2

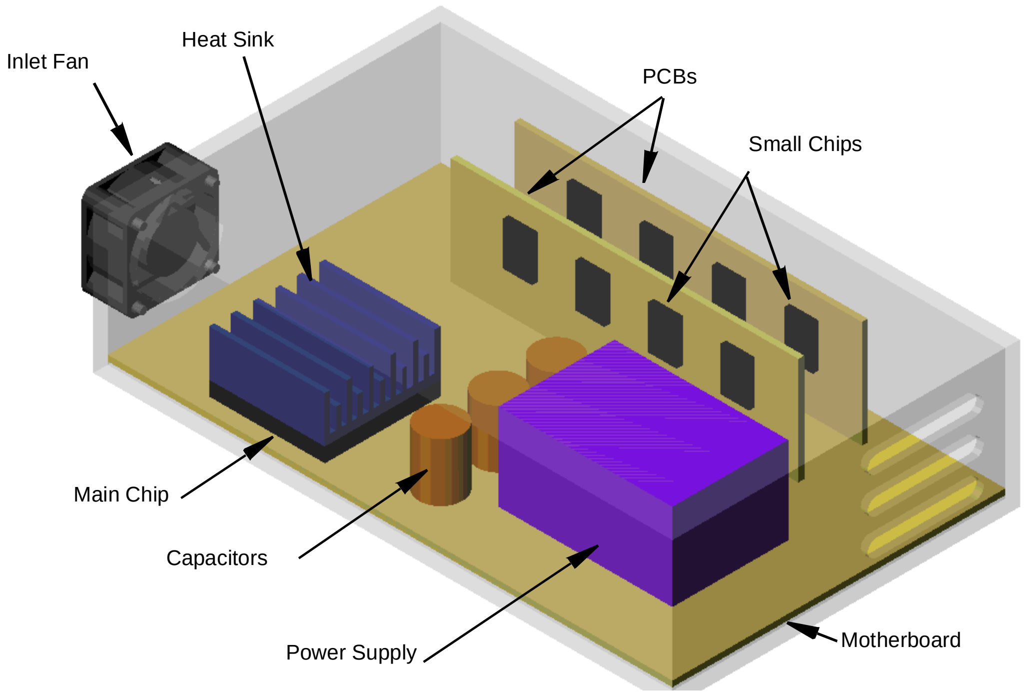
First Steps - Conjugate Heat Transfer

This First Steps - Conjugate Heat Transfer tutorial covers the basic steps required to set up a flow analysis problem including heat conduction in solids. This example is particularly pertinent to users interested in analyzing flow and heat conduction within electronics devices, although the basic principles are applicable to all thermal problems. It is assumed that you have already completed the **First Steps - Ball Valve Design** tutorial since it teaches the basic principles of using Flow Simulation in greater detail.

Open the SolidWorks Model

1 Copy the **First Steps - Electronics Cooling** folder into your working directory and ensure that the files are not read-only since Flow Simulation will save input data to these files. Click **File**, **Open**.

2 In the **Open** dialog box, browse to the `Enclosure Assembly.SLDASM` assembly located in the **First Steps - Electronics Cooling** folder and click **Open** (or double-click the assembly). Alternatively, you can drag and drop the `Enclosure Assembly.SLDASM` file to an empty area of SolidWorks window.



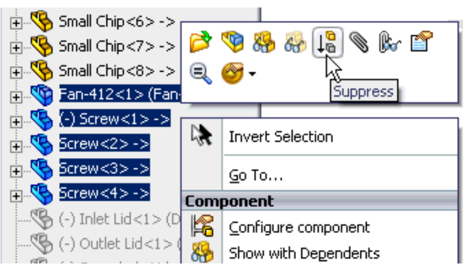
Preparing the Model

In a typical assembly there may be many features, parts or sub-assemblies that are not necessary for the analysis. Prior to creating a Flow Simulation project, it is a good practice to check the model to find components that can be removed from the analysis. Excluding these components reduces the computer resources and calculation time required for the analysis.

The assembly consists of the following components: enclosure, motherboard and two smaller PCBs, capacitors, power supply, heat sink, chips, fan, screws, fan housing, and lids. You can highlight these components by clicking them in the FeatureManager design tree. In this tutorial we will simulate the fan by specifying a **Fan** boundary condition on the inner face of the inlet lid. The fan has a very complex geometry that may cause delays while rebuilding the model. Since it is outside the enclosure, we can exclude it by suppressing it.

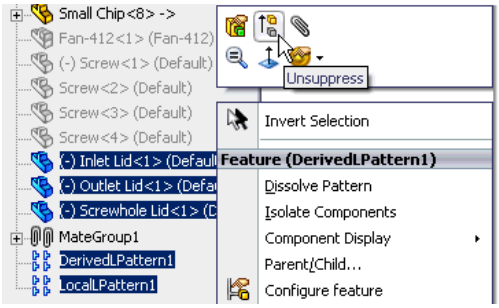
1 In the FeatureManager design tree, select the **Fan-412**, and all **Screw** components (to select more than one component, hold down the **Ctrl** key while you select).

2 Right-click any of the selected components and select **Suppress** .



Suppressing fan and its screws leaves open five holes in the enclosure. Since we are going to perform an internal analysis, all the holes must be closed with lids.

To save your time, we created the lids and included them to the model. You just need to unsupress them.



3 In the FeatureManager design tree, select the **Inlet Lid**, **Outlet Lid** and **Screwhole Lid** components and patterns DerivedLPattern1 and LocalLPattern1 (these patterns contain cloned copies of the outlet and screwhole lids).

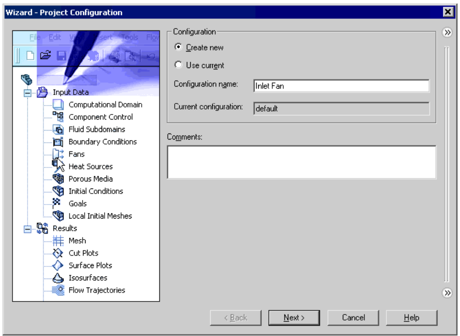
4 Right-click any of the selected components and select **Unsuppress** .

Now you can start with Flow Simulation.

Create a Flow Simulation Project

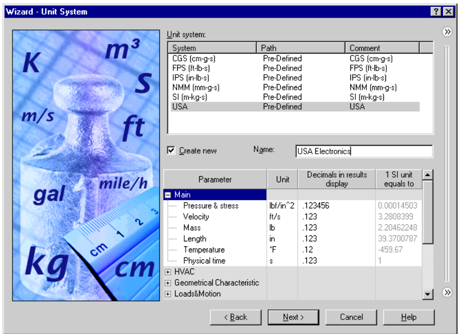
1 Click **Flow Simulation**, **Project**, **Wizard**.

2 Once inside the **Wizard**, select **Create new** in order to create a new configuration and name it `Inlet Fan`.



Click **Next**.
Now we will create a new system of units named **USA Electronics** that is better suited for our analysis.

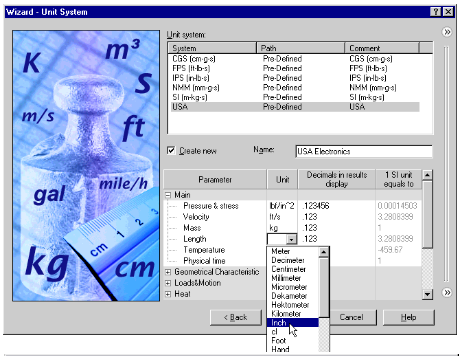
3 In the **Unit system** list select the **USA** system of units. Select **Create new** to add a new system of units to the Engineering Database and name it `USA Electronics`.



Flow Simulation allows you to work with several pre-defined unit systems but often it is more convenient to define your own *custom unit system. Both pre-defined and custom unit systems are stored in the* ***Engineering Database****. You can create the desired system of units in the* ***Engineering Database*** *or in the* ***Wizard****.*

📖 *By scrolling through the different groups in the* ***Parameter*** *tree you can see the units selected for the parameters. Although most of the parameters have convenient units such as ft/s for velocity and CFM (cubic feet per minute) for volume flow rate we will change a couple of units to that are more convenient for this model. Since the physical size of the model may be relatively small it is more convenient to choose inches instead of feet as the length unit.*

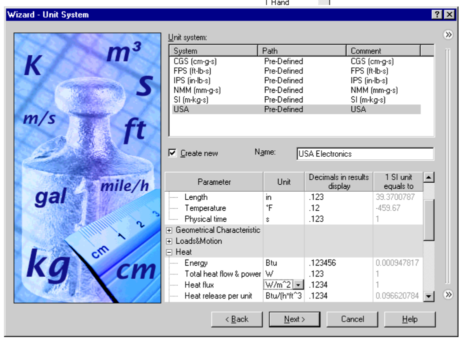
4 For the **Length** entry, double-click its cell in the **Unit** column and select **Inch**.



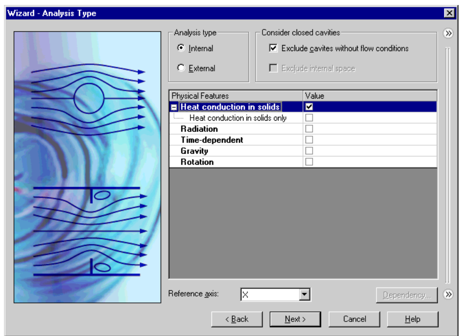
5 Next expand the **Heat** group in the **Parameter** tree.

Select **Watt**, **Watt/meter^2**, **Watt/meter^2/ Kelvin** as the units for **Total heat flow and power**, **Heat flux** and **Heat transfer coefficient** respectively, because these units are more convenient when dealing with electronic components .

Click **Next**.



6 Set the analysis type to **Internal**. Under **Physical Features** select the **Heat conduction in solids** check box.

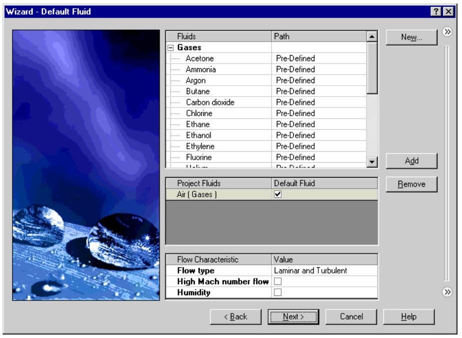


📖 *Heat conduction in solids is selected because heat is generated by several electronics components and we are interested to see how the heat is dissipated through the heat sink and other solid parts and then out to the fluid.*

Click **Next**.

7 Expand the **Gases** folder and double-click **Air**. Keep the default **Flow Characteristics**.

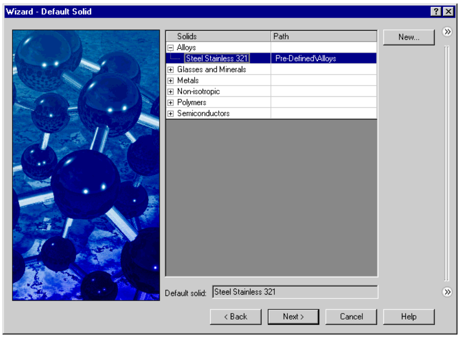
Click **Next**.



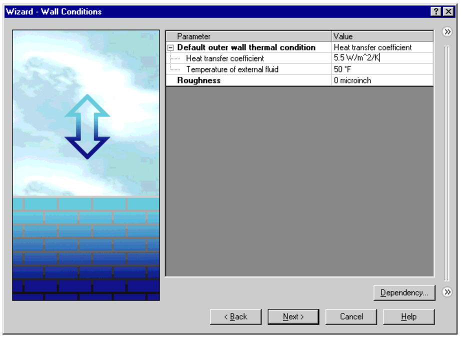
8 Expand the **Alloys** folder and click **Steel Stainless 321** to assign it as the **Default solid**.

In the ***Wizard*** *you specify the default solid material applied to all solid components in the Flow Simulation project. To specify a different solid material for one or more components, you can define a* ***Solid Material*** *condition for these components after the project is created.*

Click **Next**.



9 Select **Heat transfer coefficient** as **Default outer wall thermal condition** and specify the **Heat transfer coefficient** value of `5.5 W/m^2/K` and **Temperature of external fluid** of `50°F`. The entered value of heat transfer coefficient is automatically coverted to the selected system of units (USA Electronics).



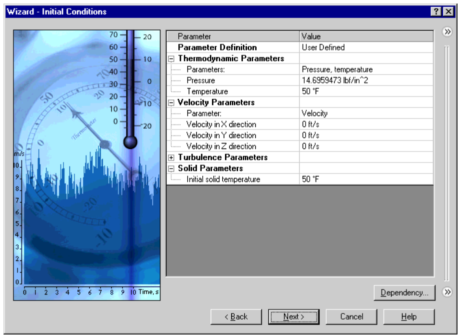
In the ***Wall Conditons*** *dialog box of the* ***Wizard*** *you specify the default conditions at the model walls. When* ***Heat conduction in solids*** *is enabled in an internal anlysis, the* ***Default outer wall thermal condition*** *parameter allows you to simulate heat exchange between the outer model walls and surrounding environment. In our case the box is located in an air-conditioned room with the air temperature of* 50°F *and heat transfer through the outer walls of the enclosure due to the convection in the room can significantly contribute to the enclosure cooling.*

Click **Next**.

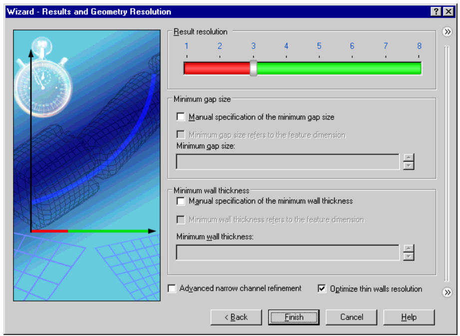
Although the initial temperature is more important for transient calculations to see how much time it takes to reach a certain temperature, in a steady-state analysis it is useful to set the initial temperature close to the expected final solution to speed up convergence. In this case we will set the initial air temperature and the initial temperature of the stainless steel (which represents the material of enclosure) to 50°F because the box is located in an air-conditioned room.

10 Set the initial fluid **Temperature** and the **Initial solid temperature** (under **Solid Parameters**) to `50°F`.

Click **Next**.



11 Accept the default **Result resolution** and keep the automatic evaluation of the **Minimum gap size** and **Minimum wall thickness**.

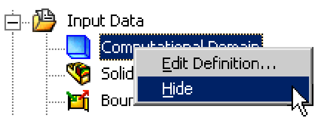


Flow Simulation calculates the default minimum gap size and minimum wall thickness using information about the overall model dimensions, the computational domain, and dimensions of faces on which you specify conditions and goals. Prior to starting the calculation, we recommend you to check the minimum gap size and minimum wall thickness to ensure that small features will be recognized. We will review this again after all the necessary conditions and goals are specified.

Click **Finish**. Now Flow Simulation creates a new configuration with the Flow Simulation project attached.

We will use the Flow Simulation Analysis tree to define our analysis, just as you use the FeatureManager design tree to design your models.

Right-click the **Computational Domain** icon and select **Hide** to hide the wireframe box.



Define the Fan

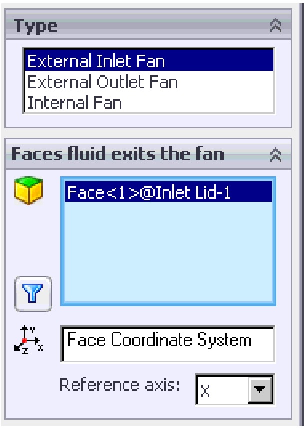
*A Fan is a type of flow boundary condition. You can specify **Fans** at selected solid surfaces, free of **Boundary Conditions** and **Sources**. At model openings closed by lids you can specify Inlet or Outlet Fans. You can also specify fans on any faces within the* *flow region as Internal Fans. A Fan is considered as an ideal device creating a flow with a certain volume (or mass) flow rate, which depends on the difference between the inlet and outlet pressures on the selected faces.*

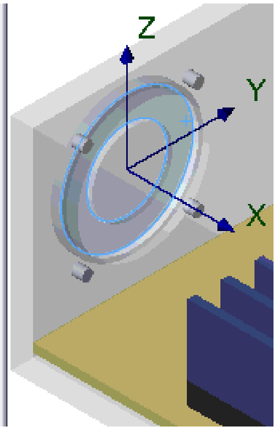
If you analyze a model with a fan, you should know the fan characteristics. In this example we use one of the pre-defined fans available in the **Engineering Database**. If you cannot find an appropriate fan in the Engineering Database, you can create your own fan in accordance with the fan specifications.

1 Click **Flow Simulation**, **Insert**, **Fan**. The **Fan** dialog box appears.

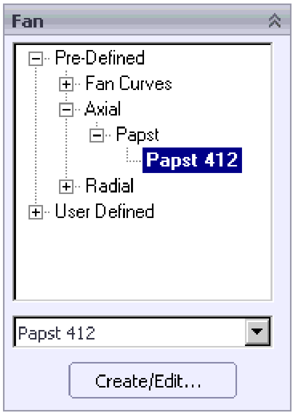
2 Select the inner face of the **Inlet Lid** part as shown. (To access the inner face, right-click the **Inlet Lid** in the graphics area and choose **Select Other**, move the pointer over items in the list of features until the inner face is highlighted, then click the left mouse button).

3 Under **Type**, select **External Inlet Fan**.



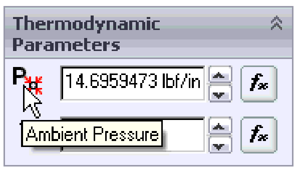


4 In the **Fan** list, under **Pre-Defined**, **Axial, Papst**, select the **Papst 412** item.



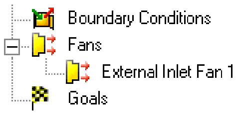
5 Under **Thermodynamic Parameters** check that the **Ambient Pressure** is the atmospheric pressure.

6 Accept **Face Coordinate System** as the reference **Coordinate system** and **X** as the **Reference axis**.

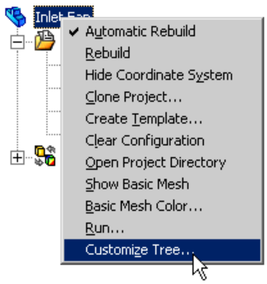


📖 ***Face coordinate system** is created automatically in the center of a planar face when you select this face as the face to apply the boundary condition or fan. The **X** axis of this coordinate system is normal to the face. The **Face coordinate system** is created only when one planar face is selected.*

7 Click **OK** ✓. The new **Fans** folder and the **External Inlet Fan 1** item appear in the Flow Simulation Analysis tree.



📖 *Now you can edit the **External Inlet Fan 1** item or add a new fan using Flow Simulation Analysis tree. This folder remains visible until the last feature of this type is deleted. You can also make a feature folder to be initially available in the tree. Right-click the project name item and select **Customize Tree** to add or remove folders.*

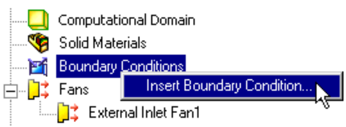


📖 *Since the outlet lids of the enclosure are at ambient atmospheric pressure, the pressure rise produced by the fan is equal to the pressure drop through the electronics enclosure.*

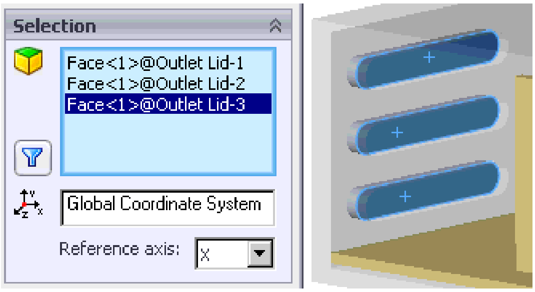
Define the Boundary Conditions

A boundary condition is required in any place where fluid enters or exits the model, excluding openings where a fan is specified. A boundary condition can be set in form of **Pressure, Mass Flow Rate**, **Volume Flow Rate** or **Velocity**. You can also use the **Boundary Condition** dialog for specifying an **Ideal Wall** condition that is an adiabatic, frictionless wall or a **Real Wall** condition to set the wall roughness and/or temperature and/or heat conduction coefficient at the selected model surfaces. For internal analyses with **Heat conduction in solids** enabled, you can also set thermal wall condition on outer model walls by specifying an **Outer Wall** condition.

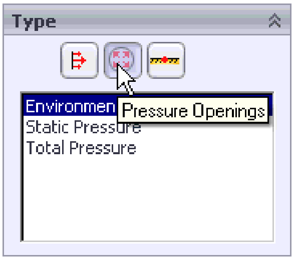
1 In the Flow Simulation analysis tree right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.



2 Select the inner faces of all outlet lids as shown.



3 Select **Pressure Openings** and **Environment Pressure**.



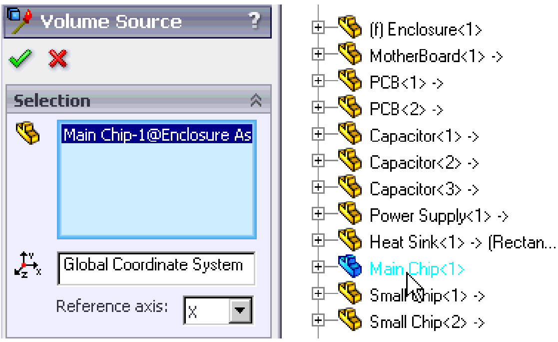
4 Keep the defaults under **Thermodynamic Parameters**, **Turbulence Parameters**, **Boundary Layer** and **Options**.Click **OK**. The new **Environment Pressure 1** item appears in the Flow Simulation Analysis tree.

The Environment pressure condition is interpreted as a static pressure for outgoing flows and as a total pressure for incoming flows.

Define Heat Sources

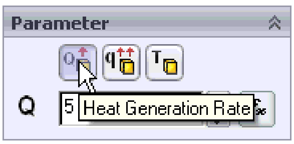
1 Click **Flow Simulation**, **Insert, Volume Source**.

2 Select the **Main Chip** from the flyout FeatureManager design tree tree to add it to the **Components to Apply the Volume Source** list.



3 Select the **Heat Generation Rate** as **Parameter**.

4 Enter 5 W in the **Heat Generation Rate** Q box.



5 Click **OK**.

6 In the Flow Simulation Analysis tree, click-pause-click the new **VS Heat Generation Rate 1** item and rename it to `Main Chip`.

📖 *Volume Heat Sources allow you to specify the heat generation rate (e.g. in Watts) or the volumetric heat generation rate (e.g. in Watts per volume) or a constant temperature boundary condition for a volume. It is also possible to specify Surface Heat Sources in terms of heat transfer rate (e.g. in Watts) or heat flux (e.g. in Watts per area).*

Click anywhere in the graphic area to clear the selection.

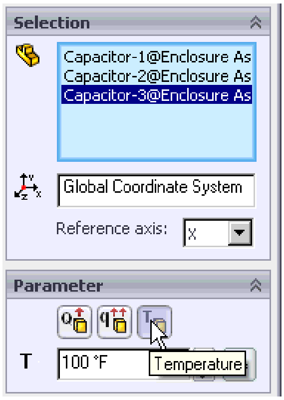
1 In the Flow Simulation analysis tree, right-click the **Heat Sources** icon and select **Insert Volume Source**.

2 In the flyout FeatureManager design tree, select all **Capacitor** components.

3 Select the **Temperature** as **Parameter** and enter 100 °F in the **Temperature** T box.

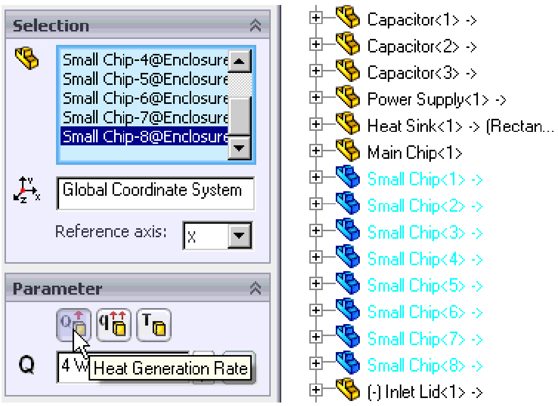
4 Click **OK**.

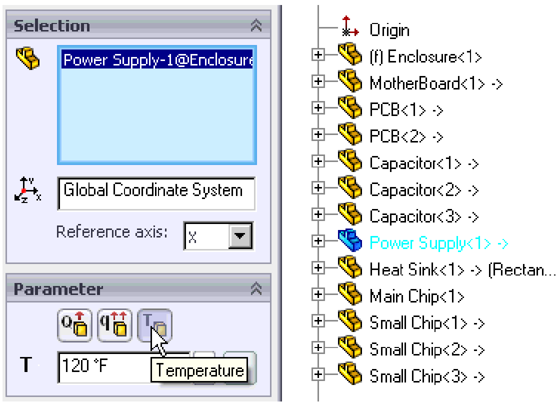
5 Click-pause-click the new **VS Temperature 1** item and rename it to `Capacitors`.



Click anywhere in the graphic area to clear the selection.

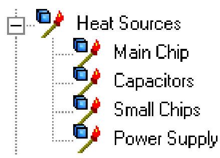
6 Following the same procedure as described above, set the volume heat sources: all chips on PCBs (**Small Chip** components) with the total heat generation rate of 4 W, **Power Supply** with the temperature of 120 °F.





7 Rename the source applied to the chips to `Small Chips` and the source for the power supply to `Power Supply`.

Click **File**, **Save**.



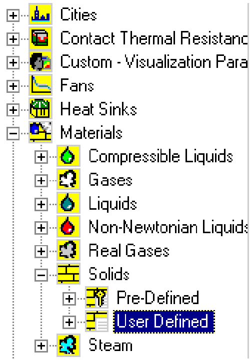
Create a New Material

The real PCBs are made of laminate materials consisting of several layers of thin metal conductor interleaved with layers of epoxy resin dielectric. As for most laminate materials, the properties of a typical PCB material can vary greatly depending on the direction - along or across the layers, i.e. it is anisotropic. The Engineering Database contains some predefined PCB materials with anisotropic thermal conductivity.

In this tutorial example anisotropic thermal conductivity of PCBs does not affect the overall cooling performance much, so we will create a PCB material having the same thermal conductivity in all directions to learn how to add a new material to the Engineering Database and assign it to a part.

1 Click **Flow Simulation**, **Tools**, **Engineering Database**.

2 In the **Database tree** select **Materials**, **Solids**, **User Defined**.



3 Click **New Item** on the toolbar.

The blank **Item Properties** tab appears. Double-click the empty cells to set the corresponding properties values.

4 Specify the material properties as follows:

Name = `Tutorial PCB`,
Comments = `Isotropic PCB`,
Density = `1120 kg/m^3`,
Specific heat = `1400 J/(kg*K)`,
Conductivity type = `Isotropic`
Thermal conductivity = `10 W/(m*K)`,
Melting temperature = `390 K`.

Property	Value
Name	Tutorial PCB
Comments	Isotropic PCB
Density	1120 kg/m^3
Specific heat	1400 J/(kg*K)
Conductivity type	Isotropic
Thermal conductivity	10 W/(m*K)
Melting temperature	390 K

We also need to add a new material simulating thermal conductivity and other thermal properties of electronic components.

5 Switch to the **Items** tab and click **New Item** on the toolbar.Specify the properties of the chips material:

Name = `Tutorial component package`,
Comments = `Component package`,
Density = `2000 kg/m^3`,
Specific heat = `120 J/(kg*K)`,
Conductivity type = `Isotropic`
Thermal conductivity = `0.4 W/(m*K)`,
Melting temperature = `390 K`.

Property	Value
Name	Tutorial component package
Comments	Component package
Density	2000 kg/m^3
Specific heat	120 J/(kg*K)
Conductivity type	Isotropic
Thermal conductivity	0.4 W/(m*K)
Melting temperature	390K

6 Click **Save** .

7 Click **File**, **Exit** to exit the database.

You can enter the material properties in any unit system you want by typing the unit name after the value and Flow Simulation will automatically convert the entered value to the SI system of units. You can also specify temperature-dependent material properties using the **Tables and Curves** *tab.*

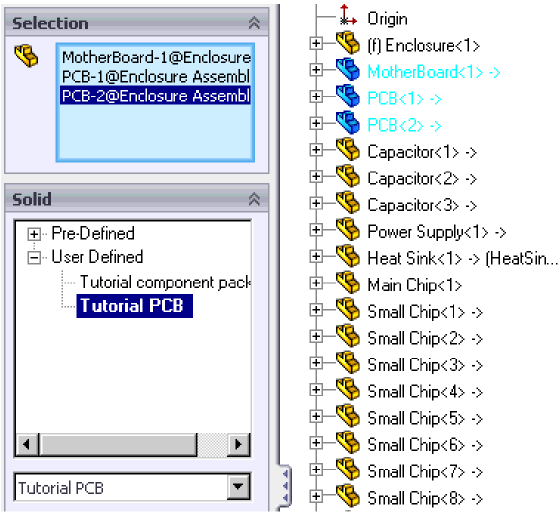
Define Solid Materials

Solid Materials are used to specify the materials for solid parts in the assembly.

1 In the Flow Simulation analysis tree, right-click the **Solid Materials** icon and select **Insert Solid Material**.

2 In the flyout FeatureManager design tree, select the **MotherBoard**, **PCB<1>** and **PCB<2>** components.

3 In the **Solid** list, expand **User Defined** and select **Tutorial PCB**.



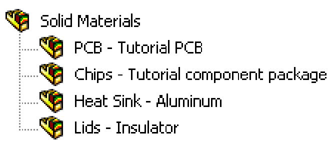
4 Click **OK**  .

5 Following the same procedure, specify solid materials for other components:

- for the **Main Chip** and all **Small Chips** assign the new **Tutorial component package** material (available under **User Defined**);
- the **Heat Sink** is made of **Aluminum** (available under **Pre-Defined**, **Metals**);
- the lids (**Inlet Lid**, **Outlet Lid**, **Screwhole Lid** and all lids in both the **DerivedLPattern1** and **LocalLPattern1** patterns) are made of the **Insulator** material (available under **Pre-Defined**, **Glasses and Minerals**).

To select a part, click it in the FeatureManager design tree or SolidWorks graphics area.

6 Change the name of each assigned solid material. The new, descriptive names should be:
`PCB - Tutorial PCB`,
`Chips - Tutorial component package`,
`Heat Sink - Aluminum`,
`Lids - Insulator`.



Click **File**, **Save**.

Define Engineering Goals

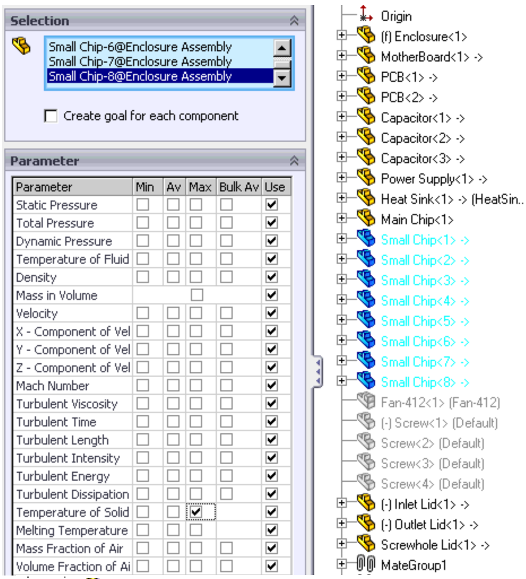
Specifying Volume Goals

1 In the Flow Simulation analysis tree, right-click the **Goals** icon and select **Insert Volume Goals**.

2 In the flyout FeatureManager design tree select all **Small Chip** components.

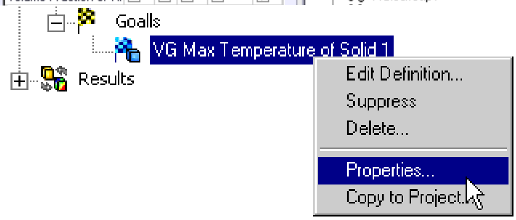
3 In the **Parameter** table, select the **Max** check box in the **Temperature of Solid** row.

4 Accept selected **Use for Conv.** (**Use for Convergence Control**) check box to use this goal for convergence control.



5 Click **OK** . The new **VG Max Temperature of Solid 1** item appears in the Flow Simulation Analysis tree.

6 Change the name of the new item to `VG Small Chips Max Temperature`. You can also change the name of the item using the **Feature Properties** dialog that appears if you right-click the item and select **Properties**.



7 Right-click the **Goals** icon and select **Insert Volume Goals**.

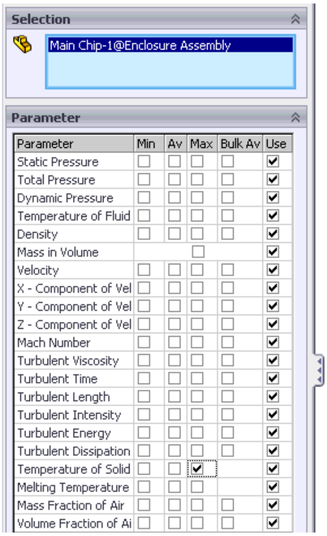
8 Select the **Main Chip** item in the flyout FeatureManager design tree.

9 In the **Parameter** table, select the **Max** check box in the **Temperature of Solid** row.

10 Click **OK** ✔.

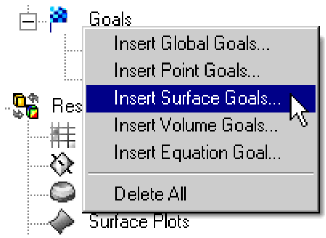
11 Rename the new **VG Max Temperature of Solid 1** item to `VG Chip Max Temperature`.

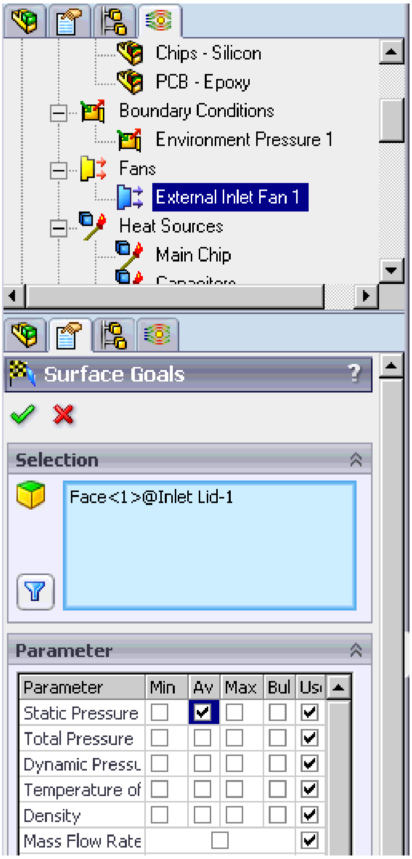
Click anywhere in the graphic area to clear the selection.



Specifying Surface Goals

1 Right-click the **Goals** icon and select **Insert Surface Goals**.



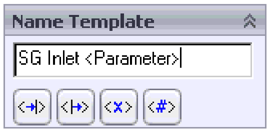


2 Click the Flow Simulation Analysis Tree tab and click the **External Inlet Fan 1** item to select the face where the goal is going to be applied.

3 In the **Parameter** table select the **Av** check box in the **Static Pressure** row.

4 Accept selected **Use for Conv.** (**Use for Convergence Control**) check box to use this goal for convergence control.

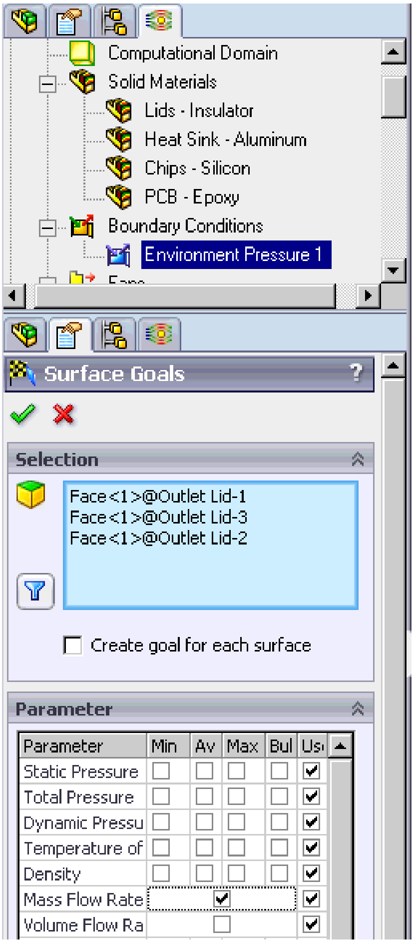
For the X(Y, Z) - Component of Force and X(Y, Z) - Component of Torque surface goals you can select the Coordinate system in which these goals will be calculated.



5 Under **Name Template**, located at the bottom of the PropertyManager, click **Inlet**  and then remove the `<Number>` field from the **Name Template** box.

6 Click **OK**  . The new **SG Inlet Av Static Pressure** goal appears.

Click anywhere in the graphic area to clear the selection.



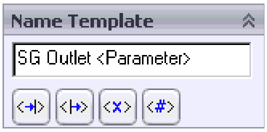
7 Right-click the **Goals** icon and select **Insert Surface Goals**.

8 Click the Flow Simulation Analysis Tree tab and click the **Environment Pressure 1** item to select the faces where the goal is going to be applied.

9 In the **Parameter** table select the first check box in the **Mass Flow Rate** row.

10 Accept selected **Use for Conv.** (**Use for Convergence Control**) check box to use this goal for convergence control.

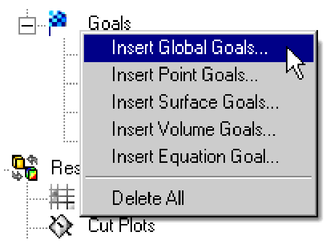
11 Under **Name Template**, located at the bottom of the PropertyManager, click **Outlet**  and then remove the `<Number>` field from the **Name Template**.



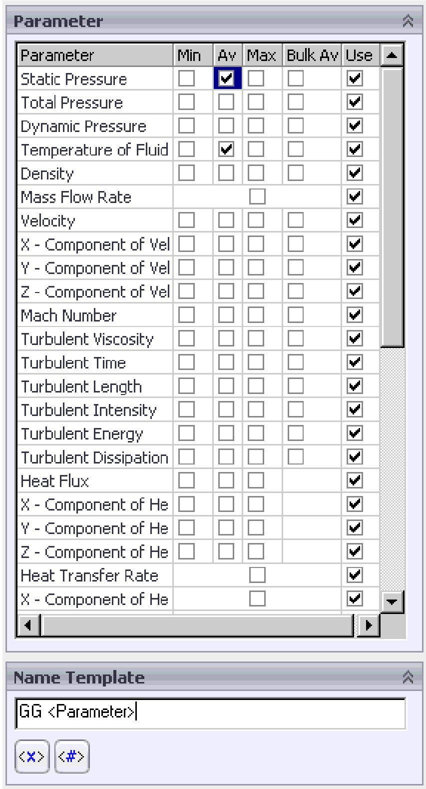
12 Click **OK**  . The **SG Outlet Mass Flow Rate** goal appears.

Specifying Global Goals

1 Right-click the **Goals** icon and select **Insert Global Goals**.



2 In the **Parameter** table select the **Av** check boxes in the **Static Pressure** and **Temperature of Fluid** rows and accept selected **Use for Conv.** (**Use for Convergence Control**) check box to use these goals for convergence control.



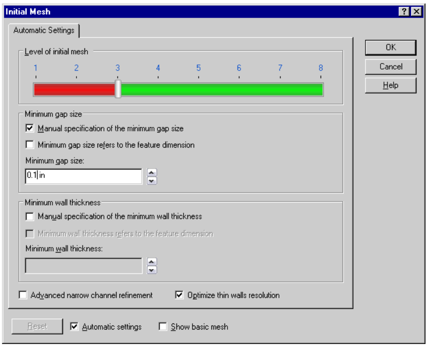
3 Remove the `<Number>` field from the **Name Template** and click **OK**  .The **GG Av Static Pressure** and **GG Av Temperature of Fluid** goals appear.

In this tutorial, the engineering goals are set to determine the maximum temperature of the heat generating components, the temperature rise in air and the pressure drop and mass flow rate through the enclosure.

Click **File**, **Save**.

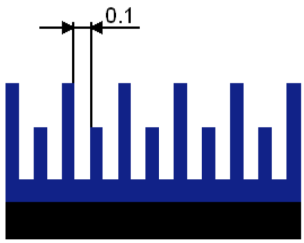
Next let us check the automatically defined geometry resolution settings for this project.

Changing the Geometry Resolution



1 Click **Flow Simulation**, **Initial Mesh**.

2 Select the **Manual specification of the minimum gap size** check box.



3 Enter `0.1 in` for the **Minimum gap size** (i.e. passage between the fins of the heat sink).

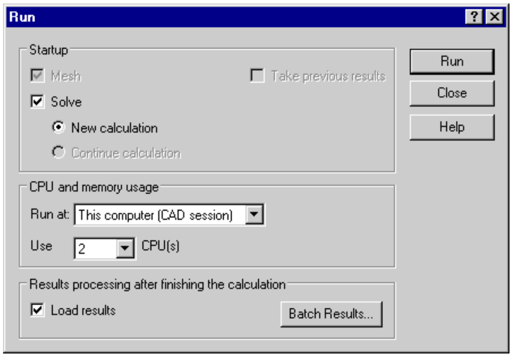
📖 *Entering values for the minimum gap size and minimum wall thickness is important when you have small features. Setting these values accurately ensures that the small features are not "passed over" by the mesh. The minimum wall thickness should be specified only if there are fluid cells on either side of a small solid feature. In case of internal analyses, there are no fluid cells in the ambient space outside of the enclosure. Therefore boundaries between internal flow and ambient space are always resolved properly. That is why you should not take into account the walls of the steel cabinet. Both the* ***minimum gap size*** *and the* ***minimum wall thickness*** *are tools that help you to create a model-adaptive mesh resulting in increased accuracy. However the minimum gap size setting is the more powerful one. The fact is that the Flow Simulation mesh is constructed so that the specified Level of initial mesh controls the minimum number of mesh cells per* ***minimum gap size****. And this number is equal to or greater than the number of mesh cells generated per* ***minimum wall thickness****. That's why even if you have a thin solid feature inside the flow region it is not necessary to specify minimum wall thickness if it is greater than or equal to the minimum gap size. Specifying the minimum wall thickness is necessary if you want to resolve thin walls smaller than the smallest gap.*

Click **OK**.

Solution

1 Click **Flow Simulation**, **Solve**, **Run**.

2 Click **Run**.

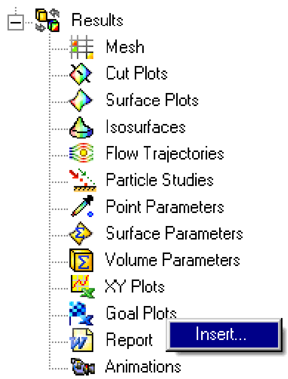


The solver takes about twenty to thirty minutes to run on a typical PC.

You may notice that different goals take different number of iterations to converge.
The goal-oriented philosophy of Flow Simulation allows you to get the answers you need in the shortest amount of time.
For example, if you were only interested in the temperature of fluid in the enclosure, Flow Simulation would have provided the result more quickly then if the solver was allowed to fully converge on all of the parameters.

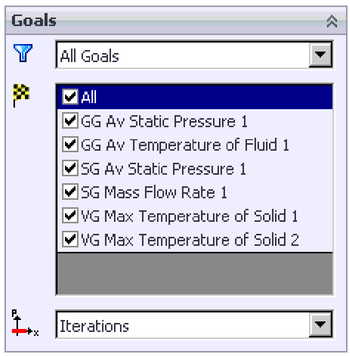
Viewing the Goals

1 Right-click the **Goal Plots** icon under **Results** and select **Insert**.



2 Click **All** in the **Goals** dialog.

3 Click **OK**.



An Excel spreadsheet with the goal results will be open. The first sheet will show a table summarizing the goals.

Enclosure Assembly.SLDASM [Inlet Fan]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence
GG Av Static Pressure	[lbf/in^2]	14.69678696	14.69678549	14.69678314	14.69678772	100	Yes
SG Inlet Av Static Pressure	[lbf/in^2]	14.69641185	14.69641047	14.69640709	14.69641418	100	Yes
GG Av Temperature of Fluid	[°F]	61.7814683	61.76016724	61.5252449	61.86764155	100	Yes
SG Outlet Mass Flow Rate	[lb/s]	-0.007306292	-0.007306111	-0.007306913	-0.007303663	100	Yes
VG Small Chips Max Temp	[°F]	91.5523903	90.97688632	90.09851988	91.5523903	100	Yes
VG Chip Max Temperature	[°F]	88.51909612	88.43365626	88.29145322	88.57515562	100	Yes

You can see that the maximum temperature in the main chip is about 89 °F, and the maximum temperature over the small chips is about 92 °F.

🕮 *Goal progress bar is a qualitative and quantitative characteristic of the goal convergence process. When Flow Simulation analyzes the goal convergence, it calculates the goal dispersion defined as the difference between the maximum and minimum goal values over the analysis interval reckoned from the last iteration and compares this dispersion with the goal's convergence criterion dispersion, either specified by you or automatically determined by Flow Simulation as a fraction of the goal's physical parameter dispersion over the computational domain. The percentage of the goal's convergence criterion dispersion to the goal's real dispersion over the analysis interval is shown in the goal's convergence progress bar (when the goal's real dispersion becomes equal or smaller than the goal's convergence criterion dispersion, the progress bar is replaced by word "Achieved"). Naturally, if the goal's real dispersion oscillates, the progress bar oscillates also, moreover, when a hard problem is solved, it can noticeably regress, in particular from the "achieved" level. The calculation can finish if the iterations (in travels) required for finishing the calculation have been performed, or if the goal convergence criteria are satisfied before performing the required number of iterations. You can specify other finishing conditions at your discretion.*

To analyze the results in more detail let us use the various Flow Simulation results processing tools. The best method for the visualization of how the fluid flows inside the enclosure is to create flow trajectories.

Flow Trajectories

1 Right-click the **Flow Trajectories** icon and select **Insert**.

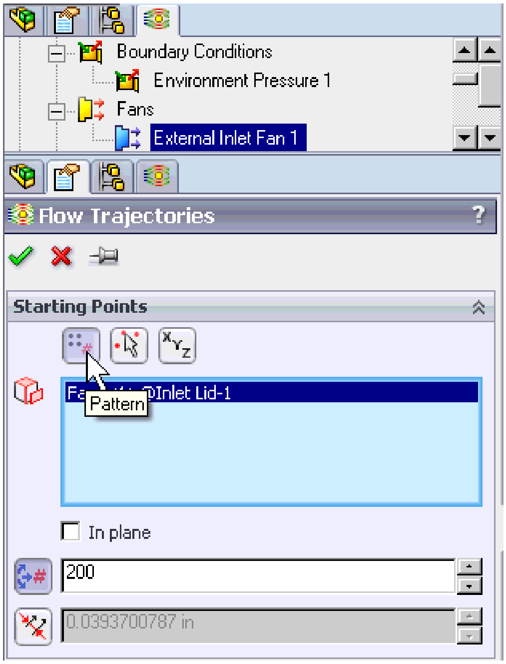
2 Click the Flow Simulation Analysis Tree tab and then click the **External Inlet Fan1** item to select the inner face of the **Inlet Lid**.

3 Set the **Number of Points** to `200`.

4 Under **Appearance**, set **Draw Trajectories as** **Bands**.

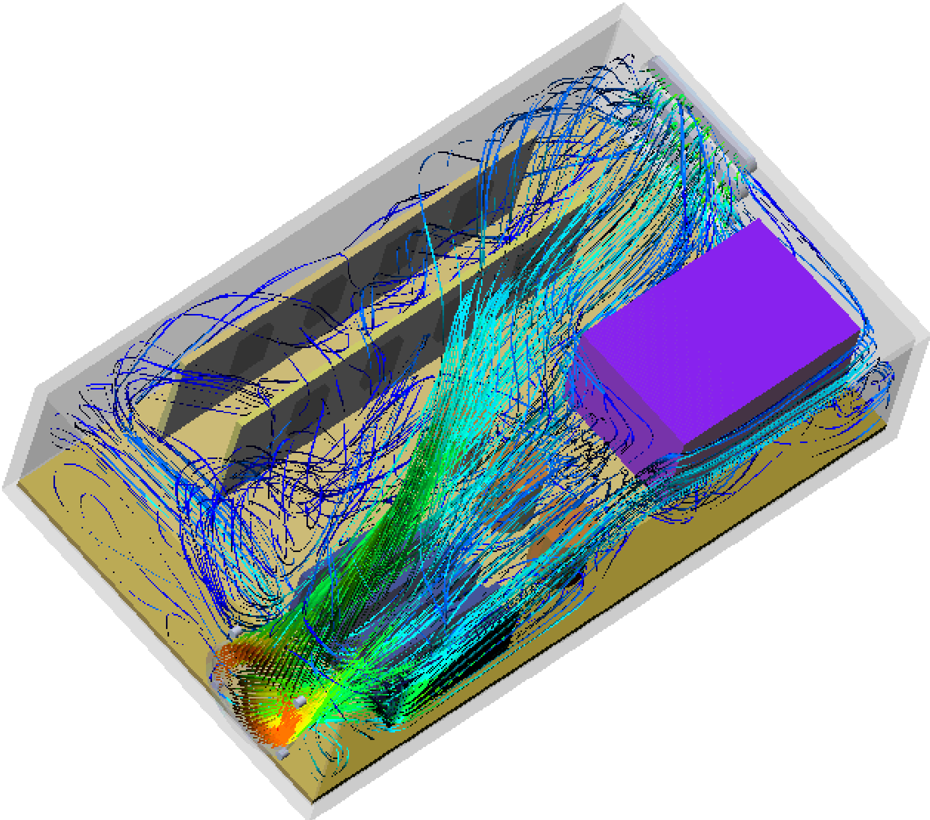
5 Make sure that **Color by Parameter** is selected and then change the parameter to **Velocity**.

If **Color by Parameter** *is selected, then the trajectories are colored in accordance with the distribution of the parameter specified. If you select* **Color** *then all flow trajectories will have a fixed color specified by you.*



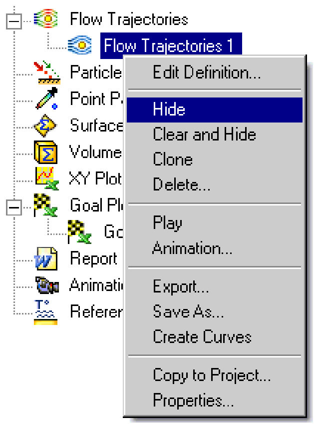
6 Click **OK** . The new **Flow Trajectories 1** item appears in the Flow Simulation Analysis tree.

This is the picture you should see.



Notice that there are only a few trajectories along the adjacent to the wall **PCB<2>** and this may cause problems with cooling of the chips placed on this PCB. Additionally the blue color indicates low velocity in front of this **PCB<2>** .

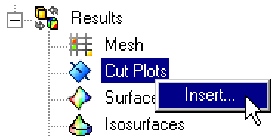
Right-click the **Flow Trajectories 1** item and select **Hide**.



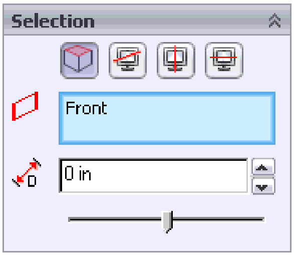
Click anywhere in the graphic area to clear the selection. Let us now examine the velocity distribution in more detail.

Cut Plots

1 Right-click the **Cut Plots** icon and select **Insert**.



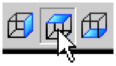
2 Set the **Front** plane as the section plane.

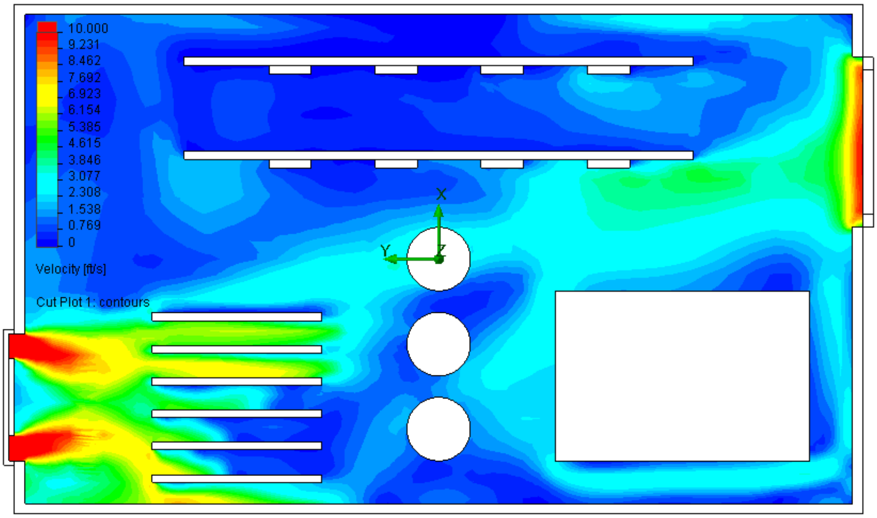


3 Under **Contours**, select **Adjust Minimum and Maximum** . Change the **Min** and **Max** values to `0` and `10 ft/s` respectively. The specified values produce a palette where it is easier to determine the value.

4 Set the **Number of levels** to `30`.

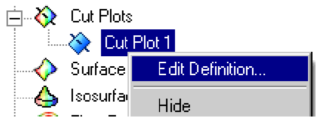
5 Click **OK** . The new **Cut Plot 1** item appears in the Flow Simulation Analysis tree.

6 Select the **Top** view  on the **Standard Views** toolbar.

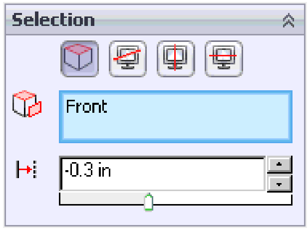


Let us now look at the fluid temperature.

7 Right-click the **Cut Plot 1** icon and select **Edit Definition**.

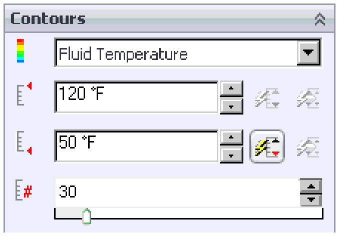


8 Change the **Offset** to `-0.3 in`.



9 Change the **Parameter** from **Velocity** to **Fluid Temperature**.

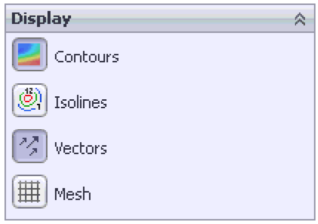
10 Change the **Min** and **Max** values to `50` and `120 F` respectively.



11 Under **Display**, select **Vectors**.

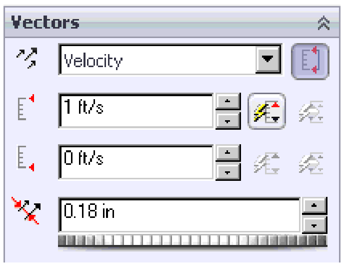
12 Under the appeared **Vectors** tab, make sure that the **Parameter** is set to **Velocity** and then select **Adjust Minimum and Maximum**.

13 Set the **Max** value to `1 ft/s`.

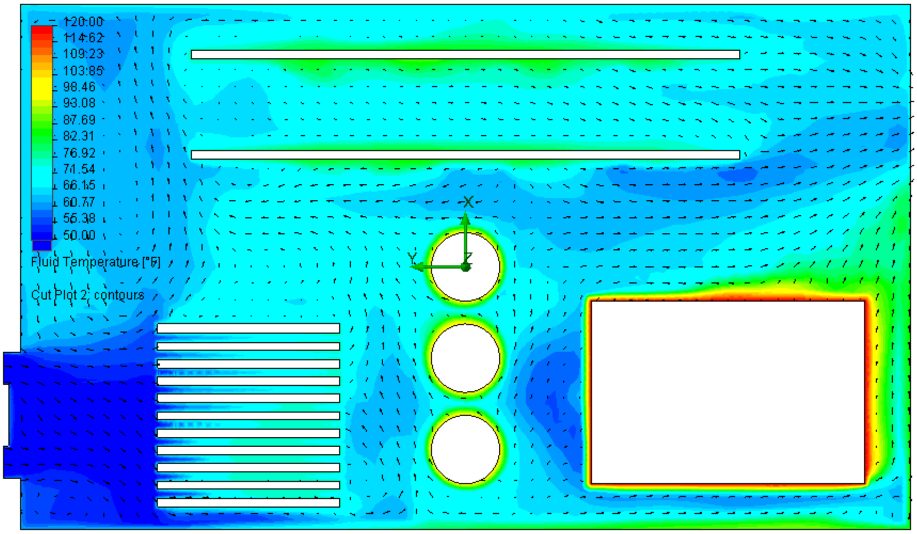


*By specifying the custom **Min** and **Max** values you can control the vector length. The vectors whose velocity exceeds the specified Max value will have the same length as the vectors whose velocity is equal to Max. Likewise, the vectors whose velocity is less than the specified Min value will have the same length as the vectors whose velocity is equal to Min. We have set 1 ft/s to display areas of low velocity.*

14 Change the **Vector Spacing** to 0.18 in.



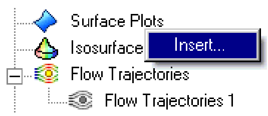
15 Click **OK** .



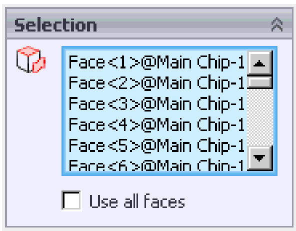
Right-click the **Cut Plot 1** item and select **Hide**. Let us now display solid temperature.

Surface Plots

1 Right-click the **Surface Plots** item and select **Insert**.

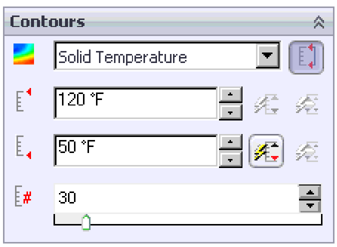


2 In the flyout FeatureManager design tree click the **Main Chip**, **Heat Sink** and all **Small Chip** components to select their surfaces.



3 Under **Contours**, change the **Parameter** to **Solid Temperature**.

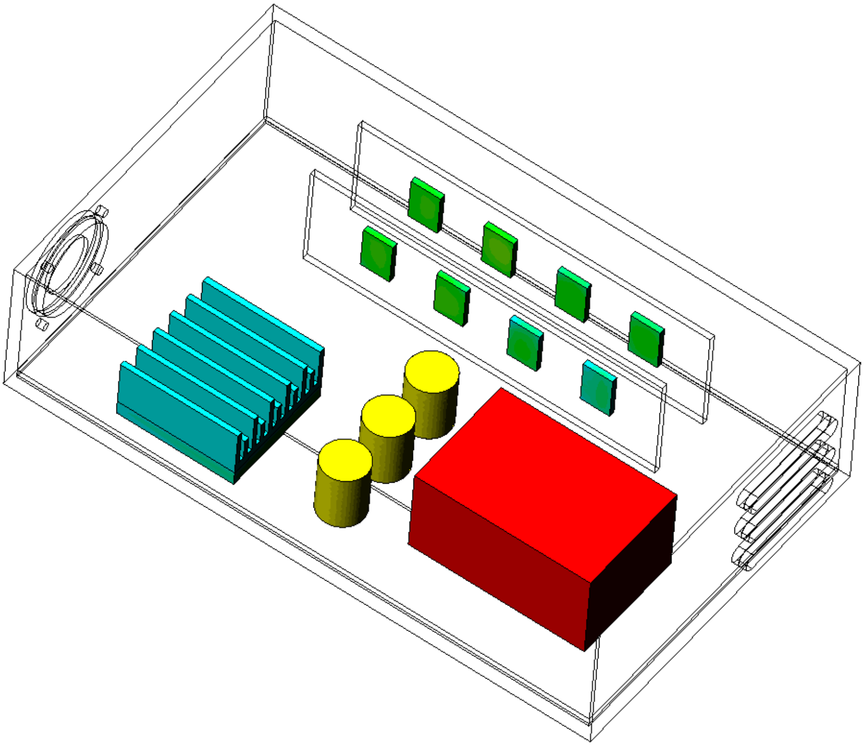
4 Change the **Min** and **Max** values to `50` and `120 F` respectively.



5 Click **OK** .

6 Repeat steps 1 and 2 and select the **Power Supply** and all **Capacitor**s components, then click **OK** .

7 On the **View** toolbar click **Wireframe**  to show only the face outlines.



You can view and analyze the results further with the post-processing tools that were shown in the **First Steps - Ball Valve Design** tutorial. Flow Simulation allows you to quickly and easily investigate your design both quantitatively and qualitatively. Quantitative results such as the maximum temperature in the component, pressure drop through the cabinet, and air temperature rise will allow you to determine whether the design is acceptable or not. By viewing qualitative results such as air flow patterns, and heat conduction patterns in the solid, Flow Simulation gives you the necessary insight to locate problem areas or weaknesses in your design and provides guidance on how to improve or optimize the design.