial x_j}\frac{\partial v}{\partial x_j}\right]$$

where $\mu_t=\rho v f_{v1}$ , $f_{v1}=\frac{\chi^3}{\chi^3+C_{v1}^3}, \chi=\frac{v}{v}$ , ρ is the density, ν is the molecular kinematic viscosity, and μ is the molecular dynamic viscosity, $\hat{S}=\Omega+2\min(0,S-\Omega)+\frac{\hat{v}}{\kappa^2 d^2}f_{v2}$ where $\Omega=\sqrt{2W_{ij}W_{ij}}$ is the magnitude of the vorticity, $d$ is the distance to the nearest wall, and

$$f_{v2}=1-\frac{\chi}{1+\chi f_{v1}},\ f_w=g\left[\frac{1+C_{w3}^6}{g^6+C_{w3}^6}\right]^{\frac{1}{6}},\ g=r+C_{w2}\left(r^6-r\right)$$

$$r=min\left[\frac{\hat{v}}{\hat{S}\kappa^2 d^2},10\right],\ W_{ij}=\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}-\frac{\partial u_j}{\partial x_i}\right),\ S_{ij}=\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)$$

$$C_{b1}=0.1355,\ \ \sigma=\frac{2}{3},\ \ C_{b2}=0.622,\ \ \kappa=0.41$$

$$C_{w2}=0.3,\ \ C_{w3}=2,\ \ C_{v1}=7.1,\ \ C_{w1}=\frac{C_{b1}}{\kappa^2}+\frac{1+C_{b2}}{\sigma}$$

### 16.4.2 Wall Functions

When a turbulence model with wall functions is selected, the near wall tangential velocity is assumed to satisfy the Law of the Wall, i.e.:

$$\frac{u_\tau}{u^*}=\frac{1}{\kappa}\ln\left(\frac{E}{1+0.3\frac{\rho_0 hu^*}{\mu}}\frac{\rho_0 yu^*}{\mu}\right) \tag{14}$$

where $h$ is the height of surface roughness, $u_\tau$ is the tangent to the wall velocity component in the nearest to the wall grid cell $y$ away from the Wall, such that, $y^+=\frac{yu^*}{\nu}$, $\kappa$=0.42, $E$=9.8, and $u^*=\sqrt{\frac{\tau_w}{\rho_o}}$, $\tau_w$ 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 $\alpha_w$ 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.

## 16.5 Radiation Models

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.

### 16.5.1 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 $\varepsilon_1$, $\varepsilon_2$ 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. $\varepsilon_1$ and $\varepsilon_2$ must be greater than zero.

### 16.5.2 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 Conditions 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.

### 16.5.3 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).

# 17 Solution Algorithms

Coolit uses the projection method (e.g. Dukowitz and Dvinsky, 1992) to solve the discretized governing equations.

## 17.1 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.

## 17.2 Projection Method

The projection method consists using the following fractional steps to iterate the governing equations:

$$\rho^n c_p \left[ \frac{T^{n+1} - T^n}{\Delta t} + V^n \cdot \nabla T^{n+1} \right] = -\nabla \cdot \lambda \nabla T^{n+1} + Q \quad (16)$$

$$\rho^n \left[ \frac{V^* - V^n}{\Delta t} + V^n \cdot \nabla V^* \right] = -\nabla p^n + \nabla \cdot 2 \mu D^* + \rho^n g \quad (17)$$

$$\nabla \cdot \Delta t \nabla \delta p - \nabla \cdot \rho^n V^* = 0 \quad (18)$$

$$V^{n+1} = V^* + \frac{\Delta t}{\rho^n} \nabla \delta p \quad (19)$$

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 and conjugate-gradient methods.

## 17.3 Convergence and Convergence Control

### 17.3.1 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.

### 17.3.2 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. |

# 18 Bibliography


Smagorinsky J. et al., Monthly Weather Rev., 93(12), 727-733, 1965.

Dacles-Mariani, J., Kwak, D., and Zilliac, G. G., Int. J. for Numerical Methods in Fluids, Vol. 30, 1999, pp. 65-82.

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.

Dukowitz J.K. and Dvinsky A.S., J. Comput. Phys. 102, 336, 1992.

Spalart P.R. and Allmaras S.R. AIAA-92-0439.

Turkel E., Fiterman A., and van Leer B., ICASE Report 93-42, Langley Research Center, 1993.

Travis J.R., Nat. Heat Transfer Conf., Denver, Colorado, Aug. 5-9, 1985.

# Index