2

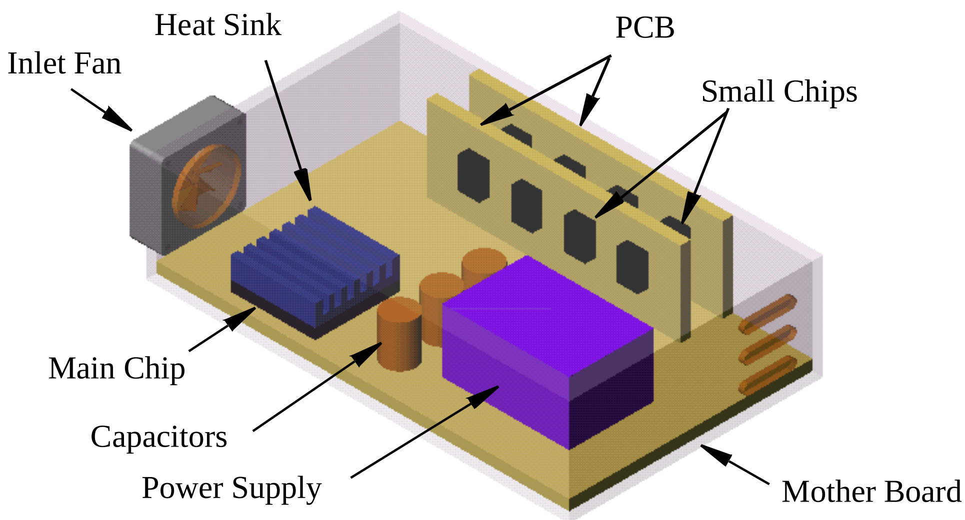
First Steps - Conjugate Heat Transfer

This First Steps - Conjugate Heat Transfer tutorial covers the basic steps to set up a flow analysis problem including conduction heat transfer in solids. This example is particularly pertinent to users interested in analyzing flow and heat transfer within electronics packages 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 COSMOSFloWorks 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 COSMOSFloWorks 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.

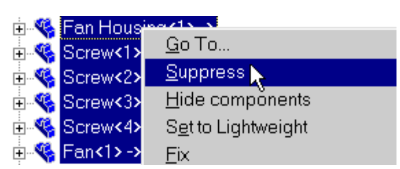


Preparing the Model

In the analysis of an assembly there may be many features, parts or sub-assemblies that are not necessary for the analysis. Prior to using COSMOSFloWorks, it is good practice to check the model to single out components that will not be involved in the analysis. Excluding these components decreases the required computer resources and calculation time.

The assembly consists of the following components: enclosure, MotherBoard and PCBs, capacitors, power supply, heat sink, chips, fan, screws, fan housing, and lids. You can view these components by clicking on the features in the SolidWorks Feature Manager. 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 very complex geometry that may cause delays while rebuilding the model. Since it is outside of the enclosure we can exclude it to hasten operations with SolidWorks.

1 In the FeatureManager, select the **Fan**, **Screws** and **Fan Housing** components (to select more than one component, hold down the **Ctrl** key while you select).



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

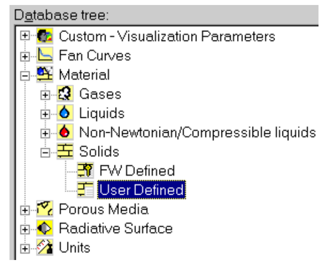
Now you can start with COSMOSFloWorks.

Create a New Material

The chips are made of Epoxy but Epoxy is not a default material in the COSMOS-FloWorks Engineering database so we must create it.

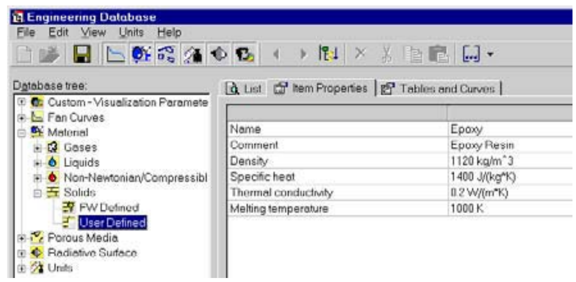
1 Click **FloWorks**, **Tools**, **Engineering Database**.

2 In the **Database tree** select **Material**, **Solids**, **User Defined**. Click **New Item** on the toolbar. The blank **Item Properties** tab appears. Double-click the empty cell to set the corresponding property value.



3 Specify the material properties as follows:

Name = `Epoxy`,
Comment = `Epoxy Resin`,
Density = `1120 kg/m3`,
Specific Heat = `1400 J/kgK`,
Thermal Conductivity = `0.2 W/mK`,
Melting Temperature = `1000 K`.



4 Click **Save**.

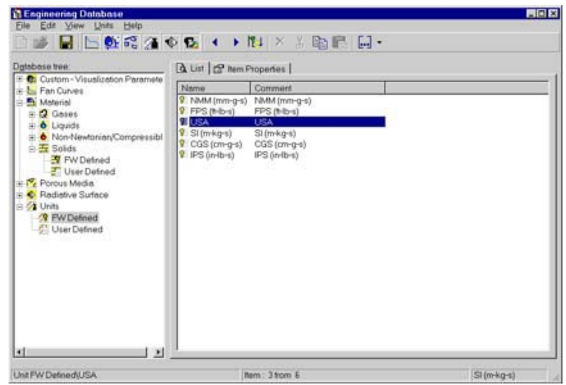
*You can enter the material properties in any unit system you want by typing the unit name after the value and COSMOSFloWorks will automatically convert the value to metric. You can also enter material properties that are temperature dependent using the **Tables and Curves** tab.*

Define a Custom Unit System

COSMOSFloWorks 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. Prior to starting the project Wizard you can create the desired system of units in the Engineering Database. To adjust the selected system of units after passing the Wizard you can use either Engineering Database or click **FloWorks**, **Units**. In this example we will adjust a pre-defined system of units before creating the project.

1 In the **Database tree** select **Units**, **FW Defined**.

2 On the **List** tab select the **USA** system of units and click **Copy** .



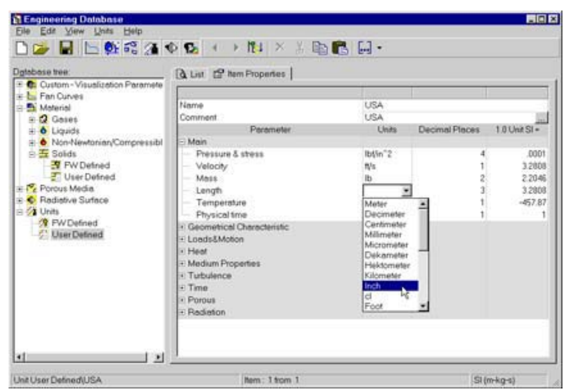
*You can modify only custom entries in the Engineering Database. To adjust the pre-defined material, porous media, unit system or fan curve you must copy it into the corresponding **User Defined** folder first and then make necessary changes.*

3 In the tree, select the **Units**, **User Defined** item and click **Paste** .

4 Click the **Item Properties** tab to adjust the USA unit system for this example.

*By scrolling through the different groups in the **Parameter** tree you can see the units selected for all 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 units that are more convenient for this model. Since the physical size of the model is relatively small it is more convenient to choose inches instead of feet as the length unit.*

5 For the **Length** entry, click on the right hand side of the units box and select inches.

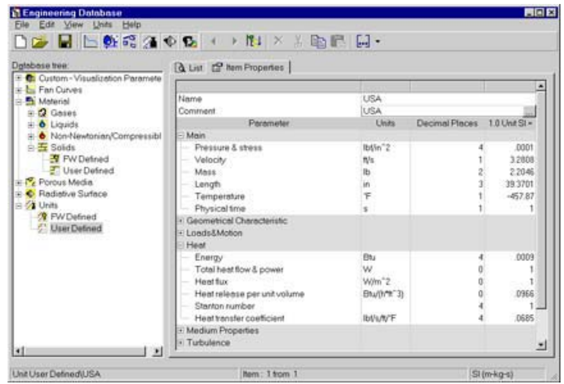


6 Next expand the **Heat** group in the **Parameter** tree.

7 Since we are dealing with electronic components it is more convenient to specify power and heat flux in **Watts** and **Watts/m^2** respectively.

8 Name the new system of units `USA Electronics`.

Click **Save** .



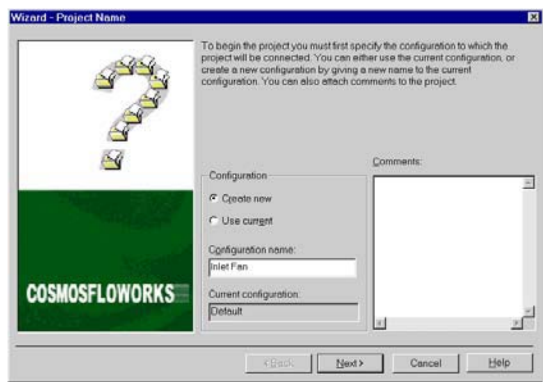
9 Close the **Engineering Database** by clicking **File**, **Exit**.

Create a COSMOSFloWorks Project

1 Click **FloWorks**, **Project**, **Wizard.**

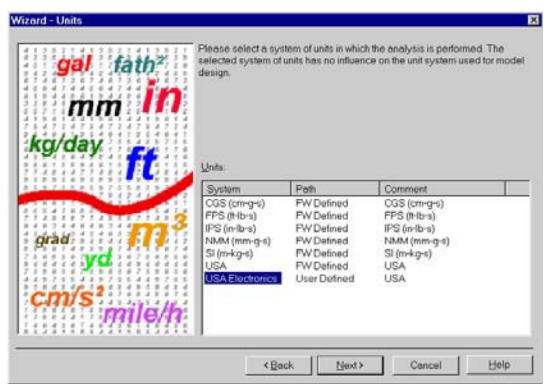
2 Once inside the **Wizard**, select **Create new** in order to create a new configuration and name it `Inlet Fan`.

Click **Next**.

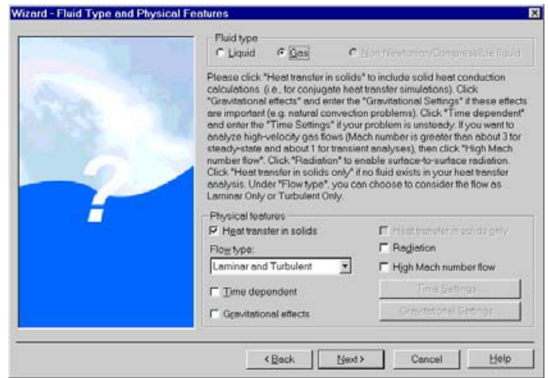


3 Choose the system of units that you have just created, **USA Electronics**. Please keep in mind that after finishing the **Wizard**, you can change the unit system anytime with **FloWorks**, **Units**.

Click **Next**.



4 Set the fluid type to **Gas**. Under physical features select the **Heat transfer in solids** check box.

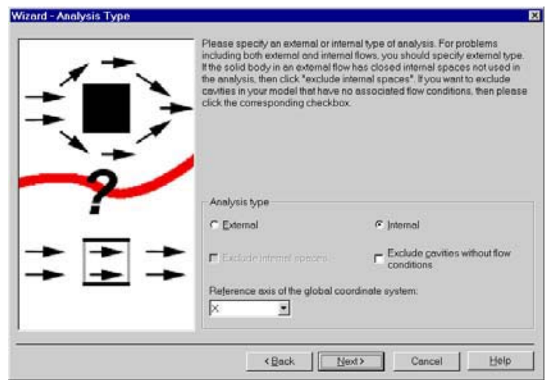


Click **Next**.

🕮 *Heat transfer in solids was selected because heat is generated by several electronics component 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. Therefore we must simulate heat conduction in the solid parts.*

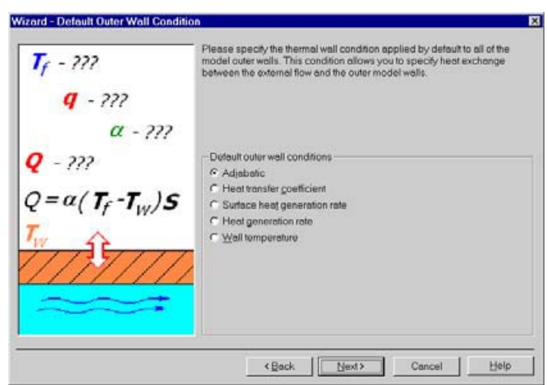
5 Set the analysis type to **Internal**.

🕮 *We want to analyze the flow* ***through*** *the structure. This is what we call an internal analysis. The opposite is an external analysis, which is the flow* ***around*** *an object. From this dialog box you can also choose to ignore cavities that are not relevant to the flow analysis without having to fill them in using SolidWorks features.*

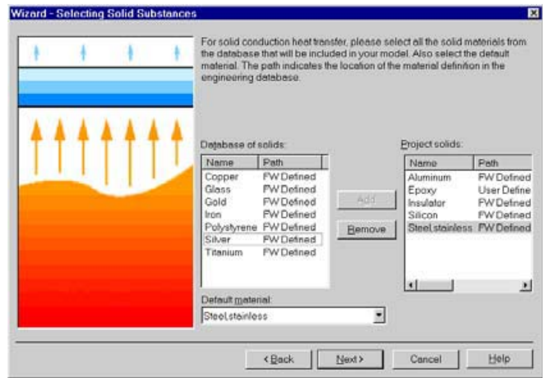


Click **Next**

6 Click **Next** accepting the adiabatic default outer wall condition.



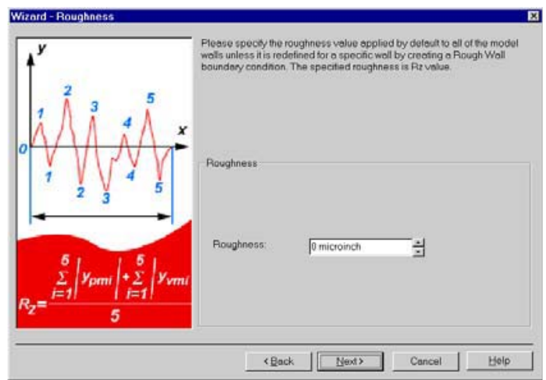
7 In the **Database of solids** list, double-click the **Aluminum**, **Epoxy**, **Insulator**, **Silicon** and **Steel, stainless** items.



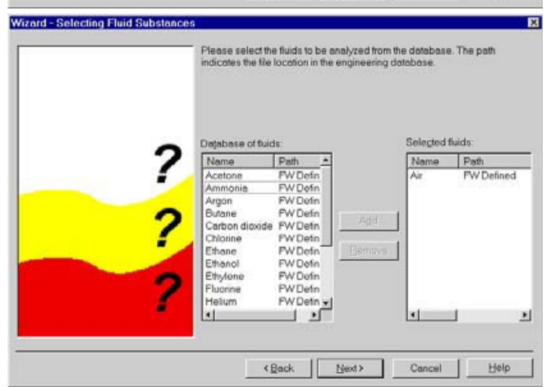
8 Select **Steel, stainless** as the **Default material**.

Click **Next**.

9 Click **Next** accepting the default zero roughness value for all model walls.



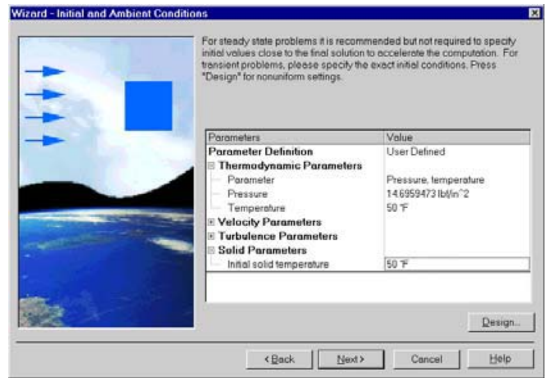
10 Choose **Air** as the fluid. You can either double-click Air or select the item in the left column and click **Add**.



Click **Next**.

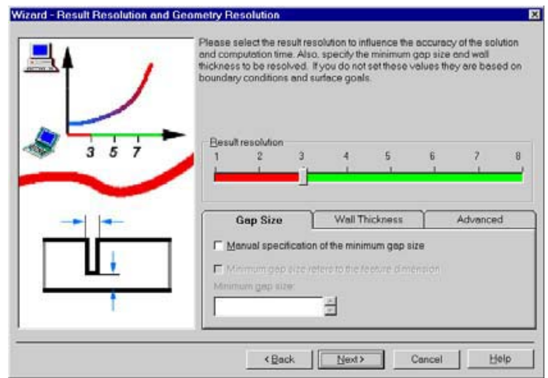
Although setting the initial temperature is more important for transient calculations to see how much time it takes to reach a certain temperature, it is useful to set the initial temperature close to the anticipated final solution to speed up convergence. In this case we will set the initial air temperature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

11 Set the initial fluid **Temperature** and the **Initial solid temperature** to 50°F.



Click **Next**.

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

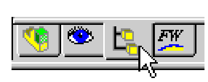


📖 *COSMOSFloWorks calculates the default minimum gap size and minimum wall thickness using information about the overall model dimensions, the computational domain, and faces on which you specify conditions and goals. Prior to starting the calculation, we recommend that you check the minimum gap size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

Click **Next**.

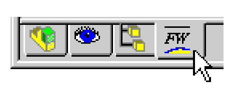
13 Click **Finish**. Now COSMOSFloWorks creates a new configuration with the COSMOSFloWorks data attached.

Click on the **SolidWorks Configuration Manager** to show the new configuration.



📖 *Notice the name of the new configuration has the name you entered in the* **Wizard**.

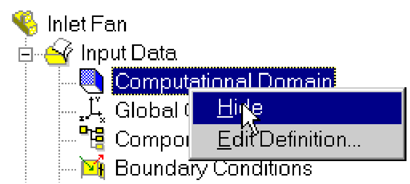
Go to the **COSMOSFloWorks design tree** and open all the icons.



We will use the COSMOSFloWorks Design Tree to define our analysis, just as the SolidWorks Feature Manager Tree is used to design your models.

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

Computational Domain *is the icon used to modify the size and visualization of the volume being analyzed as well as to specify symmetry boundary conditions and 2D flow. The wireframe box enveloping the model is the visualization of the limits of the computational domain.*

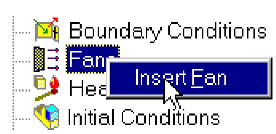


Define the Fan

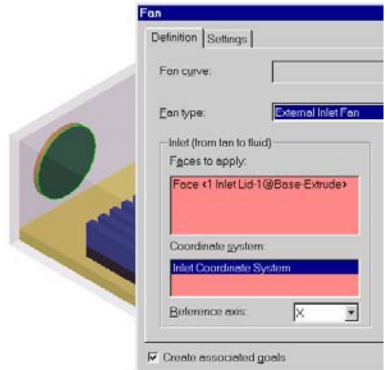
A Fan is a type of flow boundary condition. You can specify Fans at selected solid surfaces where ***Boundary Conditions*** *and* ***Sources*** *are not specified. You can specify Fans on artificial lids closing model openings as Inlet Fans or Outlet Fans. You can also specify fans on any faces arranged inside of the flow region as Internal Fans. A Fan is considered an ideal device creating a volume (or mass) flow rate depending on the difference between the inlet and outlet static pressures on the selected face. A curve of the fan volume flow rate or mass flow rate versus the static pressure difference is taken from the* ***Engineering Database****.*

If you analyze a model with a fan then you must know the fan's characteristics. In this example we use one of the pre-defined fans from the **Engineering Database**. If you cannot find an appropriate curve in the database you can create your own curve in accordance with the specification on your fan.

1 In the COSMOSFloWorks design tree, right-click the **Fans** icon and select **Insert Fan**. The **Fan** dialog box appears.



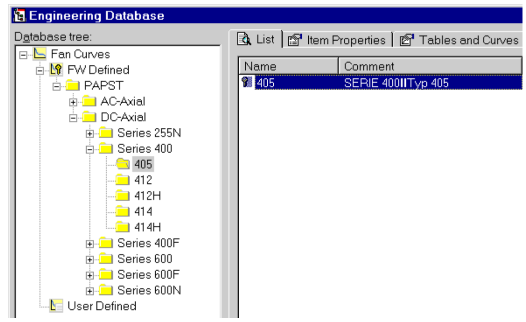
2 Select the inner face of the **Inlet Lid** part as shown. (To access the inner face, right-click the mouse to cycle through the faces under the cursor until the inner face is highlighted, then click the left mouse button).



3 Select **External Inlet Fan** as **Fan type**.

4 Click **Browse** to select the fan curve from the **Engineering database**.

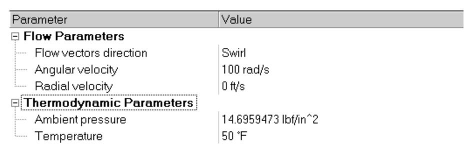
5 Select the **405** item under the **Fan Curves**, **FW Defined, PAPST, DC-Axial, Series 400, 405** item.



6 Click **OK** to return to the **Fan** dialog box.

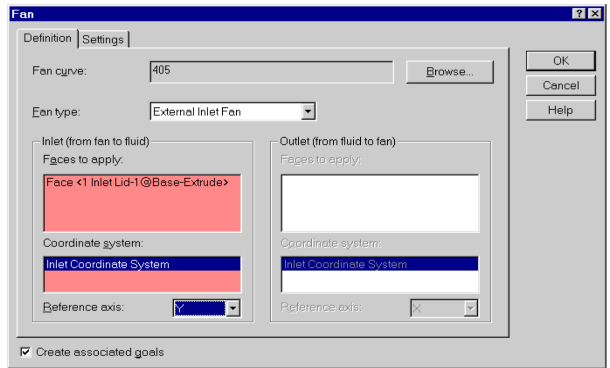
7 On the **Settings** tab expand the **Thermodynamic Parameters** item to check that the **Ambient pressure** is atmospheric pressure.

8 Expand the **Flow parameters** item and select **Swirl** in the **Flow vectors direction** list.



9 Specify the **Angular velocity** as 100 rad/s and accept the zero **Radial velocity** value.

*When specifying a swirling flow, you must choose the reference **Coordinate system** and the **Reference axis** so that the origin of the coordinate system and the swirl's center point are coincident and the angular velocity vector is aligned with the reference axis.*



10 Go back to the **Definition** tab. Hold down the **Ctrl** key and in the FeatureManager design tree select the **Inlet Coordinate System**.

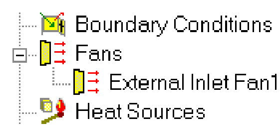
11 Select the **Global Coordinate System** item and press the **Delete** key.

12 Select **Y** in the **Reference Axis** list.

13 Accept to **Create associated goals**.

It is often convenient to specify an appropriate goal along with the specified condition. For example, if you specify a pressure opening it makes sense to define a mass flow rate surface goal at this opening. COSMOSFloWorks allows you to associate a condition type (boundary condition, fun, heat source or radiative surface) with a goal(s), which will be automatically created along with this condition. You can associate goals with a boundary condition type under the Automatic Goals item of the COSMOSFloWorks Options dialog box.

14 Click **OK**. The new **External Inlet Fan1** item appears in the COSMOSFloWorks design tree.

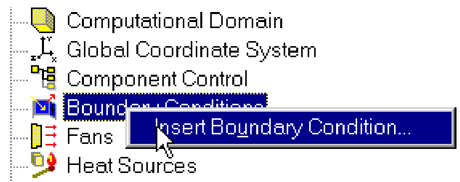


With the definition just made, we told COSMOSFloWorks that at this opening air flows into the enclosure through the fan so that the volume flow rate of air depends on the difference between the ambient atmospheric pressure and the static pressures on the fan's outlet face (inner face of the lid) in accordance with the curve shown above. 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.

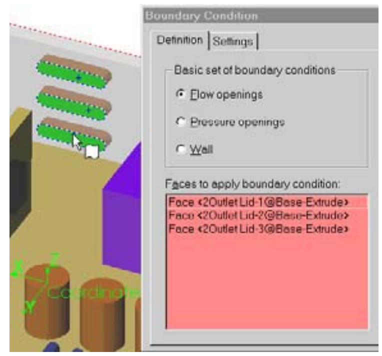
Define the Boundary Conditions

A **boundary condition** is required anywhere fluid enters or exits the system excluding openings where a fan is specified. A boundary condition can be set as a Pressure, Mass Flow, Volume Flow 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 transfer coefficient at the model surfaces. For internal analyses with "Heat transfer in solids" you can also set thermal wall condition on outer model walls by specifying an **Outer Wall** condition.

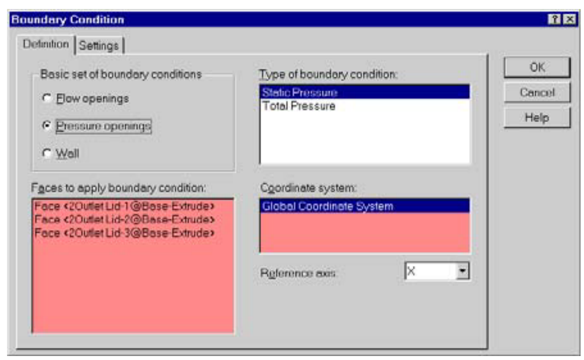
1 In the COSMOSFloWorks design tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**Click **Insert, New Boundary Condition**.



2 Select the inner face of all of the outlet lids as shown. (To access the inner face, right-click the mouse to cycle through the faces under the cursor until the inner face is highlighted, then click the left mouse button).

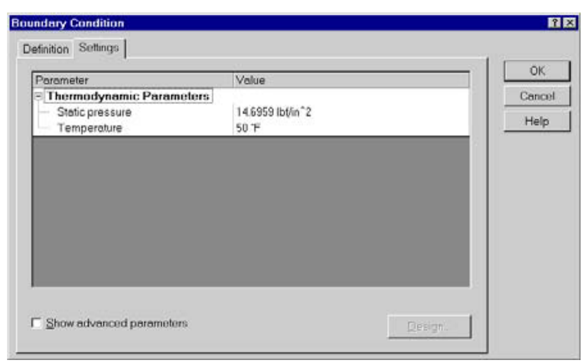


3 Select **Pressure openings** and **Static Pressure**.



4 Keep the defaults under the **Settings** tab.

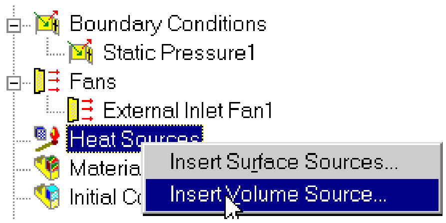
5 Click **OK**. The new **Static Pressure1** item appears in the COSMOSFloWorks design tree.



With the definition just made, we told COSMOSFloWorks that at this opening the fluid exits the model to an area of static atmospheric pressure. Within this dialog box we can also set time dependent properties for the pressure.

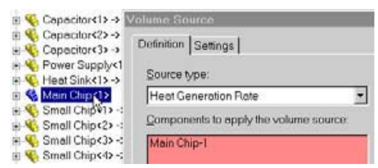
Define the Heat Source

1 In the COSMOSFloWorks design tree, right-click the **Heat Sources** icon and select **Insert Volume Source**.



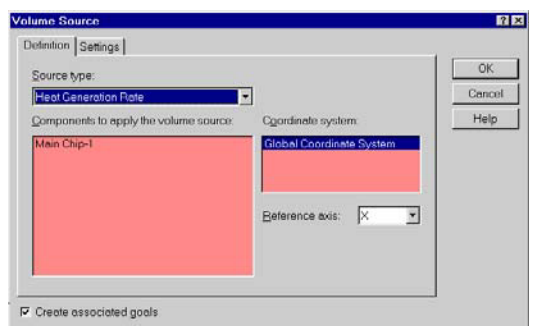
2 Since the inner faces of the outlet lids are still selected, the lids automatically appear in the **Components to apply the volume source** list. Remove all lids from the list. To remove a component, select it in the list and press the **Delete** key.

You can avoid this if before opening the **Volume Source** dialog, you click in the white area to deselect the faces.



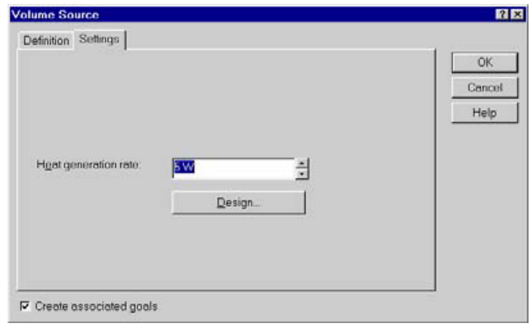
3 Select the **Main Chip** from the SolidWorks FeatureManager tree as the component to apply the volume source.

4 Select the **Source type** as **Heat Generation Rate**.



5 Click the **Settings** tab and enter `5 W` in the **Heat generation rate** box.

6 Click **OK**.



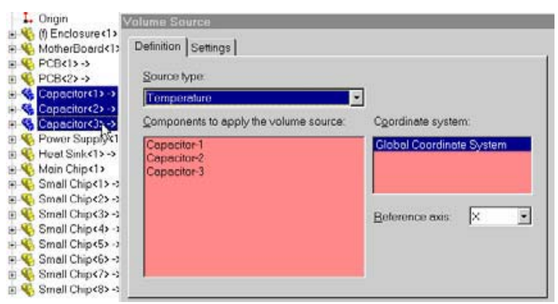
7 In the COSMOSFloWorks design tree click-pause-click the new **VS Heat Generation Rate 1** item and rename it to `Main Chip`.

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

1 In the COSMOSFloWorks design tree, right-click the **Heat Sources** icon and select **Insert Volume Source**.

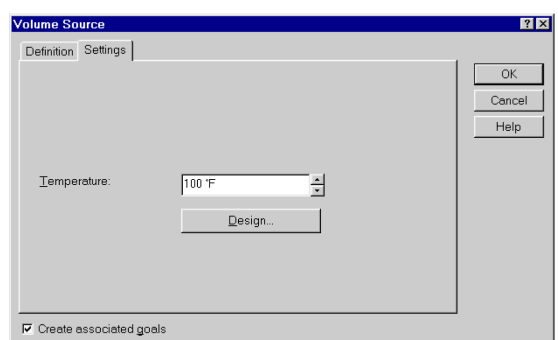
2 In the SolidWorks FeatureManager tree select all **Capacitor** components.

3 Select the **Temperature** in the **Source type** list.



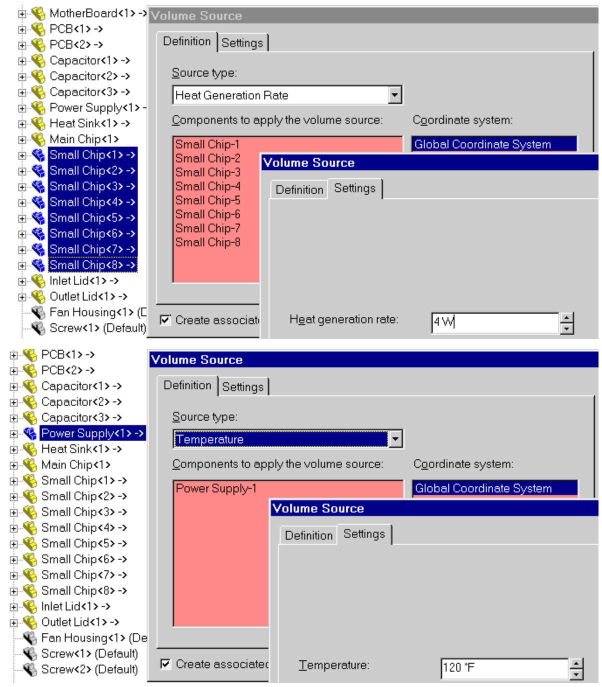
4 Click the **Settings** tab and enter 100 °F in the **Temperature** box.

5 Click **OK**.

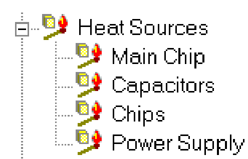


6 Click-pause-click the new **VS Temperature1** item and rename it to `Capacitors`.

7 Following the same procedure as above, set the following volume heat sources: all chips on PCB (Small Chip) - total heat generation rate of 4 W, Power Supply - temperature of 120 °F



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



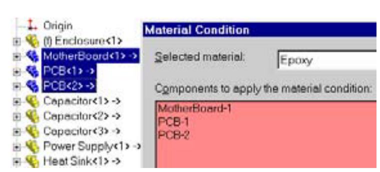
9 Click **File**, **Save**.

Define the Material Conditions

1 Right-click the **Material Conditions** icon and select **Insert Material Condition**.

2 In the SolidWorks FeatureManager select **MotherBoard**, **PCB(1)**, **PCB(2)** components.

3 Select **Epoxy** from the **Selected material** list.



4 Click **OK**.

5 Following the same procedure as above, set the following material conditions: the chips are made of **silicon**, the **heat sink** is made of **aluminum**, and the 4 Lids (Inlet Lid and three Outlet Lids) are made of **insulator** material. All four lids can be selected in the same material condition definition. Note that two of the outlet lids can be found under derived pattern (**DerivedLPattern1**) in the SolidWorks FeatureManager. Alternatively you can click on the actual part in the SolidWorks graphics area.

6 Change the name of each material condition. The new descriptive names are:
`PCB - Epoxy`,
`Heat Sink - Aluminum`,
`Chips - Silicon`, and
`Lids - Insulator`.

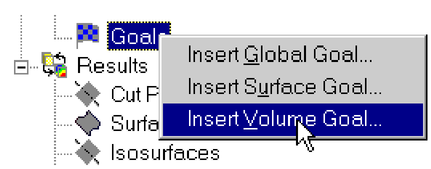
Material Conditions are used to specify the material type of solid parts in the assembly.

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

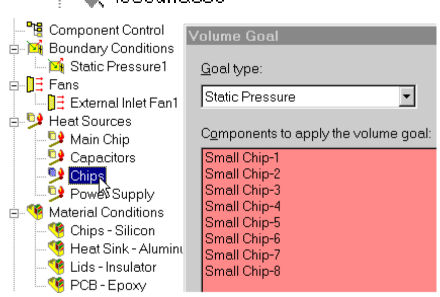
Define the Engineering Goals

Specifying Volume Goals

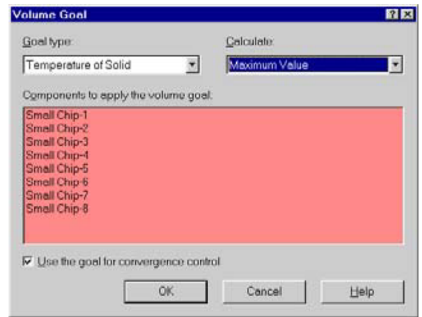
1 Right-click the **Goals** icon and select **Insert Volume Goal**.



2 Select the **Chips** item in the COSMOS-FloWorks design tree. This selects all components belonging to the **Chips** heat source.



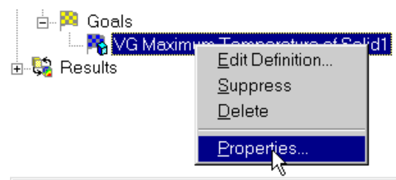
3 Select **Temperature of Solid** as the goal type and calculate **Maximum value**.



4 Accept to **Use the goal for convergence control**.

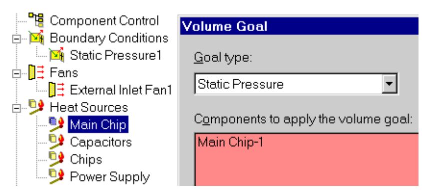
5 Click **OK**. The new **VG Maximum Temperature of Solid1** item appears in the COSMOSFloWorks design tree.

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 appearing if you right-click the item and select **Properties**.

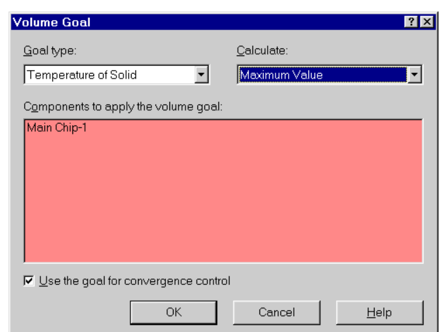


7 Right-click the **Goals** icon and select **Insert Volume Goal**.

8 Select the **Main Chip** item in the COSMOSFloWorks design tree.



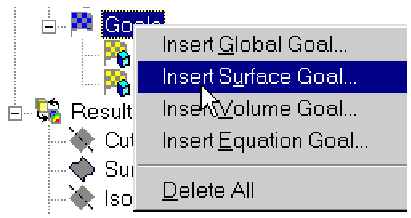
9 Select **Temperature of Solid** as the goal type and calculate **Maximum value**.



10 Click **OK**.

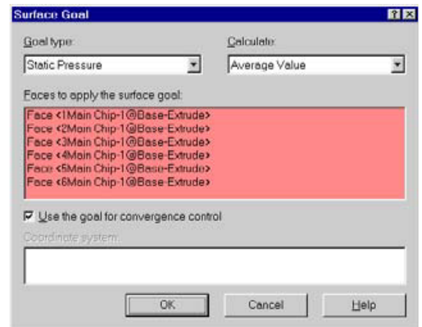
11 Rename the new **VG Maximum Temperature of Solid1** item to `VG Chip Max Temperature`.

Specifying Surface Goals

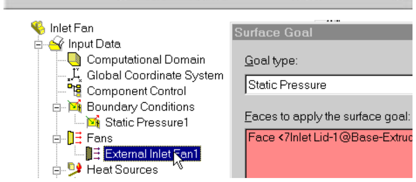


1 Right-click the **Goals** icon and select **Insert Surface Goal**.

2 Since the **Main Chip** is still selected, all its faces automatically appear in the **Faces to apply the surface goal** list. Remove all faces from the list. To remove a face, select it in the list and press the **Delete** key.

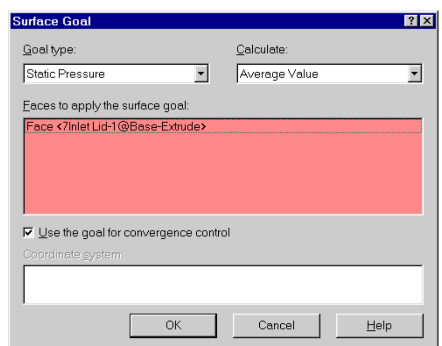


3 Click the **External Inlet Fan1** item to select the face where it is going to be applied.



4 Keep the **Static Pressure** and the **Average Value**.

5 Accept to **Use the goal for convergence control**.

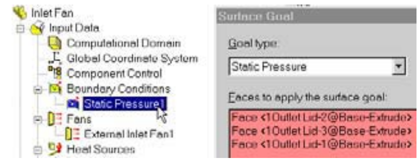


For the X(Y, Z) - Component of Force and X(Y, Z) - Component of Torque goals you can select the Coordinate system in which these goals are calculated.

6 Click **OK** and rename the new **SG Average Static Pressure1** item to `SG Av Inlet Pressure`.

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

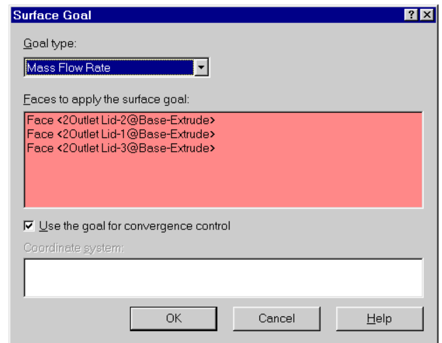
8 Click the **Static Pressure1** item to select the face where it is going to be applied.



9 Select **Mass Flow Rate** as the **Goal type**.

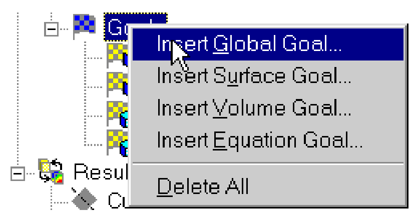
10 Accept to **Use the goal for convergence control**.

11 Click **OK** and rename the new **SG Mass Flow Rate1** item to `SG Outlet Mass Flow Rate`.



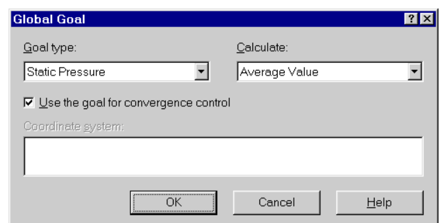
Specifying Global Goals

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



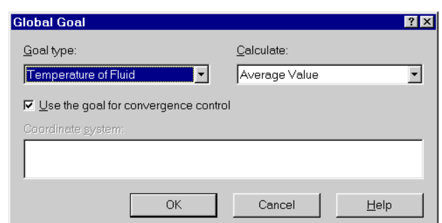
2 Keep the **Static Pressure**, **Average Value** and accept to **Use the goal for convergence control**.

3 Click **OK**. Rename the new **GG Average Static Pressure1** item to `GG Av Pressure`.



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

5 Select the **Temperature of Fluid** as the **Goal type**, keep the **Average Value** and accept to **Use the goal for convergence control**.



6 Click **OK**. Rename the new **GG Average Temperature of Fluid1** item to `GG Av Fluid Temperature`.

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

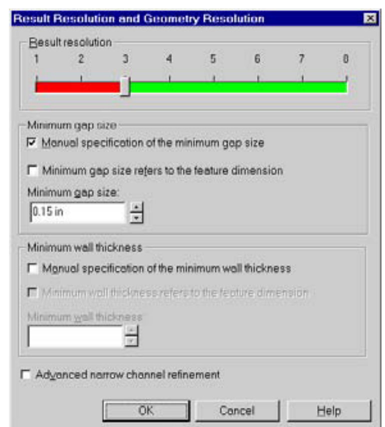
Click **File**, **Save**.

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

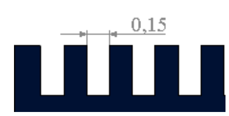
Changing the Geometry Resolution

1 Click **FloWorks**, **Initial Mesh**.

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



3 Enter `0.15"` for the minimum flow passage (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 COSMOSFloWorks mesh is constructed so that the specified Result Resolution level 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.*

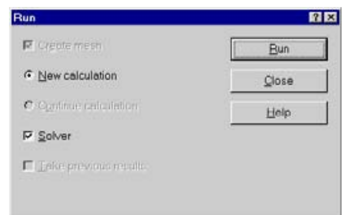
Click **OK**.

Solution

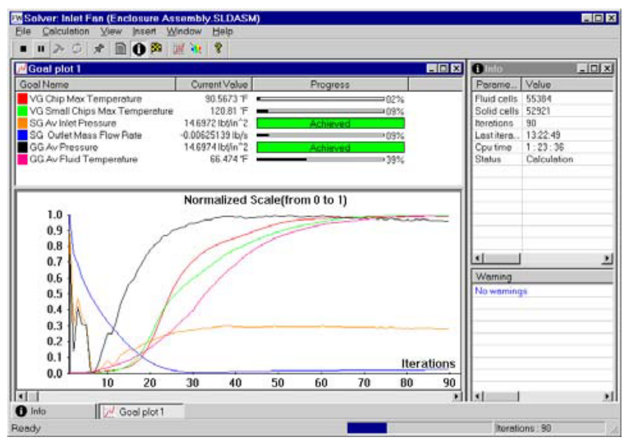
1 Click **FloWorks**, **Solve**, **Run**.

2 Click **Run**.

The solver will approximately take about 1.5 hours to run on an 850MHz platform.

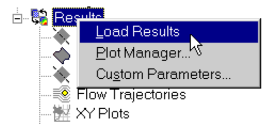


*This is the solution monitor dialog box. Notice for this tutorial that the **SG Av Inlet Pressure** and **GG Av Pressure** converged very quickly compared to the other goals. Generally different goals take more or less iterations to converge. The goal-oriented philosophy of COSMOS-FloWorks allows you to get the answers you need in the shortest amount of time. For example, if you were only interested in the pressure drop through the enclosure, COSMOS-FloWorks would have provided the result more quickly then if the solver was allowed to fully converge on all of the parameters.*



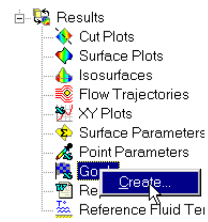
Viewing the Goals

1 Right-click the **Results** icon and select **Load Results** to activate the postprocessor.



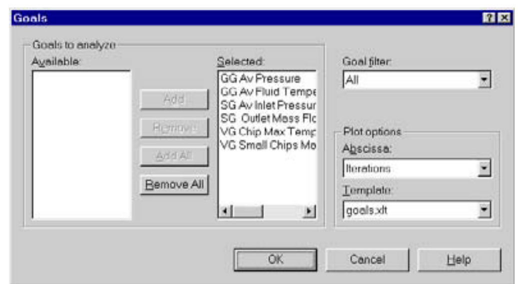
2 Select the project's results (1.fld) file and click **Open**.

3 Right-click the **Goals** icon and select **Create**.



4 Click **Add All** in the **Goals** dialog.

5 Click **OK**.



An excel workbook will open with the goal results. The first sheet will show a table summarizing the goals.

Enclosure Assembly.SLDASM [Inlet Fan]

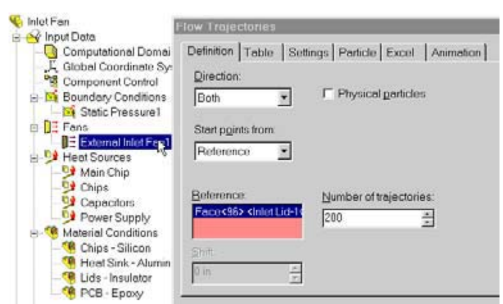
Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence
GG Av Pressure	[lbf/in^2]	14.69631	14.6963	14.6963	14.6963	100	Yes
SG Av Inlet Pressure	[lbf/in^2]	14.69615642	14.6962	14.6962	14.6962	100	Yes
GG Av Fluid Temperature	[°F]	59.4738659	59.5518	59.4699	59.6098	100	Yes
SG Outlet Mass Flow Rate	[lb/s]	-0.007634592	-0.00763519	-0.00763917	-0.00762781	100	Yes
VG Chip Max Temperature	[°F]	100.1197938	100.186	100.108	100.252	100	Yes
VG Small Chips Max Temperature	[°F]	111.1247332	112.129	111.125	113.117	100	Yes

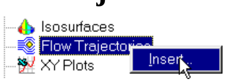
You can see that the maximum temperature in the main chip is 100.12 °F, and the maximum temperature over the small chips is 111.12 °F.

Goal's progress bar is a qualitative and quantitative characteristic of the goal's convergence process. When COSMOSFloWorks analyzes the goal's convergence, it calculates the goal's dispersion defined as the difference between the goal's maximum and minimum 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 COSMOSFloWorks 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, as well as if the goals' 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 COSMOSFloWorks post-processing tools. For the visualization of how the fluid flows inside the enclosure the best method is to create flow trajectories.

Flow Trajectories



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

2 In the COSMOSFloWorks design tree select the **External Inlet Fan1** item. This selects the inner face of the **Inlet Lid**.

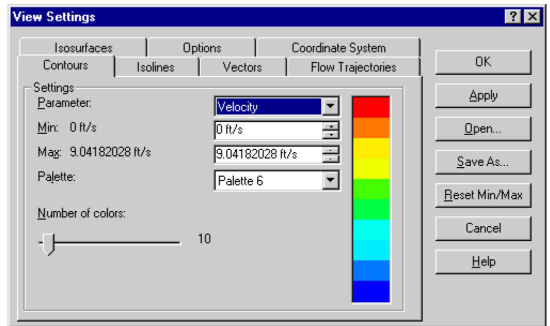
3 Set the **Number of trajectories** to 200.

4 Keep the **Reference** in the **Start points from** list.

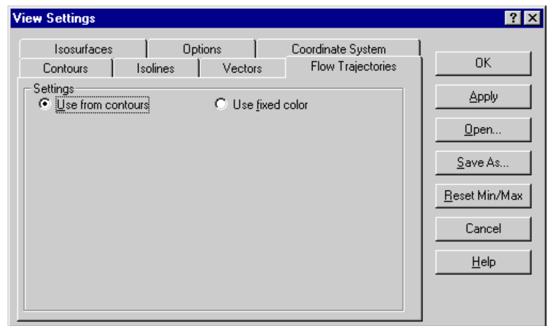
*If **Reference** is selected, then the trajectory start points are taken from this selected face.*

5 Click **View Settings**.

6 In the **View Settings** dialog box, change the **Parameter** from **Pressure** to **Velocity**.



7 Go to the **Flow Trajectories** tab and notice that the **Use from contours** option is selected.

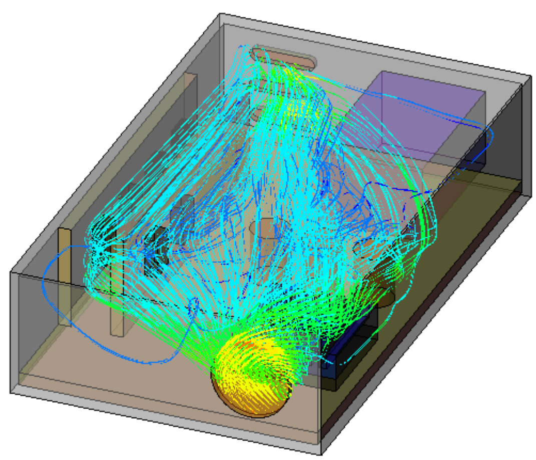


*This setting defines how trajectories are colored. If **Use from contours** is selected then the trajectories are colored with the distribution of the parameter specified on the **Contours** tab (Velocity in our case). If you select **Use fixed color** then all flow trajectories have the same color that you specify on the **Settings** tab of the **Flow Trajectories** dialog box.*

8 Click **OK** to save the changes and exit the **View Settings** dialog box.

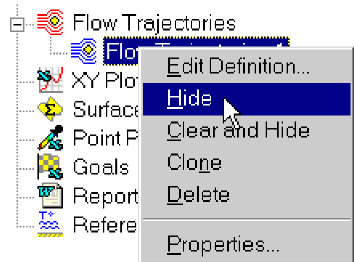
9 In the **Flow Trajectories** dialog box click **OK**. The new **Flow Trajectories 1** item appears in the COSMOSFloWorks design tree.

This is the picture you should see.



Notice that there are only a few trajectories along the **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 **PCB(2)**.

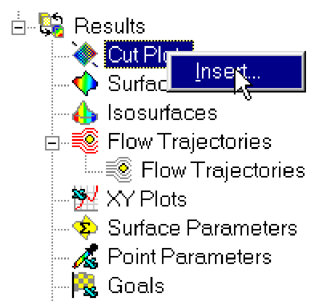
Right-click the **Flow Trajectories 1** item and select **Hide**.



Let us see the velocity distribution in more detail.

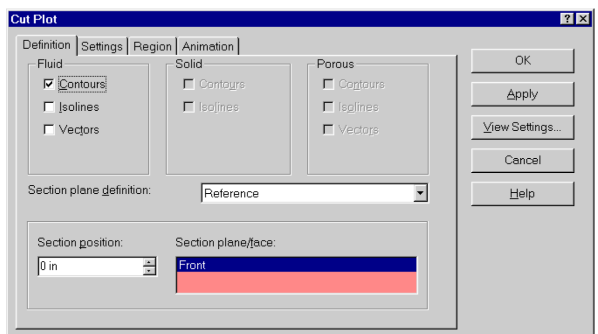
Cut Plots

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



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

3 Click **View Settings**.

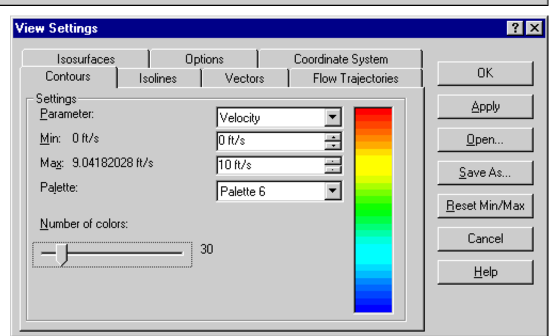


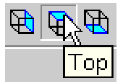
4 Change the **Min** and **Max** values to 0 and 10 respectively. The specified integer values produce a palette where it is more easy to determine the value.

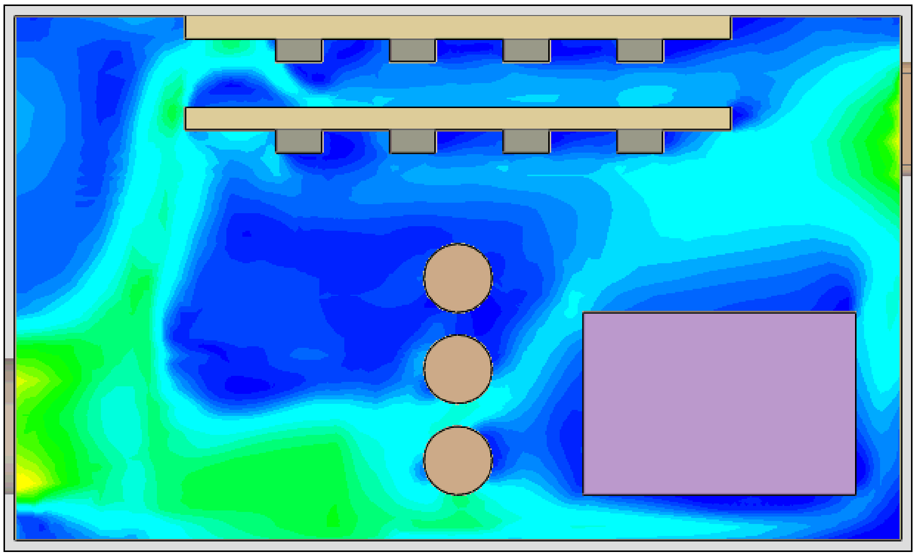
5 Set the **Number of colors** to 30.

6 Click **OK**.

7 In the **Cut Plot** dialog box click **OK**.
The new **Cut Plot 1** item appears in the COSMOSFloWorks design tree.

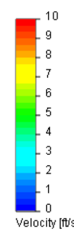


8 Select the **Top** view. 



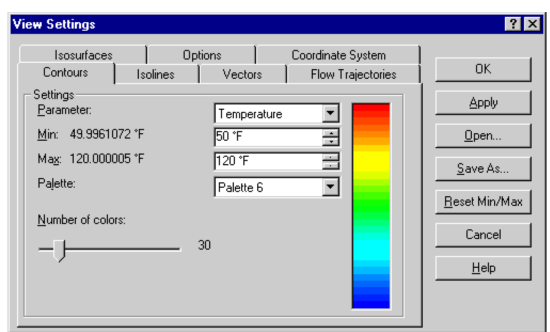
You can see that the maximum velocity region appears close to the openings; and the low velocity region is seen in the center area between the capacitors and the PCB. Furthermore the region between the PCB's has a strong flow which in all likelihood will enhance convective cooling in this region. Let us now look at the fluid temperature.

9 Double-click the palette bar in the upper left corner of the graphics area. The **View Settings** dialog appears.

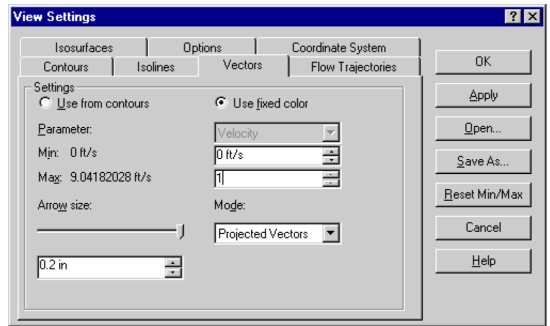


10 Change the **Parameter** from **Velocity** to **Temperature**.

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



12 Click the **Vectors** tab and change the **Arrow size** to 0.2 by typing the value in the box under the slider.



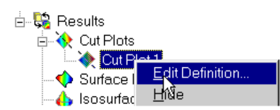
Notice that you can specify a value that is outside of the slider's range.

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

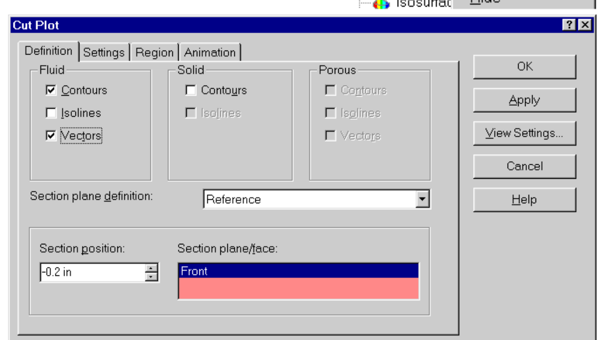
*By specifying the custom **Min** and **Max** values you can control the vector length. The vectors whose velocity exceeds the specified Max value will have the same length as the vectors whose velocity is equal to the Max. Likewise, the vectors whose velocity is less than the specified Min value will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

14 Click **OK**.

15 Right-click the **Cut Plot 1** item and select **Edit Definition**.



16 Select the **Vectors** check box.

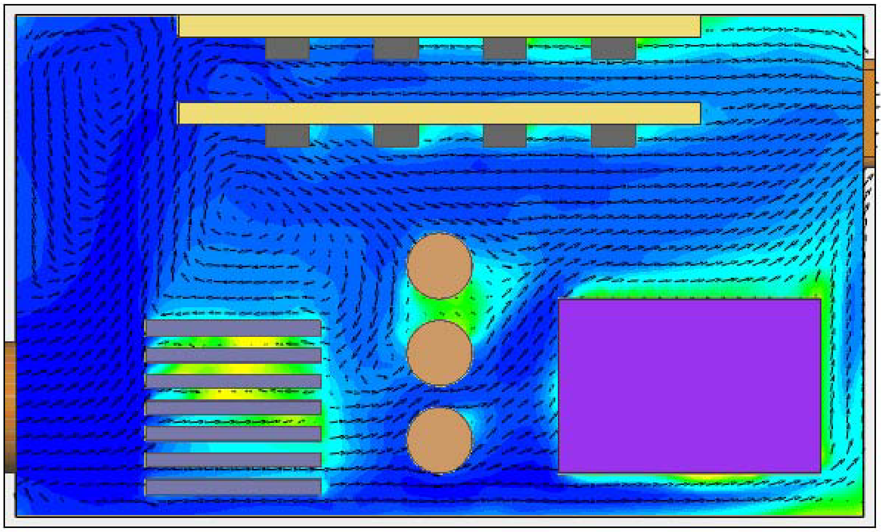


17 Change the **Section position** to `-0.2 in`.

18 Go to the **Settings** tab. Using the slider set the **Vector spacing** to 0.18 in.

19 Click **OK**.

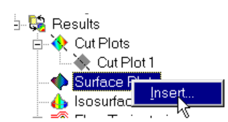
It is not surprising that the fluid temperature is high around the heat sink but it is also high in the area of low velocity denoted by small vectors.



Right-click the **Cut Plot1** item and select **Hide**. Let us now display solid temperature.

Surface Plots

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

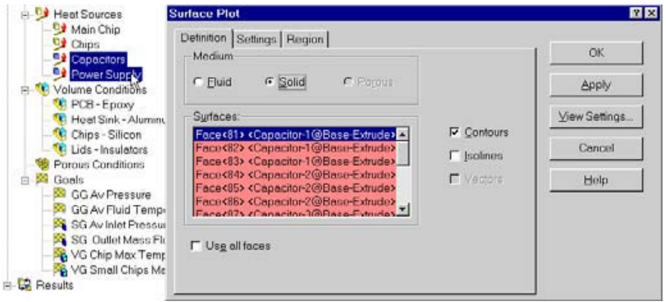


2 Click **Solid** as the **Medium**. Since the **Temperature** is the active parameter, you can display plots in solids; otherwise only the fluid medium would be available.

3 Hold down the **Ctrl** key and select the **Heat Sink - Aluminum** and **Chips - Silicon** items in the COSMOSFloWorks design tree.

4 Click **OK**. The creation of the surface plot may take a time because 76 faces need to be colored.

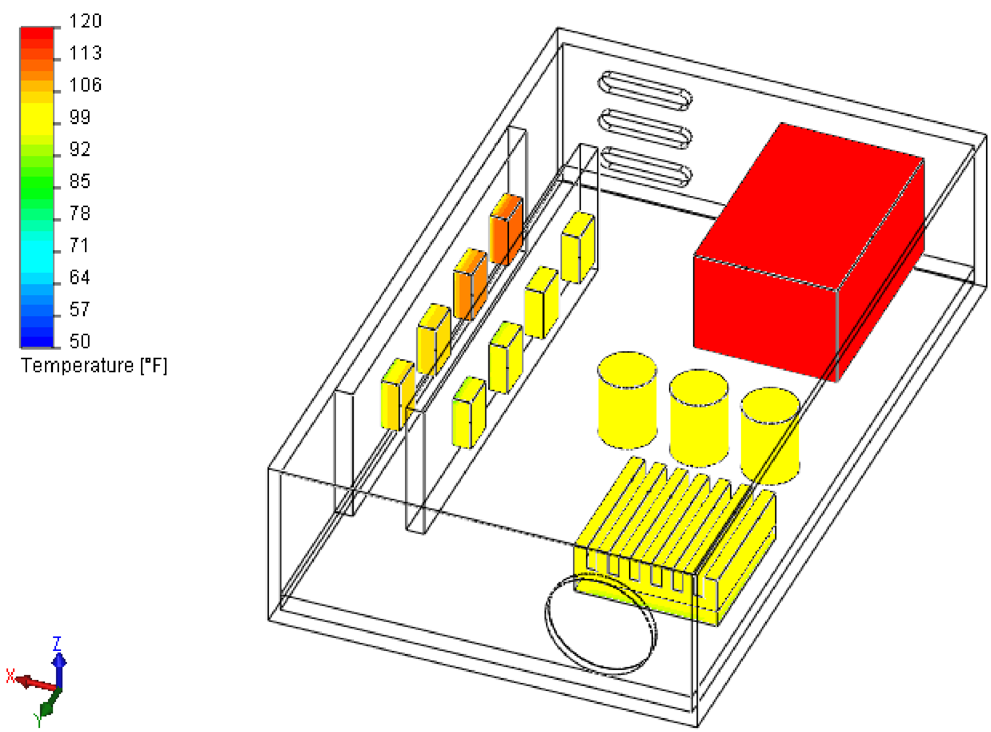
5 Repeat items 1 and 2 and select the **Power Supply** and **Capacitors** items, then click **OK**.



Now you need to hide the model because it overlaps the plots. In additional to using COSMOSFloWorks menu or COSMOSFloWorks design tree it is very convenient to use COSMOSFloWorks Toolbars.

6 Click **View**, **Toolbars**, **COSMOSFloWorks Results**, **Display**.

7 Click **Display Outlines**  to display a wireframe representation of the model. Then click **Display Model Geometry**  to hide the model.



You can further view and analyze the results with the post-processing tools that were shown in the *First Steps - Ball Valve Design* tutorial. COSMOSFloWorks 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, COSMOSFloWorks 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.