SolidWorks Flow Simulation 2011 Tutorial

3S SolidWorks

Contents

First Steps - Conjugate Heat Transfer

First Steps - Porous Media

Determination of Hydraulic Loss

Cylinder Drag Coefficient

Heat Exchanger Efficiency

Mesh Optimization

Application of EFD Zooming

Textile Machine

Non-Newtonian Flow in a Channel with Cylinders

Heated Ball with a Reflector and a Screen

Rotating Impeller

CPU Cooler

Electronic Components

Oil Catch Can

150W Halogen Floodlight

Hospital Room

Features List

This chapter contains the list of the physical and interface features of Flow Simulation as they appear in the tutorial examples. If you need to find an example of a certain feature or function usage, look for the desired feature in the left column and in its row you can see in which tutorial examples this feature is used. Usually, the first entrance of the feature in the tutorial contains the most detailed description. The tutorial examples are listed in Features List by their respective numbers. All tutorial examples are divided in three categories: First Steps, Intermediate and Advanced.

📖 *In the First Steps examples you will learn the basic principles of the Flow Simulation structure and interface.*

1 First Steps - Ball Valve Design

2 - First Steps - Conjugate Heat Transfer

3 - First Steps - Porous Media

📖 *On the Intermediate level you will learn how to solve engineering problems with Flow Simulation, using some of the most common tasks as examples.*

4 - Determination of Hydraulic Loss

5 - Cylinder Drag Coefficient

6 - Heat Exchanger Efficiency

7 - Mesh Optimization

📖 *In the Advanced examples you can see how to use a wide variety of the Flow Simulation features to solve real-life engineering problems. It is assumed that you successfully completed all First Steps examples before.*

8 - Application of EFD Zooming

9 - Textile Machine

10 - Non-Newtonian Flow in a Channel with Cylinders

11 - Heated Ball with a Reflector and a Screen

12 - Rotating Impeller

13 - CPU Cooler

14 - Electronic components

15 - Oil Catch Can

16 - 150W Halogen Floodligh

17 - Hospital Room

	First Steps			Intermediate				Advanced									
	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17
DIMENSIONALITY																	
2D flow					✔												
3D flow	✔	✔	✔	✔		✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔
ANALYSIS TYPE																	
External analysis					✔						✔		✔			✔	
Internal analysis	✔	✔	✔	✔		✔	✔	✔	✔	✔		✔		✔	✔		✔
PHYSICAL FEATURES																	
Steady state analysis	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔
Time-dependent (transient) analysis					✔												
Liquids	✔			✔	✔	✔				✔							
Gases		✔	✔			✔	✔	✔	✔		✔	✔	✔	✔	✔	✔	✔
Non-Newtonian liquids										✔							
Multi-fluid analysis																	
Mixed flows							✔										✔
Separated flows (as Fluid Subdomains)						✔										✔	
Heat conduction in solids		✔				✔		✔			✔		✔	✔		✔	
Heat conduction in solids only											✔						
Gravitational effects							✔							✔		✔	✔
Laminar only flow											✔						
Porous media			✔				✔										

	First Steps			Intermediate				Advanced									
	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17
Radiation											✔					✔	
Absorption in solids																✔	
Spectrum																✔	
Roughness									✔								
Two-phase flows (fluid flows with particles or droplets)									✔						✔		
Rotation																	
Global rotating reference frame												✔					
Local rotating regions													✔				
CONDITIONS																	
Computational domain					✔			✔			✔		✔			✔	
Symmetry					✔	✔										✔	
Initial and ambient conditions																	
Velocity parameters					✔												
Dependency					✔												
Thermodynamic parameters						✔							✔				✔
Turbulence parameters					✔												
Concentration							✔										✔
Solid parameters													✔				

	First Steps			Intermediate				Advanced									
	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17
Boundary conditions																	
Flow openings																	
Inlet mass flow	✔					✔			✔								
Inlet volume flow							✔		✔		✔			✔			✔
Outlet volume flow							✔										✔
Inlet velocity			✔	✔		✔											
Pressure openings																	
Static pressure	✔		✔	✔				✔	✔						✔		
Environment pressure		✔				✔	✔	✔				✔		✔			✔
Wall																	
Real wall									✔			✔	✔				
Boundary condition parameters	✔		✔	✔		✔	✔		✔	✔		✔					
Transferred boundary conditions								✔									
Fans		✔						✔						✔			
Contact resistances														✔			
Perforated plates														✔			
Volume conditions																	
Fluid Subdomain						✔										✔	
Initial conditions																	
Velocity parameters									✔								
Dependency									✔								
Solid parameters											✔						

	First Steps			Intermediate				Advanced									
	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17
Solid material		✔				✔		✔			✔		✔	✔		✔	
Semi-transparent																✔	
Porous medium			✔				✔										
Heat sources																	
Surface sources																	
Heat generation rate								✔			✔						✔
Volume sources																	
Temperature		✔														✔	
Heat generation rate		✔						✔					✔	✔			
Goal-dependent sources																✔	
Radiative conditions																	
Radiation sources											✔					✔	
Radiative surfaces											✔					✔	
Electronics module features (requires Electronics license)																	
Two-resistor components														✔			
Heat pipes														✔			
Printed circuit boards														✔			
PROJECT DEFINITION																	
Wizard and Navigator	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔
From template					✔												
Clone project	✔		✔	✔	✔			✔	✔	✔	✔						
General settings					✔						✔						
Copy project's features								✔									

	First Steps			Intermediate				Advanced									
	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17
GOALS																	
Global goal		✔			✔				✔			✔	✔		✔		✔
Surface goal	✔	✔	✔	✔				✔	✔	✔	✔	✔	✔	✔	✔	✔	
Volume goal		✔				✔		✔	✔		✔			✔		✔	
Point goal													✔				
Equation goal			✔		✔						✔		✔		✔		
MESH SETTINGS																	
Initial mesh																	
Automatic settings																	
Level of initial mesh								✔					✔			✔	✔
Minimum gap size	✔	✔		✔			✔	✔	✔	✔		✔	✔				✔
Minimum wall thickness							✔				✔	✔					✔
Manual adjustments																	
Control planes							✔						✔	✔			
Solid/fluid interface											✔		✔	✔			
Narrow channels							✔						✔				
Local initial mesh																	
Manual adjustments																	
Refining cells							✔							✔		✔	✔
Narrow channels							✔	✔						✔			
TOOLS																	
Dependency					✔				✔							✔	
Custom units		✔						✔									

	First Steps			Intermediate				Advanced									
	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17
Engineering database																	
User-defined items		✔	✔					✔		✔				✔	✔	✔	
Check geometry				✔													✔
Gasdynamic calculator				✔													
Toolbars						✔											
Filter								✔	✔				✔	✔		✔	
Component control							✔	✔			✔	✔	✔				
CALCULATION CONTROL OPTIONS																	
Result resolution level					✔			✔	✔			✔					
Solution adaptive mesh refinement															✔		
Calculate comfort parameters																	✔
RUNNING CALCULATION																	
Batch run					✔			✔			✔						
MONITORING CALCULATION																	
Goal plot	✔																
Preview	✔																
GETTING RESULTS																	
Cut plot	✔	✔		✔		✔	✔	✔	✔							✔	✔
Surface plot	✔	✔															
Isosurfaces	✔																✔
Flow trajectories	✔	✔	✔			✔			✔								
Particle study									✔						✔		

	First Steps			Intermediate				Advanced									
	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17
XY plot	✔																
Surface parameters	✔					✔											
Volume parameters																	✔
Goal plot		✔	✔	✔	✔	✔		✔		✔			✔	✔			
Display parameters				✔													
Results summary																	✔
Display mode																	
Show/Hide model geometry				✔		✔	✔										
Transparency	✔		✔														
Apply lighting	✔																
OPTIONS																	
Use CAD geometry							✔										
Display mesh							✔										

First Steps - Ball Valve Design

This First Steps tutorial deals with the flow of water through a ball valve assembly before and after some design changes. The objective is to show how easy fluid flow simulation can be with Flow Simulation and how simple it is to analyze design variations. These two factors make Flow Simulation the perfect tool for engineers who want to test the impact of their design changes.

Open the SolidWorks Model

1 Copy the **First Steps - Ball Valve** folder from the installation directory into your working directory and ensure that the files are not read-only since Flow Simulation will save input data to these files. Run **Flow Simulation**.

2 Click **File**, **Open**. In the **Open** dialog box, browse to the Ball Valve.SLDASM assembly located in the **First Steps - Ball Valve** folder and click **Open** (or double-click the assembly). Alternatively, you can drag and drop the Ball Valve.SLDASM file to an empty area of SolidWorks window. Make sure that the **default** configuration is the active one.

📖 *This is a ball valve. Turning the handle closes or opens the valve. The assembly mate angle controls the opening angle.*

3 Highlight the **lids** by clicking the features in the FeatureManager design tree (Lid <1> and Lid <2>).

📖 *We utilize this model for the Flow Simulation simulation without any significant changes. The user simply closes the interior volume using extrusions that we call lids. In this example the lids are made semi-transparent so you may look into the valve.*
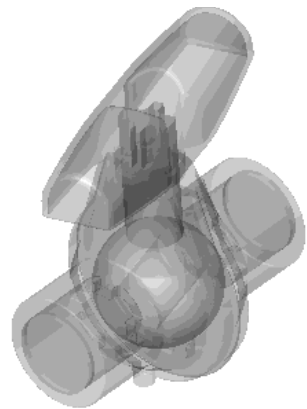
Create a Flow Simulation Project

1   In the main menu click **Flow Simulation**, **Project**, **Wizard**.

2   Once inside the Wizard, select **Create new** in order to create a new configuration and name it Project 1.
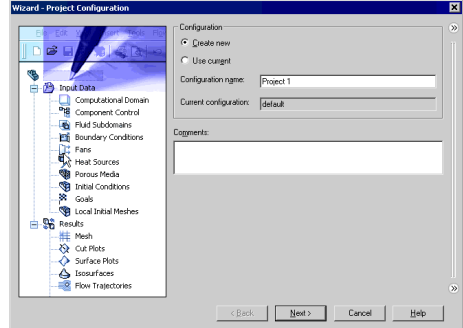
  📖 *Flow Simulation will create a new configuration and store all data in a new folder.*

    Click **Next**.

3   Choose the system of units (**SI** for this project). Please keep in mind that after finishing the Wizard you can change the unit system at any time by clicking **Flow Simulation**, **Units**.
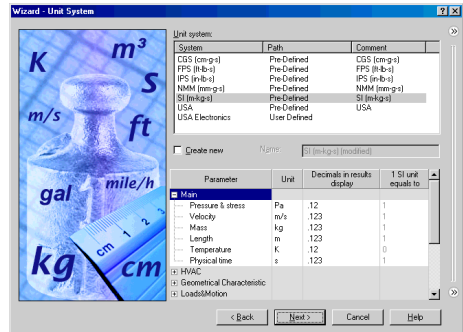
  📖 *Within Flow Simulation, there are several predefined systems of units. You can also define your own and switch between them at any time.*

    Click **Next**.

4   Keep the default **Internal** analysis type. Do not include any physical features.
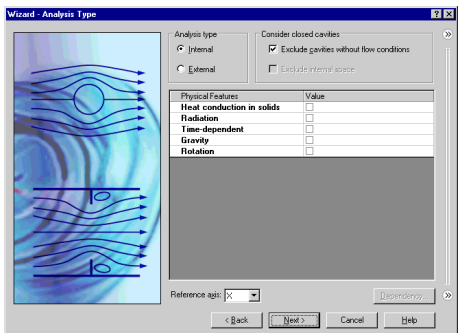
  📖 *We want to analyze the flow **through** the structure. This is what we call an internal analysis. The alternative is an external analysis, which is the flow **around** an object. In this dialog box you can also choose to ignore cavities that are not relevant to the flow analysis, so that Flow Simulation will not waste memory and CPU resources to take them into account.*

  📖 *Not only will Flow Simulation calculate the fluid flow, but can also take into account heat conduction within the solid, including surface-to-surface radiation. Transient (time-dependent) analyses are also possible. Gravitational effects can be included for natural convection cases. Analysis of rotating equipment is one more option available. We skip all these features, as none of them is needed in this simple example.*

    Click **Next**.

5 In the **Fluids** tree expand the **Liquids** item and choose **Water** as the fluid. You can either double-click **Water** or select the item in the tree and click **Add**.

📖 *Flow Simulation is capable of calculating flow of fluids of different types in the same analysis, but fluids of different types must be separated by walls. A mixing of fluids may be considered only if the fluids are of the same type.*
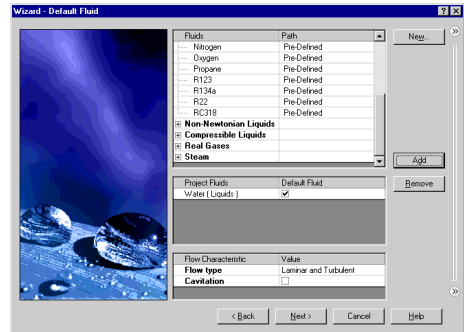
📖 *Flow Simulation has an integrated database containing properties of several liquids, gases and solids. Solids are used in conjugate heat conduction analyses. You can easily create your own materials. Up to ten liquids or gases can be chosen for each analysis run.*

📖 *Flow Simulation can analyze any flow type: Turbulent only, Laminar only or Laminar and Turbulent. The turbulent equations can be disregarded if the flow is entirely laminar. Flow Simulation can also handle low and high Mach number compressible flows for gases. For this demonstration we will perform a fluid flow simulation using a liquid and will keep the default flow characteristics.*

Click **Next**.

6 Click **Next** accepting the default wall conditions.

📖 *Since we did not choose to consider heat conduction in solids, we have an option to define a value of heat transfer for all surfaces of the model being in contact with the fluid. Keep the default **Adiabatic wall** to specify that the walls are perfectly insulated.*
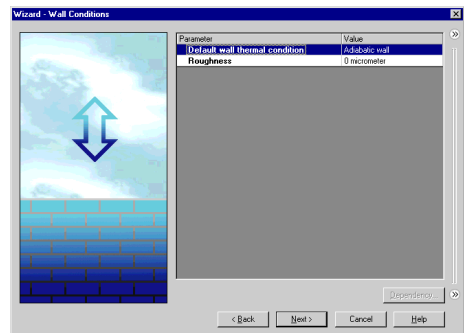
📖 *You can also specify a wall roughness value applied by default to all model walls. The specified roughness value is the R_z value.*

📖 *To set a heat transfer or roughness value for a specific wall, you can define a **Real Wall** boundary condition.*

7   Click **Next** accepting the default for the
initial conditions.

℻ *On this step we can change the default
settings for pressure, temperature and
velocity. The closer these values to the
final values determined in the analysis,
the quicker the analysis will finish.
Since we do not have any knowledge of
the expected final values, we will not
modify them for this demonstration.*
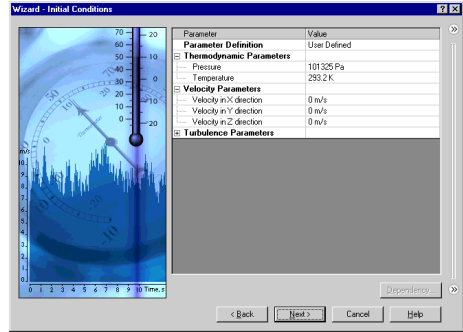
8   Accept the default for the **Result
Resolution**.
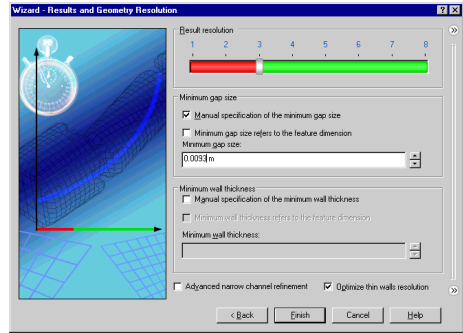
℻ *Result Resolution is a measure of the desired level of accuracy of the results. It controls
not only the resolution of the geometry by the mesh, but also sets many parameters for
the solver, e.g. convergence criteria. The higher the Result Resolution, the finer the
mesh will be and the stricter convergence criteria will be set. Thus, Result Resolution
determines the balance between results precision and computation time. Entering
values for the minimum gap size and minimum wall thickness is important when you
have small features. Accurately setting these values ensures that the small features of
the model will not be "passed over" by the mesh. For our model we type the value of
the minimum flow passage as the minimum gap size.*

Select the **Manual specification of the minimum gap size** check box. Type the value
of 0.0093 m for the **Minimum gap size**.

Click **Finish**.

Now Flow Simulation creates a new configuration with the Flow Simulation data
attached.

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

℻ *Notice that the new configuration has the
name that you entered in the Wizard.*

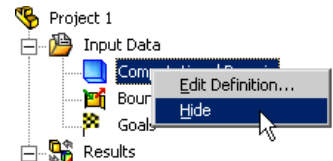Go to the **Flow Simulation Analysis Tree** and expand all the items in the tree.
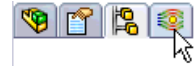
    📖 *We will use the Flow Simulation Analysis Tree to define our analysis, just as you use the FeatureManager design tree to design your models. The Flow Simulation analysis tree is fully customizable; anytime you can select which folders are shown and which folders are hidden. A hidden folder becomes visible when you add a new feature of the corresponding type. The folder remains visible until the last feature of this type is deleted.*
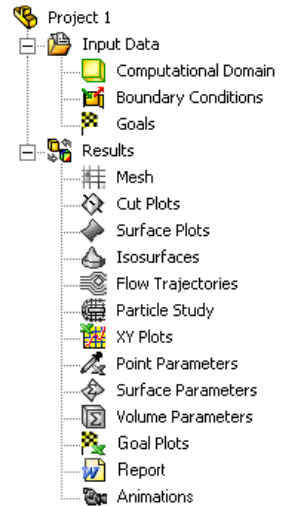
Right-click the **Computational Domain** icon and select **Hide** to hide the wireframe box.

    📖 *The Computational Domain icon is used to modify the size and visualization of the volume being analyzed. The wireframe box enveloping the model is the visualization of the limits of the computational domain.*

Boundary Conditions

A **boundary condition** is required where fluid enters or exits the model and can be specified as a Pressure, Mass Flow Rate, Volume Flow Rate or Velocity.

1 In the Flow Simulation Analysis Tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.
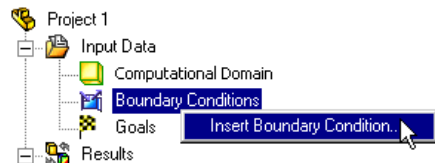
2   Select the **inner** face of the **Lid <1>** part
as shown. (To access the inner face, right-
click the **Lid <1>** in the graphics area and

choose **Select Other** 🔲, move the
mouse pointer over items in the list until
the inner face is highlighted, then click
the left mouse button).
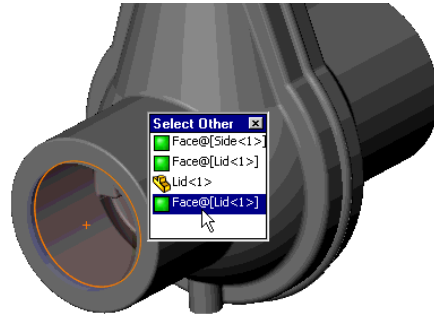
3   Select **Flow Openings** 🔁 and **Inlet
Mass Flow**.
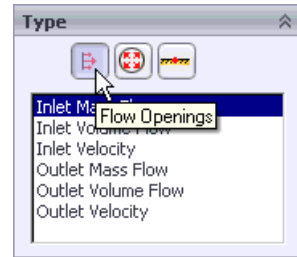
4   Set the **Mass Flow Rate Normal to Face** $\dot{m}$ to
`0.5 kg/s`.
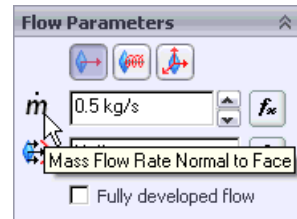
5   Click **OK** ✔. The new **Inlet Mass Flow 1** item
appears in the Flow Simulation Analysis tree.

📖 *With the definition just made, we told Flow Simulation that at this opening 0.5
kilogram of water per second is flowing into the valve. Within this dialog we can also
specify swirling of the flow, a non-uniform profile and time-dependent properties of the
flow. The mass flow rate at the outlet does not need to be specified due to the
conservation of mass; inlet mass flow rate equals outlet mass flow rate. Therefore, a
different condition must be specified, such as outlet pressure.*

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

7 In the Flow Simulation Analysis Tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.

8 Select **Pressure Openings** and **Static Pressure**.

9 Keep the defaults under **Thermodynamic Parameters**, **Turbulence Parameters**, **Boundary Layer** and **Options**.
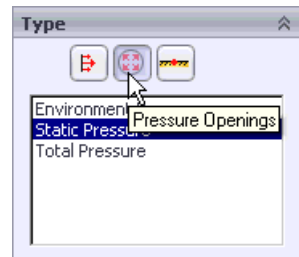
**10** Click **OK** . The new **Static Pressure 1** item appears in the Flow Simulation Analysis tree.
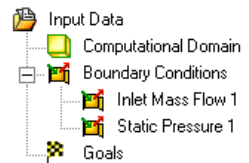
📖 *With the definition just made, we told Flow Simulation that at this opening the fluid exits the model to an area of static atmospheric pressure. Within this dialog we can also set a time-dependent properties pressure.*

Define the Engineering Goal

1 Right-click the **Goals** icon in the Flow Simulation Analysis Tree and select **Insert Surface Goals**.
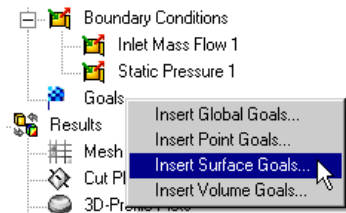
2   Click the **Flow Simulation Analysis Tree** tab and click the **Inlet Mass Flow 1** item to select the face where the goal is going to be applied.
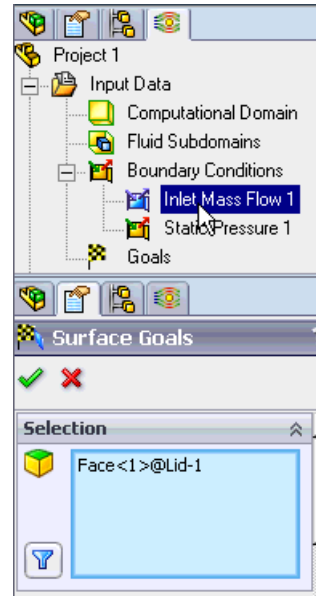
3   In the **Parameter** table, select the **Av** check box in the **Static Pressure** row. The already selected **Use for Conv.** check box means that the created goal will be used for convergence control.
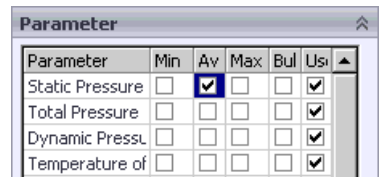
📖 *If the* **Use for Conv.** *(Use for Convergence Control) check box is not selected, the goal will not influence the calculation stopping criteria. Such goals can be used as monitoring parameters to give you additional information about processes in your model without influencing the other results and the total calculation time.*

4   Click **OK** ✔ . The new **SG Av Static Pressure 1** item appears in the Flow Simulation Analysis tree.
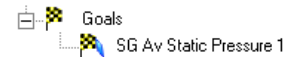
📖 *Engineering goals are the parameters of interest. Setting goals is a way of conveying to Flow Simulation what you are trying to get out of the analysis, as well as a way to reduce the time Flow Simulation needs to reach a solution. By setting a parameter as a project goal you give Flow Simulation information about parameters that are important to converge upon (the parameters selected as goals) and parameters that can be computed with less accuracy (the parameters not selected as goals) in the interest of the calculation time. Goals can be set throughout the entire domain (Global Goals), within a selected volume (Volume Goals), for a selected surface area (Surface Goals), or at given point (Point Goals). Furthermore, Flow Simulation can consider the average value, the minimum value or the maximum value of the goal. You can also define an Equation Goal that is a goal defined by an equation involving basic*

mathematical functions with existing goals and input data parameters as variables. The equation goal allows you to calculate the parameter of interest (i.e., pressure drop) and keeps this information in the project for later reference.
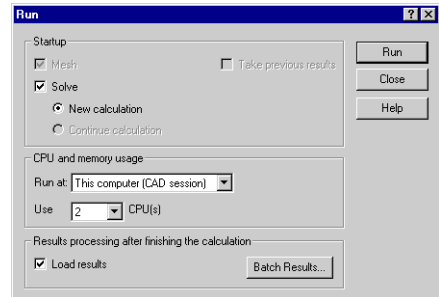
Click **File**, **Save**.

Solution

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

📖 *The already selected* **Load results** *check box means that the results will be automatically loaded after finishing the calculation.*
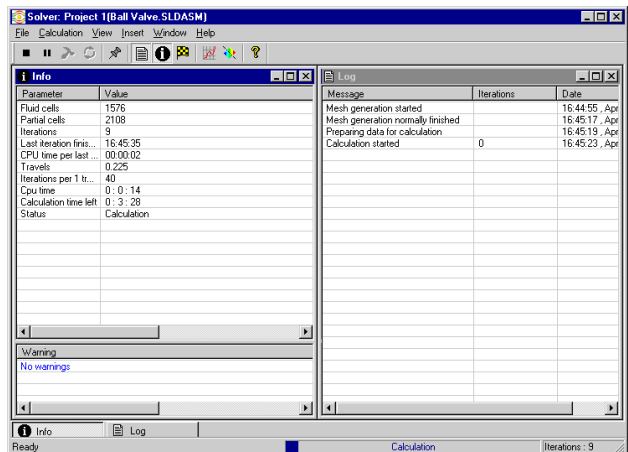
2 Click **Run**.

📖 *The solver takes less than a minute to run on a typical PC.*

Monitor the Solver

📖 *This is the solver monitor dialog box. By default, on the left is a log of each step taken in the solution process. On the right is the information dialog box with mesh information and warnings concerning the analysis. Do not be surprised when the error message "*A vortex crosses the pressure opening*" appears. We will explain this later during the demonstration.*

1   After the calculation has started and several first iterations has passed (keep your eye on the **Iterations** line in the **Info** window), click the **Suspend** ❚❚ button on the **Solver** toolbar.

📖 *We employ the Suspend option only due to extreme simplicity of the current example, which otherwise could be calculated too fast, leaving you not enough time to perform the subsequent steps of monitoring. Normally you can use the monitoring tools without suspending the calculation.*

2   Click **Insert Goal Plot** 📈 on the **Solver** toolbar. The **Add/Remove Goals** dialog box appears.

**3**   Select the **SG Average Static Pressure 1** in the **Select goals** list and click **OK**.

📖 *This is the Goals dialog box and each goal created earlier is listed in the table at top. Here you can see the current value and graph for each goal as well as the current progress towards completion given as a percentage. The progress value is only an estimate and the rate of progress generally increases with time.*
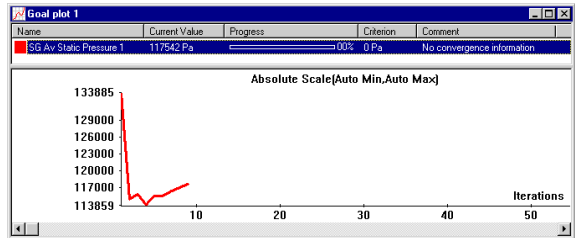
4   Click **Insert Preview** 🔷 on the **Solver** toolbar.

5   This is the **Preview Settings** dialog box. Selecting any SolidWorks plane from the **Plane name** list and pressing **OK** will create a preview plot of the solution in
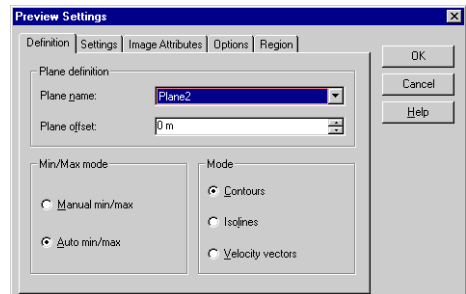
that plane. For this model `Plane2` is a good choice to use as the preview plane.

  📖 *The preview allows you to look at the results while the calculation is still running. This helps to determine if all the boundary conditions are correctly defined and gives the user an idea of how the solution will look even at this early stage. At the start of the run the results might look odd or*

change abruptly. However, as the run progresses these changes will lessen and the results will settle in on a converged solution. The result can be displayed either in contour-, isoline- or vector-representation.
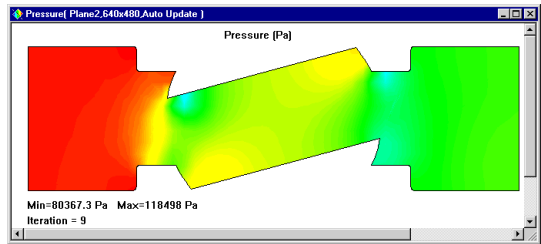
6   Click the **Suspend** ❚❚ button again to let the solver go on.

7   When the solver is finished, close the monitor by clicking **File**, **Close**.

Adjust Model Transparency

Click **Flow Simulation**, **Results**, **Display**, **Transparency** and set the model transparency to 0.75.

  📖 *The first step for results processing is to create a transparent view of the geometry, a 'glass-body'. This way you can easily see where cut planes etc. are located with respect to the geometry.*
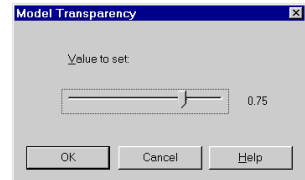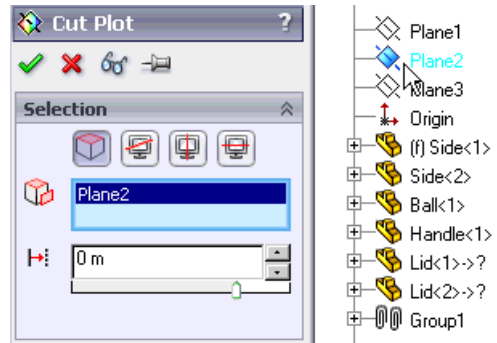

Cut Plots

1   In the Flow Simulation Analysis tree, right-click the **Cut Plots** icon and select **Insert**.
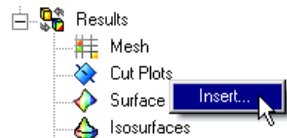
2   Specify a plane. Choose Plane 2 as
the cut plane. To do this, in the flyout
FeatureManager design tree select
Plane 2.

3   Click **OK** ✔ .

The plot you will see looks something like
this.

 📕 *A cut plot displays the distribution of the selected
parameter on a certain SolidWorks plane. It can be
represented as a contour plot, isolines, vectors, or as
arbitrary combination of the above (e.g. contours with
overlaid vectors).*
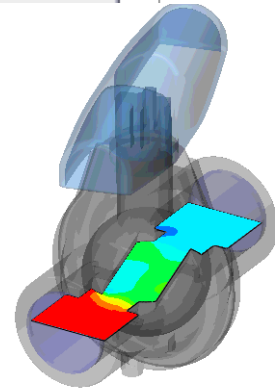
 📕 *If you want to access additional options for this and
other plots, you can double-click on the color bar. Some options
available here include changing the displayed parameter as well as
changing the min/max plot values. The best way to learn each of
these options is thorough experimentation.*

4   Change the contour cut plot to a vector cut plot. To
do this, right-click the **Cut Plot 1** icon and select
Edit Definition.
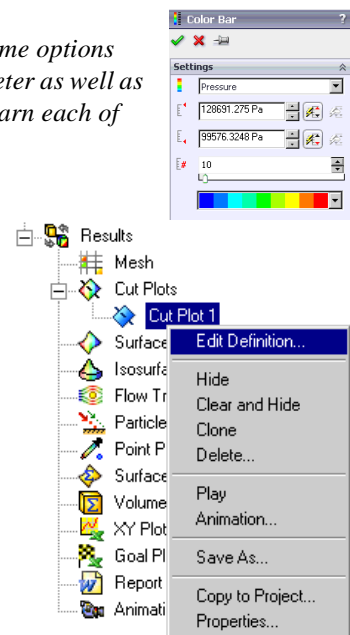
5   Under **Display**, clear **Contours** and select **Vectors** .

6   Click **OK** ✔.
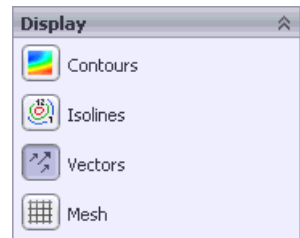
This is the plot you should see.

   📖 *The vectors size and spacing can be controlled under the* **Vectors**.
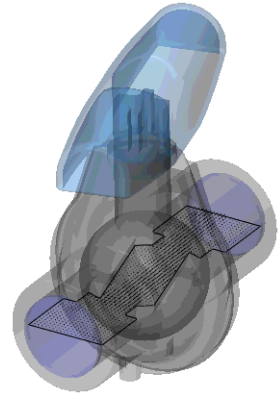
Surface Plots

Right-click the **Cut Plot 1** icon and select **Hide**.

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

2   Select the **Use all faces** check box.

   📖 *The same basic options are available for Surface Plots as for Cut Plots. Feel free to experiment with different combinations on your own.*
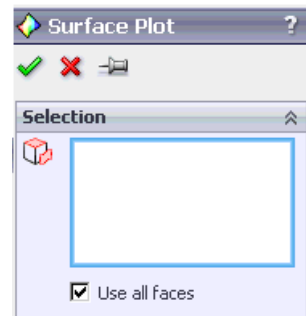
3   Click **OK** ✔ and you get the following picture:

📖 *This plot shows the pressure distribution on all faces of the valve in contact with the fluid. You can also select one or more single surfaces for this plot, which do not have to be planar.*
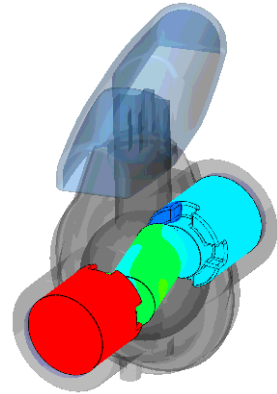
Isosurface Plots

Right-click the **Surface Plot 1** icon and select **Hide**.

1   Right-click the **Isosurfaces** icon and select **Insert**.

2   Keep the default value under **Value 1**.

3   Under **Appearance**, select **Grid** 🔴 and click **OK** ✔. This is the plot that will appear.

📖 *The Isosurface is a 3-Dimensional surface created by Flow Simulation at a constant value for a specific variable.*
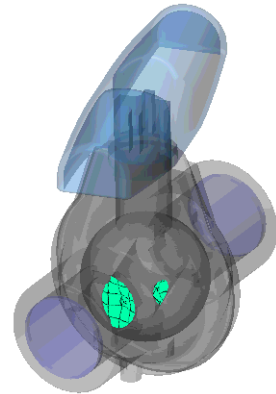
4 Right-click the **Isosurface 1** icon and select **Edit Definition**. Enable **Value 2** and specify some value in the appeared box that is different to the **Value 1**.

5 Click **OK** ✔ .
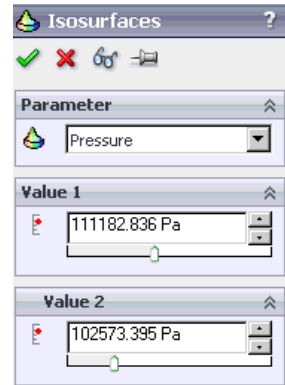
You should see something similar to this image.

📖 *The isosurface is a useful way of determining the exact 3D area, where the flow reaches a certain value of pressure, velocity or other parameter.*
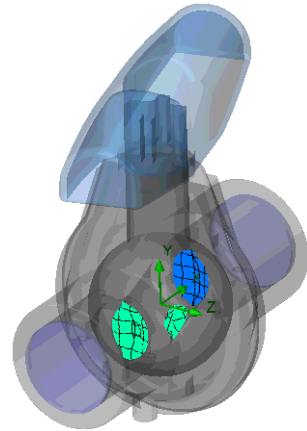
Flow Trajectory Plots

Right-click the **Isosurfaces** icon and select **Hide**.

1 Right-click the **Flow Trajectories** icon and select **Insert**.
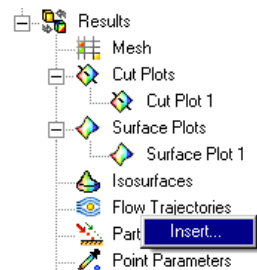
2 Click the Flow Simulation Analysis Tree tab and then click the **Static Pressure 1** item to select the inner face of the **Lid <2>**.

3 Set the **Number of Points** 🔣 to 16.

4 Under **Appearance**, set **Draw Trajectories as** 🎿 **Bands**.
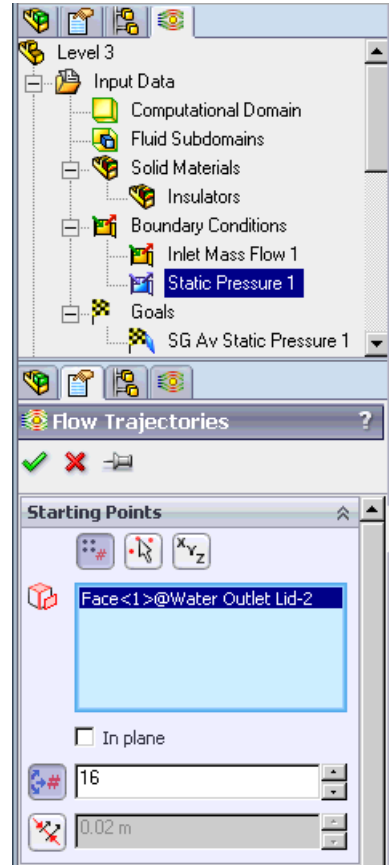
5 Click **OK** ✔ and your model should look like the following:

📖 *Using Flow trajectories you can show the flow streamlines. Flow trajectories provide a very good image of the 3D fluid flow. You can also see how parameters change along each trajectory by exporting data into Microsoft® Excel®. Additionally, you can save trajectories as SolidWorks reference curves.*

📖 *For this plot we selected the outlet lid (any flat face or sketch can be selected) and therefore every trajectory crosses that selected face. Notice the trajectories that are entering and exiting through the exit lid. This is the reason for the warning we received during the calculation. Flow Simulation warns us of inappropriate analysis conditions so that we do not need to be CFD experts. When flow both enters and exits the same*
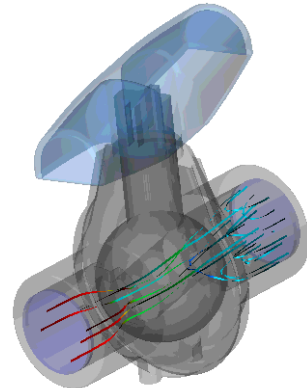
opening, the accuracy of the results will worsen. In a case like this, one would typically add the next component to the model (say, a pipe extending the computational domain) so that the vortex does not occur at opening.

XY Plots

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

We want to plot pressure and velocity along the valve. We have already created a SolidWorks sketch containing several lines.

This sketch work does not have to be done ahead of time and your sketch lines can be created after the calculation is finished. Take a look at Sketch1 in the FeatureManager design tree.
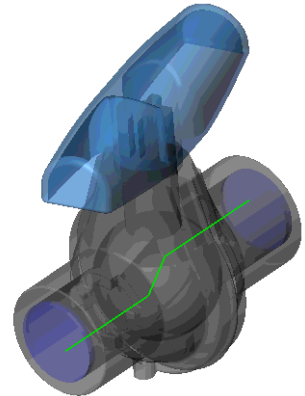
1   Right-click the **XY Plots** icon and select **Insert**.

2   Choose **Velocity** and **Pressure** as physical **Parameters**. Select **Sketch1** from the flyout FeatureManager design tree.

    Leave all other options as defaults.
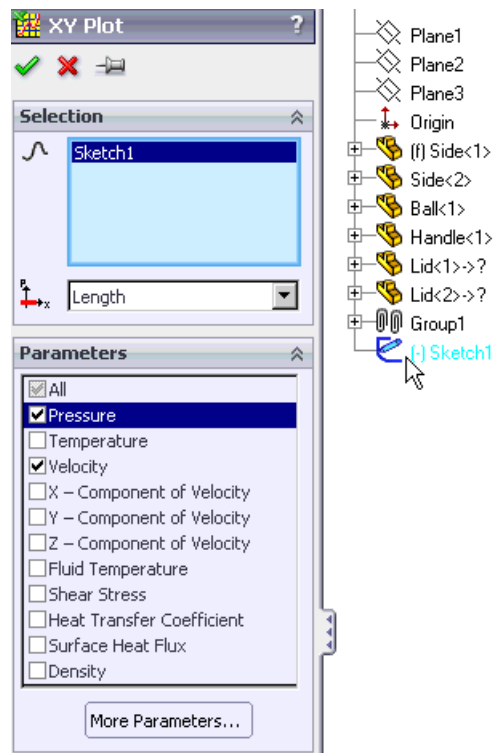
3 Click **OK** ✔. Excel will open and generate two columns of data points together with two charts for Velocity and for Pressure, respectively. One of these charts is shown below. You will need to toggle between different sheets in Excel to view each chart.

📖 *The XY Plot allows you to view any result along sketched lines. The data is put directly into Excel.*
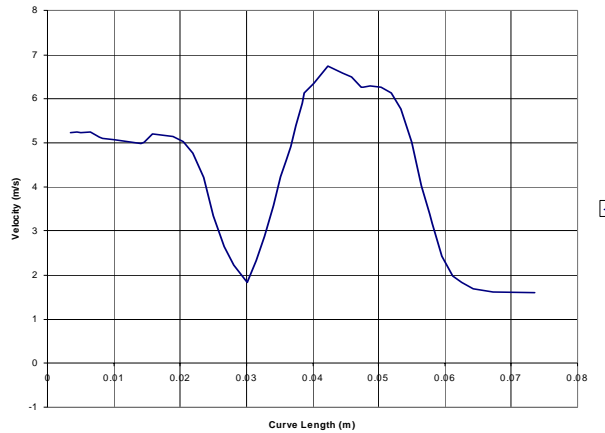
Surface Parameters

Surface Parameters is a feature used to determine the values of pressure, forces, heat fluxes as well as many other variables on any face in your model contacting the fluid. For this type of analysis, a calculation of the average static pressure drop from the valve inlet to outlet would probably be of some interest.

1 Right-click the **Surface Parameters** icon and select **Insert**.
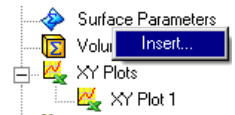
2 Click the Flow Simulation Analysis Tree tab and then click the **Inlet Mass Flow 1** item to select the inner face of the **Lid <1>**.

3 Under **Parameters**, select **All**.

**4** Click **Show**. The calculated parameters values are displayed on the pane at the bottom of the screen. Local parameters are displayed at the left side of the bottom pane, while integral parameters are displayed at the right side.

5 Take a look at the local parameters.

Local Parameter	Minimum	Maximum	Average	Bulk Average	Surface Area [m^2]
Pressure [Pa]	127597	127743	127670	127671	0.000391595
Density [kg/m^3]	997.562	997.562	997.562	997.562	0.000391595
Velocity [m/s]	1.61943	1.61943	1.61943	1.61943	0.000391595
X – Component of Velocity [m/s]	1.61943	1.61943	1.61943	1.61943	0.000391595
Y – Component of Velocity [m/s]	0	0	0	0	0.000391595
Z – Component of Velocity [m/s]	5.6542e-015	5.6542e-015	5.6542e-015	5.6542e-015	0.000391595
Fluid Temperature [K]	293.2	293.2	293.2	293.2	0.000391595

The average static pressure at the inlet face is shown to be about 127670 Pa. We already know that the outlet static pressure is 101325 Pa since we have specified it previously as a boundary condition. So, the average static pressure drop through the valve is about 27000 Pa.

6 Close the **Surface Parameters** dialog.
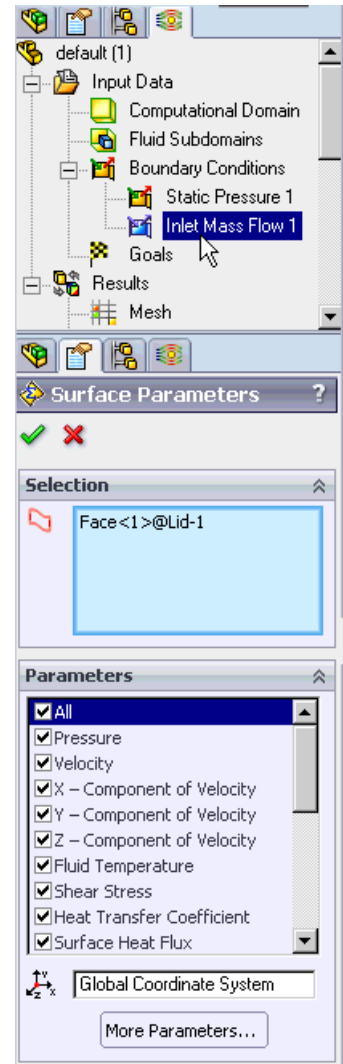
Analyze a Design Variant in the SolidWorks Ball part

This section is intended to show you how easy it is to analyze design variations. The variations can be different geometric dimensions, new features, new parts in an assembly – whatever! This is the heart of Flow Simulation and this allows design engineers to quickly and easily determine which designs have promise, and which designs are unlikely to be successful. For this example, we will see how filleting two sharp edges will influence the pressure drop through the valve. If there is no improvement, it will not be worth the extra manufacturing costs.

Create a new configuration using the SolidWorks Configuration Manager Tree.

1 Right-click the root item in the SolidWorks Configuration Manager and select **Add Configuration**.
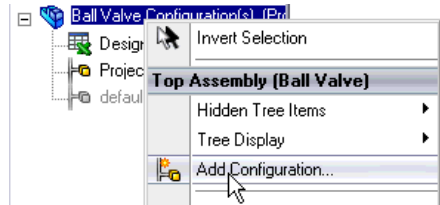
2 In the **Configuration Name** box type `Project 2`.

3 Click **OK** .
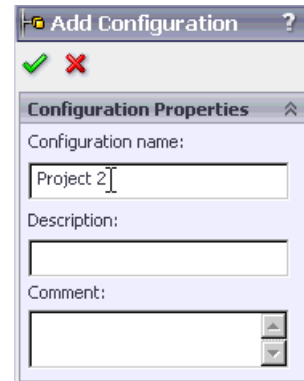
4 Go to FeatureManager design tree, right-click the **Ball** item and select **Open Part** 📂 . A new window **Ball.SLDPRT** appears.
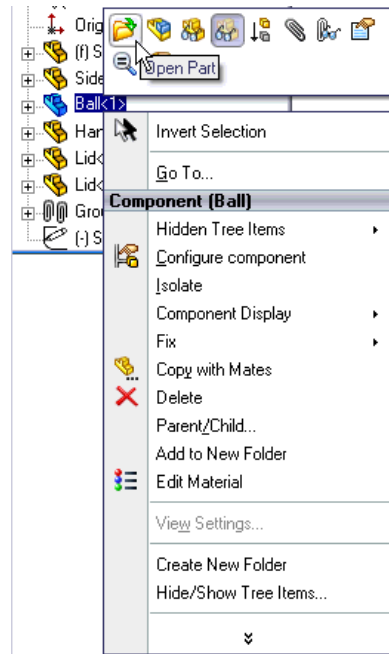
Create a new configuration using the SolidWorks Configuration Manager Tree.

1 Right-click the root item in the SolidWorks Configuration Manager and select **Add Configuration.**
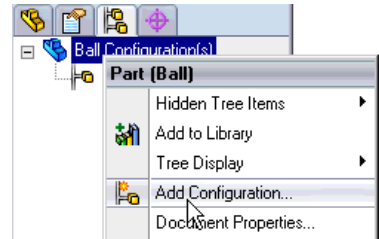
2 Name the new configuration as `1,5_fillet Ball`.
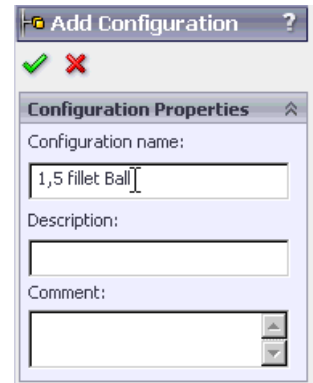
3 Click **OK** ✔ .

4 Add a 1,5 mm fillet to the shown face.
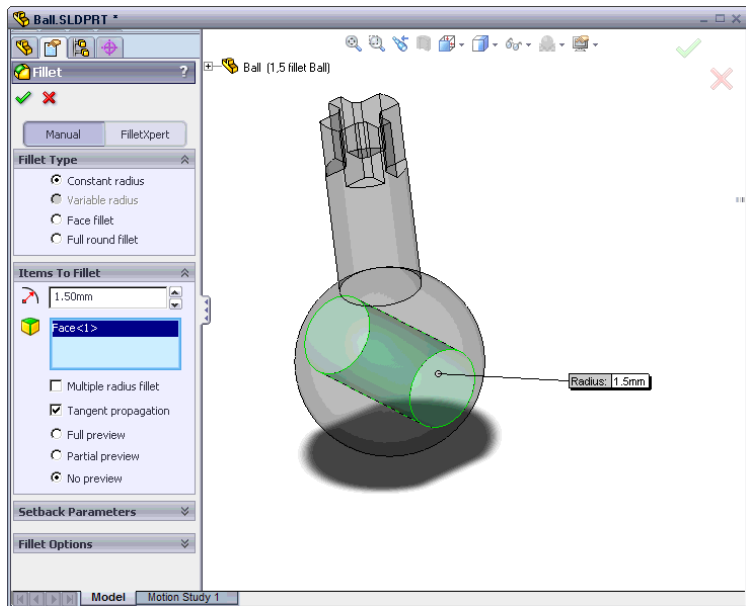
5   Switch back to the assembly window and select **Yes** in the message dialog box that appears. In the FeatureManager design tree right-click the **Ball** item and select **Component Properties**.

6   At the bottom of the **Component Properties** dialog box, under **Referenced configuration** change the configuration of the Ball part to the new filleted one.
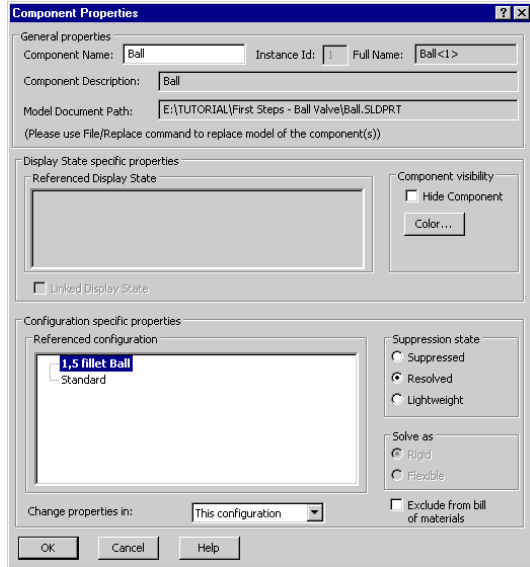
7   Click **OK** to confirm and close the dialog.

Now we have replaced the old ball with our new 1.5_fillet Ball. All we need to do now is re-solve the assembly and compare the results of the two designs. In order to make the results comparable with the previous model, it would be necessary to adjust the valve angle to match the size of the flow passage of the first model. In this example, we will not do this.

8   Activate **Project 1** by using the Configuration Manager Tree. Select **Yes** for the message dialog box that appears.
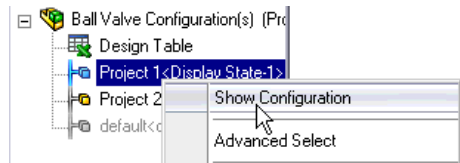
Clone the Project

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

2 Select **Add to existing**.

3 In the **Existing configuration** list select **Project 2**.

4 Click **OK**. Select **Yes** for each message dialog box that appears after you click **OK**.
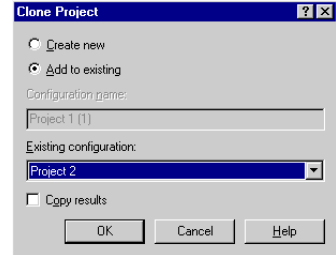
📖 *Now the Flow Simulation project we have chosen is added to the SolidWorks project which contains the geometry that has been changed. All our input data are copied, so we do not need to define our openings or goals again. The Boundary Conditions can be changed, deleted or added. All changes to the geometry will only be applied to this new configuration, so the old results are still saved.*

Please follow the previously described steps for solving and for viewing the results.

Analyze a Design Variant in the Flow Simulation Application

📖 *In the previous sections we examined how you could compare results from different geometries. You may also want to run the same geometry over a range of flow rates. This section shows how quick and easy it can be to do that kind of parametric study. Here we are going to change the mass flow to 0.75 kg/s.*

Activate the **Project 1** configuration.

1 Create a copy of the Project 1 project by clicking **Flow Simulation**, **Project**, **Clone Project**.

2 Type Project 3 for the new project name and click **OK**.
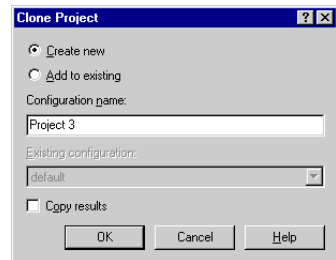
Flow Simulation now creates a new configuration. All our input data are copied, so we do not need to define our openings or goals again. The Boundary Conditions can be changed, deleted or added. All changes to the geometry will only be applied to this new configuration, so the old results remain valid. After changing the inlet flow rate value to 0.75 kg/s you would be ready to run again. Please follow the previously described steps for solving and for viewing the results.

Imagine being the designer of this ball valve. How would you make decisions concerning your design? If you had to determine whether the benefit of modifying the design as we have just done outweighted the extra costs, how would you do this? Engineers have to make decisions such as this every day, and Flow Simulation is a tool to help them make those decisions. Every engineer who is required to make design decisions involving fluid and heat transfer should use Flow Simulation to test their ideas, allowing for fewer prototypes and quicker design cycles.

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.
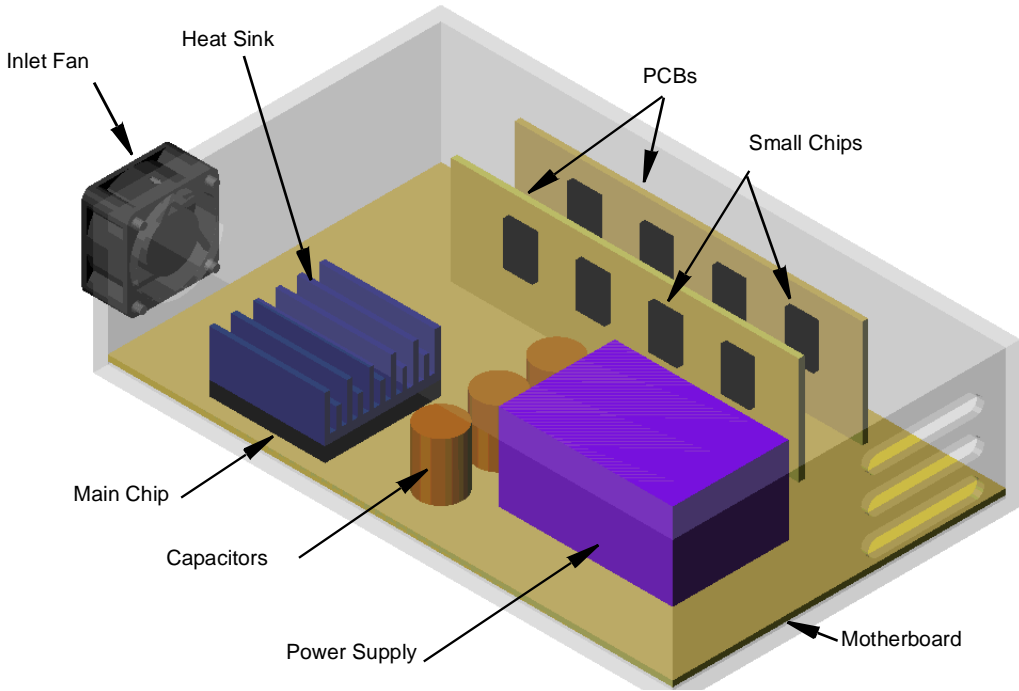
Preparing the Model

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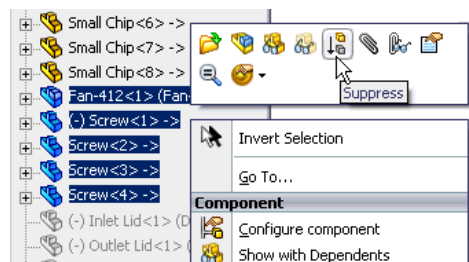
Suppressing fan and its screws leaves open five holes in the enclosure. Since we are going to perform an internal analysis, all the holes must be closed with lids.

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 🔼 .
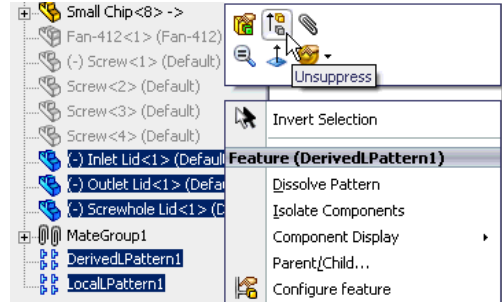
Now you can start with Flow Simulation.

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*
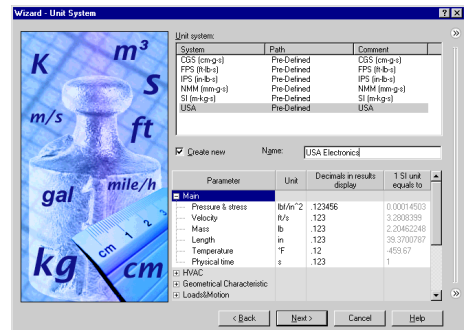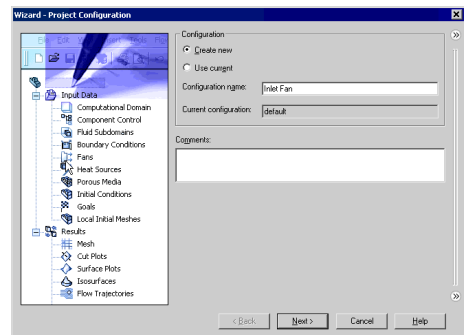
custom unit system. Both pre-defined and custom unit systems are stored in the **Engineering Database***. You can create the desired system of units in the* **Engineering Database** *or in the* **Wizard***.*

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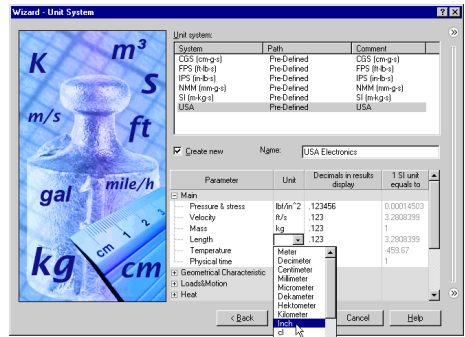
5 Next expand the **Heat** group in the **Parameter** tree.

    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 .
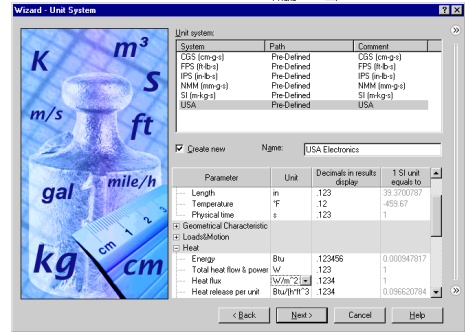
    Click **Next**.

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.*
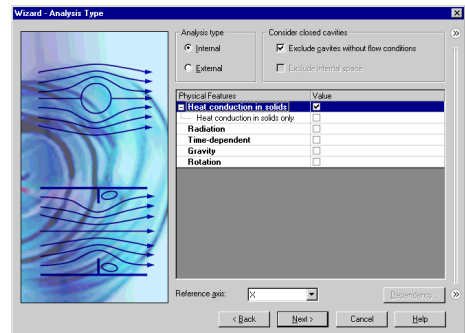
    Click **Next**.

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

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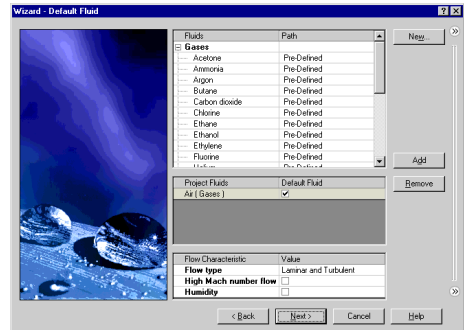
8 Expand the **Alloys** folder and click **Steel Stainless 321** to assign it as the **Default solid**.

📖 *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**.
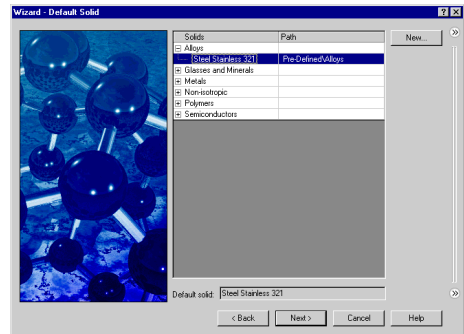
9 Select **Heat transfer coefficient** as **Default outer wall thermal condition** and specify the **Heat transfer coefficient** value of 5.5 W/m^2/K and **Temperature of external fluid** of 50°F. The entered value of heat transfer coefficient is automatically coverted to the selected system of units (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**.
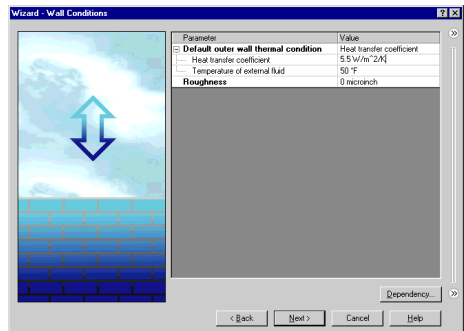
Although the initial temperature is more important for transient calculations to see how much time it takes to reach a certain temperature, in a steady-state analysis it is useful

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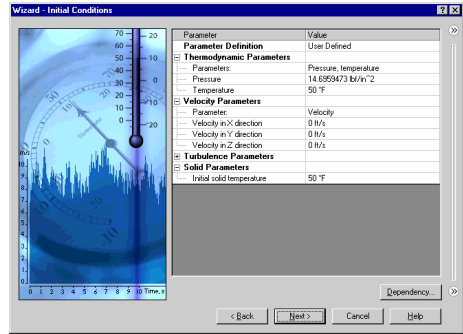
11 Accept the default **Result resolution** and keep the automatic evaluation of the **Minimum gap size** and **Minimum wall thickness**.

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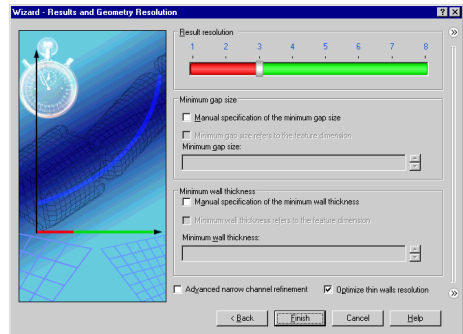
Click **Finish**. Now Flow Simulation creates a new configuration with the Flow Simulation project attached.

    📖 *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.
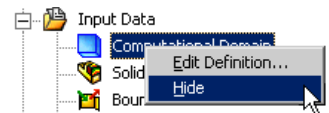
Define the Fan

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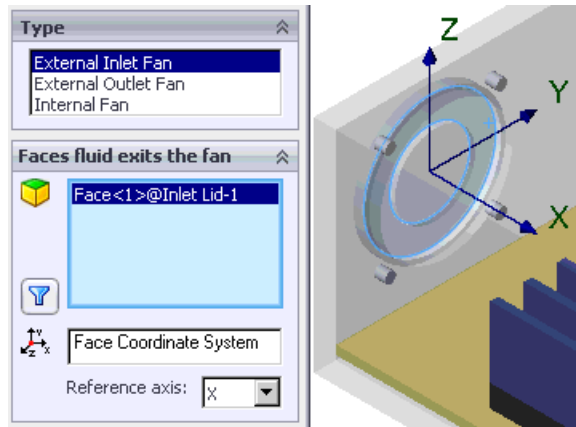
3 Under **Type**, select **External Inlet Fan**.

4 In the **Fan** list, under **Pre-Defined**, **Axial, Papst**, select the **Papst 412** item.
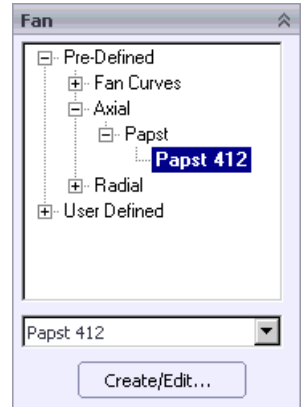
5 Under **Thermodynamic Parameters** check that the Ambient Pressure $P_\text{##}$ is the atmospheric pressure.

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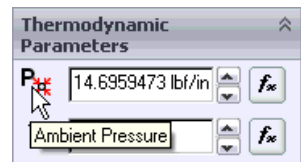
📖 *Face coordinate system is created automatically in the center of a planar face when you select this face as the face to apply the boundary condition or fan. The **X** axis of this coordinate system is normal to the face. The **Face coordinate system** is created only when one planar face is selected.*

7   Click **OK** ✔ . The new **Fans** folder and the **External Inlet Fan 1** item appear in the Flow Simulation Analysis tree.
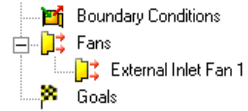
📖 *Now you can edit the **External Inlet Fan 1** item or add a new fan using Flow Simulation Analysis tree. This folder remains visible until the last feature of this type is deleted. You can also make a feature folder to be initially available in the tree. Right-click the project name item and select **Customize Tree** to add or remove folders.*
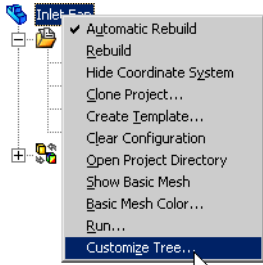
📖 *Since the outlet lids of the enclosure are at ambient atmospheric pressure, the pressure rise produced by the fan is equal to the pressure drop through the electronics enclosure.*

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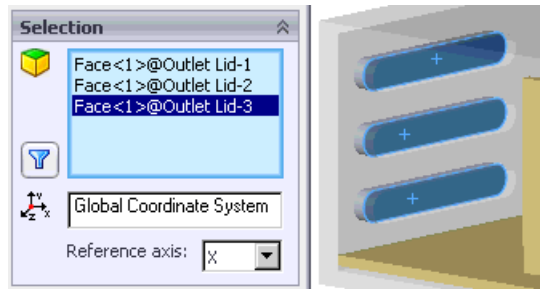
3 Select **Pressure Openings** 🔀
   and **Environment Pressure**.

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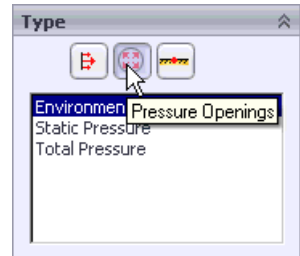
📖 *The Environment pressure condition is interpreted as a static pressure for outgoing
   flows and as a total pressure for incoming flows.*

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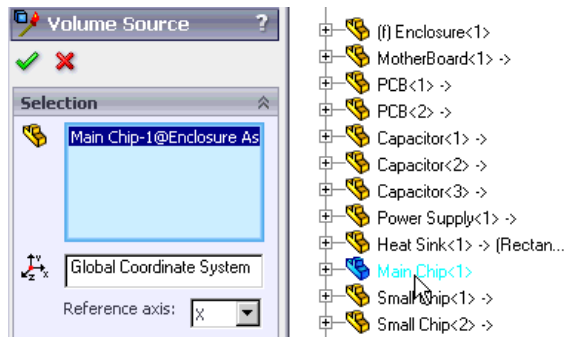
3 Select the **Heat Generation
   Rate** 🔲 as **Parameter**.

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

5 Click **OK** ✔.
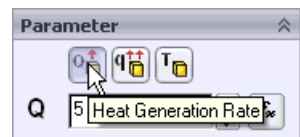
6 In the Flow Simulation Analysis tree, click-pause-click the new **VS Heat Generation Rate 1** item and rename it to `Main Chip`.
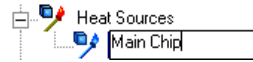
     Volume Heat Sources allow you to specify the heat generation rate (e.g. in Watts) or the volumetric heat generation rate (e.g. in Watts per volume) or a constant temperature boundary condition for a volume. It is also possible to specify Surface Heat Sources in terms of heat transfer rate (e.g. in Watts) 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** $T_{\square}$ 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`.
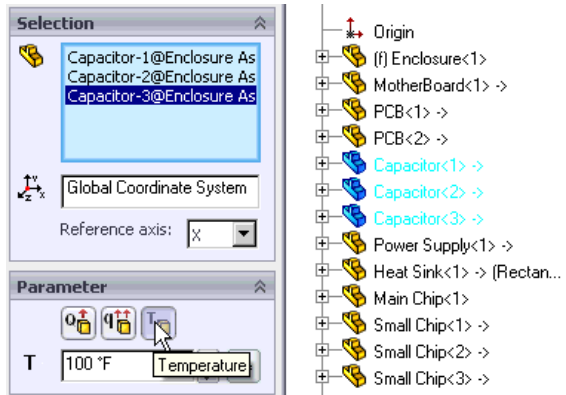
Click anywhere in the graphic area to clear the selection.

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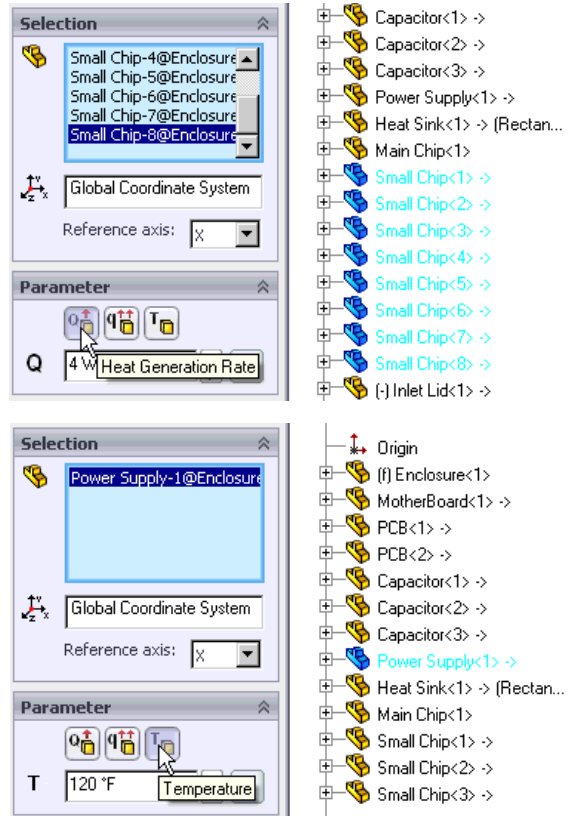
7 Rename the source applied to the chips to Small Chips and the source for the power supply to Power Supply.

Click **File**, **Save**.
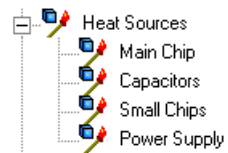
Create a New Material

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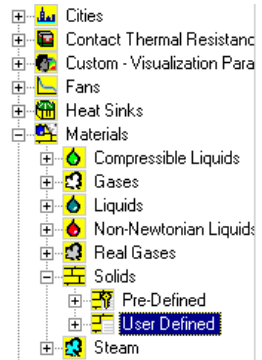
Property	Value
Name	Tutorial PCB
Comments	Isotropic PCB
Density	1120 kg/m^3
Specific heat	1400 J/(kg*K)
Conductivity type	Isotropic
Thermal conductivity	10 W/(m*K)
Melting temperature	390 K

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**.
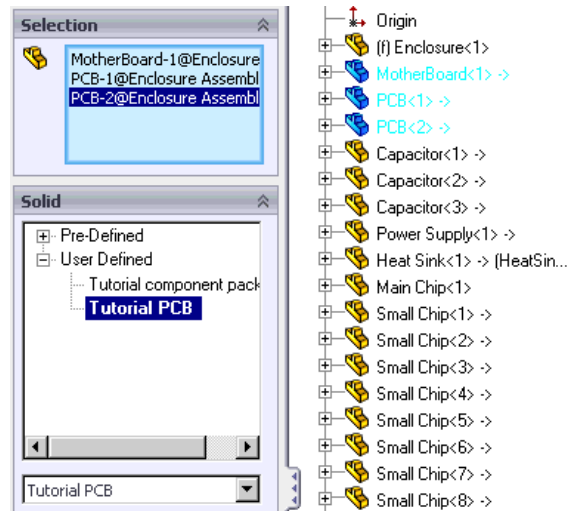
4 Click **OK** ✔ .

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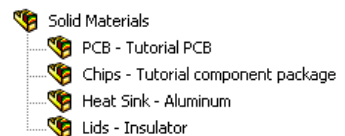
Click **File**, **Save**.

Define Engineering Goals

Specifying Volume Goals

**1** In the Flow Simulation analysis tree, right-click the **Goals** icon and select **Insert Volume Goals**.
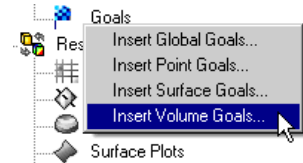
2 In the flyout FeatureManager design tree select all **Small Chip** components.

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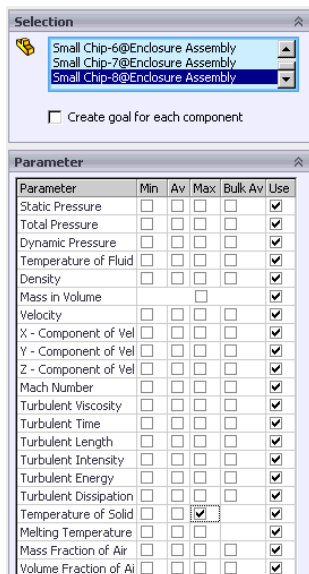
5 Click **OK**. The new **VG Max Temperature of Solid 1** item appears in the Flow Simulation Analysis tree.

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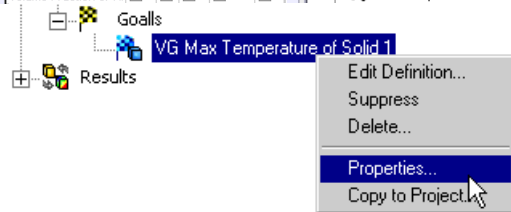
7 Right-click the **Goals** icon and select **Insert Volume Goals**.

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.

Selection			
🛇 Main Chip-1@Enclosure Assembly			

Parameter	Min	Av	Max	Bulk Av	Use
Static Pressure	☐	☐	☐	☐	☑
Total Pressure	☐	☐	☐	☐	☑
Dynamic Pressure	☐	☐	☐	☐	☑
Temperature of Fluid	☐	☐	☐	☐	☑
Density	☐	☐	☐	☐	☑
Mass in Volume				☐	☑
Velocity	☐	☐	☐	☐	☑
X - Component of Vel	☐	☐	☐	☐	☑
Y - Component of Vel	☐	☐	☐	☐	☑
Z - Component of Vel	☐	☐	☐	☐	☑
Mach Number	☐	☐	☐	☐	☑
Turbulent Viscosity	☐	☐	☐	☐	☑
Turbulent Time	☐	☐	☐	☐	☑
Turbulent Length	☐	☐	☐	☐	☑
Turbulent Intensity	☐	☐	☐	☐	☑
Turbulent Energy	☐	☐	☐	☐	☑
Turbulent Dissipation	☐	☐	☐	☐	☑
Temperature of Solid	☐	☐	☑		☑
Melting Temperature	☐	☐	☐		☑
Mass Fraction of Air	☐	☐	☐	☐	☑
Volume Fraction of Ai	☐	☐	☐	☐	☑

Design tree:
- ↳ Origin
- ⊞ 🛇 (f) Enclosure<1>
- ⊞ 🛇 MotherBoard<1> ->
- ⊞ 🛇 PCB<1> ->
- ⊞ 🛇 PCB<2> ->
- ⊞ 🛇 Capacitor<1> ->
- ⊞ 🛇 Capacitor<2> ->
- ⊞ 🛇 Capacitor<3> ->
- ⊞ 🛇 Power Supply<1> ->
- ⊞ 🛇 Heat Sink<1> -> (HeatSin..
- ⊞ 🛇 Main Chip<1>
- ⊞ 🛇 Small Chip<1> ->
- ⊞ 🛇 Small Chip<2> ->
- ⊞ 🛇 Small Chip<3> ->
- ⊞ 🛇 Small Chip<4> ->
- ⊞ 🛇 Small Chip<5> ->
- ⊞ 🛇 Small Chip<6> ->
- ⊞ 🛇 Small Chip<7> ->
- ⊞ 🛇 Small Chip<8> ->
- 🛇 Fan-412<1> (Fan-412)
- 🛇 (-) Screw<1> (Default)
- 🛇 Screw<2> (Default)
- 🛇 Screw<3> (Default)
- 🛇 Screw<4> (Default)
- ⊞ 🛇 (-) Inlet Lid<1> ->
- ⊞ 🛇 (-) Outlet Lid<1> ->

Specifying Surface Goals

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

- ⊟ 🖈 Goals
  - Insert Global Goals...
  - Insert Point Goals...
  - Insert Surface Goals... ⮜
  - Insert Volume Goals...
  - Insert Equation Goal...
  - Delete All
- 🔷 Surface Plots

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.*
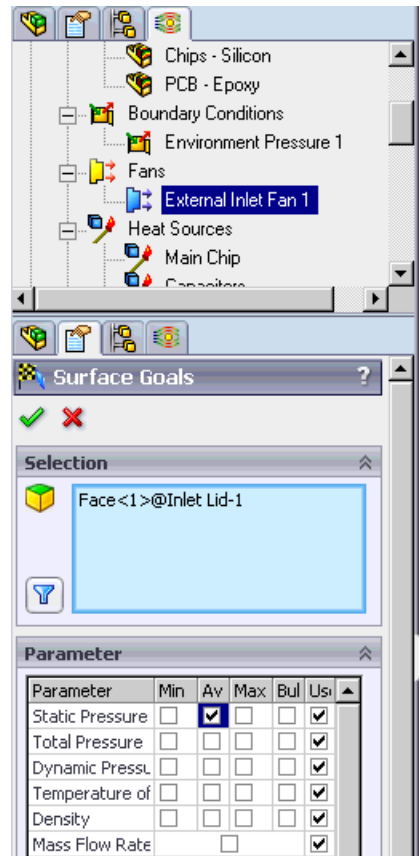
5   Under **Name Template**, located at the bottom of the PropertyManager, click **Inlet** ⟨⁺⃗⟩ and then remove the <Number> field from the **Name Template** box.
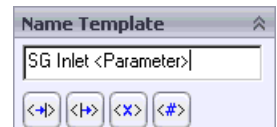
6   Click **OK** ✔ . The new **SG Inlet Av Static Pressure** goal appears.

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.
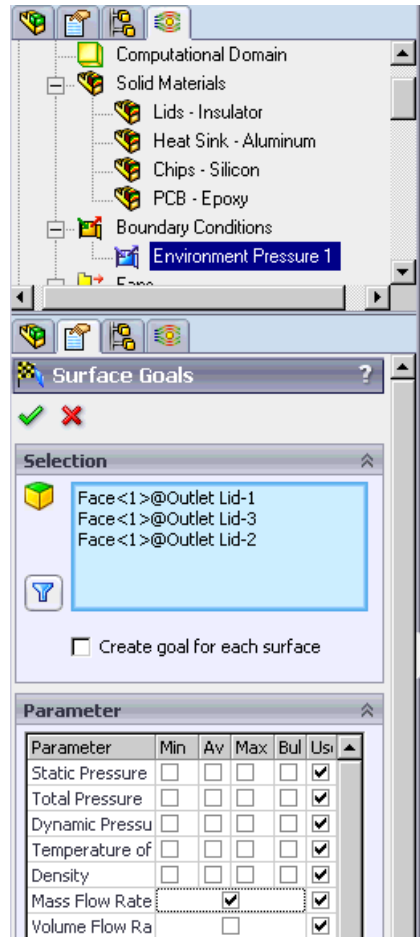
11 Under **Name Template**, located at the bottom of the PropertyManager, click **Outlet** |⟨⊦⟩| and then remove the <Number> field from the **Name Template**.
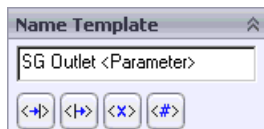
12 Click **OK** ✓ . The **SG Outlet Mass Flow Rate** goal appears.

Specifying Global Goals

1 Right-click the **Goals** icon and select **Insert Global Goals**.
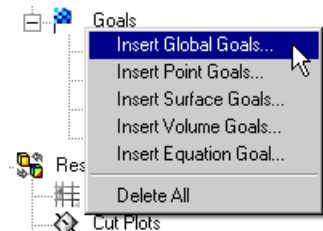
2   In the **Parameter** table select the **Av** check boxes in the **Static Pressure** and **Temperature of Fluid** rows and accept selected **Use for Conv.** (**Use for Convergence Control**) check box to use these goals for convergence control.
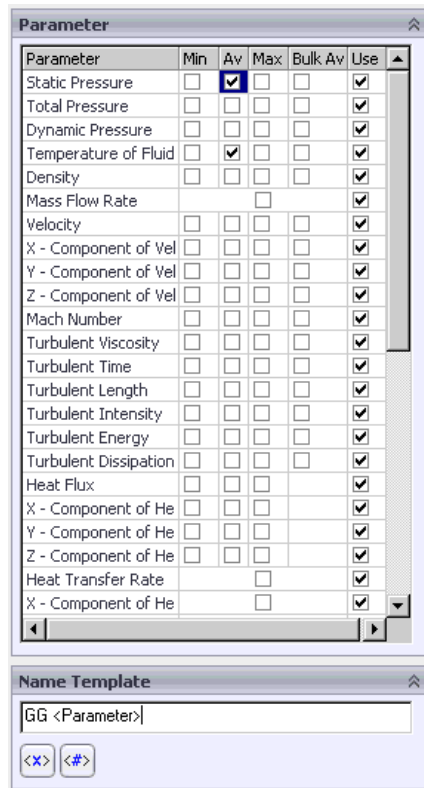
3   Remove the <Number> field from the

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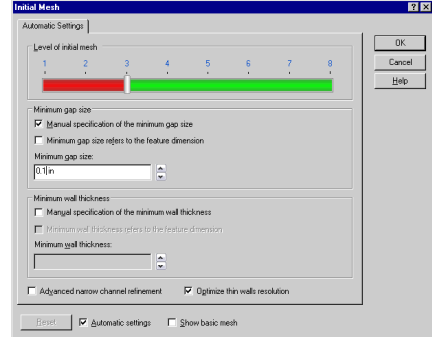
3   Enter `0.1 in` for the **Minimum gap size** (i.e. passage between the fins of the heat sink).

📖 *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.*
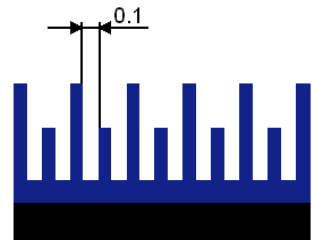
Click **OK**.

Solution

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

2   Click **Run**.

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

ℒ *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.
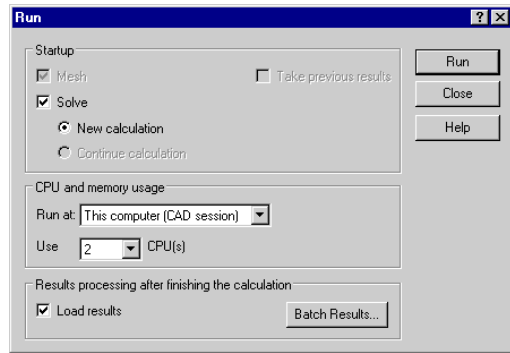
Viewing the Goals

1   Right-click the **Goal Plots** icon under **Results** and select **Insert**.
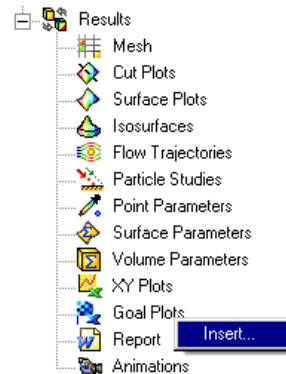
2   Click **All** in the **Goals** dialog.

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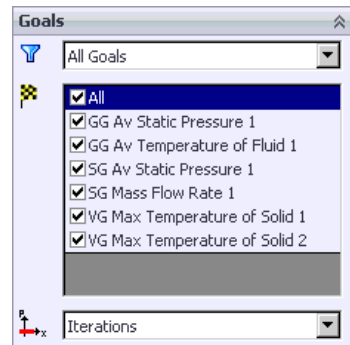
An Excel spreadsheet with the goal results will be open. The first sheet will show a table summarizing the goals.

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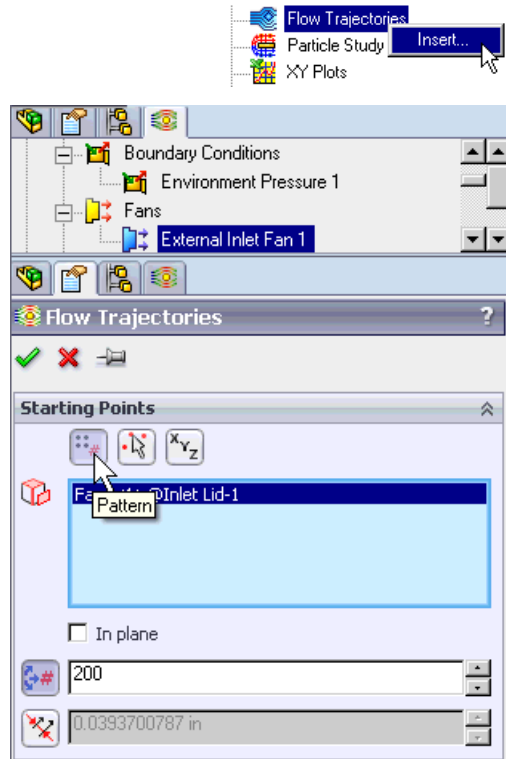
This is the picture you should see.
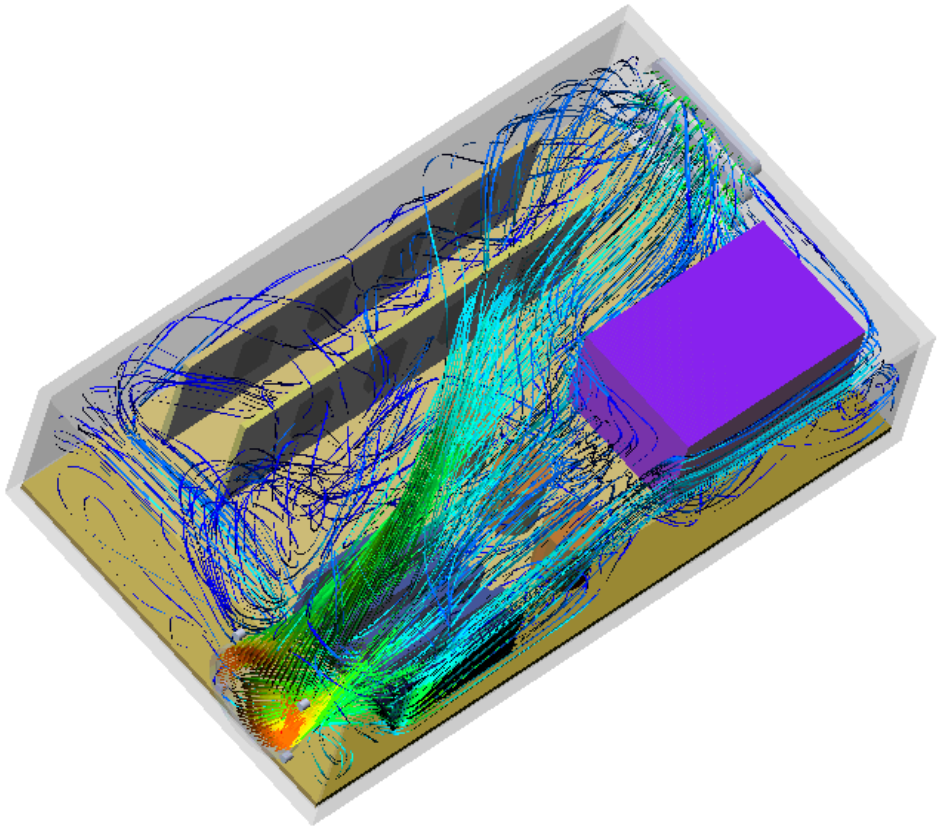
Notice that there are only a few trajectories along the adjacent to the wall **PCB<2>** and this may cause problems with cooling of the chips placed on this PCB. Additionally the blue color indicates low velocity in front of this **PCB<2>** .

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.
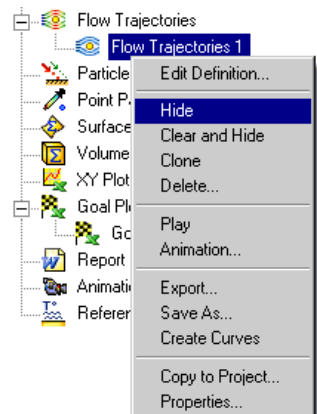
Cut Plots

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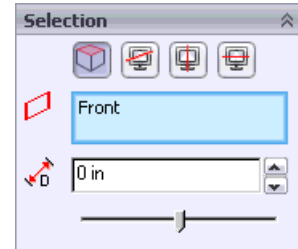
4   Set the **Number of levels** to 30.

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.
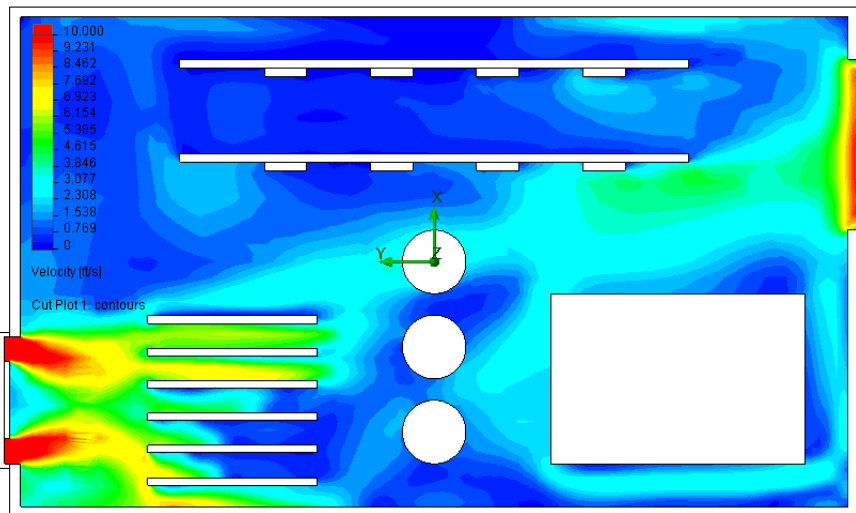
Let us now look at the fluid temperature.

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

8 Change the **Offset** ⊢⦂ to -0.3 in.
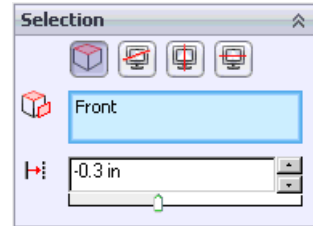
9 Change the **Parameter** from **Velocity** to **Fluid Temperature**.

10 Change the **Min** and **Max** values to 50 and 120 F respectively.
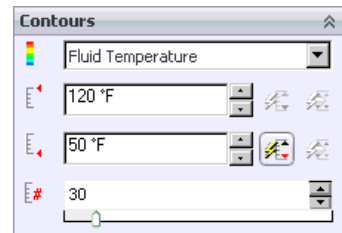
11 Under **Display**, select **Vectors** ⟋.

12 Under the appeared **Vectors** tab, make sure that the **Parameter** is set to **Velocity** and then select **Adjust Minimum and Maximum** ⟨⟩.
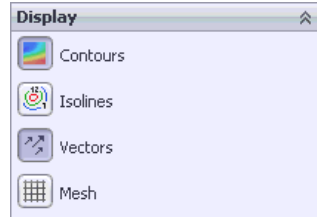
13 Set the **Max** value to 1 ft/s.

📖 *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.
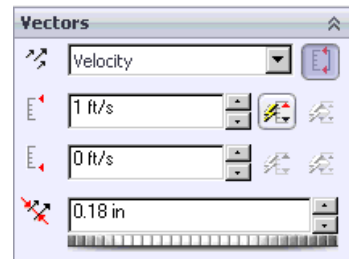
15 Click **OK** ✅ .
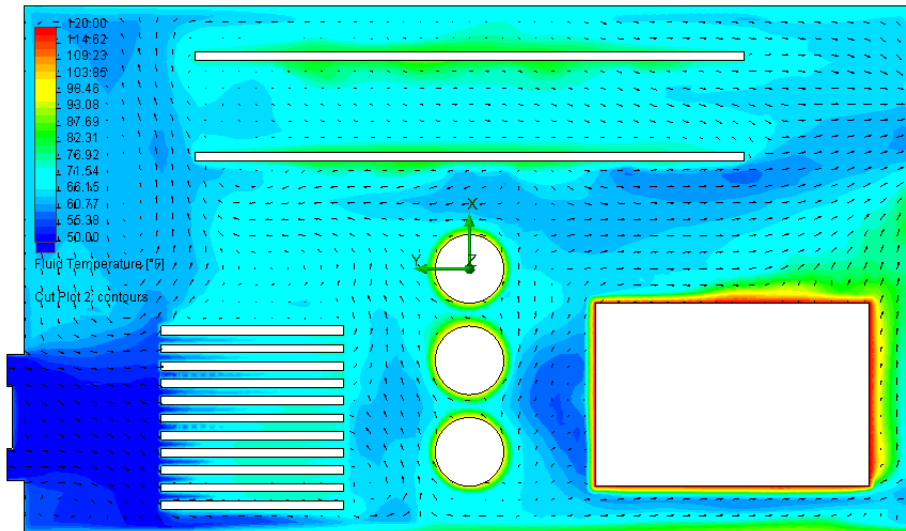
Right-click the **Cut Plot 1** item and select **Hide**. Let us now display solid temperature.

Surface Plots

1 Right-click the **Surface Plots** item and select **Insert**.
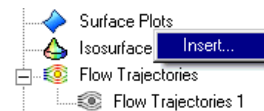
2 In the flyout FeatureManager design tree click the **Main Chip**, **Heat Sink** and all **Small Chip** components to select their surfaces.
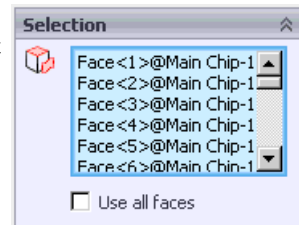
3 Under **Contours**, change the **Parameter** to **Solid Temperature**.

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

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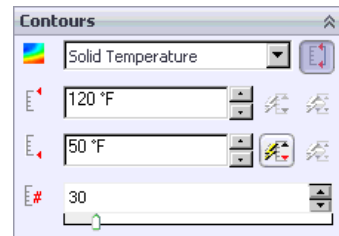
6 Repeat steps 1 and 2 and select the **Power Supply** and all **Capacitor**s components, then click **OK** ✔.

7 On the **View** toolbar click **Wireframe** 🔲 to show only the face outlines.
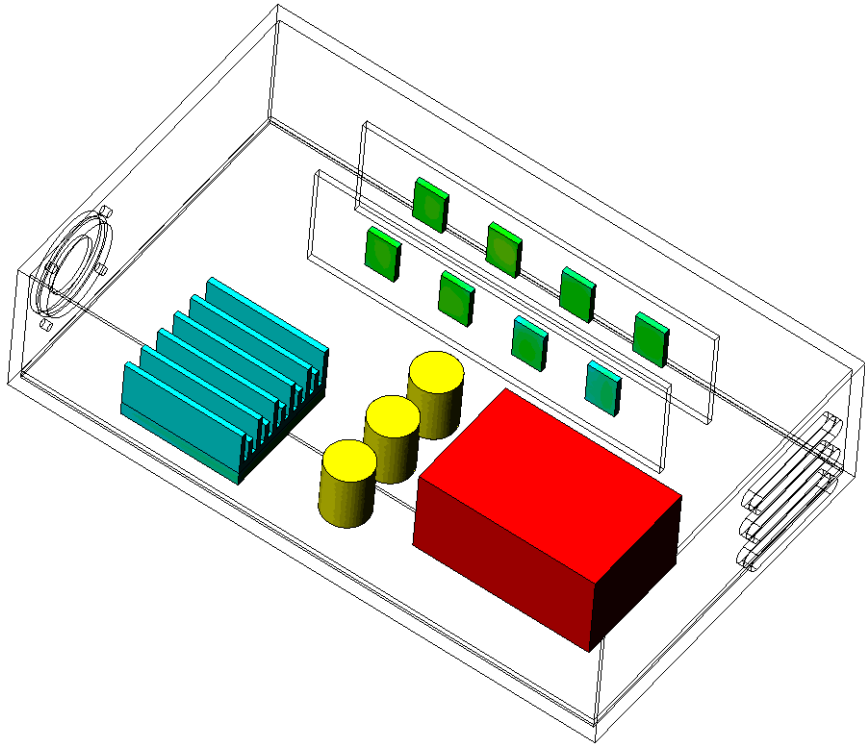
You can view and analyze the results further with the post-processing tools that were shown in the **First Steps - Ball Valve Design** tutorial. Flow Simulation allows you to quickly and easily investigate your design both quantitatively and qualitatively. Quantitative results such as the maximum temperature 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.

3

First Steps - Porous Media

In this tutorial we consider flow in a section of an automobile exhaust pipe, whose exhaust flow is resisted by two porous bodies serving as catalysts for transforming harmful carbon monoxide into carbon dioxide. When designing an automobile catalytic converter, the engineer faces a compromise between minimizing the catalyst's resistance to the exhaust flow while maximizing the catalyst's internal surface area and duration that the exhaust gases are in contact with that surface area. Therefore, a more uniform distribution of the exhaust mass flow rate over the catalyst's cross sections favors its serviceability. The porous media capabilities of Flow Simulation are used to simulate each catalyst, which allows you to model the volume that the catalyst occupies as a distributed resistance instead of discretely modeling all of the individual passages within the catalyst, which would be impractical or even impossible. Here, as a Flow Simulation tutorial example we consider the influence of the catalysts' porous medium permeability type (isotropic and unidirectional media of the same resistance to flow) on the exhaust mass flow rate distribution over the catalysts' cross sections. We will observe the latter through the behavior of the exhaust gas flow trajectories distributed uniformly over the model's inlet and passing through the porous catalysts. Additionally, by coloring the flow trajectories by the flow velocity the exhaust gas residence time in the porous catalysts can be estimated, which is also important from the catalyst effectiveness viewpoint.

Open the SolidWorks Model

1   Click **File**, **Open.**

2   In the **Open** dialog box, browse to the Catalyst.SLDASM assembly located in the **First Steps - Porous Media** folder and click **Open** (or double-click the assembly). Alternatively, you can drag and drop the Catalyst.SLDASM file to an empty area of SolidWorks window.
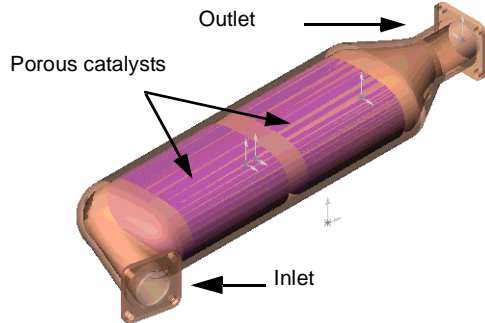
Create a Flow Simulation Project

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

Once inside the **Wizard**, select **Create new** in order to create a new configuration and name it Isotropic.
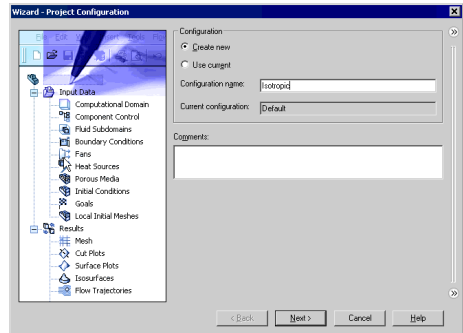
📖 *The project Wizard guides you through the definition of the project's properties step-by-step. Except for two steps (steps to define the project fluids and default solid), each step has some pre-defined values, so you can either accept these values (skipping the step by clicking **Next**) or modify them to your needs.*
These pre-defined settings are:
unit system – SI,
analysis type – internal, no additional physical capabilities are considered,
wall condition – adiabatic wall
initial conditions – pressure - 1 atm, temperature - 293.2 K.
result and geometry resolution – level 3,
*For this project these default settings suit perfectly and all what we need to do is just to select Air as the project fluid. To avoid passing through all steps we will use **Navigator** pane that provides a quick access to the Wizard's pages.*

2   Click an arrow ⏩ at the right.

3 In the **Navigator** pane click **Fluids**.
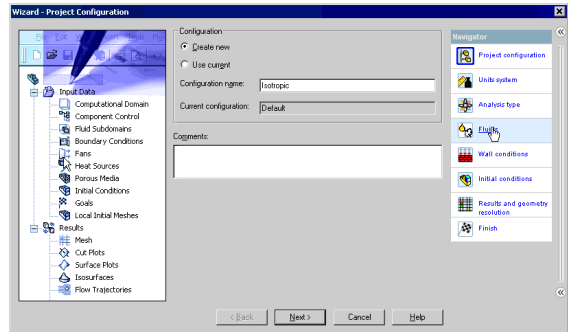
4 Open the **Gases** folder, click **Air**, then click **Add**.
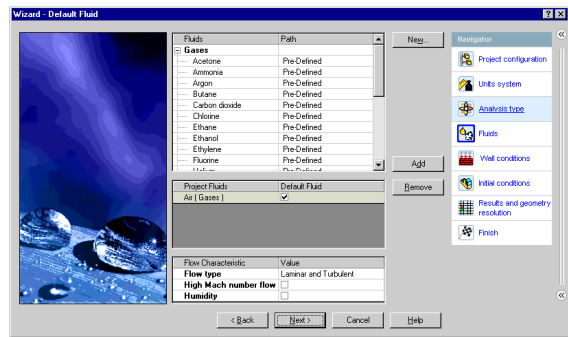
5 Since we do not need to change other properties we can close the Wizard. Click **Finish** in the Navigator panel.
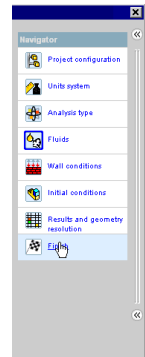
📖 *You can click Finish at any moment, but if you attempt to close Wizard without specifying all obligatory properties (such as project fluds), the Wizard will not close and the page where you need to define a missing property will be marked by the exclamation icon* ⚠.

Now Flow Simulation creates a new configuration with the Flow Simulation data attached.

In the Flow Simulation Analysis tree, right-click the **Computational Domain** icon and select **Hide** to hide the black wireframe box.

Define the Boundary Conditions

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

2 Select the inner face of the inlet lid as shown.

3 Select **Flow Openings** and **Inlet Velocity**.
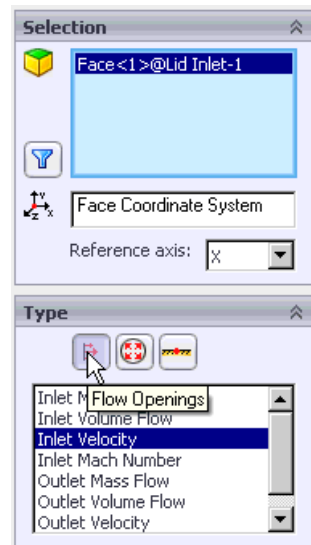
4 Set the **Velocity Normal to Face** V to 10 m/s.

5 Click **OK** .

📖 *With the definition just made, we told Flow Simulation that at this opening air is flowing into the catalyst with a velocity of 10 m/s.*
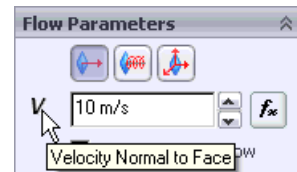
6 Select the inner face of the outlet lid as shown.

7 Right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.
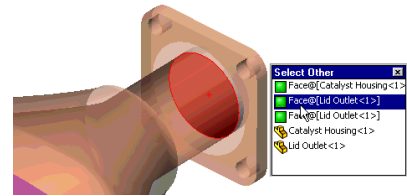
8 Select **Pressure Openings** and **Static Pressure**.

9 Keep the defaults under **Thermodynamic Parameters**, **Turbulence Parameters**, **Boundary Layer** and

   Options.Click **OK** .
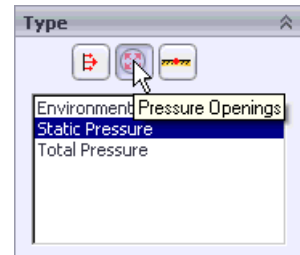
📖 *With the definition just made, we told Flow Simulation that at this opening the fluid exits the model to an area of static atmospheric pressure.*

Now we can specify porous media in this project. To define a porous medium, first we need to specify the porous medium's properties (porosity, permeability type, etc.) in the **Engineering Database** and then apply the porous medium to a component in the assembly.

Create an Isotropic Porous Medium

The material you are going to create is already defined in the Engineering Database under the Pre-Defined folder. You can skip the definition of porous material and select the pre-defined "Isotropic" material from the Engineering database when you will assign the porous material to a component later in this tutorial.

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

2 In the **Database tree** select **Porous Media**, **User Defined**.

3 Click **New Item** on the toolbar. The blank **Item Properties** tab appears. Double-click the empty cells to set the corresponding property values.

4 Name the new porous medium `Isotropic`.

5 Under **Comment**, click the **...** button and type the desired comments for this porous medium. The **Comment** property is optional, you can leave this field blank.
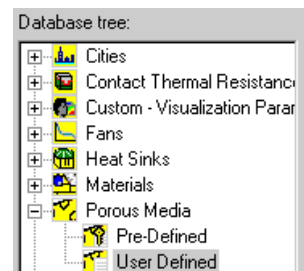
6   Set the medium's **Porosity** to `0.5`.

   📖 *Porosity is the effective porosity of the porous medium, defined as the volume fraction of the interconnected pores with respect to the total porous medium volume; here, the porosity is equal to 0.5. The porosity will govern the exhaust flow velocity in the porous medium channels, which, in turn, governs the exhaust gas residence in the porous catalyst and, therefore, the catalyst efficiency.*

7   Choose **Isotropic** as the **Permeability type**.

   📖 *First of all let us consider an **Isotropic** permeability, i.e, a medium with permeability not depending on the direction within the medium. Then, as an alternative, we will consider a **Unidirectional** permeability, i.e., the medium permeable in one direction only.*

8   Choose **Pressure drop**, **Flowrate**, **Dimensions** as the **Resistance calculation formula.**

   📖 *For our media we select the **Pressure Drop, Flowrate, Dimensions** medium resistance to flow, i.e., specify the porous medium resistance as $k = \Delta P \times S /(m \times L)$ (in units of s^{-1}), where the right-side parameters are referred to a tested parallelepiped sample of the porous medium, having the cross-sectional area S and the length L in the selected sample direction, in which the mass flow rate through the sample is equal to m under the pressure difference of ΔP between the sample opposite sides in this direction.*
In this project we will specify $\Delta P = 20$ Pa at m = 0.01 kg/s (and $\Delta P = 0$ Pa at m=0 kg/s), S = 0.01 m^2, L = 0.1m. Therefore, k = 200 s^{-1}.
Knowing S and L of the catalyst inserted into the model and m of the flow through it, you can approximately estimate the pressure loss at the model catalyst from
$\Delta P = k \times m \times L/S.$

9   For the **Pressure drop vs. flowrate** choose **Mass Flow Rate**. Click the **...** button to switch to the **Tables and Curves** tab.
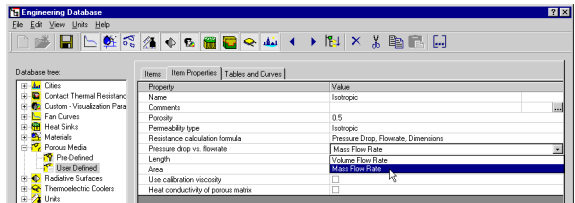
10 In the **Property** table specify the linear dependency of pressure drop vs. mass flow rate as shown.

11 Go back to the **Item Properties** tab.

12 Set **Length** to `0.1 m` and **Area** to `0.01 m²`.

13 Click **Save** 💾.

14 Click **File**, **Exit** to exit the database.
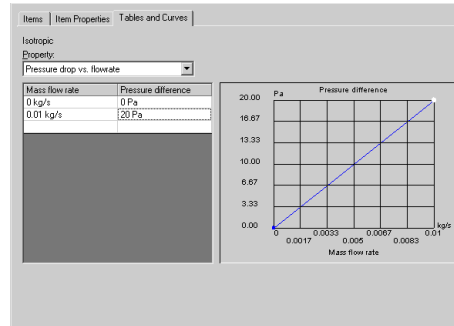
Now we will apply the specified porous medium to the model components representing the porous bodies.

A porous medium can be applied only to a component that is not treated by Flow Simulation as a solid. To consider a model's component as not belonging to a solid region, you need to disable the component in the **Component Control** dialog box. Components are automatically disabled when you assign a porous media to them by creating the **Porous Medium** condition, so you do not need to disable them manually.

Define the Porous Medium - Isotropic Type

1 Click **Flow Simulation**, **Insert**, **Porous Medium**.

2 In the flyout FeatureManager design tree, select the **Monolith<1>** and **Monolith<2>** components.
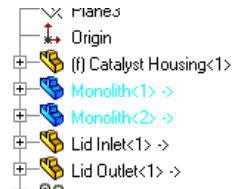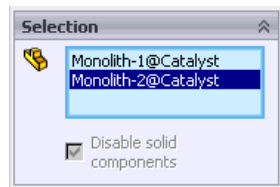
3   Expand the list of the **User Defined** porous media and
select **Isotropic**. If you skipped the definition of porous
medium, use the **Isotropic** material available under
Pre-Defined.

4   Click **OK** ✓ to complete the definition of porous media
and exit the **Porous Medium** dialog.

To obtain the total pressure drop between the model inlet and
outlet we will specify an **Equation Goal** based on two
Surface Goals.
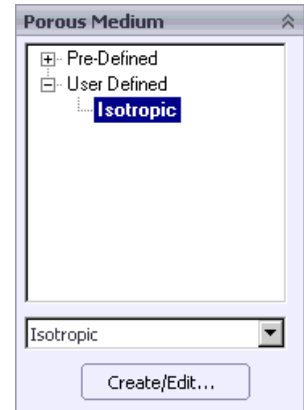
Specifying Surface Goals

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

2 Click the Flow Simulation Analysis Tree tab and click the **Inlet Velocity 1** item to select the inner face of the inlet lid.

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

4 Accept the selected **Use for Conv.** check box to use this goal for convergence control.

5 Under **Name Template**, located at the bottom

of the PropertyManager, click **Inlet** ⟨→|⟩.
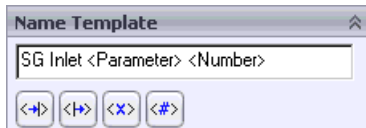
6 Click **OK** ✔ - the new
   `SG Inlet Av Total Pressure 1` goal
   appears.

7 Right-click the **Goals** icon and select **Insert Surface Goals**.
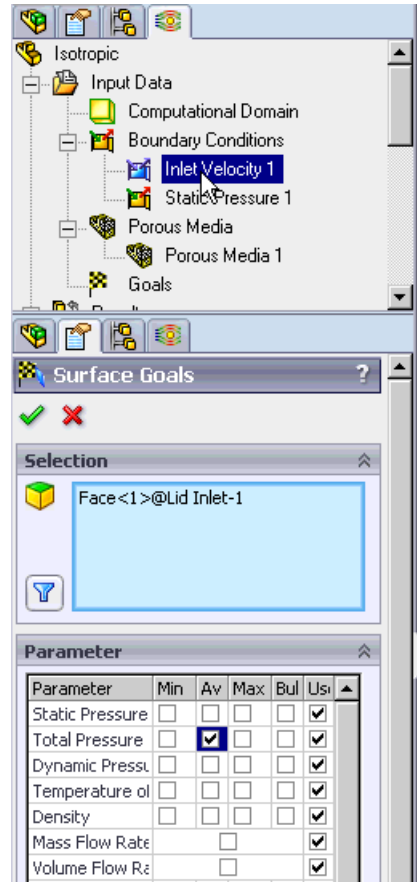
8   Click the Flow Simulation Analysis Tree tab
    and click the **Static Pressure 1** item to select
    the inner face of the outlet lid.

9   In the **Parameter** table select the **Av** check box
    in the **Total Pressure** row.

10 Accept the selected **Use for Conv.** check box to
    use this goal for convergence control.

11 Under **Name Template**, located at the bottom
    of the PropertyManager, click **Outlet** ⟨H⟩.
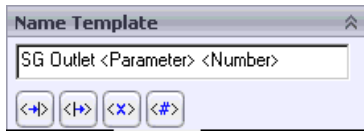
12 Click **OK** ✓ - the new
    `SG Outlet Av Total Pressure 1` goal
    appears.
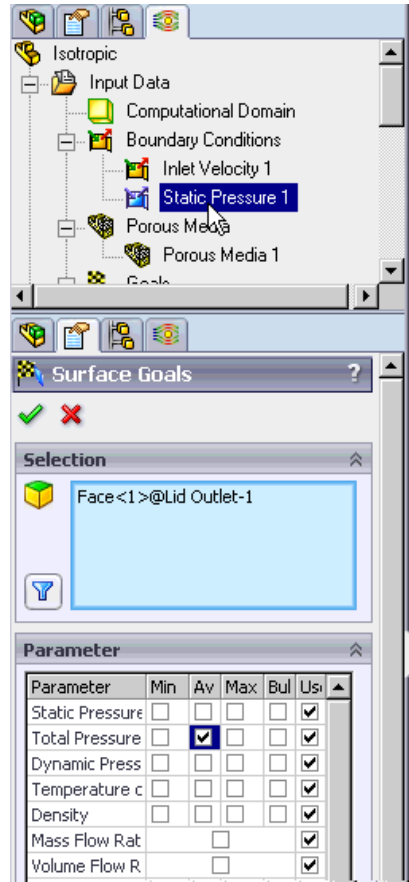
Define the Equation Goal

    📖 **Equation Goal** *is a goal defined by an analytical function of the existing goals and/or
parameters of input data conditions. This goal can be viewed as equation goal during
the calculation and while displaying results in the same way as the other goals. As
variables, you can use any of the specified goals, including another equation goals,
except for goals that are dependent on other equation goals, and parameters of the
specified project's input data features (general initial or ambient conditions, boundary
conditions, fans, heat sources, local initial conditions). You can also use constants in
the definition of the equation goal.*

1   Right-click the **Goals** icon and select **Insert Equation
    Goal.**
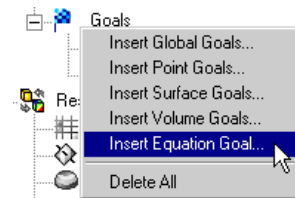
2 In the Flow Simulation Analysis tree select the **SG Inlet Av Total Pressure 1** goal. It appears in the **Expression** box.

3 Click the minus "-" button in the calculator.

4 In the Flow Simulation Analysis tree select the **SG Outlet Av Total Pressure 1** goal.

    📖 *You can use goals (including previously specified Equation Goals), parameters of input data conditions and constants in the expression defining an Equation Goal. If the constants in the expression represent some physical parameters (i.e. length, area, etc.), make sure that they are specified in the project's system of units. Flow Simulation has no information about the physical meaning of the constants you use, so you need to specify the Equation Goal dimensionality by yourself.*

5 Make sure that **Pressure & Stress** is selected in the **Dimensionality** list.

6 Click **OK**. The new **Equation Goal 1** item appears in the tree.
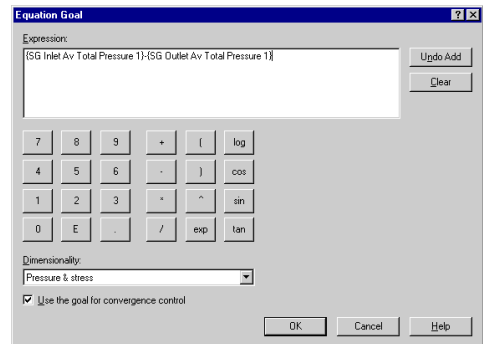
Solution

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

2 Click **Run**.

After the calculation has finished, close the **Monitor** dialog box.
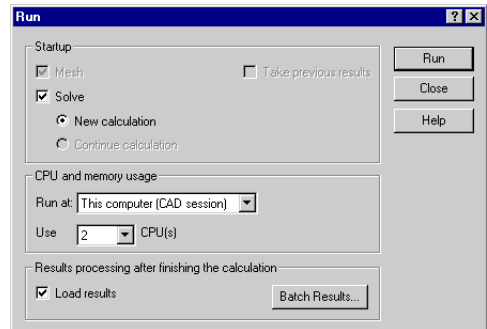
Viewing the Goals

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

2   Select the **Equation Goal 1** in the **Goals** dialog box.

3   Click **OK**.

An Excel spreadsheet with the goal results will open. The first sheet will contain a table presenting the final values of the goal.
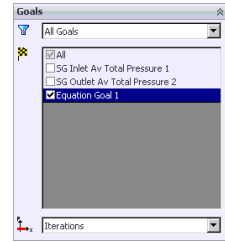
You can see that the total pressure drop is about 120 Pa.

Catalyst.SLDASM [Isotropic]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence
Equation Goal 1	[Pa]	120.0326909	121.774802	120.0326909	124.432896	100	Yes

To see the non-uniformity of the mass flow rate distribution over a catalyst's cross section, we will display flow trajectories with start points distributed uniformly across the inlet.

Flow Trajectories

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

2   Click the Flow Simulation Analysis Tree tab and then click the **Inlet Velocity 1** item. This selects the inner face of the inlet lid.

3   Under **Appearance**, make sure that **Color by Parameter**

   is selected.and then change the parameter to **Velocity**.

4   Click **Adjust Minimum/Maximum and Number of Levels** and set the **Max** value to 12 m/s.

5   Click **OK** .
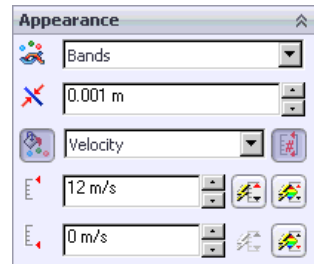
To see trajectories inside the porous media we will apply some transparency to the model.

6 Click **Flow Simulation**, **Results**, **Display**, **Transparency** and set the model transparency to 0.75**.**
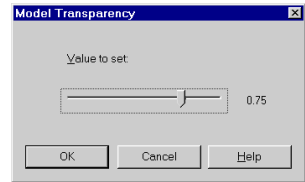
This is the picture you will see.
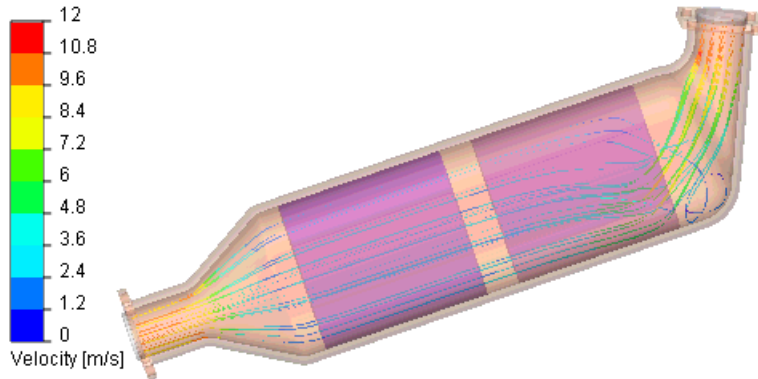
To compare the effectiveness of a unidirectional porous catalyst to an isotropic catalyst, let us calculate the project with a porous medium of unidirectional type.

Clone the Project

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

2 Enter Unidirectional as the **Configuration name**.

3 Click **OK**.
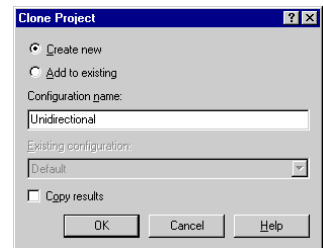
Create a Unidirectional Porous Medium

The material you are going to create is already defined in the Engineering Database under the Pre-Defined folder. You can skip the definition of porous material and select the pre-defined "Unidirectional" material from the Engineering database when you will assign the porous material to a component later in this tutorial.

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

2   In the **Database tree** select **Porous Media**, **User Defined**.

3   On the **Items** tab select the **Isotropic** item**.**

4   Click **Copy** 📋 .

5   Click **Paste** 📋 . The new **Copy of Isotropic (1)** item appears in the list.

6   Select the **Copy of Isotropic (1)** item and click the **Item Properties** tab.

7   Rename the item to Unidirectional.

8   Change the **Permeability type** to **Unidirectional**.

9   Save the database and exit.
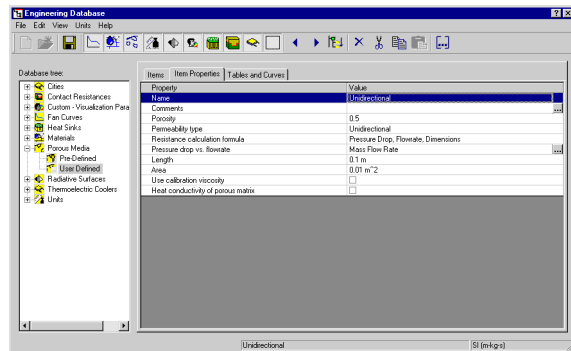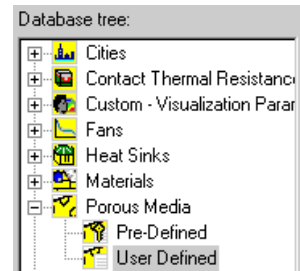
Now we can apply the new porous medium to the monoliths.

Define the Porous Medium - Unidirectional Type

1   Right-click the **Porous Medium 1** icon and select **Edit Definition**.

2 Expand the list of **User Defined** porous medium and select **Unidirectional**If you skipped the definition of the unidirectional porous medium, use the **Unidirectional** material available under **Pre-Defined**.

3 In the **Direction**, select the **Z** axis of the Global Coordinate System.

📖 *For porous media having unidirectional permeability, we must specify the permeability direction as an axis of the selected coordinate system (axis Z of the Global coordinate system in our case).*

4 Click **OK** ✔.

Since all other conditions and goals remain the same, we can start the calculation immediately
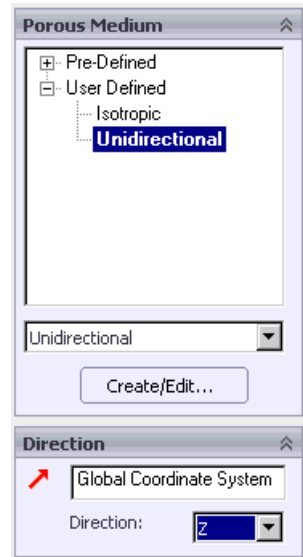
Compare the Isotropic and Unidirectional Catalysts

When the calculation is finished, create the goal plot for the **Equation Goal 1**.

Catalyst.SLDASM [Unidirectional]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence
Equation Goal 1	[Pa]	117.0848512	118.6235708	117.0761518	121.5639633	100	Yes

Display flow trajectories as described above.
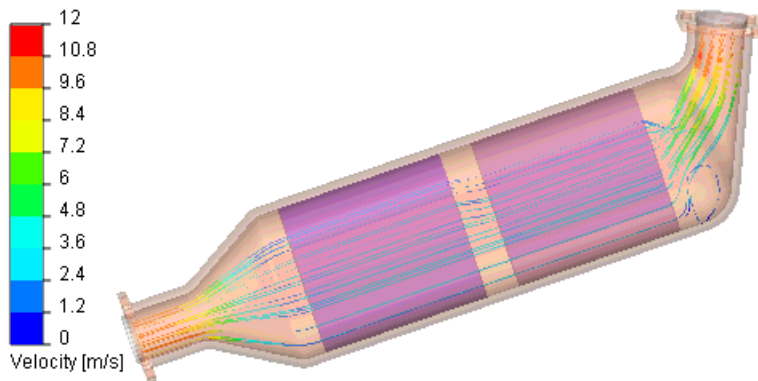
Comparing the trajectories passing through the isotropic and unidirectional porous catalysts installed in the tube, we can summarize:

Due to the asymmetric position of the inlet tube with respect to the larger tube in which the catalyst bodies are installed, the incoming flow is non-uniform. Since the incoming flow is non-uniform, the flow inside the first catalyst body is non-uniform also. It is seen that the catalyst type (isotropic or unidirectional) affects both the incoming flow non-uniformity (slightly) and, more substantially, the flow within the catalysts (especially the first catalyst body). In both the cases the gas stream mainly enters the first catalyst body-closer to the wall opposite to the inlet tube. For the isotropic case, the gas flows into the first body nearer to the wall than for the case of the unidirectional catalyst. As a result, the flow in the initial (about one-third of the body length) portion of the first catalyst body is noticeably more non-uniform in the isotropic catalyst. Nevertheless, due to the isotropic permeability, the main gas stream expands in the isotropic catalyst and occupies a larger volume in the next part of the body than in the unidirectional catalyst, which, due to its unidirectional permeability, prevents the stream from expanding. So, the flow in the last two-thirds of the first catalyst body is less non-uniform in the isotropic catalyst. Since the distance between the two porous bodies installed in the tube is rather small, the gas stream has no time to become more uniform in the volume between the catalyst bodies, although in the unidirectional case a certain motion towards uniformity is perceptible. As a result, the flow non-uniformity occurring at the first catalyst body's exit passes to the second catalyst body. Then, it is seen that the flow non-uniformity does not change within the second catalyst body.

Let us now consider the flow velocity inside the catalyst. This is easy to do since the flow trajectories' colors indicate the flow velocity value in accordance with the specified palette. To create the same conditions for comparing the flow velocities in the isotropic and unidirectional catalysts, we have to specify the same velocity range for the palette in both the cases, since the maximum flow velocity governing the value range for the palette by default is somewhat different in these cases. It is seen that, considering the catalyst on the whole, the flow velocities in the isotropic and unidirectional catalysts are practically the same. Therefore, from the viewpoint of gas residence in the catalyst, there is no difference between the isotropic and unidirectional catalysts.

We can conclude that the isotropic catalyst is more effective than the unidirectional catalyst (of the same resistance to uniform flows), since the flow in it, as a whole, is more uniform. In spite of specifying the same resistance of the catalysts to flow, the overall pressure loss is lower by about 2% in the case of employing the unidirectional catalyst. This difference is due to the different flow non-uniformity both in the catalyst bodies and out of them.

Determination of Hydraulic Loss

In engineering practice the hydraulic loss of pressure head in any piping system is traditionally split into two components: the loss due to friction along straight pipe sections and the local loss due to local pipe features, such as bends, T-pipes, various cocks, valves, throttles, etc. Being determined, these losses are summed to form the total hydraulic loss. Generally, there are no problems in engineering practice to determine the friction loss in a piping system since relatively simple formulae based on theoretical and experimental investigations exist. The other matter is the local hydraulic loss (or so-called local drag). Here usually only experimental data are available, which are always restricted due to their nature, especially taking into account the wide variety of pipe shapes (not only existing, but also advanced) and devices, as well as the substantially complicated flow patterns in them. Flow Simulation presents an alternative approach to the traditional problems associated with determining this kind of local drag, allowing you to predict computationally almost any local drag in a piping system within good accuracy.

Click **File**, **Open**. In the **Open** dialog box, browse to the **Valve.SLDPRT** model located in the **Tutorial 1 - Hydraulic Loss** folder and click **Open** (or double-click the part). Alternatively, you can drag and drop the **Valve.SLDPRT** file to an empty area of the SolidWorks window.

Model Description

This is a ball valve. Turning the handle closes or opens the valve.

The local hydraulic loss (or drag) produced by a ball valve installed in a piping system depends on the valve turning angle or on the minimum flow passage area governed by it. The latter depends also on a ball valve geometrical parameter, which is the ball-to-pipe diameter ratio governing the handle angle at which the valve becomes closed:
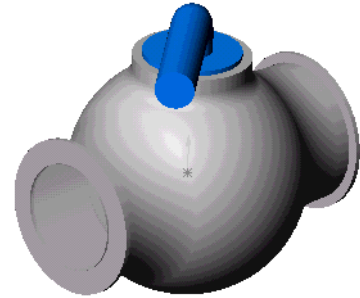
$$\theta = \arcsin\left[2\frac{D_{ball}}{D_{pipe}}\right]$$

The standard engineering convention for determining local drag is by calculating the difference between the fluid dynamic heads measured upstream of the local pipe feature (ball valve in our case) and far downstream of it, where the flow has become uniform (undisturbed) again. In order to extract the pure local drag the hydraulic friction loss in the straight pipe of the same length must be subtracted from the measured dynamic head loss.

In this example we will obtain pressure loss (local drag) in the ball valve whose handle is turned by an angle of 40$^{\text{o}}$. The **Valve** analysis represents a typical Flow Simulation internal analysis.

  Internal flow analyses deal with flows inside pipes, tanks, HVAC systems, etc. The fluid enters a model at the inlets and exits the model through outlets.

To perform an internal analysis all the model openings must be closed with lids, which are needed to specify inlet and outlet flow boundary conditions on them. In any case, the internal model space filled with a fluid must be fully closed. You simply create lids as additional extrusions covering the openings. In this example the lids are semi-transparent allowing a view into the valve.

To ensure the model is fully closed click **Flow Simulation**, **Tools**, **Check Geometry**. Then click **Check** to calculate the fluid and solid volumes of the model. If the fluid volume is equal to zero, the model is not closed.

Click **Fluid Volume** to see the volume that will be occupied by fluid in the analysis.

Uncheck **Fluid Volume**. Close the **Check Geometry** dialog box.

📖 *The **Check Geometry** tool allows you to calculate the total fluid and solid volumes, check bodies for possible geometry problems (i.e. invalid contact) and visualize the fluid area and solid body as separate models.*
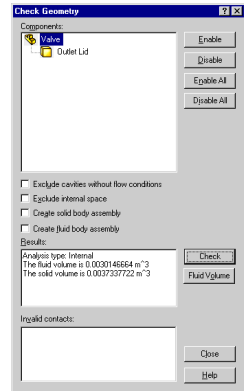
The first step is to create a new Flow Simulation project.

Creating a Project

1 Click **Flow Simulation**, **Project**, **Wizard**. The project wizard guides you through the definition of a new Flow Simulation project.

2 In the **Project Configuration** dialog box, click **Use current**. Each Flow Simulation project is associated with a SolidWorks configuration. You can attach the project either to the current SolidWorks configuration or create a new SolidWorks configuration based on the current one.

Click **Next**.
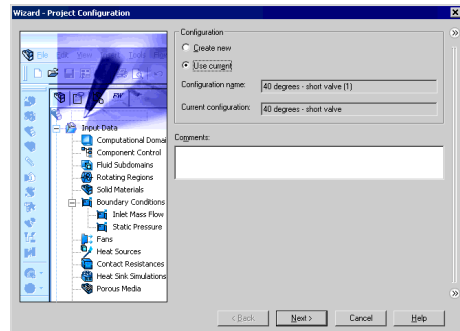
3 In the **Unit System** dialog box you can select the desired system of units for both input and output (results).
For this project use the International System **SI** by default.

Click **Next**.
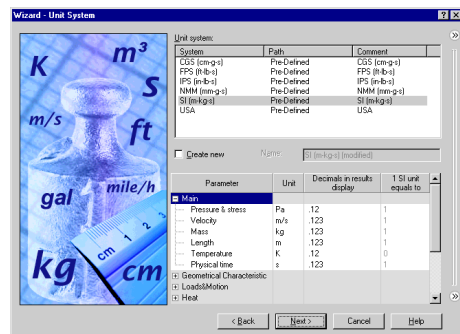
4   In the **Analysis Type** dialog box you can select either **Internal** or **External** type of the flow analysis.

To disregard closed internal spaces not involved in the internal analysis, you select **Exclude cavities without flow conditions**.

The **Reference axis of the global coordinate system** (X, Y or Z) is used for specifying data in a tabular or formula form in a cylindrical coordinate system based on this axis.

This dialog also allows you to specify advanced physical features you may want to take into account (heat conduction in solids, gravitational effects, time-dependent problems, surface-to-surface radiation, rotation).

Specify **Internal** type and accept the other default settings.
Click **Next**.
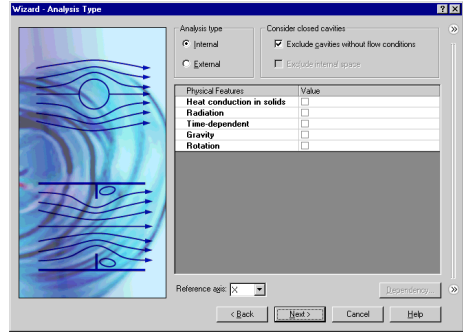
5   Since we use water in this project, open the **Liquids** folder and double-click the **Water** item.
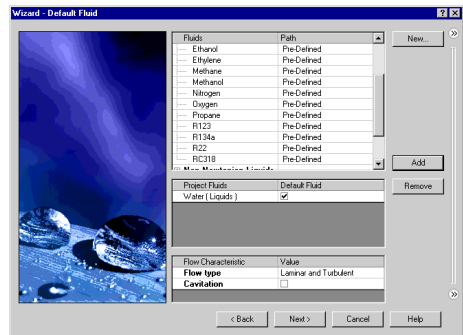
📖 **Engineering Database** *contains numerical physical information on a wide variety of gas, liquid and solid substances as well as radiative surfaces. You can also use the Engineering Database to specify a porous medium. The Engineering Database contains pre-defined unit systems. It also contains fan curves defining volume or mass flow rate versus static pressure difference for selected industrial fans. You can easily create your own substances, units, fan curves or specify a custom parameter you want to visualize.*

Click **Next**.

6 Since we do not intend to calculate heat conduction in solids, in the **Wall Conditions** dialog box you can specify the thermal wall boundary conditions applied by default to all the model walls contacting with the fluid.

For this project accept the default **Adiabatic wall** feature denoting that all the model walls are heat-insulated.

In this project we will not consider rough walls.

Click **Next**.
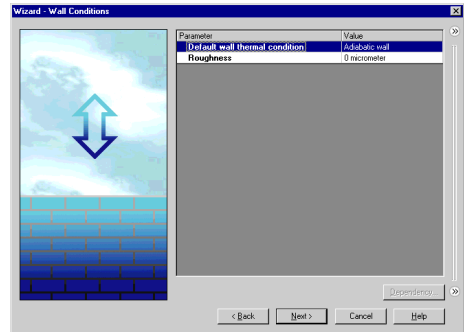
7 In the **Initial Conditions** dialog box specify initial values of the flow parameters. For steady internal problems, the specification of these values closer to the expected flow field will reduce the analysis convergence time.

📖 *For steady flow problems Flow Simulation iterates until the solution converges. For unsteady (transient, or time-dependent) problems Flow Simulation marches in time for a period you specify.*

For this project use the default values.

Click **Next**.
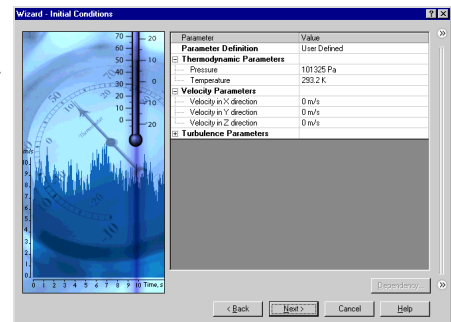
8 In the **Results and Geometry Resolution** dialog box you can control the analysis accuracy as well as the mesh settings and, through them, the required computer resources (CPU time and memory).

For this project accept the default result resolution level 3.
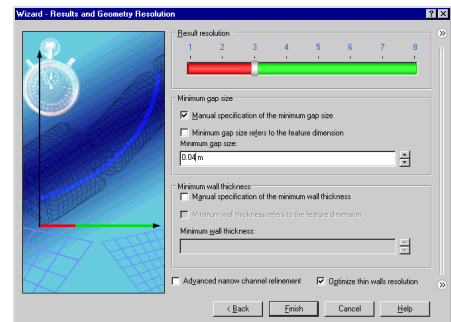
📖 **Result Resolution** *governs the solution accuracy via mesh settings and conditions of finishing the calculation that can be interpreted as resolution of calculation results. The higher the Result Resolution, the finer the mesh and the stricter the convergence criteria. Naturally, higher Result Resolution requires more computer resources (CPU time and memory).*

Geometry Resolution *(specified through the minimum gap size and the minimum wall thickness) governs proper resolution of geometrical model features by the computational mesh. Naturally, finer Geometry Resolution requires more computer resources.*

Select the **Manual specification of the minimum gap size** check box and enter `0.04 m` for the **Minimum gap size**.

📖 *Flow Simulation 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. However, this information may be insufficient to recognize relatively small gaps and thin model walls. This may cause inaccurate results. In these cases, the Minimum gap size and Minimum wall thickness must be specified manually.*

Click **Finish**.

📖 *The **Flow Simulation Analysis tree** provides a convenient specification of project data and view of results. You also can use the Flow Simulation Analysis tree to modify or delete the various Flow Simulation features.*

At the same time, a computational domain appears in the SolidWorks graphics area as a wireframe box.

📖 *The **Computational Domain** is a rectangular prism embracing the area inside which the flow and heat transfer calculations are performed.*
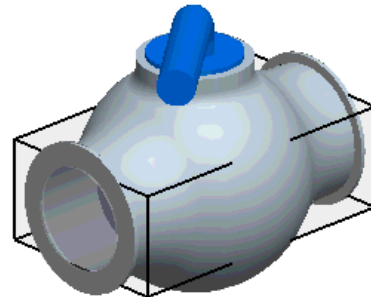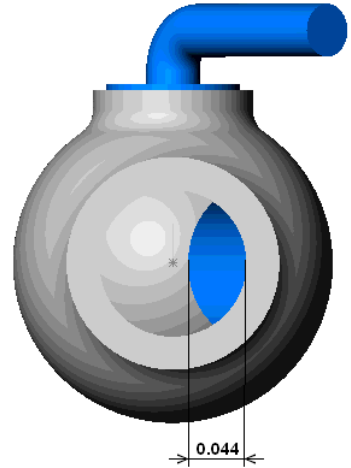
The next step is specifying **Boundary Conditions**. Boundary Conditions are used to specify fluid characteristics at the model inlets and outlets in an internal flow analysis or on model surfaces in an external flow analysis.

Specifying Boundary Conditions

1 Click **Flow Simulation**, **Insert**, **Boundary Condition**.

2 Select the **Inlet Lid** inner face (in contact with the fluid).

The selected face appears in the **Faces to Apply the**

Boundary Condition 🟧 list.
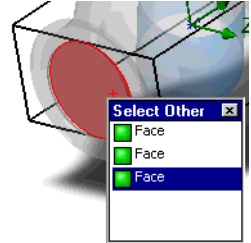
3 In the **Type of Boundary Condition** list, select the **Inlet Velocity** item.
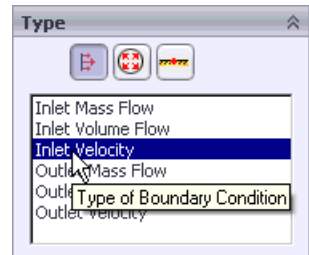
4 Click the **Velocity Normal to Face** *V* edit box and set its value equal to 1 m/s (type the value, the units will appear automatically).
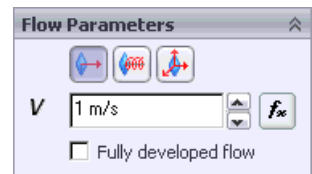
5 Accept all the other parameters and click **OK** ✅.

This simulates the water flow which enters the valve with the velocity of 1.0 m/s.

6   Select the **Outlet Lid** inner face.

7   In the graphics area, right-click outside the model and select **Insert Boundary Condition, Other**. The **Boundary Condition** dialog appears with the selected face in the **Faces to apply the boundary condition** list.
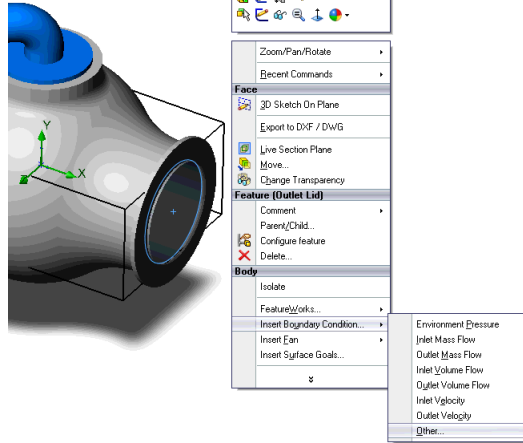
    Before the calculation starts, Flow Simulation *checks the specified boundary conditions for mass flow rate balance. The specification of boundary conditions is incorrect if the total mass flow rate on the inlets is not equal to the total mass flow rate on the outlets. In such case the calculation will not start. Also, note that the mass flow rate value is recalculated from the velocity or volume flow rate value specified on an opening. To avoid problems with specifying boundary conditions, we recommend that you specify at least one Pressure opening condition since the mass flow rate on a Pressure opening is automatically calculated to satisfy the law of conservation of mass.*

8   Click **Pressure Openings** and in the **Type of Boundary Condition** list select the **Static Pressure** item.
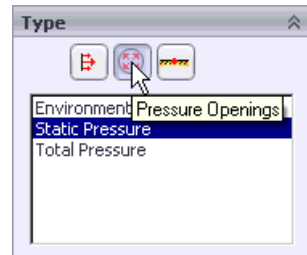
9   Accept the default values for **Static Pressure** **P** (101325 Pa), **Temperature** **T** (293.2 K) and all the other parameters.
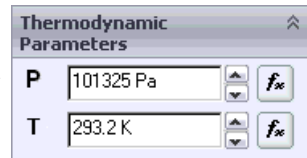
10 Click **OK** .

By specifying this condition we define that at the ball valve pipe exit the water has a static pressure of 1 atm.

The hydraulic losses are calculated through the outlet and inlet total pressure difference ΔP from the following formula:

$$\xi = \frac{\Delta P}{\rho V^2/2}$$

where ρ is the water density, and V is water velocity. Since we already know the water velocity (specified by us as 1 m/s) and the water density (998.1934 kg/m^3 for the specified temperature of 293.2 K), then our goal is to determine the total pressure value at the valve's inlet and outlet. The easiest and fastest way to find the parameter of interest is to specify the corresponding engineering goal.

Specifying Surface Goals

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

2 Select the inner faces of the inlet lid and the outlet lid (this can be done easily by holding down the **CTRL** key and clicking the corresponding boundary conditions in the Flow Simulation Analysis tree).

3 Select **Create goal for each surface** check box to create two separate goals, i.e. one for each of the selected faces.

4 In the **Parameter** table select the **Av** check box in the **Total Pressure** row.

5 Accept selected **Use for Conv.** check box to use the goals being created for convergence control.

6 Click **OK** ✔. The new **SG Av Total Pressure 1** and **SG Av Total Pressure 2** items appear in the Flow Simulation Analysis tree.
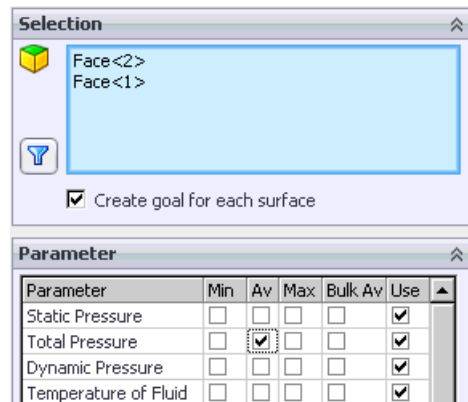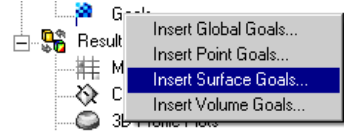
Now the Flow Simulation project is ready for the calculation. Flow Simulation will finish the calculation when the steady-state average value of total pressure calculated at the valve inlet and outlet are reached.

Running the Calculation

1   Click **Flow Simulation**, **Solve**, **Run**. The **Run** dialog box appears.

2   Click **Run** to start the calculation.

Flow Simulation automatically generates a computational mesh. The mesh is created by dividing the computational domain into slices, which are further subdivided into cells. The cells are refined as necessary to properly resolve the model geometry. During the mesh generation procedure, you can see the current step and the mesh information in the **Mesh Generation** dialog box.
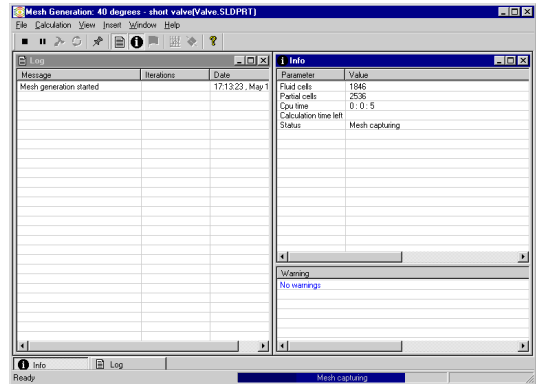
Monitoring the Calculation

After the calculation starts, the **Solver Monitor** dialog provides you with the current status of the solution. You can also monitor the goal changes and view preliminary results at selected planes. In the bottom pane of the **Info** window Flow Simulation notifies you if inappropriate results may occur. In our case, the message "**A vortex crosses the pressure opening**" appears to inform you that there is a vortex crossing the opening surface at which you specified the pressure boundary condition. In this case the vortex is broken into incoming and outgoing flow components. When flow both enters and exits an opening, the accuracy of the results is diminished. Moreover, there is no guarantee that convergence (i.e., the steady state goal) will be attained at all. Anyway, in case a vortex crosses a pressure opening the obtained results become suspect. If this warning persists we should stop the calculation and lengthen the ball valve outlet pipe to provide more space for development of the vortex. It is also expedient to attach the ball valve inlet pipe to avoid the flow disturbance caused by the valve's obstacle to affect the inlet boundary condition parameters.

Since the warning persists, click **File**, **Close** to terminate the calculation and exit the **Solver Monitor**.
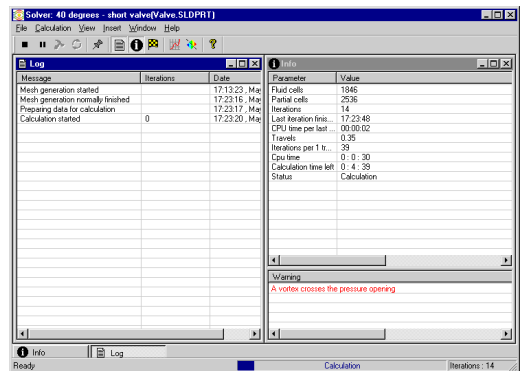
You can easily extend the ball valve inlet and outlet sections by changing the offset distance for the **Inlet Plane** and **Outlet Plane** features. Instead, we shall clone the project to the pre-defined **40 degrees - long valve** configuration**.**

Cloning the Project

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

2   Click **Add to existing**.

3   In the **Existing configuration** list, select **40 degrees - long valve**.

4   Click **OK**.

5   Flow Simulation has detected that the model was modified. Confirm the both warning messages with **Yes**.
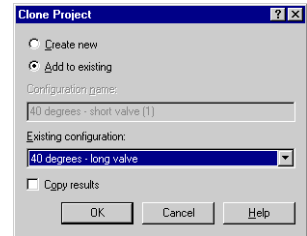
The new Flow Simulation project, attached to the **40 degrees - long valve** configuration, has the same settings as the old one attached to the **40 degrees - short valve** so you can start the calculation immediately.

In the Flow Simulation analysis tree, right-click the root **40 degrees - long valve** item and select **Run**. Then click **Run** to start the calculation.

When the calculation is finished, close the **Solver Monitor** dialog box.

Let us now see the vortex notified by Flow Simulation during the calculation, as well as the total pressure loss.
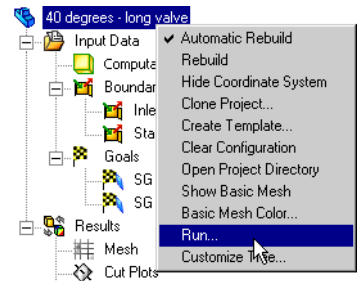
Creating a Cut Plot

1   Right-click the **Cut Plots** icon and select **Insert**. The
    Cut Plot dialog box appears.
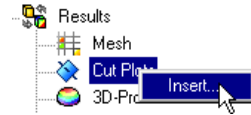
    📖 *The* **Cut Plot** *displays results of a selected parameter in a selected view section. To
    define the view section, you can use SolidWorks planes or model planar faces (with the
    additional shift if necessary). The parameter values can be represented as a contour
    plot, as isolines, as vectors, or in a combination (e.g. contours with overlaid vectors).*

2   In the flyout FeatureManager
    design tree, expand the **Valve**
    item and select **Plane2**. Its name
    appears in the **Section Plane or**

    Planar Face 📦 list.
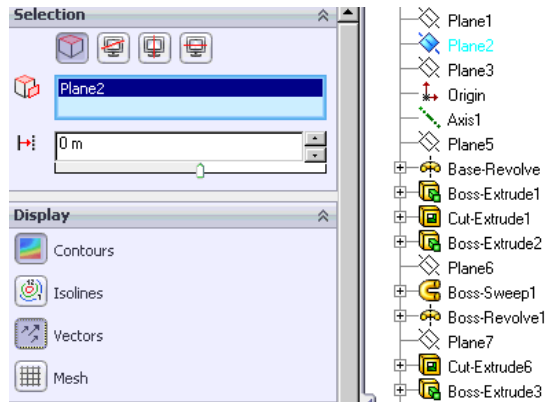
3   In the **Cut Plot** dialog box, in
    addition to displaying **Contours**

    🌈 , select **Vectors** 🗲.

4   Under **Contours** specify the parameter which values to

    show at the contour plot. In the **Parameter** 🌈 box,
    select **X - Component of Velocity**.
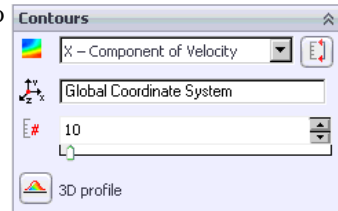
5   Under **Vectors** set the **Vector Spacing** to 0.012 m

    and set the **Arrow size** to 0.02 m.

6   In the **Cut Plot** dialog box click **OK** ✓. The new
    Cut Plot 1 item appears in the Flow Simulation
    Analysis tree.

However, the cut plot cannot be seen through the model. In order to see the plot, you can hide the model by clicking **Flow Simulation**, **Results**, **Display**, **Geometry**. Alternatively,

you can use the standard SolidWorks **Section View** option.

7 Click **View**, **Display**, **Section View**. Under **Section 1** specify **Plane2** as a **Reference Section Plane/Face** and

click **OK** .
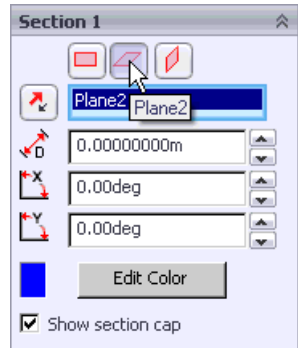
8 In the Flow Simulation Analysis tree, right-click the **Computational Domain** icon and select **Hide**.

Now you can see a contour plot of the velocity and the velocity vectors projected on the plot.
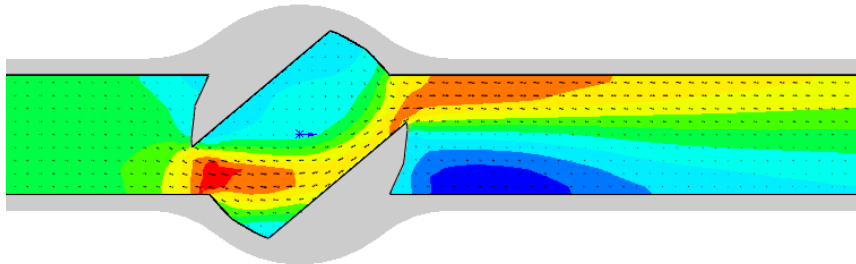
For better visualization of the vortex you can scale small vectors:

9 In the Flow Simulation Analysis tree, right-click the **Cut Plot** icon and select **Edit Definition**.
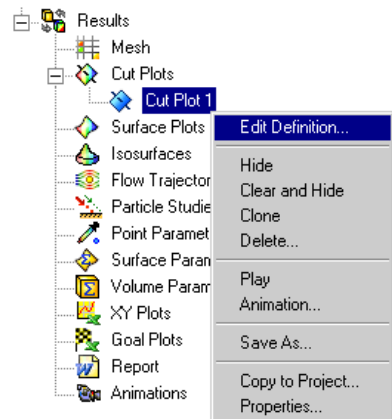
10 Under **Vectors** click the **Adjust Minimum and Maximum** and change the **Minimum** value to 2 m/s.
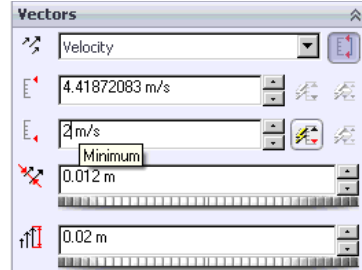
📖 *By specifying the custom **Minimum** we change the vector length range so that the vectors in areas where velocity is less than the specified **Minimum** value will appear as if it is equal to **Minimum**. This allows us to visualize the low velocity area in more detail.*

11 Click **OK** ✔ to save the changes and exit the **Cut Plot** dialog box. Immediately the cut plot is updated.
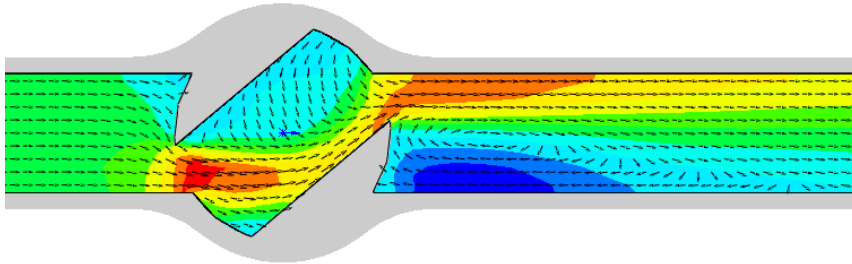
You can easily visualize the vortex by displaying the flow relative to the X axis. For that, you can display the X - Component of Velocity component in a two-color palette and set the value, separating two colors, at zero.

12 In the graphics area, double-click the palette bar or right-click on it and select **Edit Definition**.

13 Under **Settings** using the slider set **Number of Levels** to **3**.

14 In the **Maximum** box type 1.

15 In the **Minimum** box type -1.

16 Click **OK** ✔.
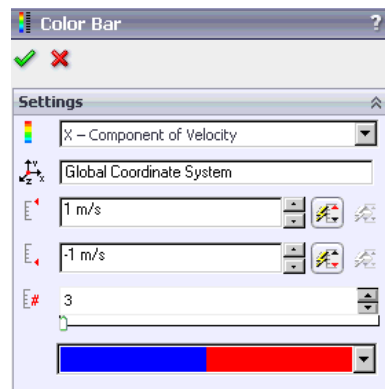
Now the distribution of the **X - Component of Velocity** component is displayed in red-blue palette so that all the positive values are in red and all the negative values are in blue. This means that the blue area show the region of reverse flow, i.e. half of the vortex.
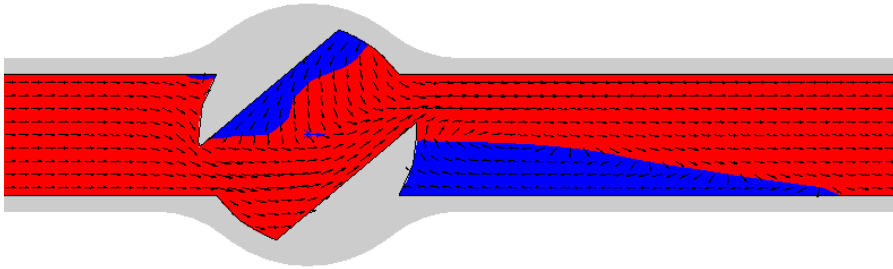
Next, we will display the distribution of total pressure within the valve.

Working with Parameter List

By default the total pressure is not included in the list of parameters available to display. To enable or disable a physical parameter for displaying you can use **Parameter List**.

1 In the Analysis tree, right-click the **Results** icon and select **Parameter List**. Expand the **Local** item and select the **Total Pressure** check box or select parameter's row and click **Enable**.

2 Click **OK** to close the **Display Parameters** dialog box.
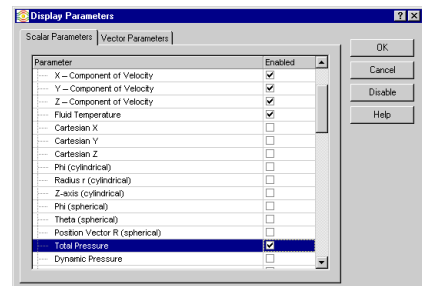
Now you can apply the total pressure for the contour plot.

3 In the palette bar click the caption with the name of the current visualization parameter and select **Total Pressure** in a dropdown list and click ✅.
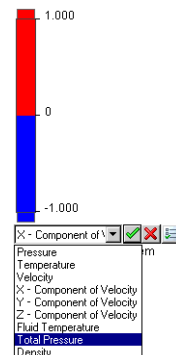
4 In the graphics area, double-click the palette bar.

5   Under **Settings** using the slider, set the **Number of Levels** $E^\#$ to about 30.

6   Click **OK** ✔ to save the changes and exit the **Color Bar** dialog box.
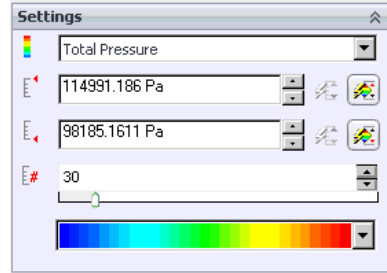
Immediately the cut plot is updated to display the total pressure contour plot.
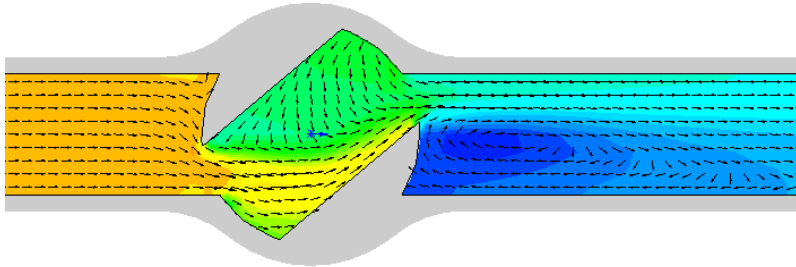
The cut plot shows you the flow pattern. To obtain the exact value of the total pressure which is required to calculate the loss, we will use the surface goal plot.

Creating a Goal Plot

  The **Goal Plot** *allows you to study how the goal value changed in the course of calculation. Flow Simulation uses Excel to display goal plot data. Each goal plot is displayed in a separate sheet. The converged values of all project goals are displayed in the* **Summary** *sheet of an automatically created Excel workbook.*

Click **View**, **Display**, **Section View** to hide the section.

1   In the Flow Simulation Analysis tree, under **Results**, right-click the **Goal Plots** icon and select **Insert**. The **Goal Plot** dialog box appears.

2 Select **All**.

3 Click **OK** ✓ . The **Goals1** Excel workbook is created.

This workbook displays how the goal changed during the calculation. You can take the total pressure value presented at the **Summary** sheet.

Valve.SLDPRT [40 degrees - long valve]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence
SG Av Total Pressure 1	[Pa]	101833.4184	101833.8984	101833.3951	101834.7911	100	Yes
SG Av Total Pressure 2	[Pa]	111386.6792	111389.5793	111384.8369	111399.0657	100	Yes

In fact, to obtain the pressure loss it would be easier to specify an Equation goal with the difference between the inlet and outlet pressures as the equation goal's expression. However, to demonstrate the wide capabilities of Flow Simulation, we will calculate the pressure loss with the Flow Simulation gasdynamic **Calculator**.

📖 *The* **Calculator** *contains various formulae from fluid dynamics which can be useful for engineering calculations. The calculator is a very useful tool for rough estimations of the expected results, as well as for calculations of important characteristic and reference values. All calculations in the Calculator are performed only in the International system of units* **SI***, so no parameter units should be entered, and Flow Simulation Units settings do not apply in the Calculator.*

Working with Calculator

1 Click **Flow Simulation**, **Tools**, **Calculator**.

2 Right-click the A1 cell in the **Calculator** sheet and select **New Formula**. The **New Formula** dialog box appears.

3 In the **Select the name of the new formula** tree expand the **Pressure and Temperature** item and select the **Total pressure loss** check box.
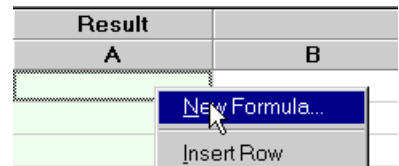
4 Click **OK**. The total pressure loss formula appears in the **Calculator** sheet.

In the **Result** (or **A**) column you see the formula name, in the next columns (**B**, **C**, etc.) you see names of the formula arguments (variables and constants). You can either type all the formula arguments' values in cells under their names in the SI units, or copy and paste them from the goals Excel worksheet table obtained through the **Goals** dialog box. The result value appears in the **Result** column cell immediately when you enter all the arguments and click another cell.

5 Specify the values in the cells as follows:

Density = 998.1934 (the water density for the specified temperature of 293.2 K),
Velocity = 1.

6 Open the **goals1** Excel workbook and copy the **Value** of **SG Av Total Pressure 1** into the Clipboard.

7 Go to the **Calculator**, click the **B2** cell and press **Ctrl+V** to paste the goal value from the Clipboard.

8 Return to Excel, copy the **Value** of **SG Av Total Pressure 2**. Go to the **Calculator,** click the **C2** cell and press **Ctrl+V**. Click any free cell. Immediately the **Total pressure loss** value appears in the **Result** column.

Total pressure loss	Total pressure loss	Total pressure at point 1	Total pressure at point 2	Density	Velocity
	-19.1411019	101833.418 Pa	111386.679 Pa	998.1934 kg/m^3	1 m/s

9 Click **File, Save**.

10 In the **Save As** dialog box browse to the folder where the ball valve model used in this example is located, enter `ball valve` for the file name, and click **Save**.

11 Click **File**, **Exit** to exit the **Calculator**.

To obtain the pure local drag, it is necessary to subtract from the obtained value the total pressure loss due to friction in a straight pipe of the same length and diameter. To do that, we perform the same calculations in the ball valve model with the handle in the 0° angle position. You can do this with the **00 degrees - long valve** configuration.

Since the specified conditions are the same for both **40 degrees - long valve** and **00 degrees - long valve** configurations, it is useful to attach the existing Flow Simulation project to the **00 degrees - long valve** configuration.

Clone the current project to the **00 degrees - long valve** configuration.
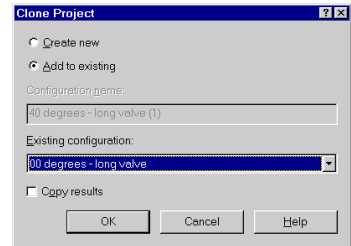
Since at zero angle the ball valve becomes a simple straight pipe, there is no need to set the **Minimum gap size** value smaller than the default gap size which, in our case, is automatically set equal to the pipe's diameter (the automatic minimum gap size depends on the characteristic size of the faces on which the boundary conditions are set). Note that using a smaller gap size will result in a finer mesh and, in turn, more computer time and memory will be required for calculation. To solve your task in the most effective way you should choose the optimal settings for the task.

Changing the Geometry Resolution

Check to see that the **00 degrees - long valve** is the active configuration.

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

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

3 Click **OK**.
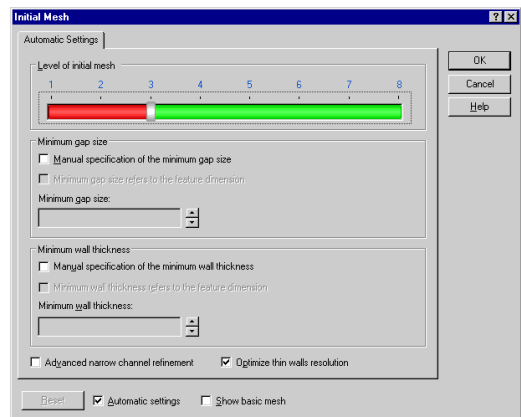
Click **Flow Simulation**, **Solve**, **Run**. Then click **Run** to start the calculation.

After the calculation is finished, create the **Goal Plot**. The **goals2** workbook is created. Go to Excel, then select the both cells in the **Value** column and copy them into the Clipboard.

Goal Name	Unit	Value	Averaged Value	Minimum Value
SG Av Total Pressure 1	[Pa]	101805.2057	101804.8525	101801.4794
SG Av Total Pressure 2	[Pa]	102023.7419		102022.7459

Now you can calculate the total pressure loss in a straight pipe.

1   Click **Flow Simulation**, **Tools**, **Calculator**.

2   In the **Calculator** menu, click **File**, **Open**. Browse to the folder where you saved the calculator file earlier in this tutorial and select the **ball valve.fwc** file. Click **Open**.

3   Click the **B4** cell and in the Calculator toolbar click 📋 to paste data from the Clipboard.

4   Save the existing value of the total pressure loss: click the **A2** cell, click 📋, then click the **A7** cell and finally click 📋 .

5   Double-click the **Name7** cell and type 40 degrees.

Total pressure loss	Total pressure loss	Total pressure at point 1
	-19.1411019	101833.418 Pa
		101805.2057
		102023.7419
40 degrees	-19.1411019	

6   Right-click the **Total pressure at point 1** cell and select **Add Relation**. The cursor 🖑 appears.

	B	C	D
	Total pressure at	Total pressure	nsity
	11	New Formula...	3.1934 kg
		Delete Formula	
		Insert Row	
		Delete Row	
		Add Relation	
		Delete Relation	
		Import from the Engineering DB...	
		Run Numeric Calculation	

7   Click the **B4** cell. The value of total pressure is now taken from the **B4** cell.

Total pressure at point 1
101833.418 Pa

🖑 101805.2057
102023.7419

8   Right-click the **Total pressure at point 2** cell and select **Add Relation**.

9   Click the **B5** cell. The value of total pressure is now taken from the **B5** cell. Immediately the total pressure value is recalculated.

Total pressure at point 1=B4
101805.206 Pa

101805.2057
🖑 102023.7419

Now you can calculate the local drag in the ball valve whose handle is set at 40°.

Total Pressure loss (40 deg)	Total Pressure loss (0 deg)	Local Drag
19.14	0.44	18.70

5

Cylinder Drag Coefficient

Flow Simulation can be used to study flow around objects and to determine the resulting lift and drag forces on the objects due to the flow. In this example we use Flow Simulation to determine the drag coefficient of a circular cylinder immersed in a uniform fluid stream. The cylinder axis is oriented perpendicular to the stream.

The computations are performed for a range of Reynolds numbers $(1,1000,10^5)$, where $\text{Re} = \dfrac{\rho U D}{\mu}$, D is the cylinder diameter, U is the velocity of the fluid stream, ρ is the density, and μ is the dynamic viscosity. The drag coefficient for the cylinder is defined as:

$$C_D = \frac{F_D}{\frac{1}{2}\rho U^2 DL}$$

where F_D is the total force in the flow direction (i.e. drag) acting on a cylinder of diameter D and length L.

The goal of the simulation is to obtain the drag coefficient predicted by Flow Simulation and to compare it to the experimental data presented in **Ref.1**.

Click **File**, **Open**. In the **Open** dialog box, browse to the **Cylinder 0.01m.SLDPRT** part located in the **Tutorial 2 - Drag Coefficient\cylinder 0.01m** folder and click **Open** (or double-click the part). Alternatively, you can drag and drop the **cylinder 0.01m.SLDPRT** file to an empty area of SolidWorks window.

The **Cylinder** analysis represents a typical Flow Simulation **External** analysis.
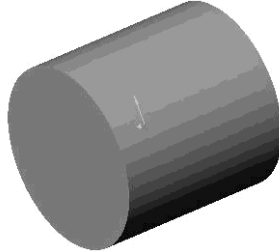
  External flows analyses deal with flows over or around a model such as flows over aircrafts, automobiles, buildings, etc. For external flow analyses the far-field boundaries are the Computational Domain boundaries. You can also solve a combined external and internal flow problem in a Flow Simulation project (for example flow around and through a building). If the analysis includes a combination of internal and external flows, you must specify External type for the analysis.

The first step is to create a new Flow Simulation project.

Creating a Project

1   Click **Flow Simulation**, **Project**, **Wizard**. The project wizard guides you through the definition of a new Flow Simulation project. In this project we will analyze flow over the cylinder at the Reynolds number of 1.

2   Select **Create new**. In the **Configuration name** box type **Re 1.** This is the name of the SolidWorks configuration that will be created for the associated Flow Simulation project.

Click **Next**.
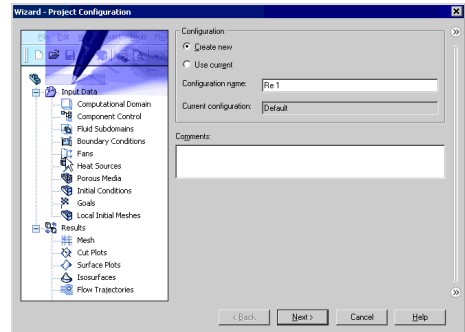
3 In the **Unit System** dialog box you can select the desired system of units for both input and output (results).

In this project we will specify the International System **SI** by default.

Click **Next**.
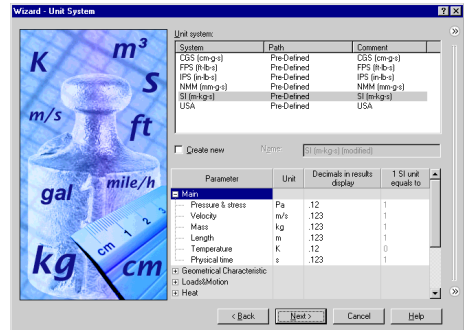
4 In the **Analysis Type** dialog box select an **External** type of flow analysis. This dialog also allows you to specify advanced physical features you want to include in the analysis. In this project we will not use any of the advanced physical features
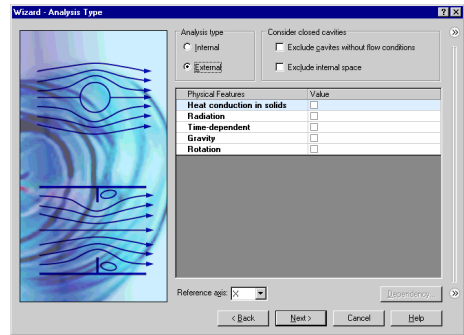
    *To disregard closed internal spaces within the body you can select **Exclude internal spaces**; however no internal spaces exist within the cylinder in this tutorial. The **Reference axis** of the global coordinate system (X, Y or Z) is used for specifying data in a tabular or formula form with respect to a cylindrical coordinate system based on this axis.*

    The flow over a cylinder is steady at a Reynolds number Re < 40 (see the cylinder Re definition above) and unsteady (time-dependent) at Re > 40. Since in this tutorial the first calculation is performed at Re=1, to accelerate the run, we perform a steady-state analysis.

Click **Next**.

5 Since we use water in this project, open the **Liquids** folder and double-click the **Water** item.
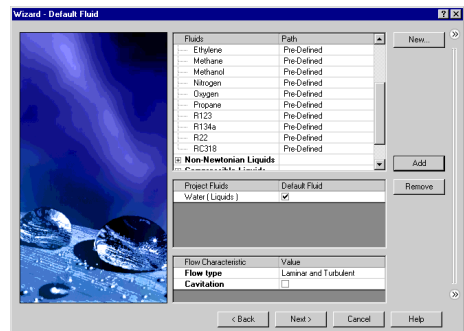
Click **Next**.

6 In the **Wall Conditions** dialog box you may specify the default thermal wall conditions applied to all the model walls in contact with the fluid.

In this project we keep the default **Adiabatic wall** setting, denoting that all the model walls are heat-insulated and accept the default zero wall roughness.

Click **Next**.

For a steady **External** problem, such as the cylinder in this tutorial, the **Initial and Ambient Conditions** dialog box asks you to specify the ambient flow conditions of the undisturbed free stream. Thus you will specify initial conditions inside the **Computational Domain** and boundary conditions at the **Computational Domain** boundaries. The ambient conditions are thermodynamic (static pressure and temperature by default), velocity, and turbulence parameters.

In this project we consider the flow under the default thermodynamic conditions (i.e., the standard atmosphere at sea level), and set the incoming stream (X-component) velocity in accordance with the desired Reynolds number.

For convenience we can use the **Dependency** box to specify the incoming flow velocity in terms of the Reynolds number.

7 Click in the **Velocity in X direction** field. The **Dependency** button is enabled.

8 Click **Dependency**. The **Dependency** dialog box appears.
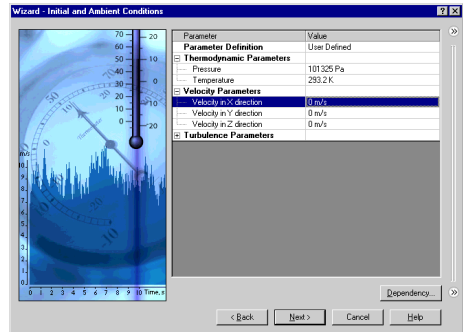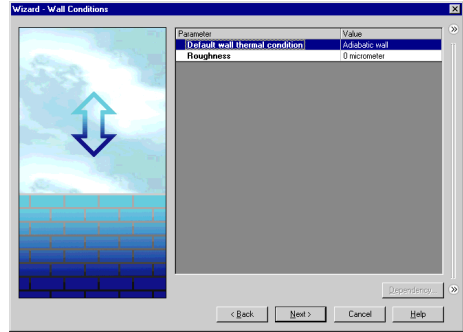
📖 *Using **Dependency** you can specify data in several ways: as a constant, as a tabular or formula dependency on x, y, z, r, θ, φ coordinates and time t (only for time-dependent analysis). The radius r is the distance from a point to the **Reference axis** selected from the reference coordinate system (the **Global Coordinate System** for all data set in the **Wizard** and **General Settings** dialog boxes), while θ and φ are the polar and azimuthal angles of spherical coordinate system, respectively. Therefore, by combination of r, θ, and φ coordinates you can specify data in cylindrical or spherical coordinate systems.*

9 In the **Dependency type** list select **Formula Definition**.

10 In the **Formula** box type the formula defining the flow velocity using the Reynolds number: `1*(0.0010115/0.01/998.19)`. Here:

> **1** – the Reynolds number (Re)
> **0.0010115** (Pa*s) - the water dynamic viscosity (μ) at the specified temperature of 293.2 K
> **0.01 (m) -** the cylinder diameter (D)
> **998.19** (kg/m^3)- the water density (ρ) at the specified temperature of 293.2 K

11 Click **OK**. You will return to the **Initial and Ambient Conditions** dialog box.

For most flows it is difficult to have a good estimation of their turbulence *a priori,* so it is recommended that the default turbulence parameters be used. The default turbulence intensity values proposed by Flow Simulation are 0.1% for external analyses and 2% for internal analyses and these values are appropriate for most cases. In this project we will specify a turbulence intensity of 1%.

12 Expand the **Turbulence parameters** item and in the **Turbulence intensity** box type 1.

Click **Next**.
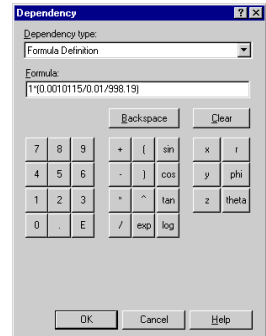
13 In the **Result and Geometry Resolution** dialog box specify the result resolution level of **7** and accept the automatically defined minimum gap size and minimum wall thickness.

Click **Finish**. The project is created and the 3D Computational Domain is automatically generated.
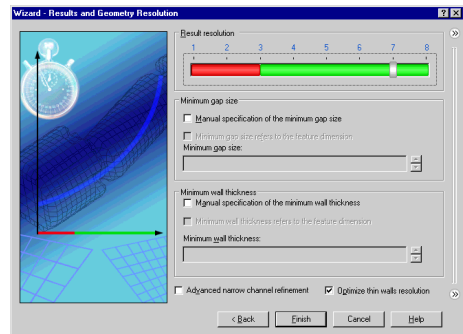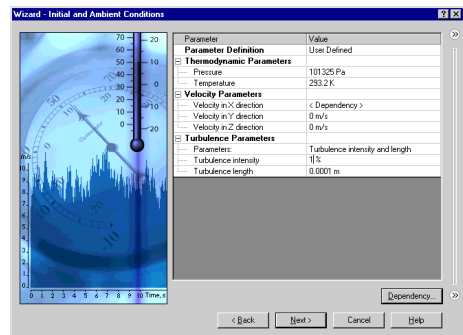
In this tutorial we are interested in determining the drag coefficient of the cylinder only, without the accompanying 3D effects. Thus, to reduce the required CPU time and computer memory, we will perform a two-dimensional (2D) analysis in this tutorial.

Specifying 2D simulation

1   In the Flow Simulation Analysis tree, expand the **Input Data** item.

2   Right-click the **Computational Domain** icon and select **Edit Definition**.

3   Under **Type** select **2D simulation** ⬚ and **XY plane** (since the Z-axis is the cylinder axis).
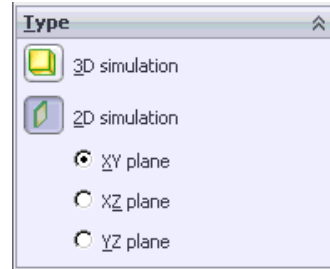
4   Automatically the **Symmetry** ⬚ condition is specified at the **Z min** ⬚ₓ and **Z max** ⬚ₓ boundaries of the Computational domain under **Size and Conditions**.

You can see that the **Z min** ⬚ₓ and **Z max** ⬚ₓ boundaries are set automatically, basing on the model dimensions.

Thus the reference cylinder length L in the cylinder drag (C_D) formula presented above is equal to $L = Z\ max\text{-}Z\ min = 0.002\ m$.
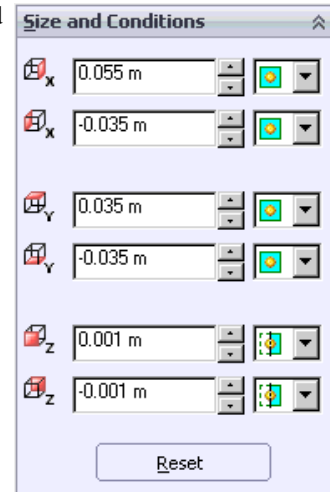
    📖 *For most cases, to study the flow field around an external body and to investigate the effects of design changes it is recommended to use the default* **Computational Domain** *size as determined by Flow Simulation. However, in this case we will compare the Flow Simulation results to experimental results and we would like to determine the drag coefficient with a high degree of accuracy. In order to eliminate any disturbances of the incoming flow at the* **Computational Domain** *boundaries due to the presence of the cylinder, we will manually set the boundaries farther away from the cylinder. The accuracy will be increased at the expense of required CPU time and memory due to the larger size of* **Computational Domain***.*

5 Under **Size and Conditions** specify the X and Y coordinates of the Computational domain boundaries as shown on the picture to the right.
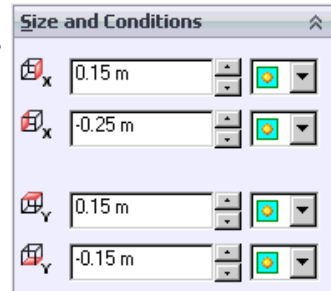
6 Click **OK** ✔.

Since the incoming flow is aligned with the X-axis direction, the cylinder drag coefficient is calculated through the X-component of the force acting on the cylinder.

The X-component of force can be determined easily by specifying the appropriate Flow Simulation goal. For this case you will specify the **X - Component of Force** as a **Global Goal**. This ensures that the calculation will not be finished until **X - Component of Force** in the entire computational domain (i.e. on the cylinder surface) is fully converged.

Specifying a Global Goal

1 Click **Flow Simulation**, **Insert**, **Global Goals**.

2 In the **Parameter** table select the first check box in the **X - Component of Force** row.

3 Accept selected **Use for Conv.** check box to use this 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. In this example the default Global Coordinate System meets the task.*

4 Click **OK** ✔. The new **GG X - Component of Force 1** item appears in the Flow Simulation Analysis tree.
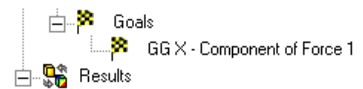
Specifying an Equation Goal

When the calculation is finished, you will need to manually calculate the drag coefficient from the obtained force value. Instead, let Flow Simulation to make all the necessary calculations for you by specifying an **Equation Goal**.

1   Click **Flow Simulation**, **Insert**, **Equation Goal.**

2   In the Flow Simulation Analysis tree select the GG X - Component of Force 1 goal. It appears in the **Expression** box.
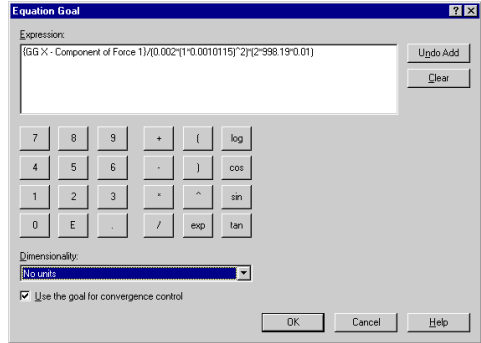
3   Use buttons in the calculator or keyboard to complete the expression as follows:

{GG X - Component of Force 1}/(0.002*(1*0.0010115)^2)*(2*998.19*0.01).

4   Select **No units** in the **Dimensionality** list and click **OK**. The new **Equation Goal 1** item appears in the Flow Simulation Analysis tree.

5   Rename the **Equation Goal 1** to Drag Coefficient.

To compare the Flow Simulation results with the experimental curve taken from **Ref.1**, we will obtain the results at a Reynolds number of 1, 10^3 and 10^5. As with Re = 1, the **Cylinder 0.01m.SLDPRT** is used to calculate the flow at the Reynolds number of 10^3. The **Cylinder 1m.SLDPRT** is used to calculate the flow at the Reynolds number of 10^5.

Cloning a Project and Creating a New Configuration

1   In the Flow Simulation Analysis tree, right-click the top **Re 1** icon and select **Clone Project**.

2   In the **Configuration name** box, type Re 1000.
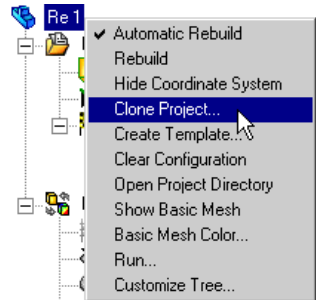
3 Click **OK**. The new **Re 1000** configuration is created with the Flow Simulation project attached.
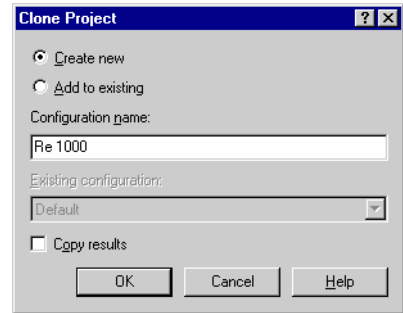
Since the new project is a copy of the **Re 1** Flow Simulation project, you only need to change the flow velocity value in accordance with the Reynolds number of 1000. Use the **General Settings** dialog box to change the data specified in the **Wizard**, except the settings for **Units** and **Result and Geometry Resolution**.

📖 *The **General Settings** always presents the current state of the project parameters. You can change **General Settings** to correct the settings made in the **Wizard** or to modify the project created with the Flow Simulation **Template** in accordance with the new project requirements.*

Changing Project Settings

1 Click **Flow Simulation**, **General Settings**. The **General Settings** dialog box appears.

2 As it has been mentioned above, since the flow over a cylinder is unsteady at Re > 40, select the **Time-dependent** physical feature for this project.

3 In the **Navigator** click **Initial and ambient conditions**.
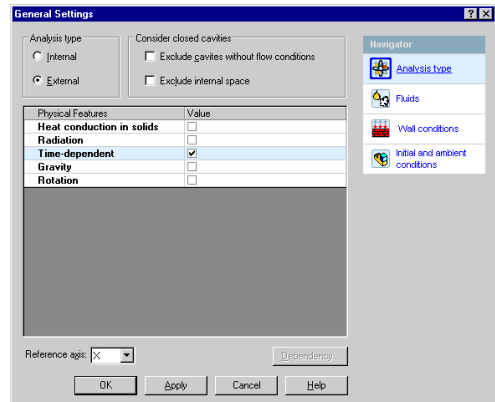
4   Click the **Velocity in X direction** field
    and then click **Dependency**.
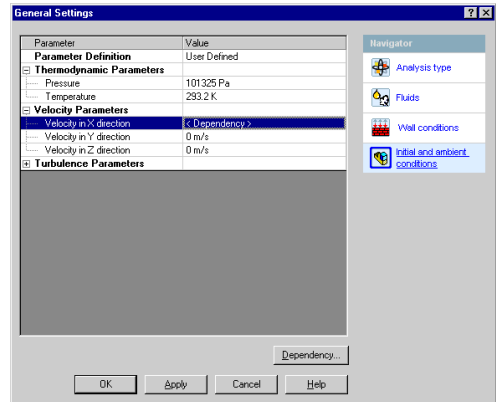
5   In the **Formula** box, type the formula
    for the new Reynolds number:
    1e3*(0.0010115/0.01/998.19).
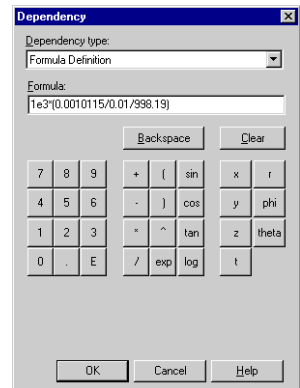
6   Click **OK** to return to the **General Settings** dialog box.

7   Click **OK** to save changes and close the **General Settings**
    dialog box.

Changing the Equation Goal

1   Right-click the **Drag Coefficient** icon under **Goals** and select **Edit Definition.**

2   In the **Expression** box type the new formula for the new Reynolds number:
    {GG X – Component of Force 1}/(0.002*(0.0010115*10^3)^2)*(2*998.19*0.01).

3   Select **No units** in the **Dimensionality** list.

4   Click **OK** to save changes and close the **Equation Goal** dialog box.

In the experiments performed with one fluid medium, the Reynolds number's large rise is usually obtained by increasing both the velocity and the model overall dimension (i.e. cylinder diameter) since it is difficult to increase only velocity by e.g. 10^5 times. Since our simulation is performed with water only, let us increase the cylinder diameter to 1 m to perform the calculation at a Reynolds number of 10^5.

Cloning a project is convenient if you want to create similar projects for the same model. The easiest way to apply the same general project settings to another model is to use the Flow Simulation *Template*.

  📖 *Template contains all of the general project settings that can be used as a basis for a new project. These settings are: problem type, physical features, fluids, solids, initial and ambient flow parameters, wall heat condition, geometry and result resolution, and unit settings. Notice that **Boundary Conditions**, **Fans**, **Initial Conditions**, **Goals** and other features accessible from the **Flow Simulation, Insert** menu, as well as results are not stored in the template. Initially, only the New Project default template is available, but you can easily create your own templates.*

Creating a Template

1 Click **Flow Simulation**, **Project**, **Create Template**. The **Create Template** dialog box appears.

2 In the **Template name** box, type Cylinder Drag.

3 Click **Save**. The new Flow Simulation template is created.
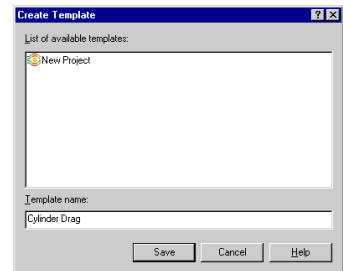
  📖 *All templates are stored as* **.fwp** *files in the* **<install_dir>/Template** *folder, so you can easily apply a template to any previously created models.*

4 Save the model.

Next, create a new project based on the *Cylinder Drag* template.

Creating a Project from the Template

Open the **Cylinder 1m.SLDPRT** file located in the **cylinder 1m** folder.

1 Click **Flow Simulation**, **Project**, **New**. The **New Flow Simulation Project** dialog box appears.

2 In the **Configuration name** box, type Re 1e5.

3 In the **List of templates**, select **Cylinder Drag**.

4 Click **OK**.
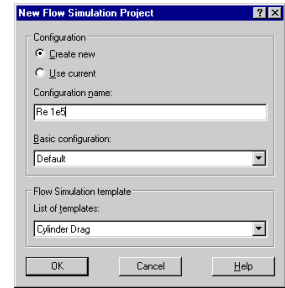
The newly created project has the same settings as the **Re 1000** project with the **cylinder 0.01m** model. The only exceptions are **Geometry Resolution** and **Computational Domain** size, which are calculated by Flow Simulation in accordance with the new model geometry.

Notice that the **2D simulation** setting and **Global Goal** are retained. Next, you can modify the project in accordance with the new model geometry.

1 Click **Flow Simulation**, **Computational Domain** and adjust the computational domain size as shown at the picture to the right.
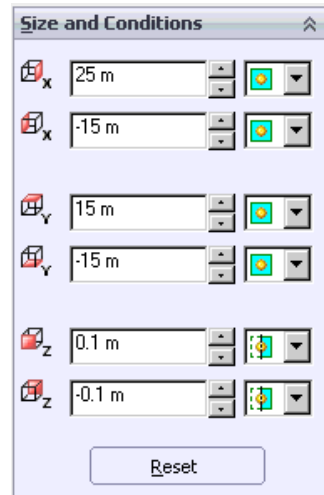
2 Click **OK** ✓.

3 Open the **General Settings** dialog box and click **Initial and ambient conditions**, click the **Velocity in X direction** field, then click **Dependency**.

4 Change the velocity X component formula as follows: 1e5*(0.0010115/1/998.19).

Click **OK** to return to the General Settings dialog box.

By default, Flow Simulation determines the default turbulence length basis equal to one percent of the model overall dimension (i.e. cylinder diameter). Since the **Re 1e5** project was created from the template, it inherited the turbulence length value calculated for the small cylinder ($d = 0.01$m). For the **cylinder 1m** we need to change this value.
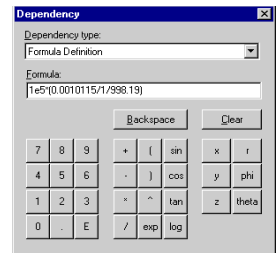
5 In the **General Settings** dialog box expand the **Turbulence parameters** item. Type 0.01 m in the **Turbulence length** field.
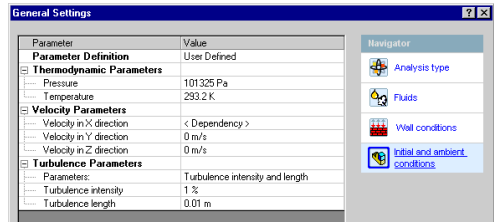
6 Click **OK**.

7 Create the **Equation Goal** for the drag coefficient of the cylinder as it was described before. In the Expression box enter the formula:

{GG X - Component of Force 1}/(0.2*(0.0010115*10^5)^2)*(2*998.19*1).

8 Select **No units** in the **Dimensionality** list.

9 Click **OK**. Rename the **Equation Goal 1** to Drag Coefficient.

Now you can solve all of the projects created for both the cylinders.

Solving a Set of Projects

Flow Simulation allows you to automatically solve a set of projects that exist in any currently opened document.

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

2 Select the **Solve** check box in the **All projects** row to select **Solve** for all projects (**Re 1**, **Re 1000**, **Re 1e5**). Also select the **Close Monitor** check box in the **All projects** row. When the **Close Monitor** check box is selected, Flow Simulation automatically closes the **Solver Monitor** window when the calculation finishes.
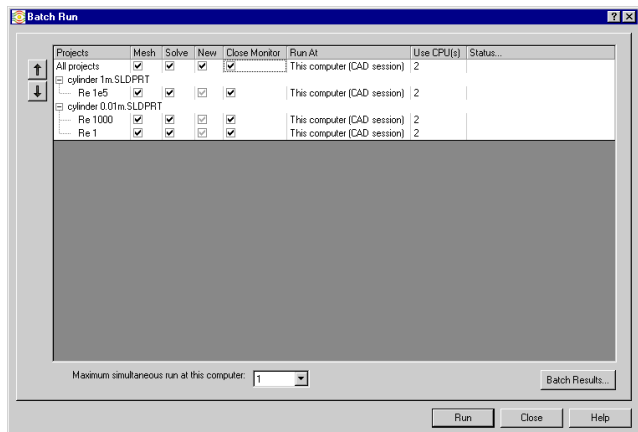
3 Click **Run**.

Getting Results

After all calculations are complete, go to the **cylinder 0.01m** model and activate the **Re 1000** configuration. Create **Goal Plot** to obtain the **Drag Coefficient** value:

1 Click **Flow Simulation**, **Results**, **Load\Unload Results**.

2   In the **Load Results** dialog box, keep the default project's results file (2.fld) and click **Open**.

3   In the Flow Simulation Analysis tree, under **Results**, right-click the **Goal Plots** icon and select **Insert**. The **Goal Plot** dialog box appears.

4   Select **All**.

5   Click **OK** . The **Goals1** Excel workbook is created. Switch to Excel to obtain the value.
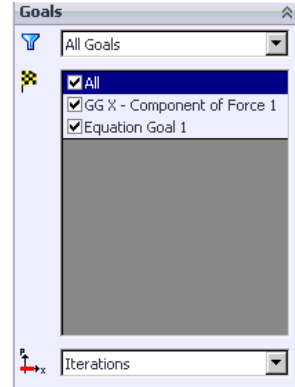
cylinder 0.01m.SLDPRT [Re 1000]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value
GG X - Component of Force	[N]	0.000118661	0.000112731	0.000102502	0.000122439
Drag Coefficient	[ ]	1.157683117	1.099830977	1.00002738	1.194542459

6   Activate the **Re 1** configuration and load results. Create the goal plot for both the goals.

cylinder 0.01m.SLDPRT [Re 1]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value
GG X - Component of Force	[N]	1.14446E-09	1.16833E-09	1.12811E-09	1.8234E-09
Drag Coefficient	[ ]	11.1656052	11.3984466	11.00608012	17.78943946

7   Switch to the **cylinder 1m** part, activate the **Re 1e5** configuration, load results and create the goal plot for both the goals.

cylinder 1m.SLDPRT [Re 1e5]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value
GG X - Component of Force	[N]	0.44242974	0.442499134	0.429568256	0.451472343
Drag Coefficient	[ ]	0.431644044	0.431711746	0.419096102	0.440466204

Even if the calculation is steady, the averaged value is more preferred, since in this case the oscillation effect is of less perceptibility. We will use the averaged goal value for the other two cases as well.

You can now compare Flow Simulation results with the experimental curve.
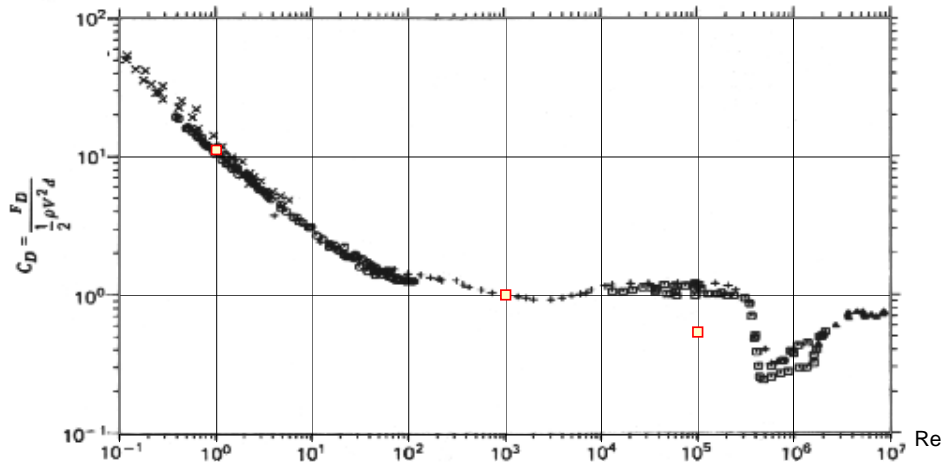
Ref. 1 Roland L. Panton, "Incompressible flow" Second edition. John Wiley & sons Inc., 1995

6

Heat Exchanger Efficiency

Flow Simulation can be used to study the fluid flow and heat transfer for a wide variety of engineering equipment. In this example we use Flow Simulation to determine the efficiency of a counterflow heat exchanger and to observe the temperature and flow patterns inside of it. With Flow Simulation the determination of heat exchanger efficiency is straightforward and by investigating the flow and temperature patterns, the design engineer can gain insight into the physical processes involved thus giving guidance for improvements to the design.

A convenient measure of heat exchanger performance is its "efficiency" in transferring a given amount of heat from one fluid at higher temperature to another fluid at lower temperature. The efficiency can be determined if the temperatures at all flow openings are known. In Flow Simulation the temperatures at the fluid inlets are specified and the temperatures at the outlets can be easily determined. Heat exchanger efficiency is defined as follows:

$$\varepsilon = \frac{actual\ heat\ transfer}{maximum\ possible\ heat\ transfer}$$

The actual heat transfer can be calculated as either the energy lost by the hot fluid or the energy gained by the cold fluid. The maximum possible heat transfer is attained if one of the fluids was to undergo a temperature change equal to the maximum temperature difference present in the exchanger, which is the difference in the inlet temperatures of the hot and cold fluids, respectively: $(T_{hot}^{inlet} - T_{cold}^{inlet})$. Thus, the efficiency of a counterflow

heat exchanger is defined as follows: $\varepsilon = \dfrac{T_{hot}^{inlet} - T_{hot}^{outlet}}{T_{hot}^{inlet} - T_{cold}^{inlet}}$ - if hot fluid capacity rate is less

than cold fluid capacity rate or $\varepsilon = \dfrac{T_{cold}^{outlet} - T_{cold}^{inlet}}{T_{hot}^{inlet} - T_{cold}^{inlet}}$ - if hot fluid capacity rate is more than

cold fluid capacity rate, where the capacity rate is the product of the mass flow and the

specific heat capacity: $C = \dot{m}c$ (**Ref.2**)

The goal of the project is to calculate the efficiency of the counterflow heat exchanger. Also, we will determine the average temperature of the heat exchanger central tube's wall. The obtained wall temperature value can be further used for structural and fatigue analysis.

Open the Model

Click **File**, **Open**. In the **Open** dialog box, browse to the **Heat Exchanger.SLDASM** assembly located in the **Tutorial 3 - Heat Exchanger** folder and click **Open** (or double-click the assembly). Alternatively, you can drag and drop the **Heat Exchanger.SLDASM** file to an empty area of SolidWorks window.

Cold water = 0.02 kg/s

$T_{inlet} = 293.2$ K

Warm water

Air

Steel

Hot air = 10 m/s

$T_{inlet} = 600$ K
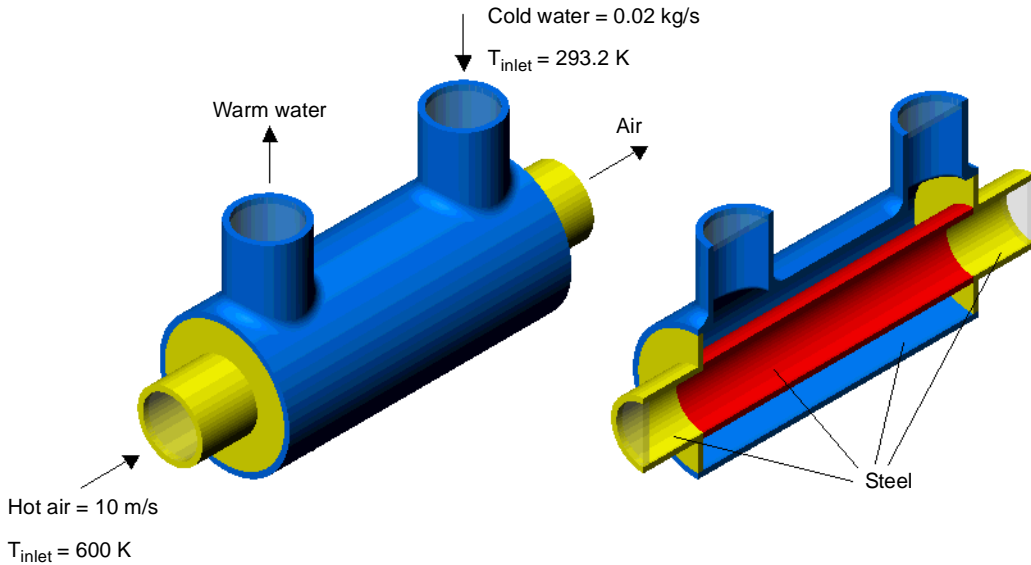
Creating a Project

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

2 Select **Create new**. In the **Configuration name** box type Level 3. The 'Level 3' name was chosen because this problem will be calculated using **Result Resolution** level 3.

Click **Next**.
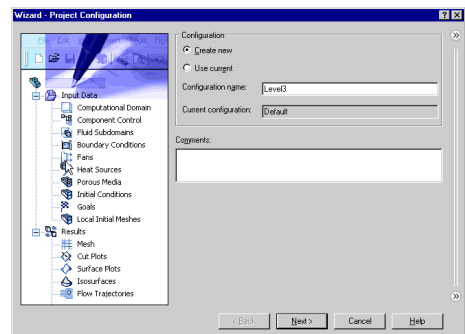
3 In the **Units** dialog box select the desired system of units for both input and output (results). For this project we will use the International System **SI** by default.
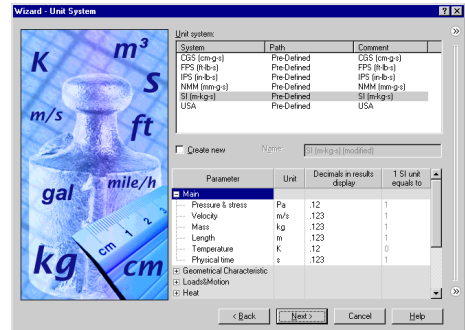
Click **Next**.

4 In the **Analysis Type** dialog box among **Physical features** select **Heat conduction in solids**.

📖 *By default, Flow Simulation will consider heat conduction not in solids, but only within the fluid and between the walls and the fluid (i.e., convection). Selecting the* **Heat conduction in solids** *option enables the combination of convection and conduction heat transfer, known as conjugate heat transfer. In this project we will analyze heat transfer between the fluids through the model walls, as well as inside the solids.*
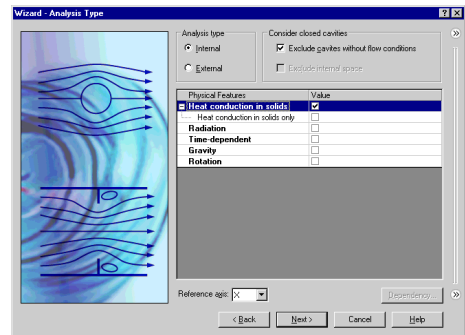
Click **Next**.

5 Since two fluids (water and air) are used in this project, expand the **Liquids** folder and add **Water** and then expand the **Gases** folder and add **Air** to the **Project Fluids** list. Check that the **Default fluid type** is **Liquids**.
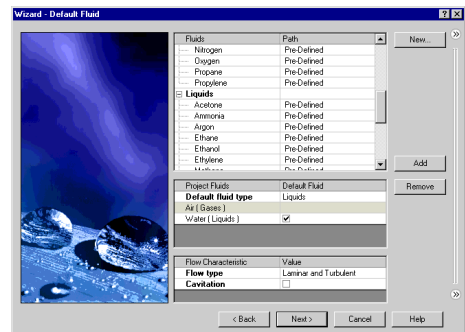
Click **Next**.

6 Since we have selected the **Heat conduction in solids** option at the **Analysis Type** step of the Wizard, the **Default Solid** dialog box appears. In this dialog you specify the default solid material applied to all solid components. To assign a different material to a particular assembly component you need to create a **Solid Material** condition for this component.

If the solid material you wish to specify as the default is not available in the **Solids** table, you can click **New** and define a new substance in the **Engineering Database**. The tube and its cooler in this project are made of stainless steel.

Expand the **Alloys** folder and click **Steel Stainless 321** to make it the default solid material.

Click **Next**.
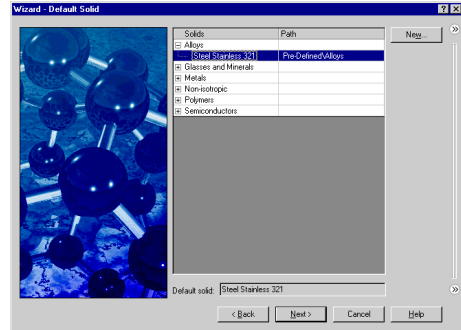
 📖 *If a component has been previously assigned a solid material by the SolidWorks'* *Materials Editor, you can import this material into Flow Simulation and apply this* *solid material to the component in the Flow Simulation project by using the* **Insert Material from Model** *option accessible under* **Flow Simulation**, **Tools**.

7 In the **Wall Condition** dialog box, select **Heat transfer coefficient** as **Default outer wall thermal condition**.

 📖 *This condition allows you to define the heat transfer from the outer model walls to an external fluid (not modeled) by specifying the reference fluid temperature and the heat transfer coefficient value.*
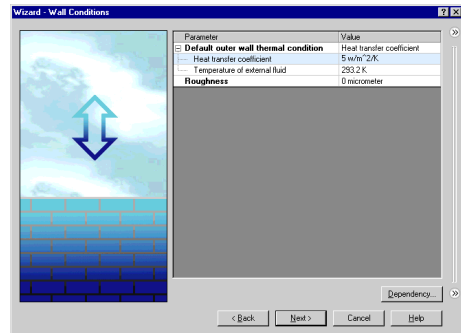
Set the **Heat transfer coefficient** value to 5 W/m^2/K.

In this project we do not consider walls roughness.

Click **Next**.

8 In the **Initial Conditions** dialog box under **Thermodynamics parameters** enter 2 atm in the **Value** cell for the **Pressure** parameter. Flow Simulation automatically converts the entered value to the selected system of units.
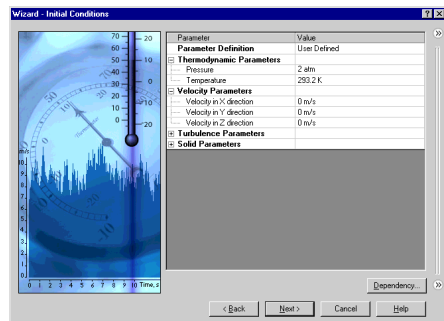
Click **Next** accepting the default values of other parameters for initial conditions.

9 In the **Results and Geometry Resolution** dialog box we accept the default result resolution level 3 and the default minimum gap size and minimum wall thickness.
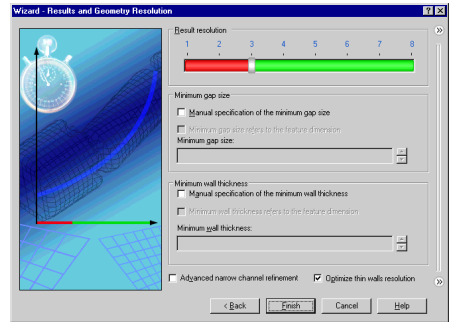
Click **Finish**.

After finishing the **Wizard** you will complete the project definition by using the Flow Simulation Analysis tree. First of all you can take advantage of the symmetry of the heat exchanger to reduce the CPU time and memory required for the calculation. Since this model is symmetric, it is possible to "cut" the model in half and use a symmetry boundary condition at the plane of symmetry. This procedure is not required, but is recommended for efficient analyses.

Symmetry Condition

1 In the Flow Simulation Analysis tree, expand the **Input Data** item.

2 Right-click the **Computational Domain** icon and select **Edit Definition**.

3 Under **Size and Conditions** select the **Symmetry** condition at the **X max** boundary and type 0 in the **X max** edit-box

  📖 *To resize the domain manually, select the* **Computational Domain** *item in the Flow Simulation analysis tree, and in the graphics area click and drag the arrow handles at the sides of the computational domain frame to the desired positions, then adjust the exact coordinates in the appearing callouts..*
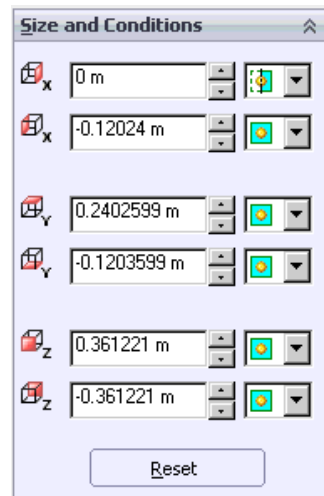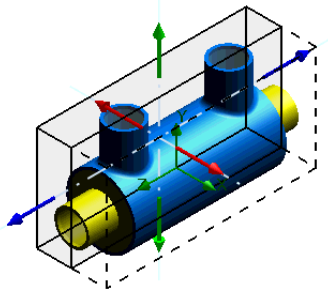
4   Click **OK** ✔ .
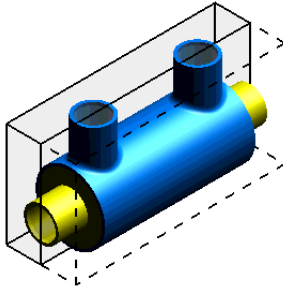
Specifying a Fluid Subdomain

Since we have selected **Liquids** as the **Default fluid type** and **Water** as the **Default fluid** in the Wizard, we need to specify another fluid type and select another fluid (air) for the fluid region inside the tube through which the hot air flows. We can do this by creating a **Fluid Subdomain**. When defining a **Fluid Subdomain** parameters we will specify **Gas** as the fluid type for the selected region, **Air** as the fluid and the initial temperature of 600 K and flow velocity of 10 m/s as the initial conditions in the selected fluid region.

1   Click **Flow Simulation**, **Insert**, **Fluid Subdomain**.

2   Select the **Flange 1** inner face (in contact with the fluid). Immediately the fluid subdomain you are going to create is displayed in the graphics area as a body of blue color.

📖 *To specify the fluid subdomain within a fluid region we must specify this condition on the one of the faces lying on the region's boundary - i.e. on the boundary between solid and fluid substances. The fluid subdomain specified on the region's boundary will be applied to the entire fluid region. You may check if the region to apply a fluid subdomain is selected properly by looking at the fluid subdomain visualization in the graphics area.*
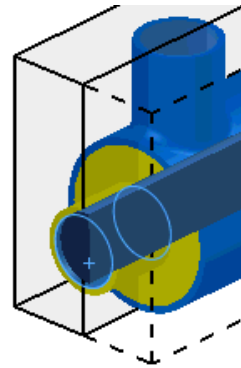
3   Accept the default **Coordinate System** and the **Reference axis**.
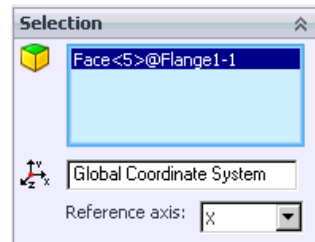
4 In the **Fluid type** list select **Gases / Real Gases / Steam**. Because **Air** was defined in the Wizard as one of the **Project fluids** and you have selected the appropriate fluid type, it appears as the fluid assigned to the fluid subdomain.

   📖 *In the **Fluids** group box, Flow Simulation allows you to specify the fluid type and/or fluids to be assigned for the fluid subdomain as well as flow characteristics, depending on the selected fluid type.*
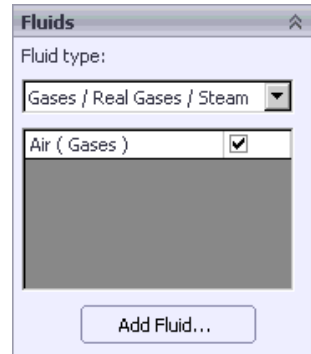
5 Under **Flow Parameters** in the **Velocity in Z Direction** V_z box enter -10.

   📖 *Flow Simulation allows you to specify initial flow parameters, initial thermodynamic parameters, and initial turbulence parameters (after a face to apply the Fluid Subdomain is selected). These settings are applied to the specified fluid subdomain.*
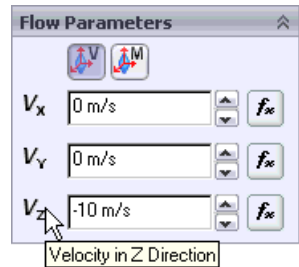
6 Under **Thermodynamic parameters** in the **Static Pressure** P box enter 1 atm. Flow Simulation automatically converts the entered value to the selected system of units.
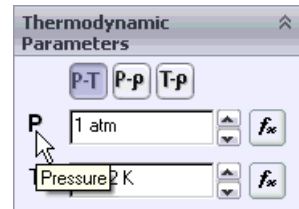
7 Under **Thermodynamic parameters** in the **Temperature** T box enter 600.

   These initial conditions are not necessary and the parameters of the hot air inlet flow are defined by the boundary condition, but we specify them to improve calculation convergence.
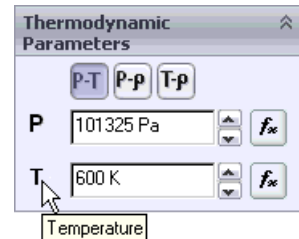
8 Click **OK** ✔. The new **Fluid Subdomain 1** item appears in the Analysis tree.

9   To easily identify the specified condition you can give a more descriptive name for the **Fluid Subdomain 1** item. Right-click the **Fluid Subdomain 1** item and select **Properties**. In the **Name** box type `Hot Air` and click **OK**.
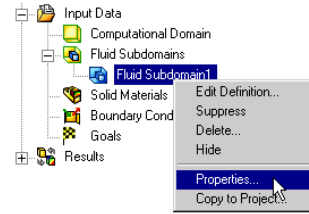
You can also click-pause-click an item to rename it directly in the Flow Simulation Analysis tree.

Specifying Boundary Conditions

1   Right-click the **Boundary Conditions** icon in the Flow Simulation Analysis tree and select **Insert Boundary Condition**. The **Boundary Condition** dialog box appears.

2   Select the **Water Inlet Lid** inner face (in contact with the fluid).

The selected face appears in the **Faces to Apply the Boundary Condition** list.

3   Accept the default **Inlet Mass Flow** condition and the default **Coordinate System** and **Reference axis**.
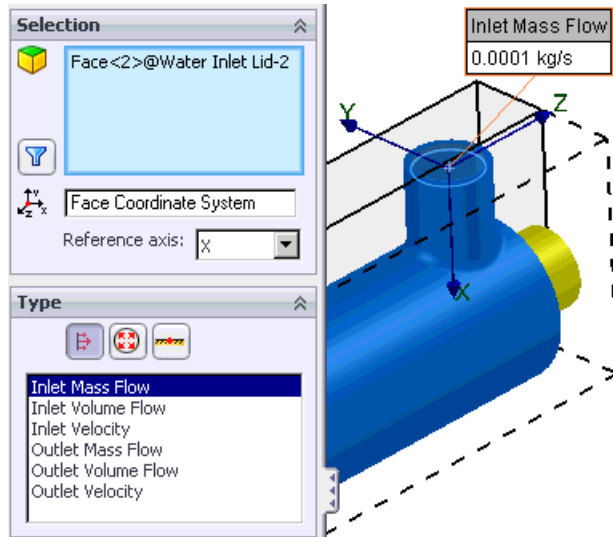
📖 *If you specify a new boundary condition, a callout showing the name of the condition and the default values of the condition parameters appears in the graphics area. You can double-click the callout to open the quick-edit dialog.*

4   Click the **Mass Flow Rate Normal to Face** $\dot{m}$ box and set its value equal to `0.01 kg/s`. Since the symmetry plane halves the opening, we need to specify a half of the actual mass flow rate.

5   Click **OK** ✔. The new **Inlet Mass Flow 1** item appears in the Analysis tree.
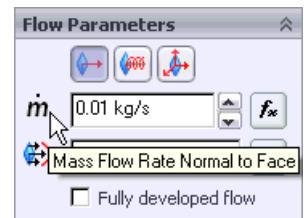
This boundary condition specifies that water enters the steel jacket of the heat exchanger at a mass flow rate of 0.02 kg/s and temperature of 293.2 K.

6 Rename the **Inlet Mass Flow 1** item to `Inlet Mass Flow - Cold Water`.
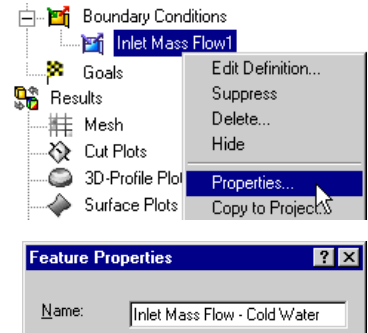
Next, specify the water outlet **Environment Pressure** condition.

7 In the Flow Simulation Analysis tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.

8 Select the **Water Outlet Lid** inner face (in contact with the fluid). The selected face appears in the

   Faces to Apply the Boundary Condition 📦 list.
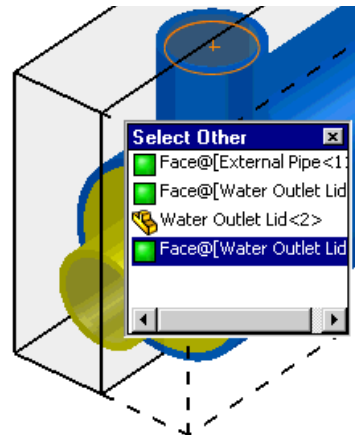
9 Click **Pressure Openings** 🔘 and in the **Type of Boundary Condition** list select the **Environment Pressure** item.
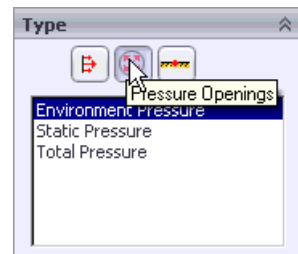
10 Accept the value of **Environment Pressure** (202650 Pa), taken from the value specified at the **Initial Conditions** step of the **Wizard**, and the default values of

   Temperature T (293.2 K) and all other parameters.
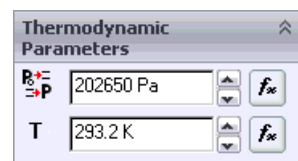
11 Click **OK** ![checkmark]. The new **Environment Pressure 1** item appears in the Flow Simulation Analysis tree.

12 Rename the **Environment Pressure 1** item to Environment Pressure – Warm Water.

> ⊟ ⬜ Boundary Conditions
>   ⬜ Inlet Mass Flow - Cold Water
>   ⬜ Environment Pressure - Warm Water
>   ❌ Goals
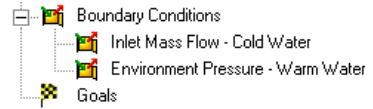
Next we will specify the boundary conditions for the hot air flow.

13 In the Flow Simulation Analysis tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.

14 Select the **Air Inlet Lid** inner face (in contact with the fluid).

The selected face appears in the **Faces to Apply the**

Boundary Condition ![icon] list. Accept the default

Coordinate System ![icon] and **Reference axis**.

15 Under **Type** select the **Inlet Velocity** condition.

> **Select Other**
> ⬜ Face@[Flange1<1
> ⬜ Face@[Air Inlet Lid
> 🔖 Air Inlet Lid<1>
> ⬜ Face@[Air Inlet Lid
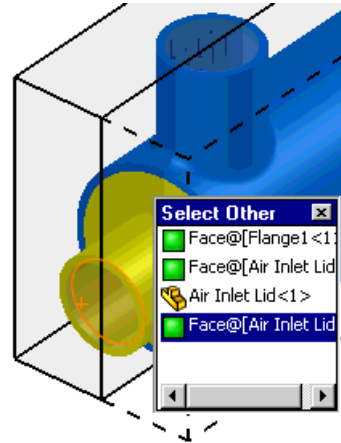
16 Click the **Velocity Normal to Face** ![V] box and set its value equal to 10 (type the value, the units will appear automatically).

> **Flow Parameters**
> V [10 m/s] f_x
> Velocity Normal to Face
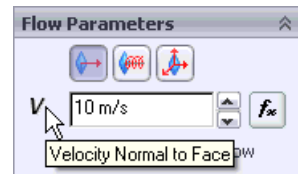
17 Expand the **Thermodynamic Parameters** item. The default temperature value is equal to the value specified as the initial temperature of air in the **Fluid Subdomain** dialog box. We accept this value.

> **Thermodynamic Parameters**
> ~P [101325 Pa] f_x
> T   [600 K] f_x
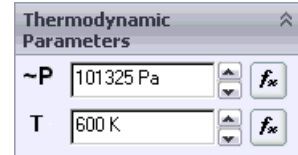
18 Click **OK** ![checkmark]. The new **Inlet Velocity 1** item appears in the Analysis tree.

This boundary condition specifies that air enters the tube at the velocity of 10 m/s and temperature of 600 K.

19 Rename the **Inlet Velocity 1** item to Inlet Velocity – Hot Air.

Next specify the air outlet **Environment Pressure** condition.

20 In the Flow Simulation Analysis tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**. The **Boundary Condition** dialog box appears.

21 Select the **Air Outlet Lid** inner face (in contact with the fluid).

The selected face appears in the **Faces to Apply the**

Boundary Condition ⬛ list.
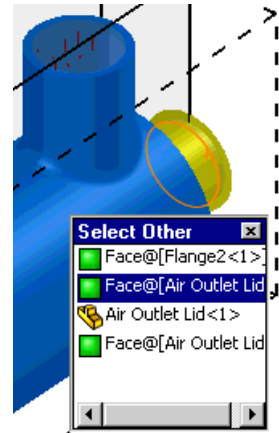
22 Click **Pressure Openings** 🔘 and in the **Type of Boundary Condition** list select the **Environment Pressure** item.
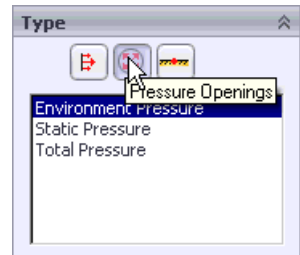
23 Check the values of **Environment Pressure** ⇨P (101325

Pa) and **Temperature** T (600 K). If they are different, correct them. Accept the default values of other parameters.

Click **OK** ✅ .
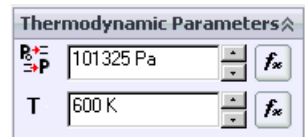
24 Rename the new item **Environment Pressure 1** to Environment Pressure – Air.
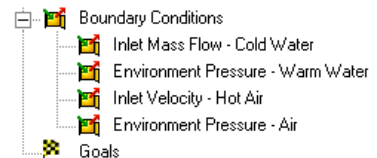
This project involving analysis of heat conduction in solids. Therefore, you must specify the solid materials for the model's components and the initial solid temperature.

Specifying Solid Materials

Notice that the auxiliary lids on the openings are solid. Since the material for the lids is the default stainless steel, they will have an influence on the heat transfer. You cannot suppress or disable them in the **Component Control** dialog box, because boundary conditions must be specified on solid surfaces in contact with the fluid region. However, you can exclude the lids from the heat conduction analysis by specifying the lids as insulators.

1   Right-click the **Solid Materials** icon and select **Insert Solid Material**.

2   In the flyout FeatureManager design tree, select all the lid components. As you select the lids, their names appear in the **Components to Apply the Solid Material** 🐿 list.

3   In the **Solid** group box expand the list of **Pre-Defined** materials and select the **Insulator** solid in the **Glasses & Minerals** folder.

4   Click **OK** ✔. Now all auxiliary lids are defined as insulators.
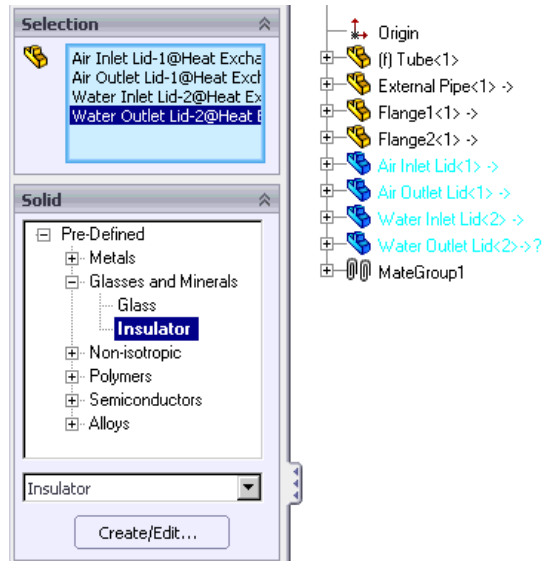
📖 *The thermal conductivity of the* **Insulator** *substance is zero. Hence there is no heat transferred through an insulator.*

5   Rename the **Insulator Solid Material 1** item to `Insulators`.

Specifying a Volume Goal

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 the **Tube** part.

3 In the **Parameter** table select the **Av** check box in the **Temperature of Solid** row. Accept the selected **Use for Conv.** check box to use this goal for convergence control.

4 In the **Name template** type VG Av T of Tube.

5 Click **OK** ✔.
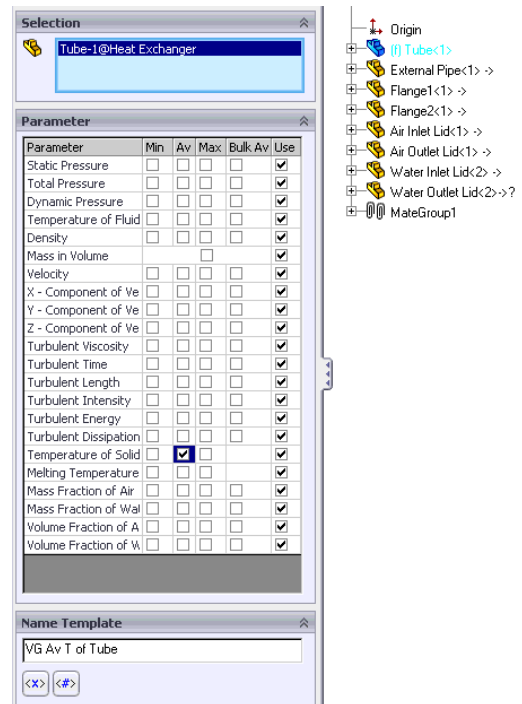
Running the Calculation

1 Click **Flow Simulation**, **Solve**, **Run**. The **Run** dialog box appears.

2 Click **Run**.

After the calculation finishes you can obtain the temperature of interest by creating the corresponding **Goal Plot**.

Viewing the Goals

In addition to using the Flow Simulation Analysis tree you can use Flow Simulation Toolbars and SolidWorks CommandManager to get fast and easy access to the most frequently used Flow Simulation features. Toolbars and SolidWorks CommandManager are very convenient for displaying results.

Click **View**, **Toolbars**, **Flow Simulation Results**. The **Flow Simulation Results** toolbar appears.

Click **View**, **Toolbars**, **Flow Simulation Results Features**. The **Flow Simulation Results Features** toolbar appears.

Click **View**, **Toolbars**, **Flow Simulation Display**. The **Flow Simulation Display** toolbar appears.

The SolidWorks CommandManager is a dynamically-updated, context-sensitive toolbar, which allows you to save space for the graphics area and access all toolbar buttons from one location. The tabs below the CommandManager is used to select a specific group of commands and features to make their toolbar buttons available in the CommandManager. To get access to the Flow Simulation commands and features, click the **Flow Simulation** tab of the CommandManager.

If you wish, you may hide the Flow Simulation toolbars to save the space for the graphics area, since all necessary commands are available in the CommandManager. To hide a toolbar, click its name again in the **View**, **Toolbars** menu.

1   Click **Generate goal plot** on the **Results Main** toolbar or CommandManager. The **Goal Plot** dialog box appears.

2   Select the goals of the project (actually, in our case there is only one goal) .

3   Click **OK** . The **Goals1** Excel workbook is created.
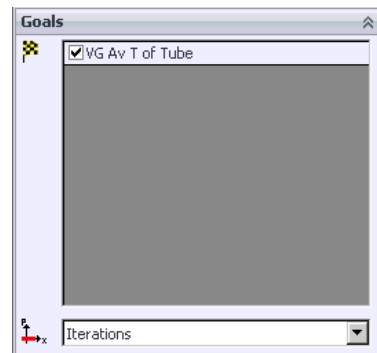
You can view the average temperature of the tube on the **Summary** sheet.

heat exchanger.SLDASM [level 3]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence
VG Av T of Tube	[K]	343.2771368	342.7244032	341.792912	343.2771368	100	Yes

Iterations: 40
Analysis interval: 20

Creating a Cut Plot

1 Click **Cut Plot** on the **Flow Simulation Results Features** toolbar. The **Cut Plot** dialog box appears.

2 In the flyout FeatureManager design tree select **Plane3**.

3 In the **Cut Plot** dialog, in addition to displaying **Contours** , select **Vectors** .
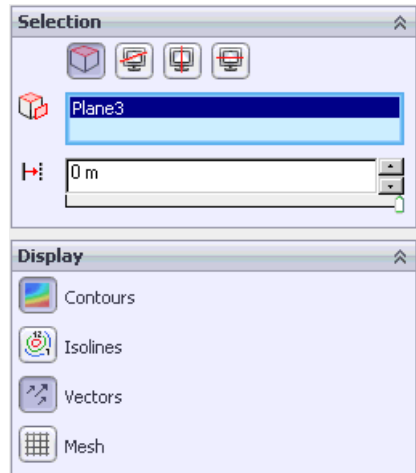
4 Under **Contours** specify the parameter which values to show at the contour plot. In the **Parameter** box, select **Temperature**.

5 Using the slider set the **Number of Levels** to maximum.

6 Under **Vectors** click the **Adjust Minimum and Maximum** and change the **Maximum** velocity to 0.004 m/s.
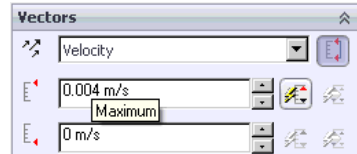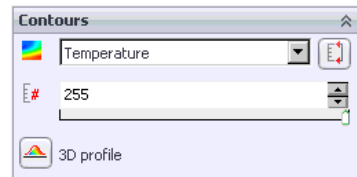
7 Click **OK** . The cut plot is created but the model overlaps it. Click the **Right** view on the **Standard Views** toolbar.

8 Click **Geometry** 🐝 on the **Flow Simulation Display** toolbar to hide the model.
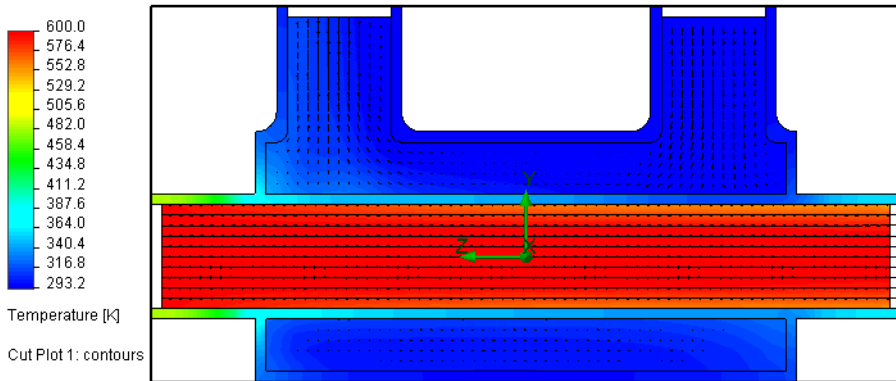
Let us now display the flow development inside the exchanger.

Flow Simulation allows you to display results in all four possible panes of the SolidWorks graphics area. Moreover, for each pane you can specify different View Settings.

9   Click **Window**, **Viewport**, **Two View - Horizontal**.

10 To restore the view orientation in the top pane, click **Right** view on the **Standard Views** toolbar.
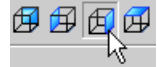
11 Click the bottom pane and select the **Isometric** view on the **Standard Views** toolbar.

The gray contour around the pane border indicates that the view is active.

12 On the **Flow Simulation Display** toolbar,

click **Geometry** 🐝 , then on the **View** toolbar click **Hidden Lines**

Visible 🔲 to show the face outlines. Click the top pane and set the same display mode for it by clicking **Hidden**

Lines Visible 🔲 again.
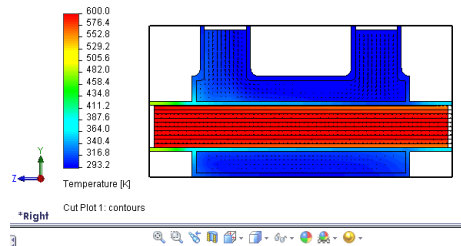
To see how the water flows inside the exchanger we will display the **Flow Trajectories**.
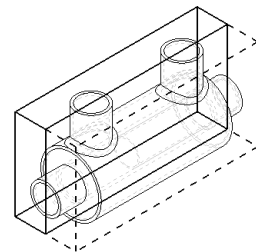
 Click the bottom pane to make it the active pane.

Displaying Flow Trajectories

1 Click **Flow Trajectories** 🍥 on the **Flow Simulation Results Features** toolbar. The **Flow Trajectories** dialog appears.

2 Click the Flow Simulation Analysis tree tab and select the **Inlet Mass Flow – Cold Water** item.

This selects the inner face of the **Water Inlet Lid** to place the trajectories start points on it.
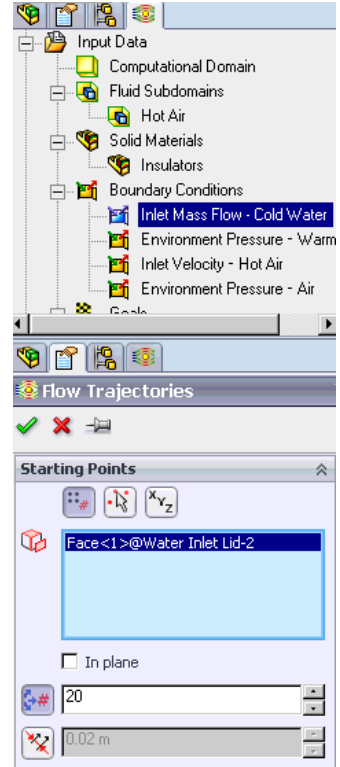
3 Under **Appearance**, in the **Color by Parameter** 🔲 list, select **Velocity**.

4 Click the **Adjust Minimum/Maximum and Number of Levels** 📧 and set **Maximum** ⌐ velocity to 0.004 m/s.

5 Click **OK** ✔. Trajectories are created and displayed.
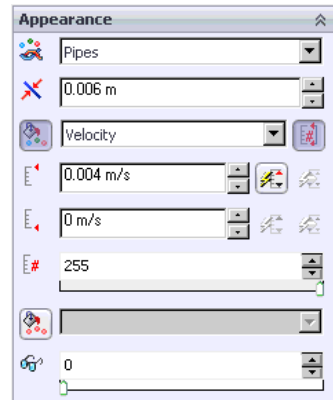
By default the trajectories are colored in accordance with the distribution of the parameter specified in the

Color by Parameter list. Since you specified velocity, the trajectory color corresponds to the velocity value. To define a fixed color for flow

trajectories click **Color** and select a desired color.
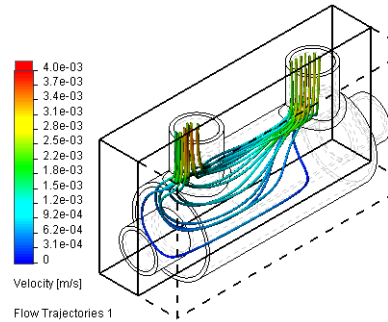
  Notice that in the top pane the temperature contours are still displayed.

Since we are more interested in the temperature distribution let us color the trajectories with the values of temperature.

1 In the velocity palette bar click the caption with the name of the current visualization parameter and select **Temperature** in a dropdown list.

2 Click . Immediately the trajectories are updated.
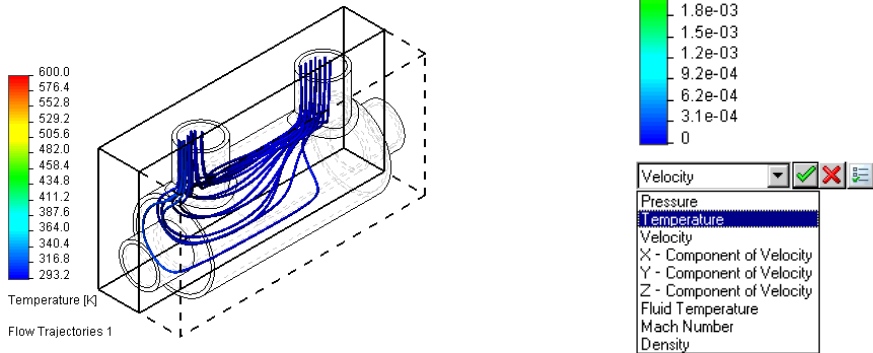
The water temperature range is less than the default overall (**Global**) range (293 – 600), so all of the trajectories are the same blue color. To get more information about the temperature distribution in water you can manually specify the range of interest.

Let us display temperatures in the range of **inlet-outlet** water temperature.

The water minimum temperature value is close to 293 K. Let us obtain the values of air and water temperatures at outlets using Surface Parameters. You will need these values to calculate the heat exchanger efficiency and determine the appropriate temperature range for flow trajectories visualization.

  Surface Parameters *allows you to display parameter values (minimum, maximum, average and integral) calculated over the specified surface. All parameters are divided into two categories: Local and Integral. For local parameters (pressure, temperature, velocity etc.) the maximum, minimum and average values are evaluated.*

Computation of Surface Parameters

1 Click **Surface Parameters** ⬦ on the **Flow Simulation Results Features** toolbar. The **Surface Parameters** dialog appears.

2 Click the Flow Simulation Analysis tree tab and select the **Environment Pressure - Warm Water** item to select the inner face of the **Water Outlet Lid**.
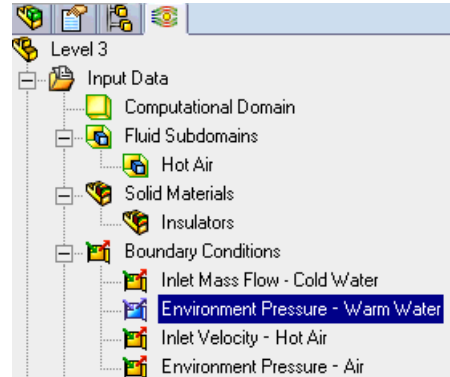
3 Select **Consider entire model** to take into account the **Symmetry** condition to see the values of parameters as if the entire model, not a half of it, was calculated. This is especially convenient for such parameters as mass and volume flow.
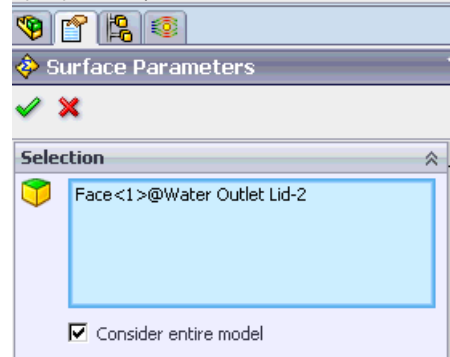
4 Under **Parameters**, select **All**.

5 Click **Show**. The calculated parameters values are displayed on the pane at the bottom of the screen. Local parameters are displayed at the left side of the bottom pane, while integral parameters are displayed at the right side.
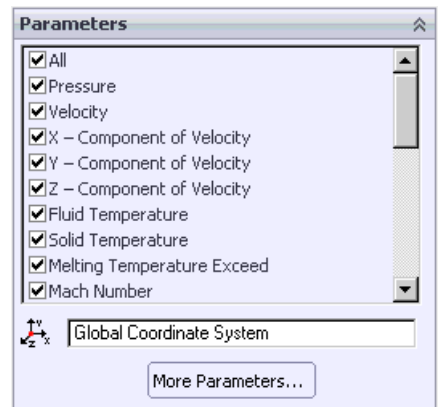
6 Take a look at the local parameters.

Local Parameter	Minimum	Maximum	Average	Bulk Average
Pressure [Pa]	202650	202650	202650	202650
Density [kg/m^3]	990.994	997.357	995.695	995.534
Velocity [m/s]	0.000273491	0.00374912	0.00246499	0.00300914
X – Component of Velocity [m/s]	-0.000169079	0.000170163	-2.83344e-005	-4.91866e-005
Y – Component of Velocity [m/s]	0.000231458	0.00374815	0.00245914	0.00300585
Z – Component of Velocity [m/s]	-0.000177123	0.00014593	1.36949e-005	4.30211e-005
Mach Number [ ]	0	0	0	0
Surface Heat Flux [W/m^2]	-0	-0	0	
Fluid Temperature [K]	294.008	314.304	299.778	300.357
Solid Temperature [K]	296.155	305.591	300.171	300.765
Melting Temperature Exceed [K]	-1386.99	-1377.56	-1383.09	-1382.39

You can see that the average water temperature at the outlet is about 300 K.

Now let us determine the temperature of air at the outlet.

7 Click the **Environment Pressure - Air** item to select the inner face of the **Air Outlet Lid**.

8 At the bottom pane, click **Refresh** ✦.
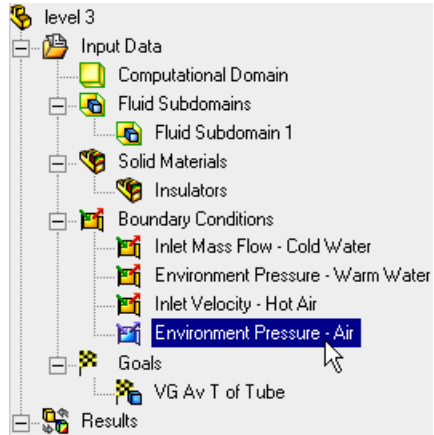
9 Look at the local parameters at the left side of the bottom pane.

You can see that the average air temperature at the outlet is about 585 K.

Local Parameter	Minimum	Maximum	Average	Bulk Average
Pressure [Pa]	101325	101325	101325	101325
Density [kg/m^3]	0.588229	0.667233	0.604387	0.602515
Velocity [m/s]	8.33569	10.1413	9.72103	9.7692
X — Component of Velocity [m/s]	-0.170256	0.164865	-0.0389458	-0.0381724
Y — Component of Velocity [m/s]	-0.164565	0.169146	0.00122779	0.00110737
Z — Component of Velocity [m/s]	-10.1412	-8.33412	-9.7206	-9.76881
Mach Number [ ]	0.0181421	0.0208413	0.0202105	0.0202842
Surface Heat Flux [W/m^2]	-0	-0	0	
Fluid Temperature [K]	528.939	599.981	584.611	586.315
Solid Temperature [K]	357.211	361.034	359.134	359.125
Melting Temperature Exceed [K]	-1325.94	-1322.12	-1324.02	-1324.02

10 The values of integral parameters are displayed at the right side of the bottom pane. You can see that the mass flow rate of air is 0.046 kg/s. This value is calculated with the

Integral Parameter	Value	X-component	Y-component	Z-component
Heat Transfer Rate [W]	0			
Mass Flow Rate [kg/s]	-0.0460101			
Volume Flow Rate [m^3/s]	-0.076427			
Surface Area [m^2]	0.0106466	-0.000216528	-9.24096e-006	0.0100263
Total Enthalpy Rate [W]	-27508.6			
Uniformity Index [ ]	1.95368			
CAD Fluid Area [m^2]	0.00787803			

Consider entire model option selected, i.e. taking into account the **Symmetry** condition.

11 Click **OK** ✓ to close the dialog box.

Calculating the Heat Exchanger Efficiency

The heat exchanger efficiency can be easily calculated, but first we must determine the fluid with the minimum capacity rate (C=$\dot{m}c$). In this example the water mass flow rate is 0.02 kg/s and the air mass flow rate is 0.046 kg/s. The specific heat of water at the temperature of 300 K is about five times greater than that of air at the temperature of 585 K. Thus, the air capacity rate is less than the water capacity rate. Therefore, according to *Ref.2*, the heat exchanger efficiency is calculated as follows:

$$\varepsilon = \frac{T_{hot}^{inlet} - T_{hot}^{outlet}}{T_{hot}^{inlet} - T_{cold}^{inlet}},$$

where T_{hot}^{inlet} is the temperature of the air at the inlet, T_{hot}^{outlet} is the temperature of the air at the outlet and T_{cold}^{inlet} is the temperature of the water at the inlet.

We already know the air temperature at the inlet (600 K) and the water temperature at the inlet (293.2 K), so using the obtained values of water and air temperatures at outlets, we can calculate the heat exchanger efficiency:

$$\varepsilon = \frac{T_{hot}^{inlet} - T_{hot}^{outlet}}{T_{hot}^{inlet} - T_{cold}^{inlet}} = \frac{600 - 584}{600 - 293.2} = 0.052$$

Specifying the Parameter Display Range

1 In the temperature palette bar click the maximum value and type `300 K` in an edit box

2 Click ✅. Immediately the trajectories are updated.
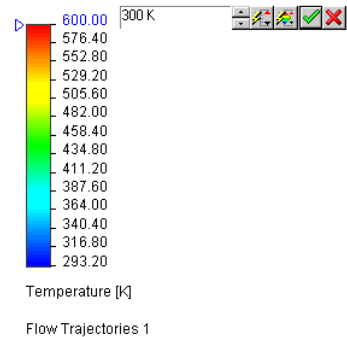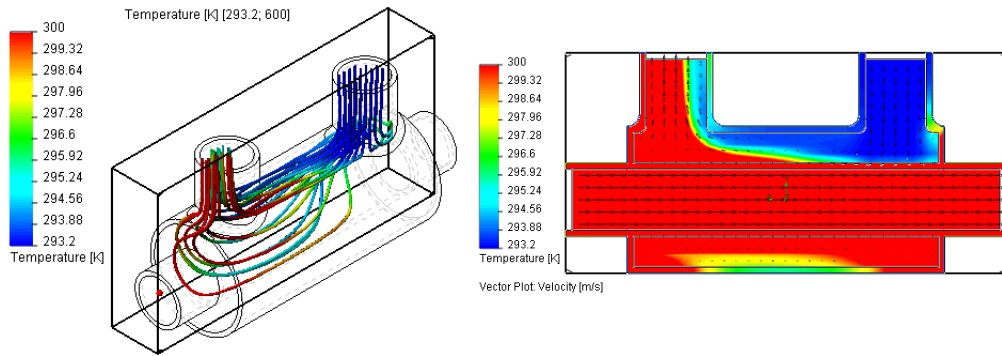
As you can see, Flow Simulation is a powerful tool for heat-exchanger design calculations.

Ref. 2 J.P. Holman. "Heat Transfer" Eighth edition.

7

Mesh Optimization

The goal of this tutorial example is to demonstrate various meshing capabilities of Flow Simulation allowing you to better adjust the computational mesh to the problem at hand. Although the automatically generated mesh is usually appropriate, intricate problems with thin and/or small, but important, geometrical and physical features can result in extremely high number of cells, for which the computer memory is too small. In such cases we recommend that you try the Flow Simulation options allowing you to manually adjust the computational mesh to the solved problem's features to resolve them better. This tutorial teaches you how to do this.

The **Ejector in Exhaust Hood** example aims to:

- Settle the large aspect ratio between the minimum gap size and the model size by adjusting the initial mesh manually.
- Resolve small features by specifying local mesh settings.

Problem Statement

The ejector model is shown on the picture. Note that the ejector orifice's diameter is more than 1000 times smaller than the characteristic model size determined as the computational domain's overall dimension.

Exhaust

Baffles

Ejector

Opening

5,79 ft

Ejected chlorine orifice

∅ 0.0044 ft
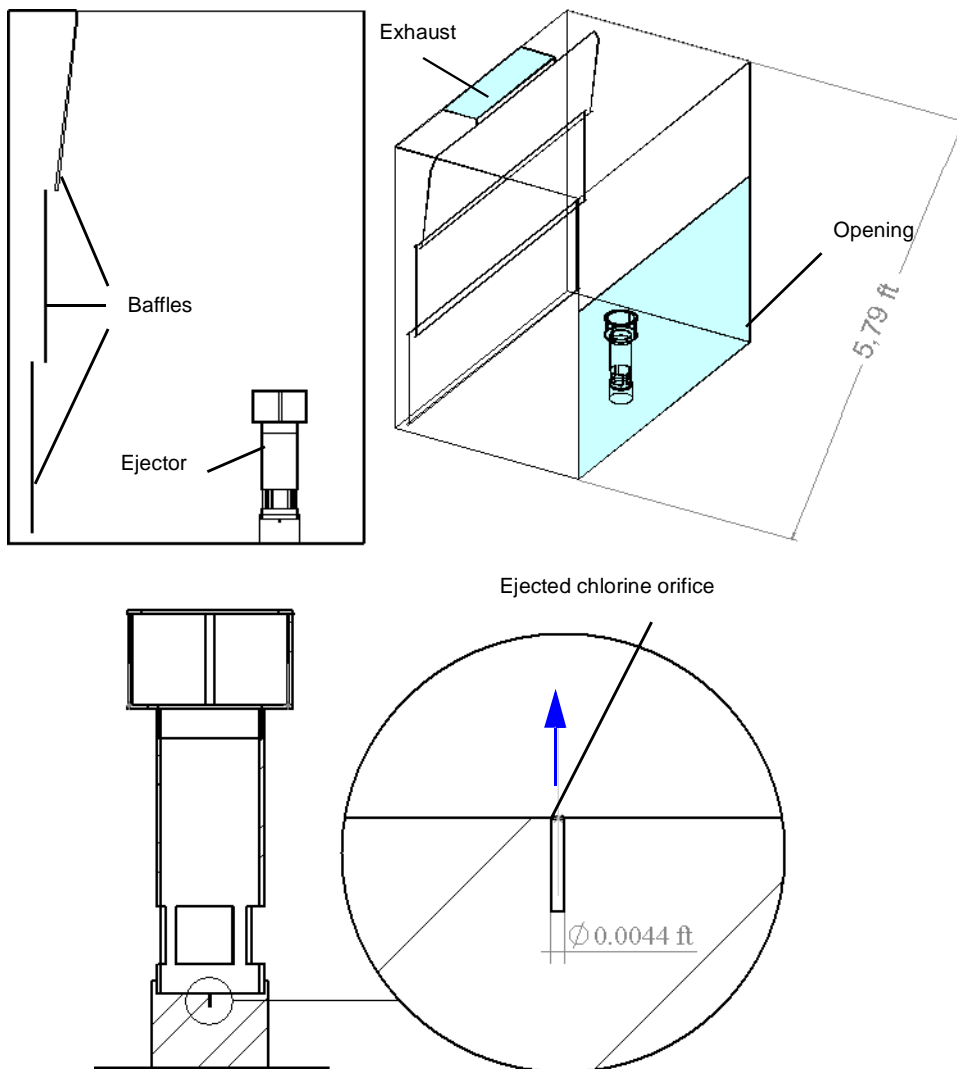
SolidWorks Model Configuration

Copy the **Tutorial 4 – Mesh Optimization** folder into your working directory and ensure that the files are not read-only since Flow Simulation will save input data to these files. Open the **Ejector in Exhaust Hood.SLDASM** assembly.

Project Definition

Using the Wizard create a new project as follows:

Project Configuration	*Use current*
Unit system	*USA*
Analysis type	*Internal; Exclude cavities without flow conditions*
Physical features	*Gravity; Default gravity (Y component: -32.1850394 ft/s^2)*
Fluids substances	*Air, Chlorine*
Wall Conditions	*Adiabatic wall, default smooth walls*
Initial Conditions	Initial gas concentration: *Air – 1, Chlorine - 0*
Result and Geometry Resolution	*Default result resolution level 3; Default geometry resolution: automatic minimum gap size and minimum wall thickness, other options by default*

  📖 *When you enable gravitation, pay attention that the hydrostatic pressure is calculated with respect to the global coordinate system, as follows:*
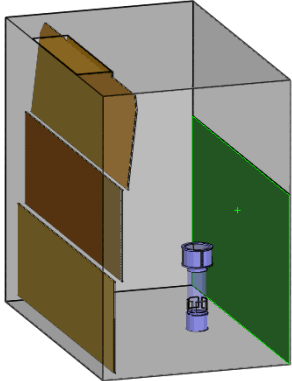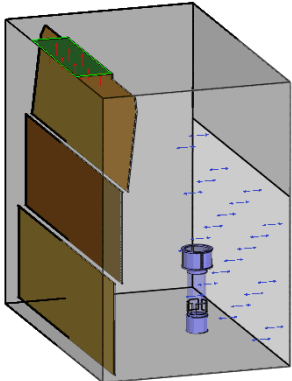
$P_{hydrostatic} = \rho(g_x{*}x + g_y{*}y + g_z{*}z)$, *where ρ – reference density, g_i - component of the gravitational acceleration vector and x, y, z - coordinates in the global coordinate system.*

Conditions

At first, let us specify all the necessary boundary conditions because they influence the automatic initial mesh through the automatic minimum gap size, which depends on the characteristic size of the faces on which the boundary conditions are set.
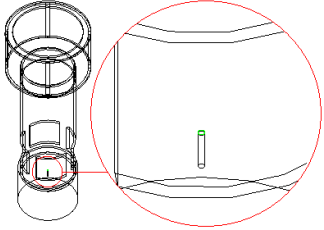
📕 *Flow Simulation calculates the default minimum gap size using information about the faces where boundary conditions (as well as sources, fans) and goals are specified. Thus, it is recommended to set all conditions before you start to analyze the mesh.*

The first two boundary conditions are imposed on the exhaust hood's inlet and outlet.

Inlet Boundary Condition	***Environment Pressure:*** *Default values (14.6959 lbf/in^2, gas substance – Air) of the* **Environment pressure** *and* **Temperature** *(68.09 °F) at the box's* **Lid for Face Opening***;*	
Outlet Boundary Condition	***Outlet Volume Flow:*** *Outlet volume flow rate of 1000 ft^3/min at the box's* **Exhaust Lid***.*	

If you open the **Initial Mesh** dialog box (click **Flow Simulation**, **Initial Mesh**) and select the **Manual specification of the minimum gap size** option, you can see that the current automatic minimum gap size is 0.5 ft, which is the width of the outlet opening (if you have opened the **Initial Mesh** dialog box, click **Cancel** to discard changes).

The next inlet volume flow rate condition defines the gas ejected from the bottom of the **Ejector** component.

Inlet Boundary Condition	***Inlet Volume Flow:*** *Inlet chlorine (**Substance concentrations**: Chlorine – 1; Air – 0) volume flow rate of 0.14 ft³/min at the lid that closes the orifice (make sure that you have selected the upper face of the lid).*	
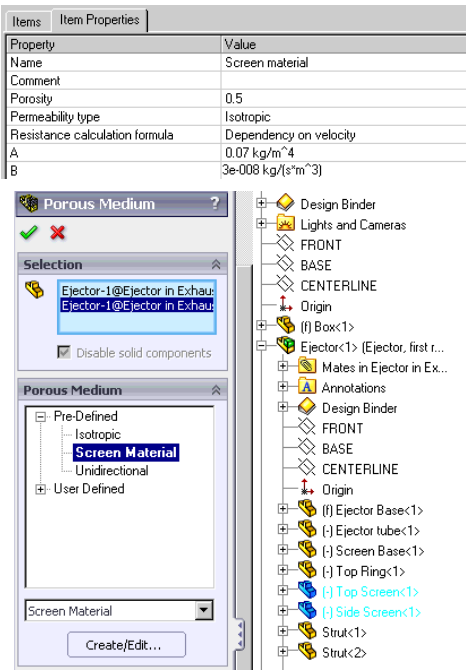
If you now look at the automatic minimum gap size value (click **Flow Simulation**, **Initial Mesh, Manual specification of the minimum gap size**), you can see that it is close to the orifice diameter - 0.0044528 ft.

   📖 *The Minimum gap size is a parameter governing the computational mesh, so that a certain number of cells per the specified gap should be generated. To satisfy this condition the corresponding parameters governing the mesh are set by Flow Simulation (number of basic mesh cells, small solid features refinement level, narrow channel resolution, etc.). Note that these parameters are applied to the whole computational domain, resolving all its features of the same geometric characteristics (not only to a specific gap).*

Since the minimum gap size value influences the mesh in the entire computational domain, the large aspect ratio between the model and the minimum gap size value will produce a non-optimal mesh: not only will all small gaps be resolved, but there will also be many small cells in places where they are not necessary. As a result, an extremely large mesh will be produced, which may result in overly large computer memory requirements exceeding the computers' available resources. Moreover, if the aspect ratio between the model and the minimum gap size is more than 1000, Flow Simulation may not adequately resolve such models with the automatically generated mesh anyway.

Finally, let us create the ejector's porous media and apply it to the ejector's top and side screens.

The material you are going to create is already defined in the Engineering Database under the Pre-Defined folder. You can skip the definition of the porous material, then when creating the porous condition, select the pre-defined "Screen Material" from the Engineering database.

Porous Media	*Screen material:* *Porosity: 0.5,* *Permeability type:* *Isotropic,* *Dependency on velocity:* $A = 0.07 \, kg/m^4$, $B = 3e\text{-}008 \, kg/(s*m^3)$. *Components to apply*: *Top Screen* *Side Screen*	

To see advantages of the local mesh and refinement options better, now let us try to generate the computational mesh governed by the automatic mesh settings. The resulting mesh consists of more than 1100000 cells, and cannot be processed by old computers due to the computer memory restriction (you may get a warning message about insufficient memory)

Manual Specification of the Minimum Gap Size

We can distinguish two very different parts of the model: a relatively big cavity having several thin walls within and no small solid features, and the ejector's region containing some very fine geometrical features. Therefore, the mesh required to properly resolve the ejector and the mesh appropriate for the rest of the model should be also very different. Since the ejector region is a part of the entire computational domain, we need to specify such settings for the automatic mesh generation that the model's geometry outside the ejector's region will be resolved without excessive mesh splitting.

The minimum gap size value, automatically defined from the dimensions of the ejector's **Top Screen** and **Side Screen** components, is too small and results in excessive mesh splitting.

To define an appropriate minimum gap size we need to examine all narrow flow passages outside the ejector's region:

- Boundary conditions;
- The passages connecting the ejector's internal volume with the model's cavity;
- The narrow flow passages between the baffles.

After reviewing the model we can accept the width of the gap between the middle and upper baffles as the minimum gap size. To avoid excessive mesh splitting, we will specify the same value for the minimum wall thickness.

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

2 Use the slider to set the **Level of the initial mesh** to **5**.

3 Select the **Manual specification of the minimum gap size** checkbox and enter `0.067 ft` in the **Minimum gap size** box.

4 Select the **Manual specification of the minimum wall thickness** checkbox and enter `0.067 ft` in the **Minimum wall thickness** box.

0.067 ft
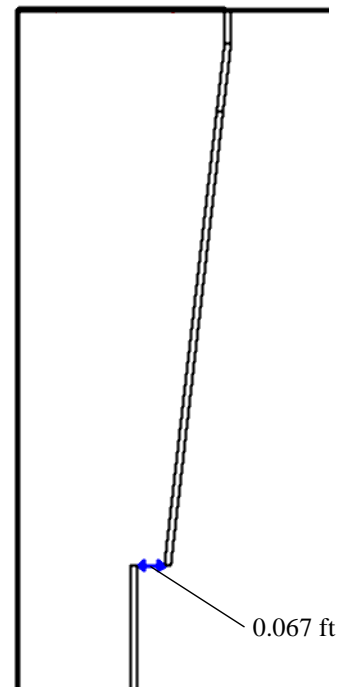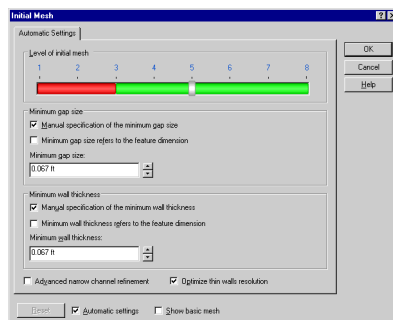
5   Click **OK**.

To see the resulting mesh:

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

2   Clear the **Solve** check box in order to generate the mesh only.

3   Click **Run**.

After the mesh generation finishes you can obtain the resulting mesh by creating a **Cut Plot** based on the CENTERLINE with the **Mesh** option selected.
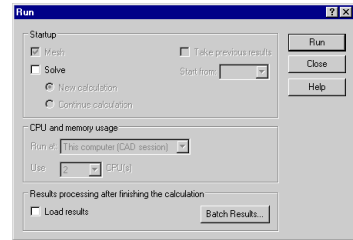
The resulting mesh has significantly less cells than the mesh generated automatically with the default values of **Minimum gap size** and **Minimum wall thickness**. The total number of cells is less than 200 000.
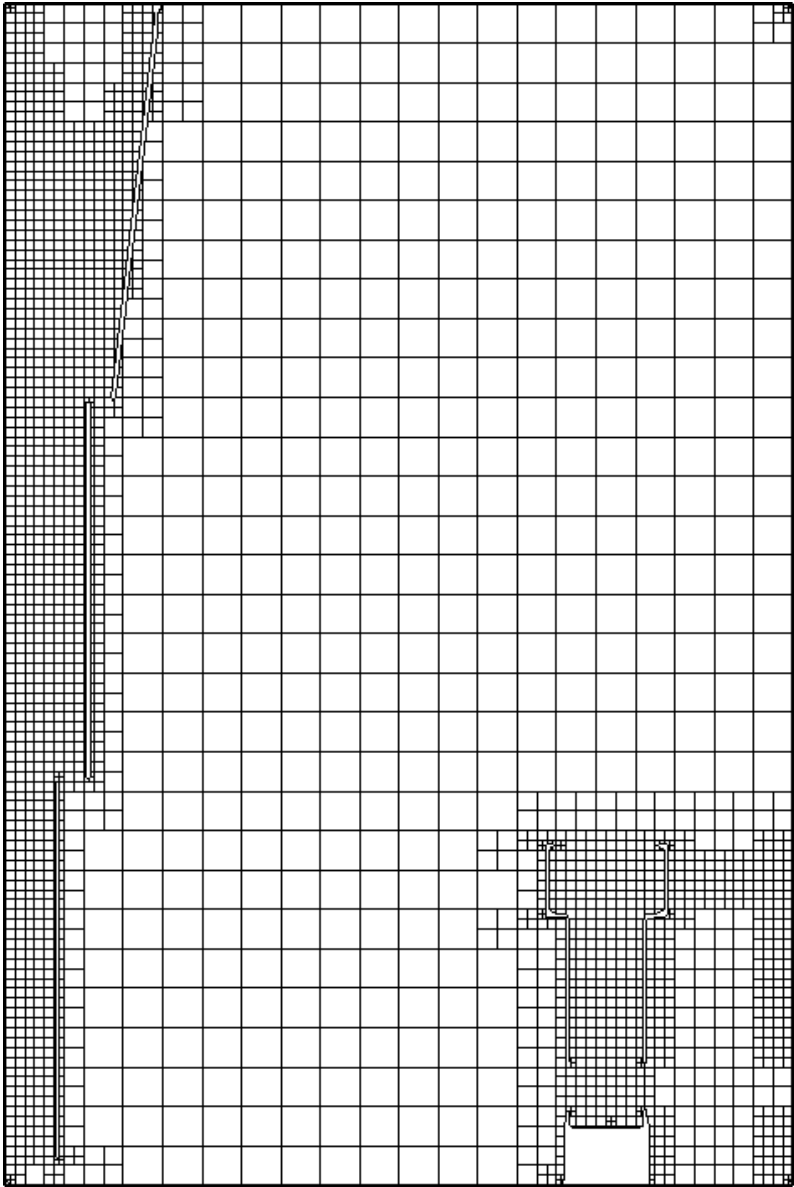
Switching off the Automatic Mesh Definition

We have successfully reduced the number of cells, yet using the mesh of the higher level. The higher level mesh provides better refinement in the regions with small geometrical features. However, we actually do not need such a fine mesh in some regions where the flow field changes slowly. We can further decrease the number of cells by switching off the automatic definition of the mesh generation settings and adjusting these settings manually. The decreased number of cells will provide us a computer memory reserve needed to better resolve fine geometrical features of the ejector.

Click **Flow Simulation**, **Project**, **Rebuild**.

1   Click **Flow Simulation**, **Initial Mesh**. Switch off the automatic mesh settings by clearing the **Automatic settings** check box. The **Initial Mesh** dialog box controls the basic mesh and the initial mesh within the entire computational domain unless local initial mesh settings are specified.

□ *The mesh is named **Initial** since it is the mesh the calculation starts from and it could be further refined during the calculation if the solution-adaptive meshing is enabled. The initial mesh is constructed from the **Basic** mesh by refining the basic mesh cells in accordance with the specified mesh settings. The Basic mesh is formed by dividing the computational domain into slices by parallel planes which are orthogonal to the Global Coordinate System's axes.*

The Initial Mesh's parameters are currently set by Flow Simulation in accordance with the previously specified automatic mesh settings, including **Minimum gap size** and **Minimum wall thickness**.

2   Go to the **Narrow channel** tab and set the **Narrow channels refinement level** to 1. This allows us to reduce the number of cells in the channels between the baffles and the wall of the **Box**.

□ *The **Narrow channels refinement level** specifies the smallest size of the cells in model's flow passages with respect to the basic mesh cells. So if N = 0...7 is the specified **Narrow channels refinement level**, the minimum size of the cells obtained due to the mesh refinement is 2^N times smaller (in each direction of the Global Coordinate System, or 8^N times by volume) than the basic mesh cell's size.*
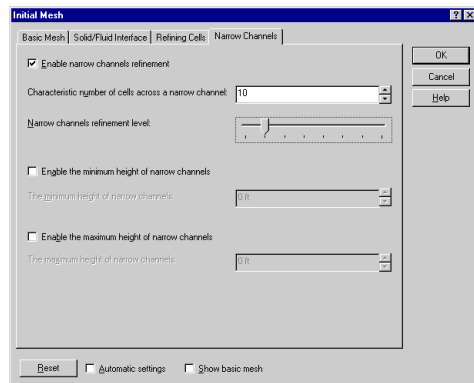
To see the resulting mesh create the mesh again (without the following calculation).

The resulting mesh is shown below. It has about 75 000 cells.
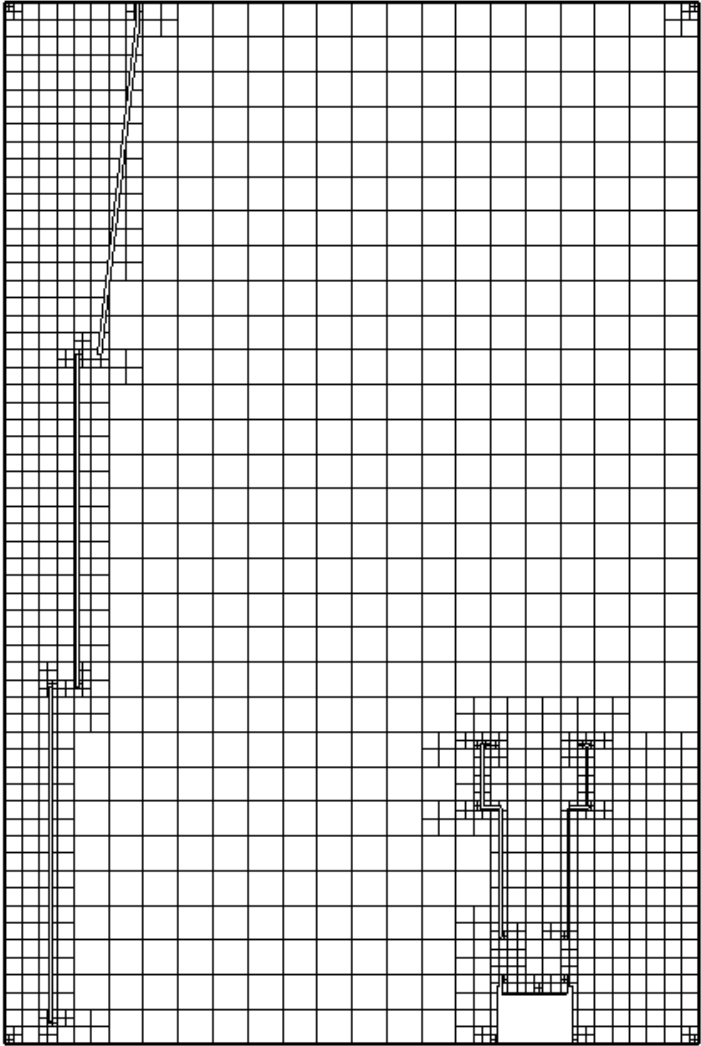
Using the Local Initial Mesh Option

The ejector's geometry is resolved reasonably well. However, if you generate the mesh and zoom in to the ejector's orifice region, you will see that the gas inlet face is still unresolved. The resolution of the boundary condition surface is very important for correctly imposing the boundary condition. To resolve the gas inlet face properly we will use the **Local Initial Mesh** option.
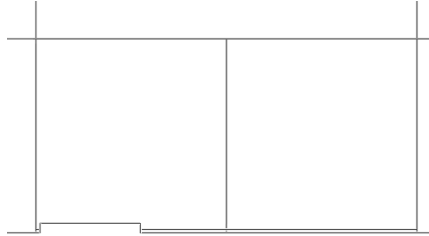
The local initial mesh option allows you to specify an initial mesh in a local region of the computational domain to better resolve the model geometry and/or flow peculiarities in this region. The local region can be defined by a component of the assembly, disabled in the Component Control dialog box, or specified by selecting a face, edge or vertex of the model. Local mesh settings are applied to all cells intersected by a component, face, edge, or a cell enclosing the selected vertex.

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

2   Select the inlet face of the ejector's orifice or click the **Inlet Volume Flow 1** boundary condition in the Flow Simulation Analysis tree to select the face on which this boundary condition is applied.
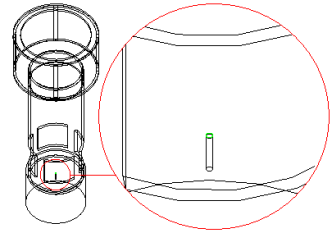
3   Clear the **Automatic settings** check box and switch to the **Refining cells** tab.

4   Select the **Refine all cells** checkbox and use the slider to set the **Level of refining all cells** to its maximum value of **7**.

5   Click **OK**.
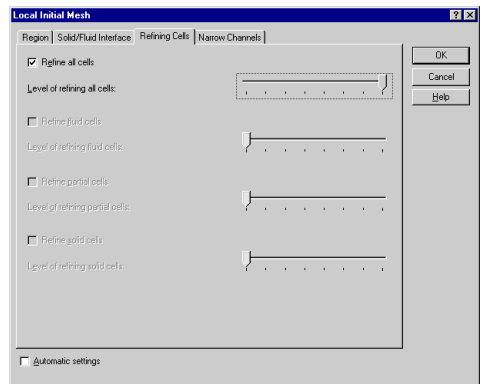
To see the resulting mesh create the mesh again (without the following calculation).

Now we have specified to refine all cells at the ejector's orifice inlet face up to the maximum level. The locally refined mesh is shown below.
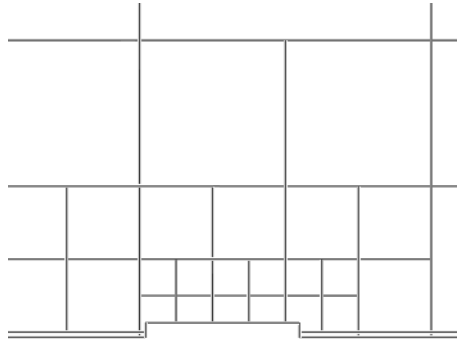
Specifying Control Planes

The basic mesh in many respects governs the generated computational mesh. The proper basic mesh is necessary for the most optimal mesh.

You can control the basic mesh in several ways:

- Change number of the basic mesh cells along the X, Y, Z-axes.
- Shift or insert basic mesh planes.
- Stretch or contract the basic mesh cells locally by changing the relative distance between the basic mesh planes.

    📖 *The local mesh settings do not influence the basic mesh but are basic mesh sensitive: all refinement levels are set with respect to the basic mesh cell.*

You may notice that the mesh resolving the ejector's orifice inlet face is not symmetric. It can has a negative effect on the specified boundary condition. We will add a control plane to shift the boundary between cells so that it will pass through the center of the inlet face.

1 In the **Initial Mesh** dialog box, go to the **Basic Mesh** tab.

2 Click **Add Plane**. The **Create Control Planes** dialog box appears.

3 In the **Creating mode** list select **Reference geometry**.

4 Under **Parallel to** select **XY**.

5 Zoom in to the ejector's orifice area and select edge of the inlet face in the graphics area. The control plane will pass through the middle of the edge parallel to the Global Coordinate System plane selected in the **Parallel to** group. Please check that the value of offset along the Z axis, appeared in the **Control planes** list, is equal to 0.703125 ft. If not, it means that you have mistakenly selected another geometry feature. In this case, right-click on the
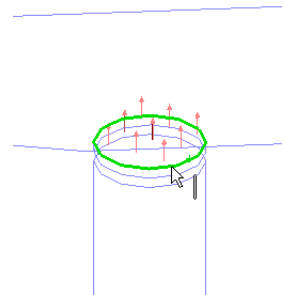
Control planes list and select **Delete All**, then try to select the edge of the inlet face again.

6   Click **OK**. The **Z2** control plane appears in the **Control intervals** table.

📖 *You can visualize the basic mesh before solving the problem. To see the basic mesh, click **Show basic mesh** in the **Initial Mesh** dialog box or click **Flow Simulation**, **Project**, **Show Basic Mesh**.*

7   Click **OK** to save changes and close the **Initial Mesh** dialog box.

Then, generate the initial mesh to check whether the thin walls and the other geometry are resolved.

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

2   Clear the **Solve** check box in order to generate the mesh only.

3   Clear the **Load results** check box.

4   Click **Run**.
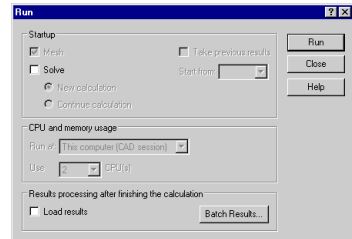
Prior to visualizing the initial computational mesh, let us switch the Flow Simulation option to use the meshed geometry instead of the SW model's geometry to visualize the results.

By default, Flow Simulation shows the SolidWorks model's geometry when displaying the results. Depending on how exactly the model has been resolved with the computational mesh, the SolidWorks model's geometry may differ from the geometry used in the calculation. To display the real captured geometry the **Use CAD geometry** option is reserved.

5   Click **Tools**, **Options**, then click **Third Party**.

6   On the **Flow Simulation Options** tab, under **General Options**, select the **Display mesh** check box.

7   Under **View Options** clear the **Use CAD geometry (Default)** check box.

8   Click **OK**.

Next load the file with the initial computational mesh: right-click the **Results** icon and select **Load Results**, then select the **1.cpt** file and click **Open**. Note that the total number of cells is about 75 000.

The calculation results, including the current computational mesh, are saved in the .fld files, whereas the initial computational mesh is saved separately in the .cpt files. Both of the files are saved in the project folder, whose numerical name is formed by Flow Simulation and must not be changed.

Create a cut plot based on the CENTERLINE with the **Mesh** option selected. Create a second cut plot based on the ejector's orifice inlet face with the **Offset** of -0.00025 ft relative to the selected face and the same settings as the first cut plot.

Now you can see that the generated mesh is symmetrical relative to the center of the inlet face.

Creating a Second Local Initial Mesh

With the specified mesh settings the ejector's geometry will be resolved properly. But we need to create the mesh successfully resolving not only fine geometrical features, but the small flow peculiarities as well. In the **Ejector Analysis** project such peculiarities can be found within the internal volume of the ejector, where the thin stream of chlorine is injected from the ejector's orifice. Therefore the mesh within the ejector's region must be split additionally. To refine the mesh only in this region and avoid excessive splitting of the mesh cells in other parts of the model, we apply a local initial mesh at the component surrounding this region. The component was created specially to specify the local initial mesh.

Set to resolved the **LocalMesh2** component. Click **Close** after Flow Simulation shows you a warning message. Note that this component was created so that there is a small distance between the boundaries of the component and the solid feature of interest (i.e., the ejector). Because the local settings are applied only to the cells whose centers lie within the selected model component, it is recommended to have the component's boundaries offset from the solid component's walls.

After resolving the **LocalMesh2** component an error message appears informing you that the inlet volume flow condition is not in contact with the fluid region. The problem disappears after disabling the component in the **Component Control** dialog box to treat it as a fluid region.

Click **Flow Simulation**, **Component Control** and disable the **LocalMesh2** component. Click **OK**.

Rebuild the project by clicking **Flow Simulation**, **Project**, **Rebuild**.

📖 *You can also disable components directly from the* **Local Initial Mesh** *dialog box by selecting the* **Disable solid components** *option on the* **Region** *tab.*
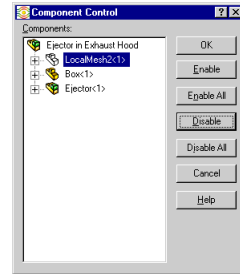
Next specify the local mesh settings for the ejector's region.

1   Select the **LocalMesh2** component.

2   Click **Flow Simulation**, **Insert**, **Local Initial Mesh**.

3   Clear the **Automatic settings** check box and switch to the **Narrow Channels** tab.

4   Specify the **Characteristic number of cells across a narrow channel** equal to 15.

5   Use the slider to set the **Narrow channels refinement level** to **3**.

6   Click **OK**.
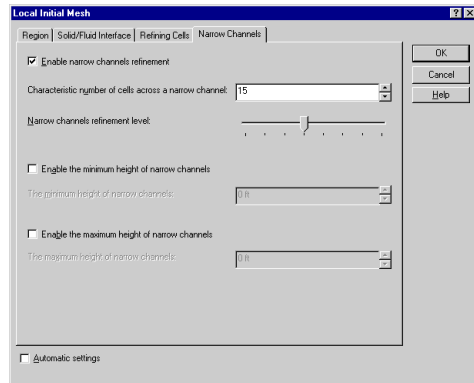
📖 *The settings on the **Narrow Channels** tab controls the mesh refinement in the model's flow passages. **Characteristic number of cells across a narrow channel** box specify the number of initial mesh cells (including partial cells) that* Flow Simulation *will try to set across the model's flow passages in the direction normal to solid/fluid interface . If possible, the number of cells across narrow channels will be equal to the specified characteristic number, otherwise it will be close to the characteristic number. If this condition is not satisfied, the cells lying in this direction will be split to satisfy the condition.*

Rebuild the project. Create the mesh again (without the following calculation) and load the **1.cpt** file.

Click **Flow Simulation**, **Results**, **Display, Geometry** to hide the model.

Finally, let us compare how the final mesh resolves the solid geometry and the fluid region within the ejector with only about 100 000 cells in contrast with 1 100 000 cells generated by the automatic mesh settings.

8
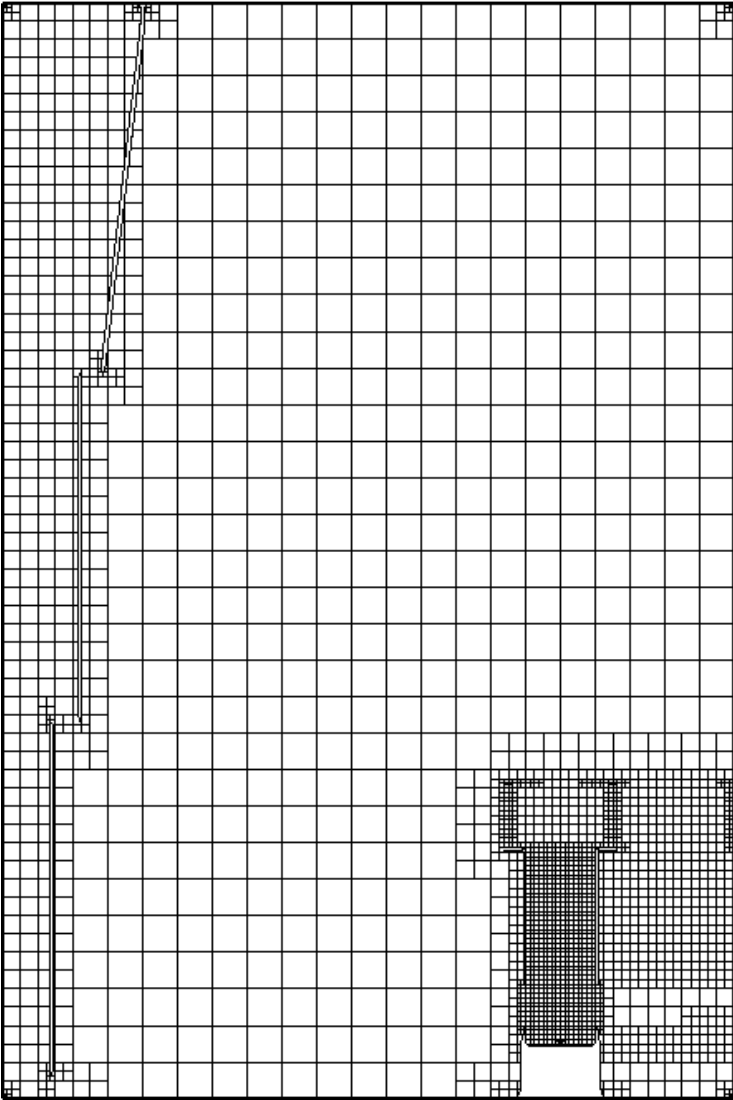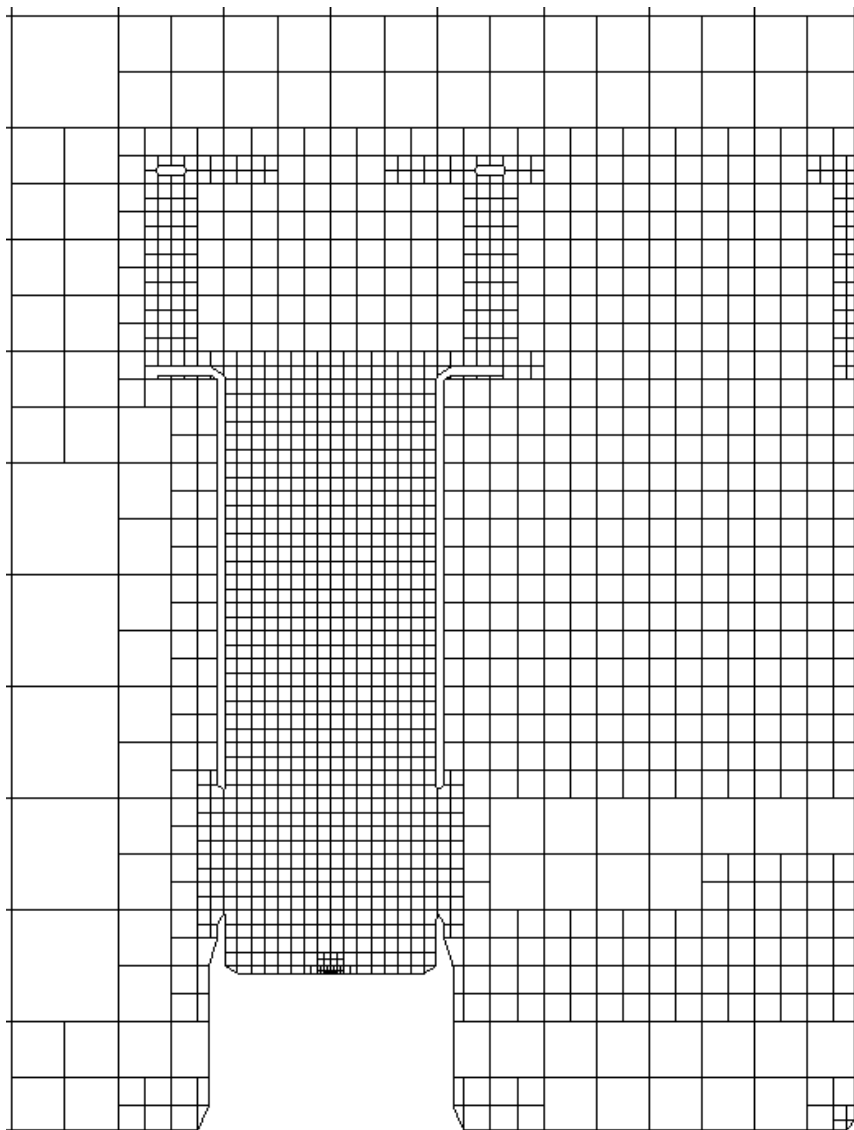
Application of EFD Zooming

Problem Statement

The Flow Simulation PE capability of EFD Zooming is demonstrated as an engineering tutorial[1] example of selecting a better heat sink shape for a main chip taking into account other electronic components in an electronic enclosure.

The assembly model of the electronic enclosure including the main chip's heat sink under consideration is shown in picture. The fan installed at the enclosure inlet blows air through the enclosure to the outlet slots with the goal of cooling the heated electronic elements (having heat sources inside). The planar main chip is attached to a motherboard made of an insulator. To cool the main chip better, its opposite plane surface is covered by a heat sink cooled by the air stream from the fan.

1.This example can be run in Flow Simulation PE only.
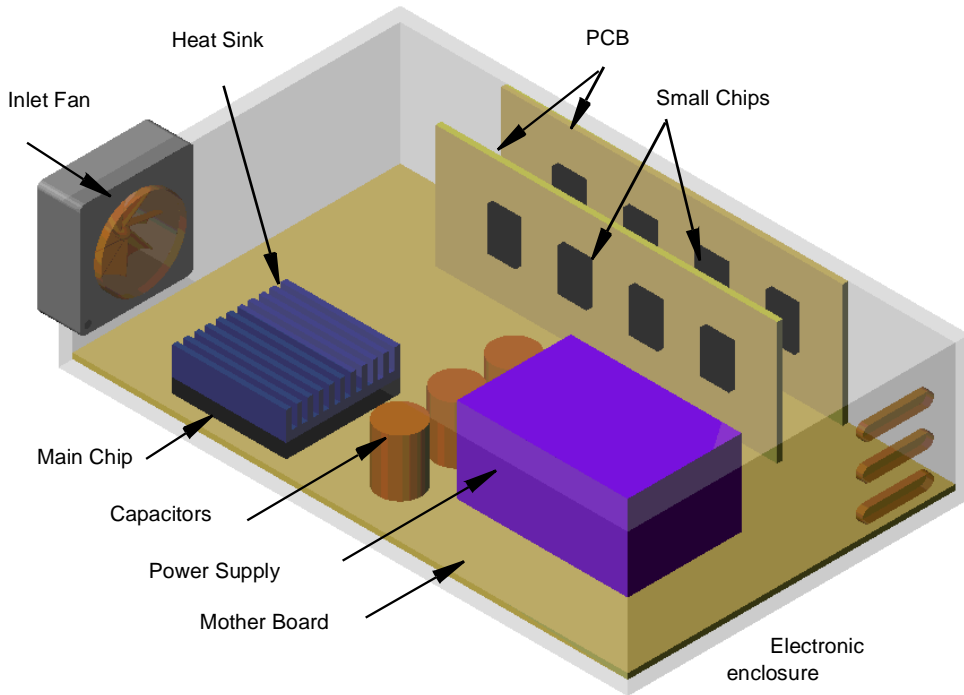
The problem's engineering aim is to determine the temperature of the main chip when using one of two heat sink designs. All other conditions within the enclosure will be invariable. As a result, we will find out the difference in cooling capability between these two competing shapes.
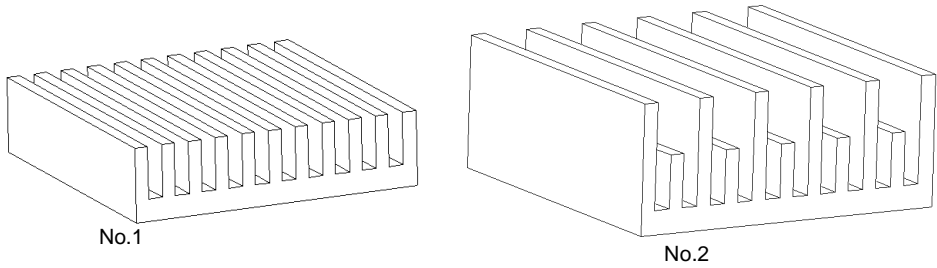
The heat sink's competing shapes (No.1 and No.2)

As you can see, all components within the electronic enclosure except the main chip's heat sink are specified as coarse shapes without small details, since they do not influence the main chip's temperature which is the aim of the analysis (the enclosure model was preliminary simplified to this level on purpose). On the contrary, the heat sink of each shape is featured by multiple thin (thickness of 0.1 in) fins with narrow (gaps of 0.1 in) channels between them.

Two Ways of Solving the Problem with Flow Simulation

Flow Simulation allows us to simplify the solution of this problem. Two possible techniques are listed below.

In the first and more direct way, we compute the entire flow inside the whole electronic enclosure for each heat sink shape with using the **Local Initial Mesh** option for constructing a fine computational mesh in the heat sink's narrow channels and thin fins. Naturally, the **Heat conduction in solids** option is enabled in these computations.

In the other, two-stage way (EFD Zooming using the **Transferred Boundary Condition** option), we solve the same problem in the following two stages:

1 computing the entire flow inside the whole electronic enclosure at a low result resolution level without resolving the heat sink's fine features (so, the parallelepiped envelope is specified instead of the heat sink's comb shape) and disabling the **Heat conduction in solids** option;

2 computing the flow over the real comb-shaped heat sink in a smaller computational domain surrounding the main chip, using the **Transferred Boundary Condition** option to take the first stage's computation results as boundary conditions, specifying a fine computational mesh in the heat sink's narrow channels and thin fins to resolve them, and enabling the **Heat conduction in solids** option.
The first stage's computation is performed once and then used for the second stage's computations performed for each of the heat sink's shapes.

The EFD Zooming Approach

Let us begin from the second (EFD Zooming) approach employing the **Transferred Boundary Condition** option. Then, to validate the results obtained with this approach, we will solve the problem in the first way by employing the **Local Initial Mesh** option.

First Stage of EFD Zooming

In accordance with the 1st stage of EFD Zooming aimed at computing the entire flow inside the electronic enclosure, it is not necessary to resolve the flow's small features, i.e. streams between the heat sink's fins, at this stage. Therefore, we suppress the heat sink's comb shape feature in the assembly model, obtaining the parallelepiped envelope instead.
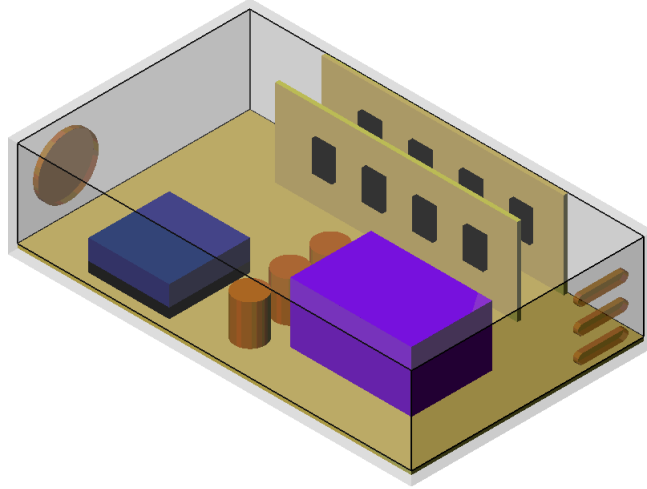
A parallelepiped heat sink is used at the 1st stage of EFD Zooming.

The model simplification at this stage allows us to compute the electronic enclosure's flow by employing the automatic initial mesh settings with a lower level of initial mesh (we use 4) and accepting the automatic settings for the minimum gap size and the minimum wall thickness. Moreover, at this stage it is also not necessary to compute heat conduction in solids, since we do not compute the main chip temperature at this stage. Instead, we specify surface heat sources of the same (5W) heat transfer rates at the main chip and heat sink (parallelepiped) faces and at the small chips' faces (they are heated also in this example) to simulate heating of the air flow by the electronic enclosure. This is not obligatory, but removing the heat conduction in solids at this stage saves computer resources. As a result, the computer resources (memory and CPU time) required at this stage are substantially reduced.

Project for the First Stage of EFD Zooming

SolidWorks Model Configuration

Click **File, Open**. In the **Open** dialog box, browse to the **Enclosure Assembly.SLDASM** assembly located in the **Tutorial PE1 - EFD Zooming** 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. Make sure that the **Zoom – Global - L4** configuration is the active one. Note that heat sink (HeatSink.SLD**PRT**) is the parallelepiped obtained by suppressing the heat sink's cuts.

Project Definition

Using the **Wizard** create a new project as follows:

Project name	*Use current: Zoom – Global - L4*
Unit system	*USA*
Analysis type	*Internal; Exclude cavities without flow conditions*
Physical features	*No physical features are selected*
Fluid	*Air*
Wall Conditions	*Adiabatic wall, Default smooth walls*
Initial Conditions	*Default conditions*
Result and Geometry Resolution	*Result resolution level set to 4, other options are default*

For this project we use the automatic initial mesh and the default computational domain.

  📖 *Note that **Level of initial mesh** is set to 4 in accordance with the **Result resolution level** specified in the Wizard. The **Result Resolution** defines two parameters in the created project, namely, the **Level of initial mesh** and the **Results resolution level**. The **Level of initial mesh** is accessible from the **Initial Mesh** dialog box and governs the initial mesh only. The **Results resolution level** is accessible from the **Calculation Control Options** dialog box and governs the*

*refinement of computational mesh during calculation and the calculation finishing conditions. The **Geometry Resolution** options, which also influence the initial mesh, can be changed in the **Initial Mesh** box, and/or their effects can be corrected in the **Initial Mesh** and **Local Initial Mesh** dialog boxes.*
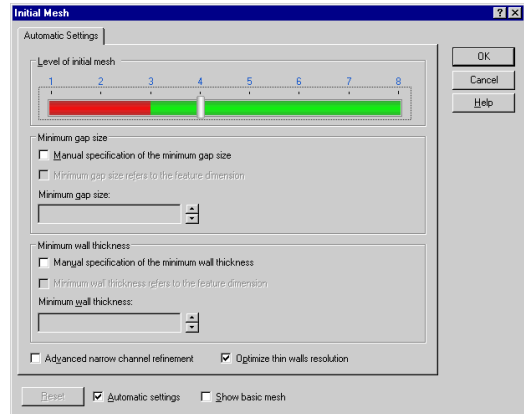
Unit System

After passing the Wizard, first we will adjust the system of units. The new custom system of units is based on the selected USA pre-defined system, but uses Watts for power, and inches for length.

1   Click **Flow Simulation**, **Units**.

2   Specify **Inch** for the **Length** and **Watt** for the **Total Heat flow & power**.

3   Click **Save**.
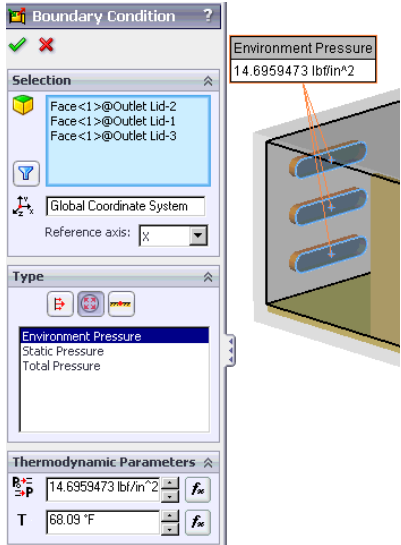
4   In the **Save to Database** dialog box, expand the **Units** group and select the **User Defined** item.

5   Name the new system of units `Electronics`.

6   Click **OK** to return to the **Unit System** dialog box.

7   Click **OK**.
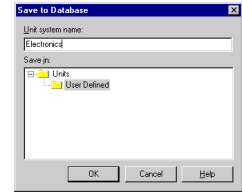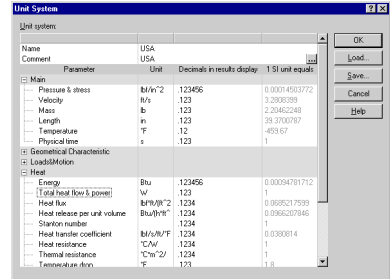
Conditions

We specify External Inlet Fan at the inlet, Environment Pressure at three outlets. For more detailed explanation of how to set these conditions please refer to the **First Steps - Conjugate Heat Transfer** tutorial.

Inlet Boundary Condition	*External Inlet Fan: Pre-Defined \Fan Curves\ PAPST\ DC-Axial\ Series 400\ 405\ 405 with default settings (ambient pressure of 14.6959 lbf/in², temperature of 68.09 °F) set at the **Inlet Lid**;*	

Outlet Boundary Condition	*Environment Pressure: Default thermodynamic parameters (ambient pressure of 14.6959 lbf/in^2, temperature of 68.09 °F) for the Environment pressure at the **Outlet Lids**.*	

Heat Sources

As mentioned earlier in this chapter, to simulate the flow heating by the electronic enclosure, we specify surface heat sources of the same (5W) heat transfer rates at the main chip and the heat sink (parallelepiped) faces and at the small chips' faces. Since we do not consider heat conduction in solids in this project, the surface source can be applied only to faces in contact with fluid. Follow the steps below to create the sources on the necessary faces:

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

In the Flyout FeatureManager Design Tree, select the **Heat Sink** and **Main Chip** components. Flow Simulation automatically selects all faces of the Heat Sink and Main Chip components. Faces that are not in contact with fluid must be removed from the **Faces to Apply the Surface Source** 🟡 list.

📖 Click **Filter Faces** 🔽 . Select **Keep outer and fluid-contacting faces**, and click **Filter**. *It is convenient to select all faces of the component by selecting this component in the Flyout FeatureManager Design Tree, though finding and removing unnecessary faces from the selection manually (one by one) may require excessive time, especially when there are many faces to remove. The **Filter** allows you to remove unnecessary faces of specified type from the list of selected faces.*
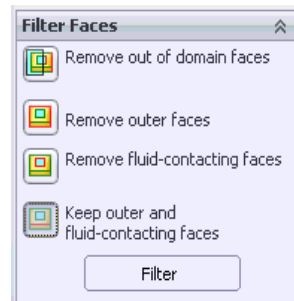
2 Under **Parameter**, set the **Heat Generation Rate** **Q** to 5 W.

   📖 *The specified heat source value (Heat Transfer Rate) is distributed among the selected faces in proportion to their areas.*

3 Click **OK** ✓ .
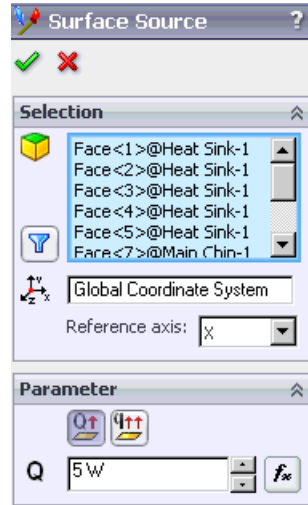
Following the same procedure, create a surface source of 5 W on the fluid-contacting surfaces of small chips.
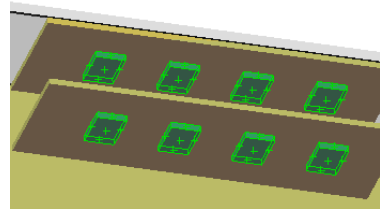
Goals

Specify the surface goals of mass flow rate at the inlet and outlet.

Run the calculation. After the calculation is finished you can start the second stage of EFD Zooming to focus on the main chip.

Save the model.

Second Stage of EFD Zooming

At the 2nd stage of EFD Zooming aimed at determining the main chip's temperature, we compute the flow over the heat sink in a smaller computational domain surrounding the main chip, using the **Transferred Boundary Condition** option to take the first stage's

computation results as boundary conditions. To compute the solids temperature, we enable the **Heat conduction in solids** option. Since at this stage the computational domain is reduced substantially, a fine computational mesh with an affordable number of cells can be constructed in the heat sink's narrow channels and thin fins, even when considering heat conduction in solids during computation.
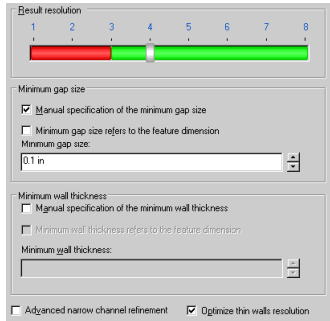
Project for the Second Stage of EFD Zooming

SolidWorks Model Configuration

Activate the **Zoom - SinkNo1 - L4** configuration. Note that heat sink's cuts are resolved now.

Project Definition

Using the **Wizard** create a new project as follows:

Project name	*Use current: Zoom - SinkNo1 - L4*
Unit system	*Electronics*
Analysis type	*Internal*
Physical features	*Heat conduction in solids is enabled*
Fluid	*Air*
Default solid	*Metals/Aluminum*
Wall Condition	*Default condition (Adiabatic); Default smooth walls (0 microinches)*
Initial Conditions	*Default initial conditions (in particular, the initial solid temperature is 68.09 °F)*
Result and Geometry Resolution	*Result resolution level set to 4; Minimum gap size = 0.1 in, automatic minimum wall thickness; other options are default.*

Here, we use the automatic initial mesh by specifying the **Result resolution** level (**Level of initial mesh**) of 4, but in contrast to the first stage's computation, we specify manually the minimum gap size of 0.1 in to resolve the fine features of heat sink.

Next, we will reduce the computational domain to focus on the main chip, i.e. perform EFD Zooming.

Computational Domain

When reducing the computational domain for EFD Zooming purposes, it is necessary to take into account that the first stage's computation results will serve as the boundary conditions at this domain's boundaries. Therefore, to obtain reliable results in the second stage's computations, we have to specify computational domain boundaries (as planes parallel to the X-, Y-, Z-planes of the Global Coordinate system) satisfying the following conditions:

1   the flow and solid parameters at these boundaries, taken from the first stage's computation, must be as uniform as possible;

2   the boundaries must not lie too close to the object of interest, since the object's features were not resolved at the first stage's computation. The computational domain must be large enough not to receive influence from more complex features of the newly added object;

3   the boundary conditions transferred to or specified at the boundaries must be consistent with the problem's statements (e.g., if in the problem under consideration the mother board is made of a heat-conducting material, then it is incorrect to cut the mother board with computational domain boundaries, since this will yield an incorrect heat flux from the chip through the mother board).

In this project we specify the following computational domain boundaries satisfying the above-mentioned requirements. Click **Flow Simulation**, **Computational Domain** to adjust the computational domain size as follows:

- X_{min} = -2.95 in (entirely lies inside the electronic enclosure side wall made of aluminum, this material does not influence the main chip's temperature since it is insulated from the chip by the heat-insulating mother board and the air flow, its boundary condition is automatically specified as the 68.09 °F temperature specified as the initial condition for all solids),

- X_{max} = 0.7 in (the boundary conditions in the fluid region of this boundary are transferred from the first stage's computation results, the same boundary conditions as at X_{min} = -2.95 in are automatically specified at this boundary's upper solid part lying in the electronic enclosure's aluminum wall, and the same boundary conditions as at Z_{min} = -1 in are automatically specified at the lower solid part lying in the mother board),

- Y_{min} = -1 in, Y_{max} = 4 in (the boundary conditions at these boundaries are specified in the same manner as at X_{max} = 0.7 in, as well as at the boundaries' side parts also lying in the aluminum wall),

- Z_{min} = -1.1 in (entirely lies inside the mother board specified as a heat insulator, therefore the adiabatic wall boundary condition is automatically specified at this boundary),

- $Z_{max} = 1.2$ in (entirely lies inside the electronic enclosure's aluminum upper wall, therefore the same boundary condition, as at Xmin = -2.95 in, are automatically specified at this boundary).
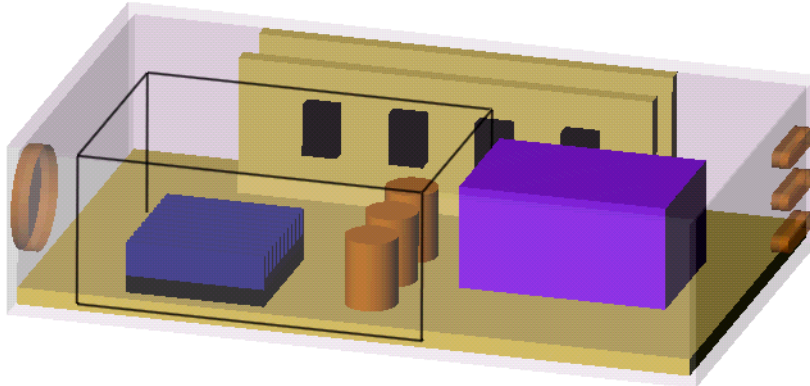
The reduced computational domain.

Conditions

First, we specify Transferred Boundary Conditions.

1 Click **Flow Simulation**, **Insert**, **Transferred Boundary Condition**.

2 Add the **Xmax**, **Ymax** and **Ymin Computational Domain boundaries** to the **Boundaries to apply the transferred boundary condition** list. To add a boundary, select it and click **Add**, or double-click a boundary.

3 Click **Next**.

4 At **Step 2**, click **Browse** to select the Flow Simulation project whose results will be used as boundary conditions for the current **Zoom – SinkNo1 - L4** project.

📖 *You can select a calculated project of any currently open model, or browse for the results (.fld) file.*
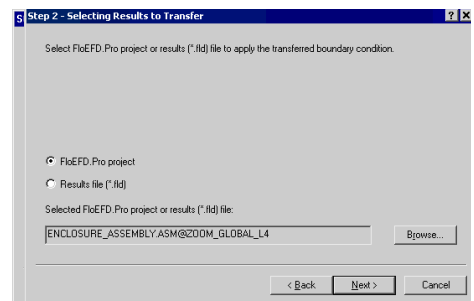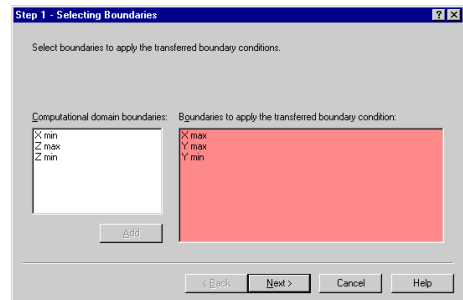
5 In the **Browse for Project** dialog select the **Zoom – Global - L4** configuration and click **OK**.

6 Click **Next**.

7 At **Step 3**, accept **Ambient** as the **Boundary condition type**.

📖 *The Ambient boundary condition consists of specifying (by taking results of a previous calculation) flow parameters at the boundary's section lying in the fluid, so they will act during the calculation in nearly the same manner as ambient conditions in an external analysis. If Heat Conduction in Solids is enabled, then the solid temperature is specified at this boundary's section lying in the solid (by taking results of a previous calculation).*
The heat flux at this boundary, which will be obtained as part of the problem solution, can be non-zero.
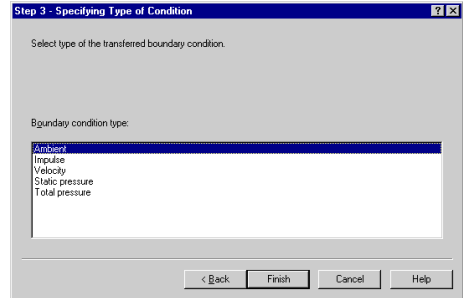
8 Click **Finish**.

Specify the other conditions as follows:

Heat Sources

Volume Source of 5W heat generation rate in the main chip.
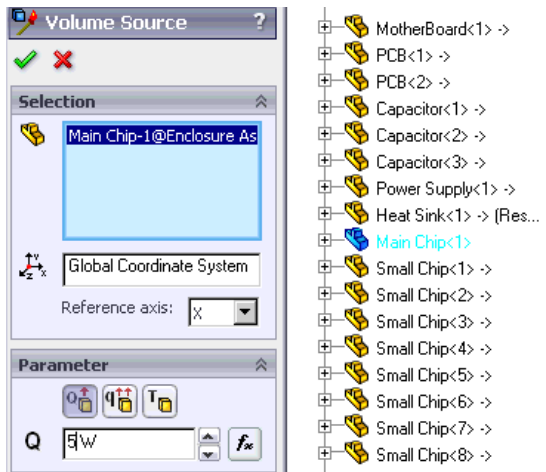
Solid Materials

a) **Main Chip** is made of silicon
(Pre-Defined/Semiconductors);
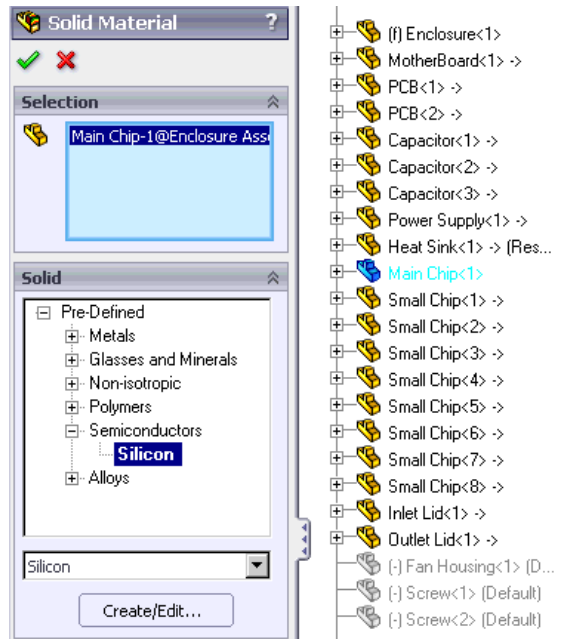
b) **MotherBoard** and **Enclosure** are
made of insulator (Pre-Defined/Glasses & Minerals);
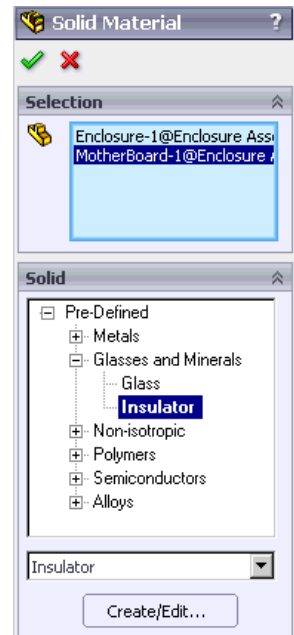
c) all other parts (e.g. the heat sink) are made of aluminum.

Goals

Specify the Volume Goals of maximum and average temperatures of the main chip and the heat sink.

Run the calculation.
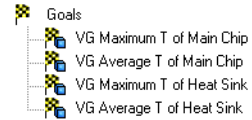
The obtained computational results are presented in tables and pictures below. These results were obtained with the heat sink's shape N.1.

If you look at the computational mesh you can see that it has two cells for each of the heat sink's channels, and two cells for each of the sink's fins.
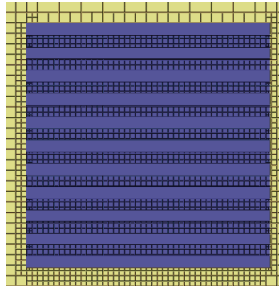
The mesh cut plot obtained for the heat sink No.1 at *Y=-0.3 in*.

📖 *In fact, the **Minimum gap size** and **Minimum wall thickness** influence the same parameter, namely, the characteristic cell size. By default, Flow Simulation generates the basic mesh in order to have a minimum of two cells per the specified **Minimum gap size**. The number of cells per the **Minimum gap size** depends non-linearly on the **Level of initial mesh** and cannot be less than two. In turn, the **Minimum wall thickness** condition induces Flow Simulation to create the basic mesh having two cells (two cells are enough to resolve a wall) per the specified **Minimum wall thickness** (regardless of the specified initial mesh level). That's why, if the **Minimum wall thickness** is equal to or greater than the **Minimum gap size**, then the former does not influence the resulting mesh at all.*

Changing the Heat Sink

Let us now see how employing the heat sink's shape No. 2 changes the computational results. To do this, we change the heat sink configuration to the No.2 version, whereas all the EFD Zooming Flow Simulation project settings of 2nd stage are retained. There is no need to perform the EFD Zooming computation of 1st stage again, as we may use its results in this project too.

The easiest way to create the same Flow Simulation project for the new model configuration is to clone the existing project to this configuration.

Clone Project to the Existing Configuration

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

2 Click **Add to existing**.

3 In the **Existing configuration** list select **Zoom - SinkNo2 - L4**.

Click **OK**. After clicking OK, two warning messages appear asking you to reset the computational domain and to rebuild the computational mesh. Select **No** to ignore the resizing of computational domain, and **Yes** to rebuild the mesh.
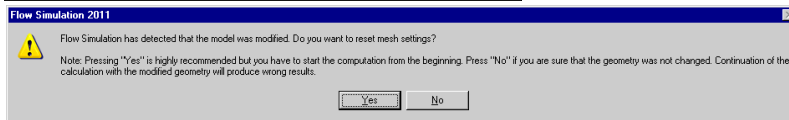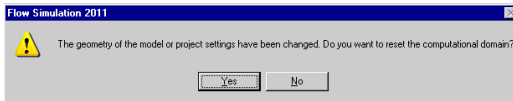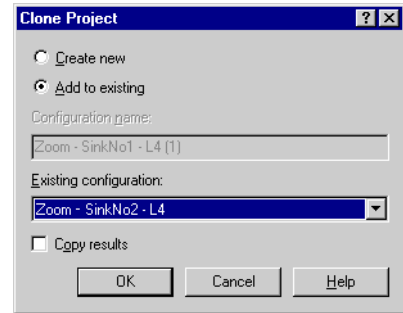
After cloning the project you can start the calculation immediately.

The obtained results are presented in tables and pictures below. It is seen that due to the new shape of the heat sink the main chip's temperature is reduced by about 15 °F. That is caused by both the increased area of the heat sink's ribs and streamlining the flow in the heat sink's narrow channels between the ribs (in heat sink No.1 about half of the channel is occupied by a counterflow vortex).

The Local Initial Mesh Approach

To validate the results obtained with the EFD Zooming approach, let us now solve the same problems employing the Local Initial Mesh option. To employ this option, we add a parallelepiped surrounding the main chip to the model assembly and then disable it in the **Component Control** dialog box. This volume represents a fluid region in which we can specify computational mesh settings differing from those in the other computational domain, using the **Local Initial Mesh** option.
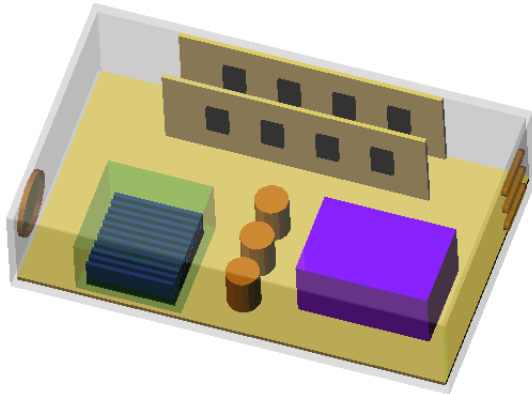
The electronic enclosure configuration with the additional part for applying the Local Initial Mesh op

Flow Simulation Project for the Local Initial Mesh Approach (Sink No1)

To create the project we clone the **Zoom – SinkNo1 - L4** to the existing **LocalMesh – SinkNo1 - N2** configuration, but in contrast to the previous cloning we reset the computational domain to the default size so the computational domain encloses the entire model.

Activate **Zoom – SinkNo1 - L4** configuration.

Open the **Clone Project** dialog, click **Add to existing** and, in the **Existing configuration** list select the **LocalMesh – SinkNo1 - N2** as the configuration to which Flow Simulation will attach the cloned project.

After clicking **OK**, confirm with **Yes** both the appearing messages.
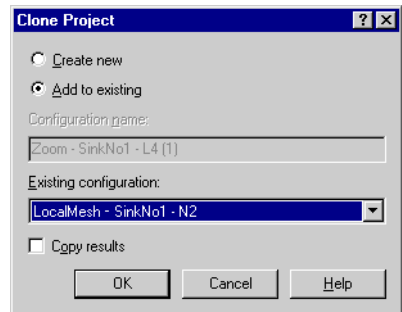
Conditions

First remove the inherited transferred boundary condition. Right-click the **Transferred Boundary Condition1** item in the tree and select **Delete**.
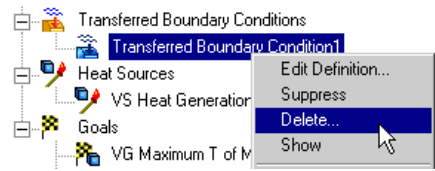
Next, copy the boundary conditions from the **Zoom – Global - L4** configuration using the **Copy Feature** tool.

1   Activate **Zoom – Global - L4** configuration.

2   Click **Flow Simulation**, **Tools**, **Copy Features**. The **Copy Features** dialog box appears.

3   Switch to the **Flow Simulation analysis tree** tab, hold down the **Ctrl** key and in the Flow Simulation Analysis tree select **Environment Pressure 1** and **External Inlet Fan 1** items. These features appear in the **Features to copy** list.

4   Select **LocalMesh – SinkNo1 - N2** as the **Target Project**.

5   Click **OK** ✔ .

6   Activate **LocalMesh – SinkNo1 - N2** configuration.
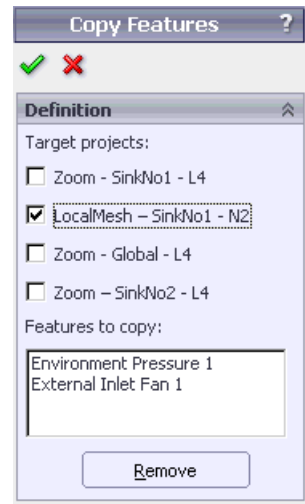
Heat Sources

To the already existing volume source of the 5W specified in the main chip, add the total 5W heat generation rate in the small chips.
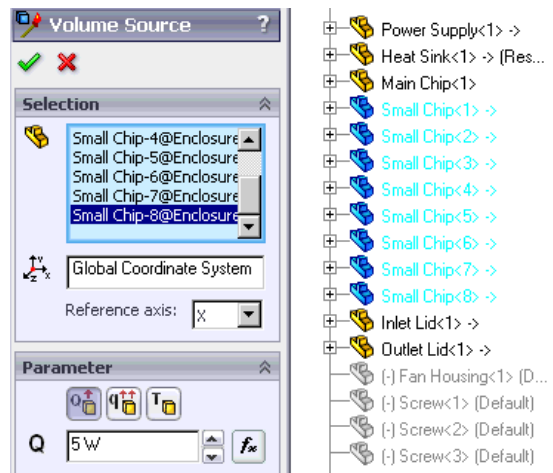
Solid Materials

The following material definitions were inherited from the previous project so you do not need to create them again, but you need to edit the **Silicon Solid Material 1** to include small chips and to edit **Insulator Solid Material 1** to include inlet and outlet lids:

a) the **Main Chip** and small chips are made of silicon;

b) the **MotherBoard**, the **Enclosure**, the **Inlet Lid** and the **Outlet Lid**s are made of insulator;

c) **PCB1** and **PCB2** are made of user defined **Tutorial PCB** material, which is added to the Engineering Database in the **First Steps - Conjugate Heat Transfer** tutorial example.

d) all other parts are made of the default aluminum.

Goals

Keep the cloned volume goals of maximum and average temperatures of the main chip and the heat sink.

Level of Initial Mesh

Click **Flow Simulation**, **Initial Mesh** to adjust the automatic initial mesh settings.

Set the **Level of initial mesh** to 3. Since heat conduction in solids is enabled, setting the **Level of initial mesh** to 4 together with the local mesh settings will produce large number of cells resulting in longer CPU time. To decrease the calculation time for this tutorial example we decrease the **Level of initial mesh** to 3. Note that the **Result resolution level** is still equal to 4 as it was specified in the Wizard. To see the value of the result resolution level, click **Flow Simulation**, **Calculation Control Options**, and then click **Reset.** To close the **Reset** dialog box, click **Cancel**.
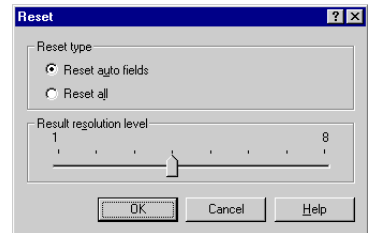
Click **Flow Simulation**, **Project**, **Rebuild**.

Specifying Local Initial Mesh Settings

To apply the local mesh setting to a region we need a component representing this region to be disabled in the **Component Control** dialog box.

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

2 In the FeatureManager Design Tree, select the **LocalMesh** component.
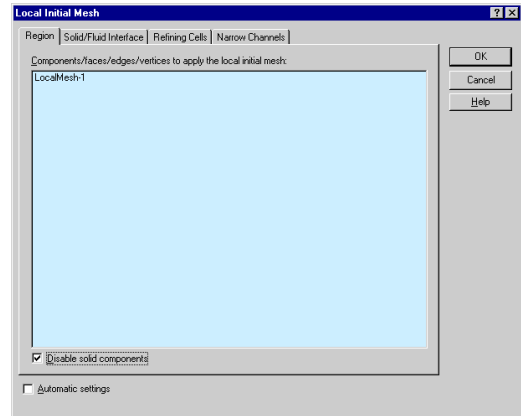
3 Click the **Disable solid components** check box.

4 Clear the **Automatic settings** check box.

5 Go to the **Narrow Channels** tab and set the **Characteristic number of cells across a narrow channel** = *2* and **Narrow channels refinement level** = *4*.

6 Click **OK**.
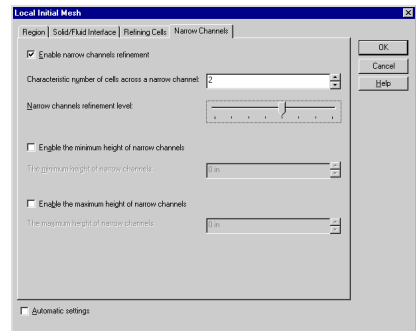
📖 *The **Narrow Channels** term is conventional and used for the definition of the model's flow passages in the normal-to-solid/ fluid-interface direction. The procedure of refinement is applied to each flow passage within the computational domain unless you specify for Flow Simulation to ignore the passages of a specified height with the **Enable the minimum height of narrow channels** and **Enable the maximum height of narrow channels** options. The **Characteristic number of cells across a narrow channel** (let us denote it as **Nc**) and **Narrow channels refinement level** (let us denote it as **L**) both influence the mesh in narrow channels in the following way: the basic mesh in narrow channels will be split to have the specified **Nc** number per a channel, if the resulting cells satisfy the specified **L**. In other words, whatever the specified **Nc**, a narrow channel's cells cannot be smaller in 8^L (2^L in each direction of the Global Coordinate System) times than the basic mesh cell. This is necessary to avoid the undesirable mesh splitting in superfine channels that may cause increasing the number of cells to an excessive value.*

In our case, to ensure the 2 cells across a channel criterion, we increased the **Narrow channels refinement level** to 4.
We perform these settings for both of the heat sinks under consideration.

Flow Simulation Project for the Local Initial Mesh Approach (Sink No2)

Clone the active **LocalMesh – SinkNo1 - N2** to the existing **LocalMesh – SinkNo2 - N2** configuration. While cloning confirm the message to rebuild the mesh.

Using the **Batch Run** calculate both projects.
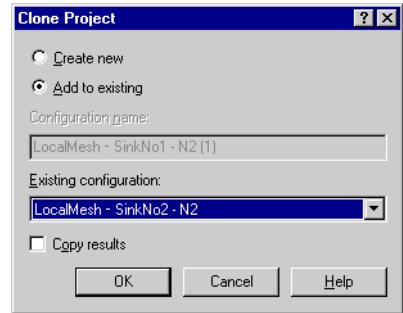
Results

The computational results obtained for both of the heat sinks are presented below in comparison with the results obtained with the EFD Zooming approach. It is seen that computations with the local mesh settings yield practically the same results as the EFD Zooming approach, therefore validating it.

The computed maximum and average main chip and heat sink temperatures when employing the different heat sinks.

	Parameter	Heat sink No.1		Heat sink No.2	
		Zoom - SinkNo1 - L4	LocalMesh - SinkNo1 - N2	Zoom - SinkNo2 - L4	LocalMesh - SinkNo2 - N2
Main chip	t_{max}, °F	111.1	114.1	96.4	99.4
	t_{aver}, °F	110.8	113.8	96.1	99.2
Heat sink	t_{max}, °F	111	114.1	96.3	99.4
	t_{aver}, °F	110.6	113.7	95.9	99
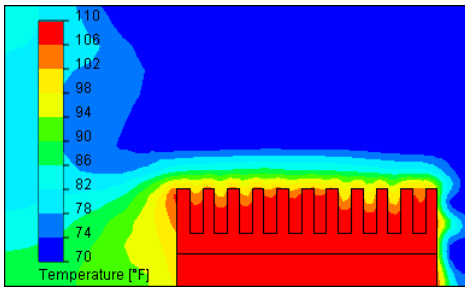
EFD Zooming   Local Mesh
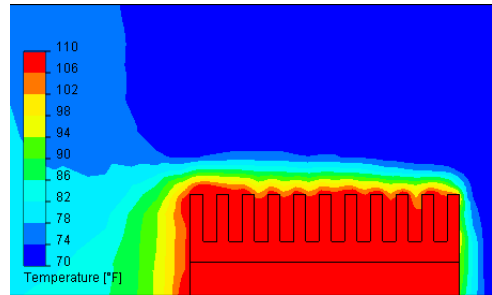
The temperature cut plots obtained for heat sink No.1 at *Y=2.19 in* (Top plane) with the EFD Zooming (left) and Local Mesh (right) approaches.
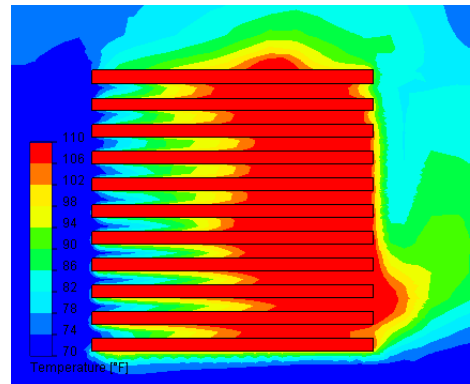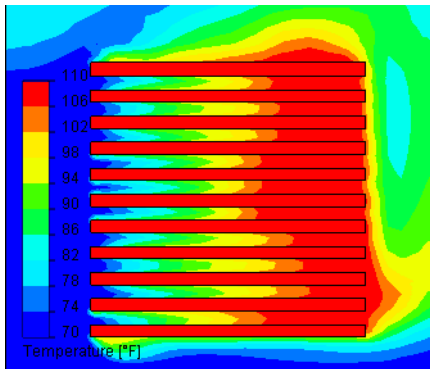
The temperature cut plots obtained for heat sink No.1 at *Z= -0.32 in* (Front plane) with the EFD Zooming (left) and Local Mesh (right) approaches.
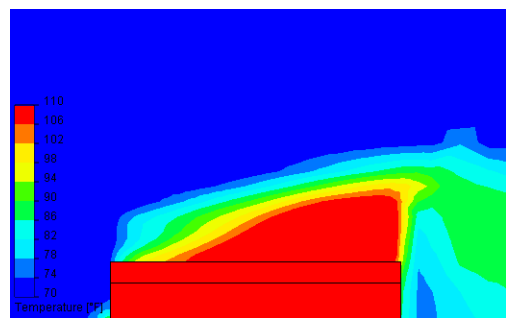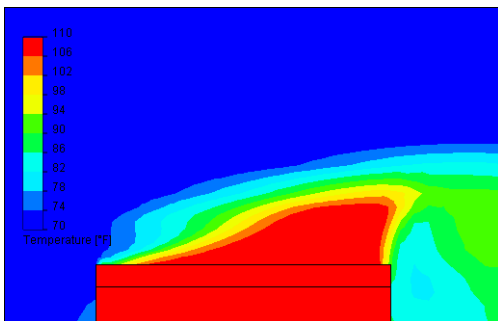
The temperature cut plots obtained for heat sink No.1 at *X= -1.53 in* (Right plane) with the EFD Zooming (left) and Local Mesh (right) approaches.
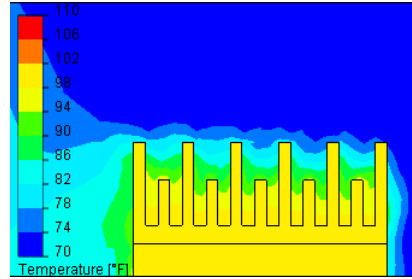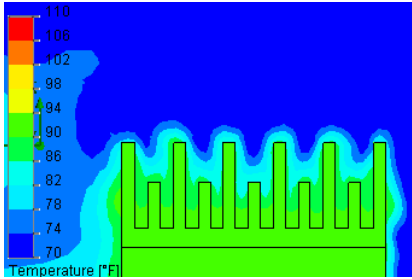
The temperature cut plots obtained for heat sink No.2 at *Y=2.19 in* (Top plane) with the EFD Zooming (left) and Local Mesh (right) approaches.
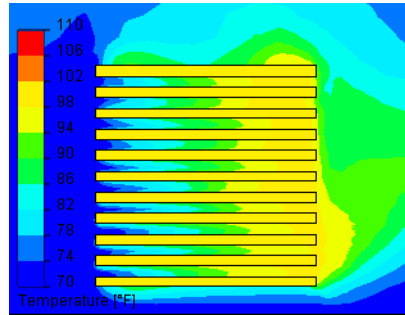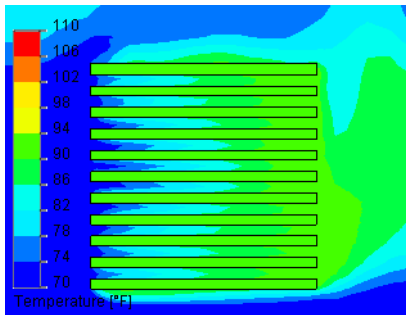
The temperature cut plots obtained for heat sink No.2 at *Z=-0.32 in* (Front plane) with the EFD Zooming (left) and Local Mesh (right) approaches.
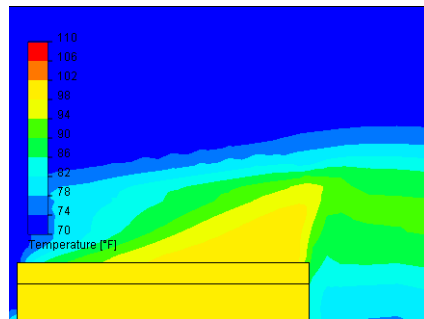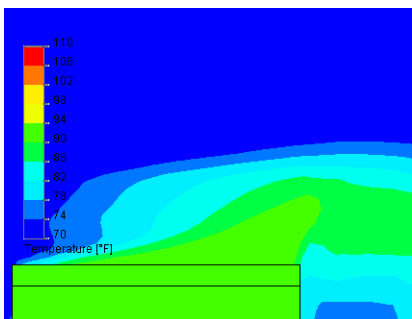
The temperature cut plots obtained for heat sink No.2 at *X= -1.53 in* (Right plane) with the EFD Zooming (left) and Local Mesh (right) approaches.

Textile Machine

Problem Statement

The simplified textile machine used by this tutorial is described as a closed hollow cylinder having a cylindrical stator with a narrow inlet tube. A thin-walled cone rotates at a very high speed. The air flows over the rotating cone before leaving through the outlet pipe. Due to the shear stress, the rotating cone swirls the air. The swirling air motion orients the fibers, for the correct formation of yarn.

In this example[1] a hollow cylinder with the following dimensions were used: 32 mm inner diameter and 20 mm inner height. Air is injected into an inlet tube of 1 mm diameter at a mass flow rate of 0.0002026 kg/s. The cone thickness is 1 mm and the cone's edge is spaced at 3 mm from the bottom of the main cylinder. The cone rotates at a speed of 130000 RPM. The static pressure of 96325 Pa is specified at the cylinder's outlet tube exit.

Flow Simulation analyzes the air flow without any fiber particles. The influence of the fiber particles on the air flow was assumed to be negligible. Small polystyrene particles were injected into the air stream using the results processing Flow Trajectory feature to study the air flows influence on the fibers. A 40 m/s tangential velocity of air is specified as an initial condition to speed up convergence and reduce the total CPU time needed to solve the problem.

1.This example can be run in Flow Simulation PE only.

Outlet

P = 96325 Pa

Housing

Rotating wall

ω = 130000 RPM

Inlet mass flow rate of
0.0002026 kg/s

Stator

1mm
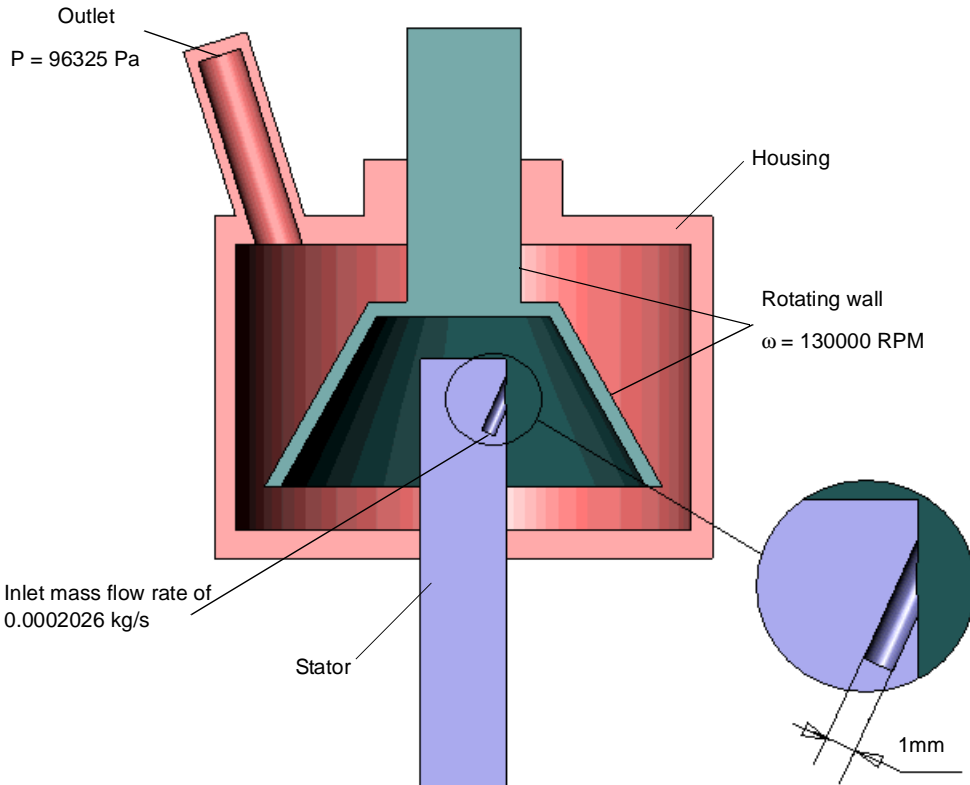
SolidWorks Model Configuration

Copy the **Tutorial Advanced 2 - Rotating Walls** folder into your working directory and
ensure that the files are not read-only since Flow Simulation will save input data to these
files. Open the **Textile Machine.SLDASM** assembly.
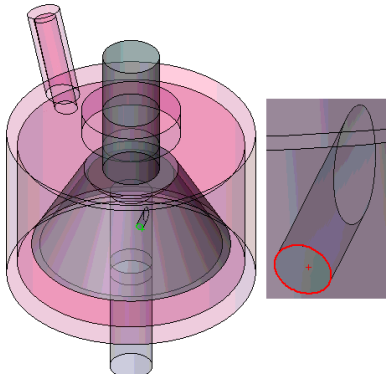
Project Definition
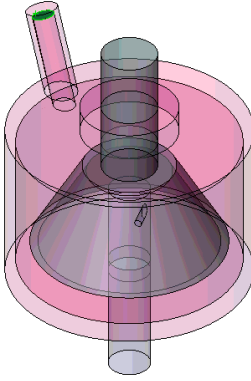
Using the **Wizard** create a new project as follows:

Project name	*Create new: 130000rpm*
Unit system	*SI; select mm (Millimeter) for Length and RPM (Rotations Per Minute) for Angular Velocity under Loads&Motion*
Analysis type	*Internal;* *Exclude cavities without flow conditions*
Physical features	*No physical features are selected*
Fluid	*Air*
Wall Conditions	*Adiabatic wall, default smooth walls*
Initial Conditions	*Default conditions*
Result and Geometry Resolution	*Result resolution level set to 4;* *Minimum gap size = 1 mm, automatic minimum wall thickness, other options are default*

Boundary Conditions

Specify the inlet and outlet boundary conditions as follows:

Inlet Boundary Condition	***Inlet Mass Flow = 0.0002026:*** *Inlet mass flow rate of 0.0002026 kg/s normal to the inlet face of the **Stator**; To do this, you may need to hide the **Initial Velocity 1** and **Initial Velocity 2** components.*	

Outlet Boundary Condition	***Outlet Static Pressure = 96325 Pa:*** *Static pressure of 96325 Pa at the outlet face of the **Housing** (the other parameters are default).*	

Specifying Rotating Walls

The influence of parts and components rotation on the flow can be simulated in Flow Simulation in two ways. With the Rotating Region feature you can assign a rotating reference frame to a selected fluid region. This allows to simulate the rotation of components of complex geometry, such as fans, pump wheels, impellers, etc. In this tutorial we consider rotation of a component with a relatively simple geometry. All surfaces of the textile machine rotor are surfaces of revolution such as cones or cylinders. For this kind of rotating geometry the Moving Wall boundary condition is better suited and usually provides more accurate results.

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

2   Select **Wall**, then **Real Wall**.

3   In the Flyout FeatureManager Design Tree select the **Rotor** component. All the rotor's faces are selected. However, the top face is out of the computational domain and must be excluded.
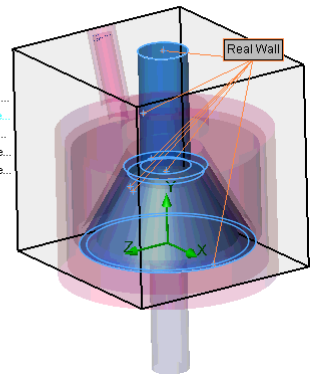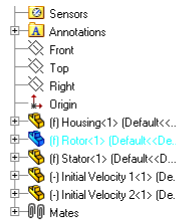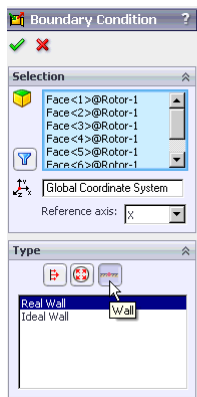
4 Click **Filter Faces** 🔽 . Select **Remove out of domain faces** 🔲 , and click **Filter**.
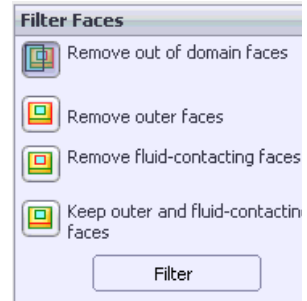
5 Select **Wall Motion**.

6 Select **Y** as the rotation **Axis**.

7 Specify the **Angular Velocity** 🌀 of 130000 RPM.

8 Click **OK** ✔ and rename the new **Real Wall 1** item to Rotating Wall = 130 000 rpm.
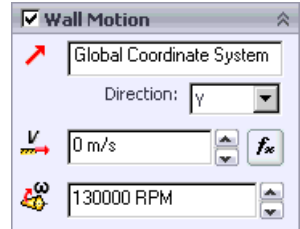
Initial Conditions - Swirl

To speed up the convergence, a 40 m/s tangential velocity of air is specified as an initial condition within the housing. The **Initial Velocity 1** and **Initial Velocity 2** auxiliary components are used to define a fluid domain.

1 Click **Flow Simulation**, **Insert**, **Initial Condition**.

2 In the Flyout FeatureManager Design Tree select the **Initial Velocity 1** and **Initial Velocity 2** components.

3 Select the **Disable solid components** option. Flow Simulation will treat these components as a fluid region.

4 Select **Y** in the **Reference axis** list.
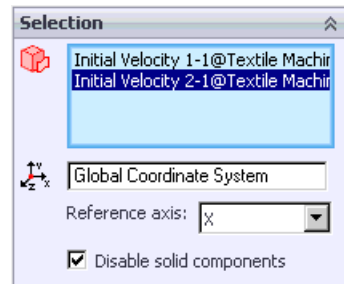
5 Under **Flow Parameters**, select the **Dependency** for the **Velocity in X direction** by clicking the 𝑓ₓ button. The **Dependency** dialog box appears.

6 In the **Dependency type** list, select **Formula Definition**.
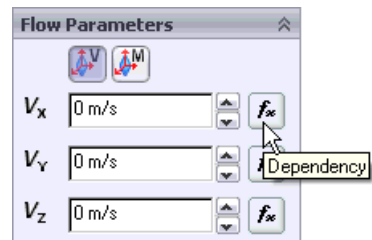
7 In the **Formula** box, type the formula defining the velocity in X direction: `40*cos(phi)`.
Here *phi* is the polar angle φ defined as shown on the picture below.
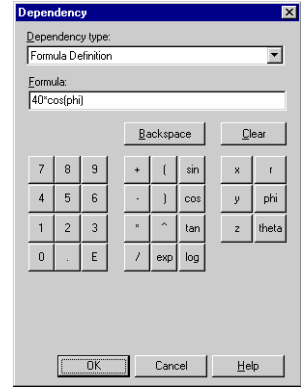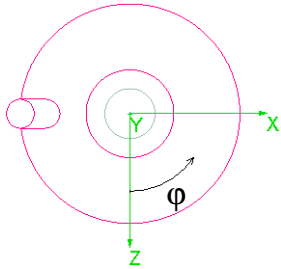
8 Click **OK**. You will return to the **Initial Condition** PropertyManager.

9 Click **Dependency** to the right of the **Velocity in Z direction** box and specify formula for the Z component of velocity: `-40*sin(phi)`.

10 Click **OK**.

11 Under **Thermodynamic Parameters,** change the **Pressure P** to `99800` Pa.

12 Click **OK** ✔ .

13 Click-pause-click the new **Initial Condition1** item and rename it to `vel = 40 m\s`.
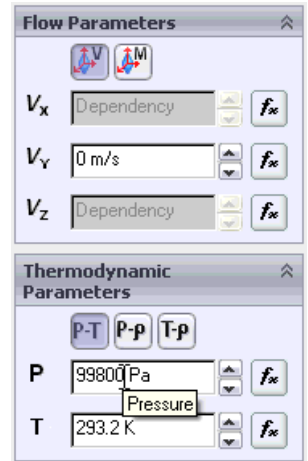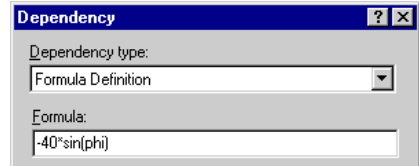
Specifying Goals

Since the rotating cone swirls the air, it make sense to specify the air velocity as a goal to ensure the calculation stops when the velocity is converged. In addition, let us specify the static pressure surface goal at the inlet and the mass flow rate surface goal at the outlet as additional criteria for converging the calculation.

Specify the following project goals:

GOAL TYPE	GOAL VALUE	FACE/COMPONENT
Global Goal	Average Velocity	
Surface Goal	Mass Flow Rate	Outlet face(click the outlet static pressure boundary condition item to select the outlet face)
Surface Goal	Average Static Pressure	Inlet face(click the inlet mass flow rate boundary condition item to select the inlet face)
Volume Goal	Average Velocity	**Initial Velocity 1** (select the component in the Flyout FeatureManager Design Tree)
Volume Goal	Average Velocity	**Initial Velocity 2** (select the component in the Flyout FeatureManager Design Tree)

Calculate the project.

Results - Smooth Walls

The calculated flow velocity field and velocity Y-component field at Z = 0 (XY section) are shown in the pictures below. It can be seen that the maximum flow velocity occurs near the inlet tube and near the rotating cone's inner surface at the cone's edge.

Velocity in the XY section at Z = 0.

Flow velocity   Y-component of flow velocity
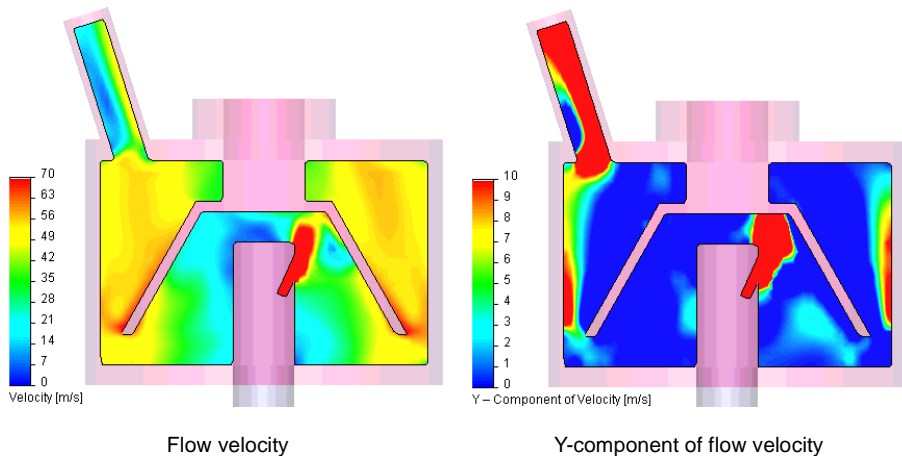
It is interesting that the vertical (i.e. along the Y axis) velocity in the region close to the rotating cone's internal and external surfaces is directed to the cylinder bottom. Also, this velocity component is nearly zero in the gap between the rotating cone and the bottom of the cylinder, and positive (i.e. directed to the top) in the vicinity of the cylinder's side walls. As a result, small particles carried by the air into the region between the lower edge of the rotating cone and the bottom of the cylinder cannot leave this region due to the small vertical velocity there. On the other hand, larger particles entering this region may bounce from the cylinder's bottom wall (in this example the ideal, i.e. full reflection is considered) and fly back to the region of high vertical velocity. Then they are carried by the air along the cylinder's side walls to the cylinder's top wall where they remain in this region's vortex.

Displaying Particles Trajectories and Flow Streamlines

1   In the Flow Simulation Analysis tree, right-click the **Flow Trajectories** icon and select **Insert**.

2   Click the Flow Simulation Analysis tree tab and then click the inlet boundary condition icon (**Inlet Mass Flow = 0.0002026**) to select the inlet face from which the particles are injected.

3   Set the **Number of Points** to 10.

4   Under **Appearance**, set the **Draw Trajectories As** to **Lines with Arrows** and change **Color by Parameters** to **Velocity**.

5   Under **Constraints.** select the **Forward** direction and increase the **Maximum Length** of trajectories to 15 m.

📖 *The **Maximum length** option limits the length of the trajectory to the specified value. We increase this value to show better the flow vorticity.*
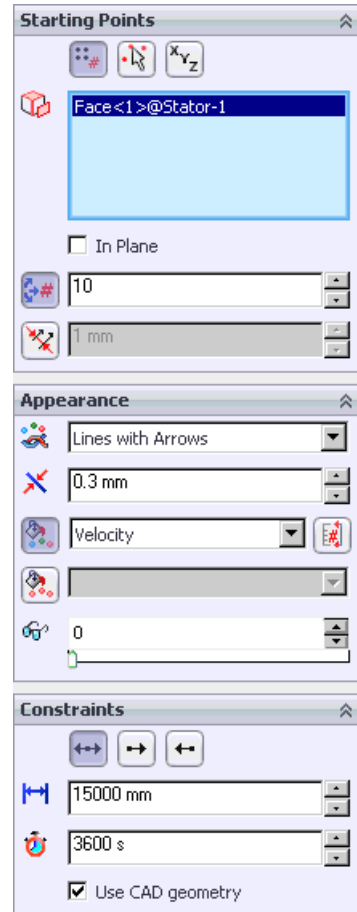
6 Click **OK** ✓ to display flow streamlines.

7 To specify the parameter display range click the maximum value in the velocity palette bar and type 70 m/s in an edit box.
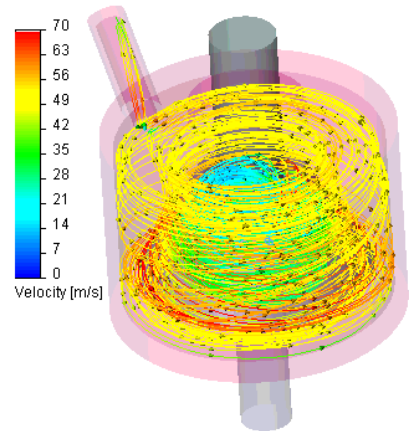
To display particles trajectories, we need to specify initial particle properties (temperature, velocity and diameter), particle's material and the wall condition (absorption or reflection).

1 In the Analysis tree, right-click the **Particle Studies** icon and select **Wizard**.

2 Keep the default name for the Particle Study and click **Next** ⊕.

3 Click the inlet boundary condition icon (**Inlet Mass Flow = 0.0002026**) in the tree to select the inlet face from which the particles are injected.

4 Set the **Number of Points** 🔁# to 5.

5 Under the **Particle Properties**, set the **Diameter** ∅ equal to 0.005 mm and change the **Material** 🔵 to the **Solids** and in the list select the **Polystyrene** (**Materials, Solids, Pre-Defined, Polymers**).

📖 *We leave unchanged the default zero values of relative velocity and temperature, which means that the velocity and temperature of particles are equal to those of the incoming flow. We also leave the default value of mass flow rate, since it is used only to estimate mass rates of erosion or accumulation, which we are not going to take into account.*

6 Click **Next** ⊕.
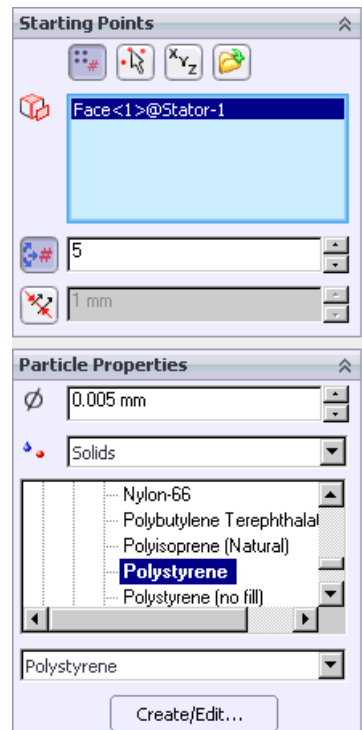
7 Change the **Default Wall Condition** to **Reflection**.

8 Click **Next** ⊕ twice.

9 Under **Default Appearance**, set **Draw Trajectories as** 🎣 **Lines with Arrows**.

10 Under **Constraints,** increase the **Maximum Length** |←→| of trajectories to 15 m.

11 Click **OK** ✔. A new **Particle Study 1** item with one sub-item (**Injection 1**) appear in the **Analysis tree**.

12 Right-click the created **Injection 1** item and select **Clone**. The **Injection 2** item will be created. For this item, increase the particle size by editing the **Diameter** to 0.015 mm.

13 Right-click the **Particle Study 1** item and select **Run**.

14 Select **Injection 1** and click **Show** to view the particles' trajectories.

15 When finished examining the trajectiries from the first injection, hide the **Injection 1** trajectories and show the **Injection 2** trajectories.
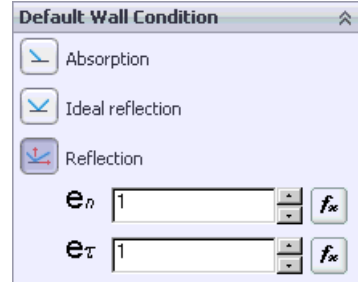
Modeling Rough Rotating Wall

In the previous calculation zero roughness was used for the walls of the rotating cone's internal and external surfaces. To investigate an influence of the rotating cone wall's roughness, let us perform the calculation with the rotating cone's internal and external surfaces' at 500 μm roughness under the same boundary conditions.

Create a new configuration by cloning the current project, and name it 130000rpm - rough wall.
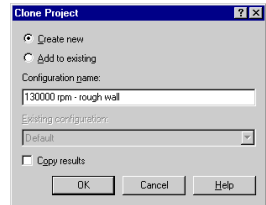
Adjusting Wall Roughness

1 Right-click the **Rotating Wall = 130 000 rpm** item and select **Edit Definition**.

2 Under **Wall parameters**, select **Adjust Wall Roughness** 📏.

3 Specify the wall roughness of 500 micrometers.

4 Click **OK** ✔ .

Run the calculation.
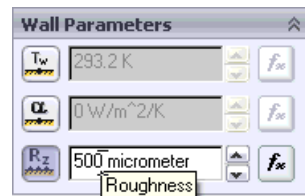
Results - Rough Walls

The calculated fields of flow velocity and Y-component of velocity in different section are shown below and reveal practically no change in the vertical velocity of the flow. As a result, the flying particles' trajectories are nearly identical to those in the case of smooth walls. It is seen that increase in the roughness from 0 to 500 μm increases the vortex flow's tangential velocity.

Velocity in the XY section at Z = 0 (roughness = 500 μm)

Flow velocity

Flow velocity's Y-component
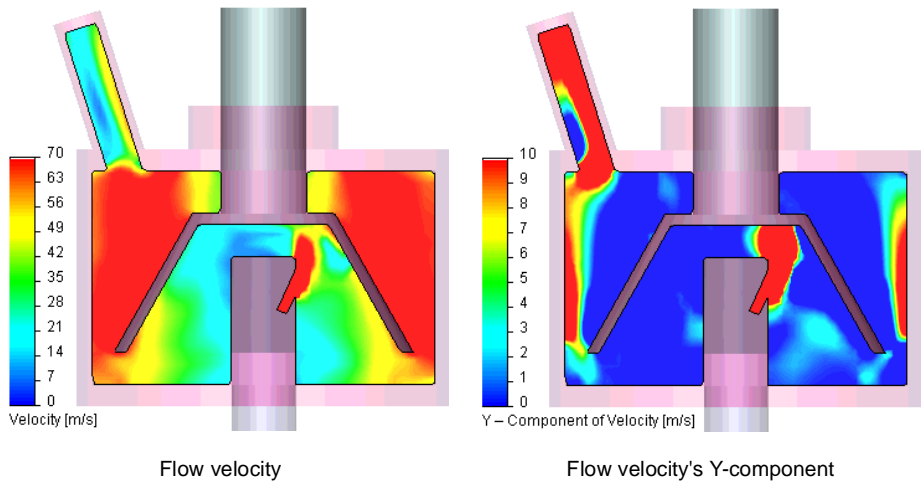
Velocity in the ZX section at Y = 2 mm

roughness = 0 μm

roughness = 500 μm
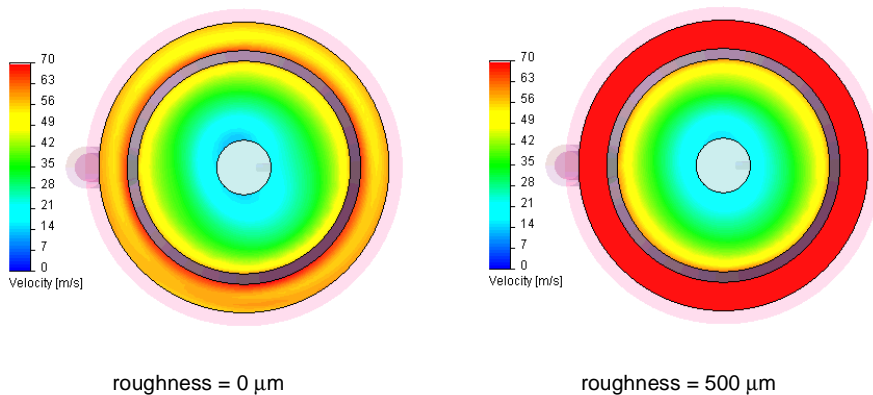
Flow streamlines
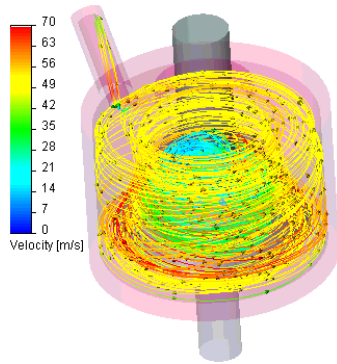
Smooth wall
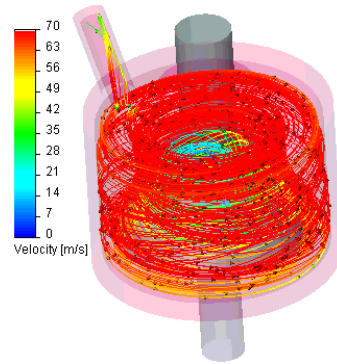
Rough wall

Trajectories of 5 μm particles
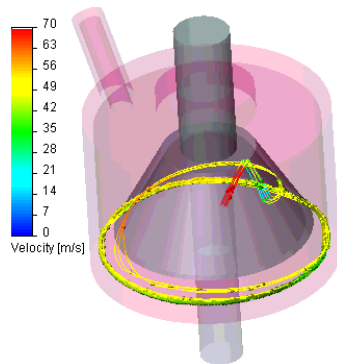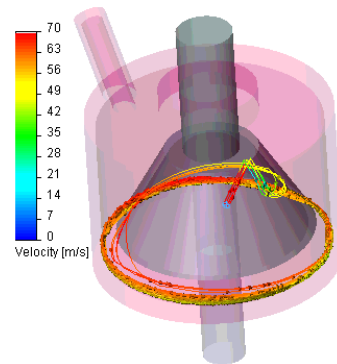
Smooth wall

Rough wall

Trajectories of 15 μm particles
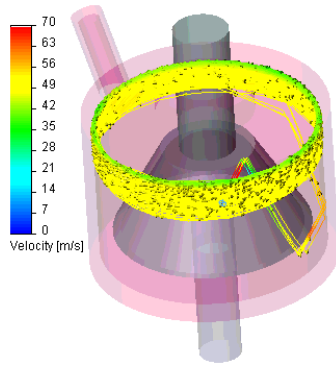
Smooth wall
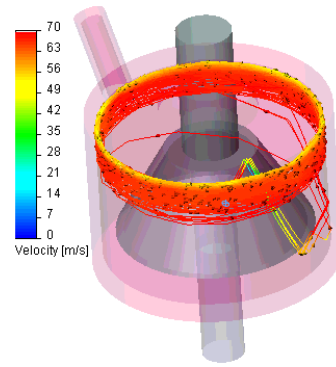
Rough wall

Non-Newtonian Flow in a Channel with Cylinders

Problem Statement

Let us consider a non-Newtonian liquid's 3D flow[1] through a rectangular-cross-section channel encumbered with seven circular cylinders arranged asymmetrically with respect to the channel's midplane shown in **Ref. 1**. Following **Ref. 1**, let us consider the 3% aqueous solution of xanthan gum as a non-Newtonian liquid. Its viscosity approximately obeys the power law $\eta = K \cdot \left(\dot{\gamma} \right)^{n-1}$ with a consistency coefficient of K = 20 Pa×s^n and a power-law index of n = 0.2, whereas its other physical properties (density, etc.) are the same as in water (since the solution is aqueous).

The problem's goal is to determine the total pressure loss in the channel. Also, to highlight the influence of the 3% xanthan gum addition to water on the channel's total pressure loss, we will calculate the flow of water using the same volume flow rate within the channel.

The Flow Simulation calculations are performed with the uniform liquid velocity profile at the channel inlet, the liquid's volume flow rate is 50 cm^3/s. The static pressure of 1 atm is specified at the channel outlet. The calculation's goal is the channel's resistance to the flow, i.e., the total pressure drop ΔP_o between the channel inlet and outlet.
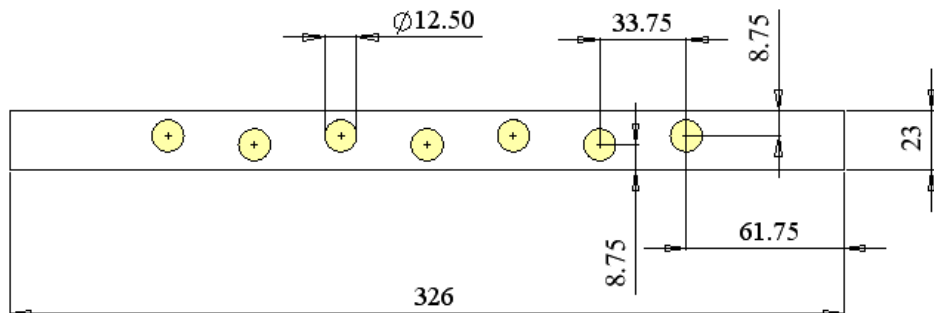
1. This example can be run in Flow Simulation PE only.

SolidWorks Model Configuration

Copy the **Tutorial Advanced 3 - Non-Newtonian Flow** folder into your working directory and ensure that the files are not read-only since Flow Simulation will save input data to these files. Open the **Array of Cylinders.sldprt** part.

Specifying Non-Newtonian Liquid

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

2   In the **Database tree**, select **Materials, Non-Newtonian Liquids, User Defined**.

3   Click **New Item** ☐ in the toolbar. The blank **Item Properties** tab appears. Double-click the empty cell to set the corresponding property value.

4   Specify the material properties as shown in the table below:

Name	XGum
Density	1000 kg/m^3
Specific heat	4000 J/(kg*K)
Thermal conductivity	0.6 W/(m*K)
Viscosity	Power law model
Consistency coefficient	20 Pa*s^n
Power law index	0.2

Save and exit the database.

Project Definition
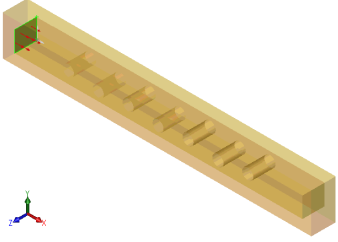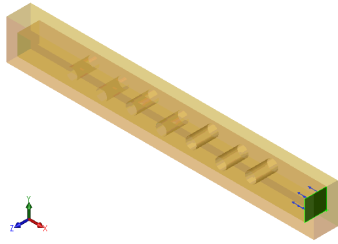
Using the **Wizard** create a new project as follows:

Project name	*Create new: XGS*
Unit system	*CGS modified: Pa (Pascal) for the Pressure & Stress*
Analysis type	*Internal; Exclude cavities without flow conditions*
Physical features	*No physical features are selected (default)*
Fluid	*XGum (non-Newtonian liquids); Flow type: Laminar only (default)*

Wall Conditions	*Adiabatic wall, default smooth walls, default slip condition*
Initial Conditions	*Default conditions*
Result and Geometry Resolution	*Default result resolution level 3; Minimum gap size=0.25 cm, no other changes*

Conditions

Specify boundary conditions as follows:

Inlet Boundary Condition	**Inlet Volume Flow 1:** *50 cm^3/s Volume flow rate normal to face; default temperature (20.05 °C) at the face;*	
Outlet Boundary Condition	**Static Pressure 1***: Default value (101325 Pa) for the Static pressure at the face;*	

Specifying Goals

Specify surface goals for the **Average Total Pressure** at the inlet and outlet.

Specify an equation goal for the total pressure drop between the channel's inlet and outlet.
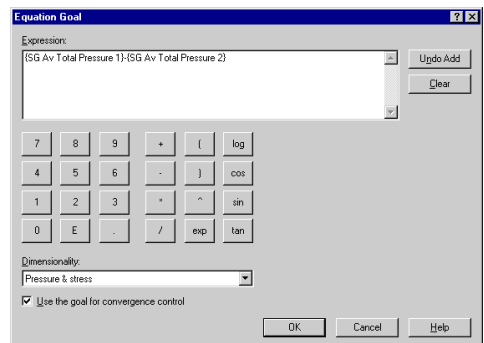
Run the calculation. When the calculation is finished, create the goal plot to obtain the pressure drop between the channel's inlet and outlet.

Array of Cylinders.SLDPRT [XGS]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]
SG Av Total Pressure 1	[Pa]	105622.4926	105622.4125	105620.3901	105627.4631	100
SG Av Total Pressure 2	[Pa]	101329.0109	101329.0091	101329.0051	101329.0109	100
Pressure Drop	[Pa]	4293.481659	4293.4034	4298.457377	4291.380166	100

It is seen that the channel's total pressure loss is about 4 kPa.

Comparison with Water

Let us now consider the flow of water in the same channel under the same conditions (at the same volume flow rate).

Create a new configuration by cloning the current project, and name it Water.
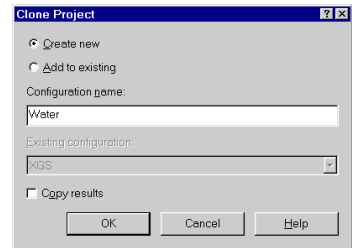
Changing Project Settings

1 Click **Flow Simulation**, **General Settings**.

2 On the **Navigator** click **Fluids**.

3 In the **Project Fluids** table, select **XGum** and click **Remove**. Answer **OK** to the appearing warning message.

4 Select **Water** in **Liquids** and click **Add**.

5 Under **Flow Characteristics**, change **Flow type** to **Laminar and Turbulent**.

6 Click **OK**.
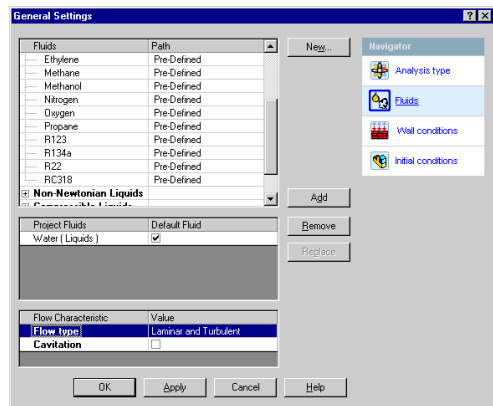
Run the calculation. After the calculation is finished, create the goal plot.

As shown in the results table above, the channel's total pressure loss is about 60 Pa, i.e.

Array of Cylinders.SLDPRT [water]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]
SG Av Total Pressure 1	[Pa]	101395.004	101395.0214	101394.8731	101395.1171	100
SG Av Total Pressure 2	[Pa]	101329.3912	101329.3378	101329.3084	101329.3912	100
Pressure Drop	[Pa]	65.6128767	65.68357061	65.76566097	65.55243288	100

60...70 times lower than with the 3% aqueous solution of xanthan gum, this is due to the water's much smaller viscosity under the problem's flow shear rates.
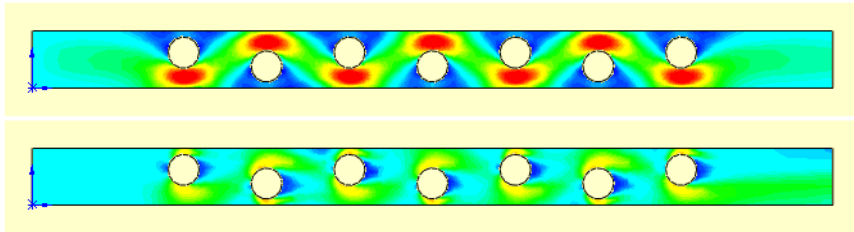
The XGS (above) and water velocity distribution in the range from 0 to 30 cm/s.

1   Georgiou G., Momani S., Crochet M.J., and Walters K. *Newtonian and Non-Newtonian Flow in a Channel Obstructed by an Antisymmetric Array of Cylinders*. Journal of Non-Newtonian Fluid Mechanics, v.40 (1991), p.p. 231-260.

11

Heated Ball with a Reflector and a Screen

Problem Statement

Let us consider a ball with diameter of 0.075 m, which is continuously heated by a 2 kW heat source. The ball radiates heat to a concentrically arranged hemispherical reflector with the inner diameter of 0.256 m, and through a glass cover of the same inner diameter to a circular screen with the diameter of 3 m arranged coaxially with the reflector at the 1 m distance from the ball. All parts except the glass cover are made of stainless steel. The ball surface and the screen surface facing the ball are blackbody. The other side of the screen side is non-radiating. The goal of the simulationl is to see how the presence of reflector and its emissivity influence the ball and screen temperatures. To do that, the following three cases are considered[1]:

- Case 1: the reflector inner surface is whitebody;
- Case 2: all reflector surfaces are blackbody;
- Case 3: the reflector is removed.

The steady-state problem is solved with the **Heat conduction in solids** option selected, so that conduction within all parts is calculated. Considering the convective heat transfer negligibly low (as in highly rarefied air), we also select the **Heat conduction in solids only** option. With this option, we do not need to specify a fluid for the project, and it is calculated without considering any fluid flow at all, thus saving the CPU time and limiting the heat transfer between parts to radiation only. The initial temperature of the parts is assumed to be 293.2 K.

Let us consider the solutions obtained with Flow Simulation for each of the cases under consideration.

1.This example can be run in Flow Simulation PE only.

SolidWorks Model Configuration

Copy the **Tutorial Advanced 4 - Surface-to-surface Radiation** folder into your working directory and ensure that the files are not read-only since Flow Simulation will save input data to these files. Open the **Heated Ball Assembly.SLDASM** assembly.
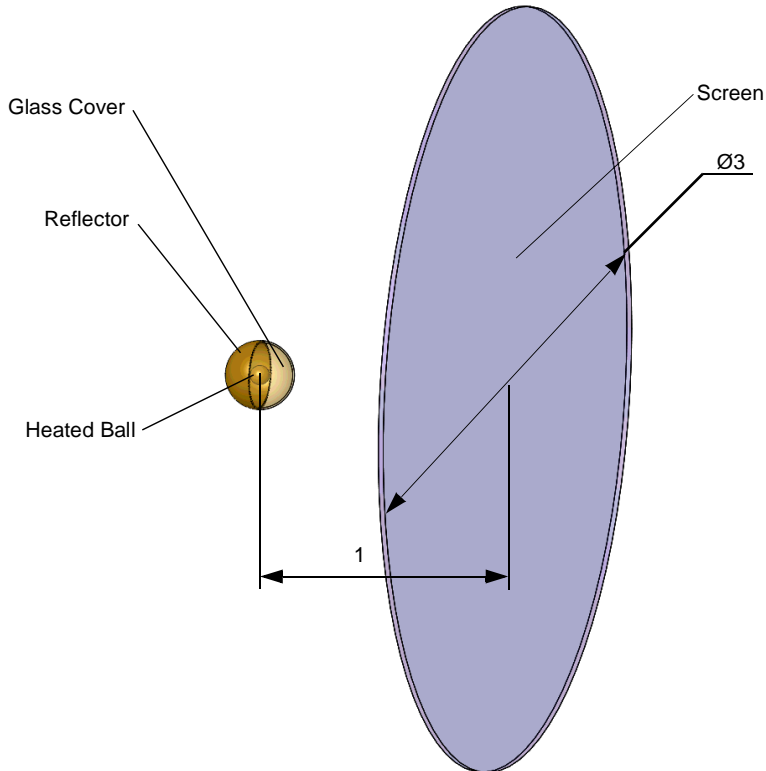
The heated ball with the reflector and screen.

Case 1: The reflector inner surface is whitebody

Project Definition

Using the **Wizard** create a new project as follows:

Project name	*Create new: Case 1*
Unit system	*SI*
Analysis type	*External*
Physical features	*Heat conduction in solids, Heat conduction in solids only, Radiation, Environment radiation: Environment temperature = 293.2 K;*
Default Solid	*Alloys/Steel Stainless 321*
Wall conditions	*Default wall radiative surface: Non-radiating surface;*
Initial and Ambient Conditions	*Default initial solid temperature of 293.2 K*
Result and Geometry Resolution	*Set result resolution level to 3; Automatic minimum gap size, Manual Minimum wall thickness = 0.007 m; other options are default.*

Definition of the Computational Domain

Specify the computational domain size as follows:

X max = 1.4 m	Y max = 1.6 m	Z max = 1.6 m
X min = -0.2 m	Y min = -1.6 m	Z min = -1.6 m

Adjusting Automatic Mesh Settings

Click **Flow Simulation**, **Initial Mesh**. Clear
the **Automatic settings** check box to
switch off the automatic initial mesh
settings, switch to the **Solid/Fluid Interface**
tab and change the **Curvature refinement
level** to 5. Click **OK**.
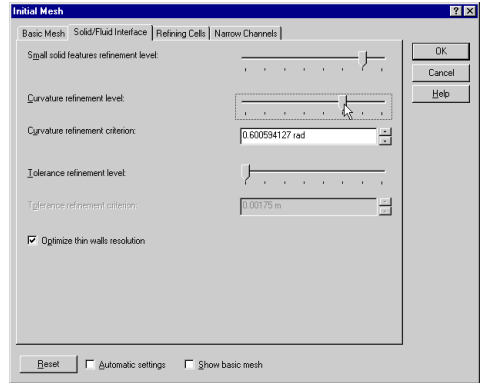
Defining Radiative Surfaces

Follow the steps below to specify the radiative surfaces:

1   Click **Flow Simulation**, **Insert**, **Radiative Surface**.

2   Under **Type**, expand the list of **Pre-Defined** radiative
    surfaces and select **Blackbody wall**.
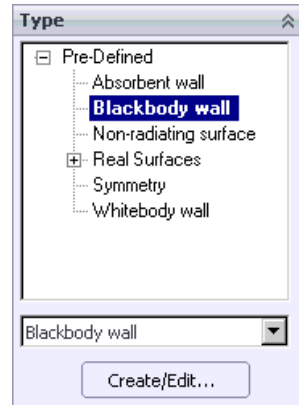
3   In the Flyout FeatureManager Design Tree select the
    Heated Sphere component. Next, select the surface of
    Screen facing the **Heated Sphere**.

4   Click **OK** ✔ . Rename the new **Radiative Surface 1**
    item to Blackbody Walls
    .
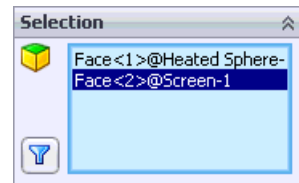
    Click anywhere in the graphic area to clear the selection.

5 Click **Flow Simulation**, **Insert**, **Radiative Surface**.

6 Under **Type**, expand the list of **Pre-Defined** radiative surfaces and select **Whitebody wall**.

7 Select the inner surface of **Reflector**.

8 Click **OK** ✔ . Change the name of the new radiative surface to Whitebody Wall.
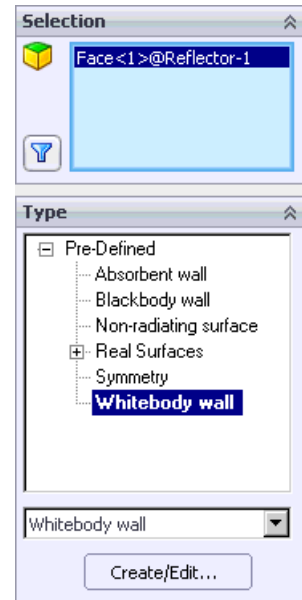
Specifying Bodies and Materials Transparent to the Heat Radiation

Assign the **Glass** material to the glass cover and specify it as transparent to radiation.

1 Click **Flow Simulation**, **Insert**, **Solid Material**.

2 In the Flyout FeatureManager Design Tree select the **Glass** component.

3 Under **Solid** expand the list of **Pre-Defined** solid materials and select **Glass** under **Glasses and Minerals**.
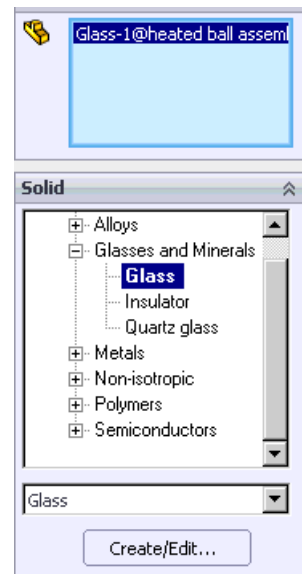
4   Under **Radiation Transparency** select **Transparent**

   ![icon], then select **Thermal only**.

Radiation Transparency ≫

▯ Opaque

▯ Transparent

   ⦿ Thermal only

   ○ Solar only

   ○ Thermal and solar
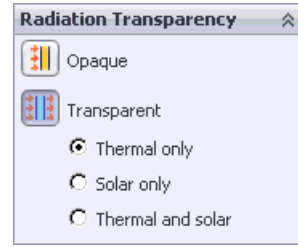
📖 *You can separately specify a solid material transparency to the solar radiation and transparency to thermal radiation from all other sources, including heated bodies. Since there are no sources of solar radiation in the project, we can select **Thermal only** to make the material fully transparent to all radiation in the project.*

5   Click **OK**. Flow Simulation now treats this solid material and all bodies it is applied to as fully transparent to the thermal radiation.

Specifying Heat Source and Goals

Specify the surface heat source of the heat generation rate at the sphere surface:

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

2   In the Flyout FeatureManager Design Tree select the **Heated Sphere** component.

Parameter ≫

Q   2000 W   *fx*

3   Select **Heat Generation Rate** 🔲 as the source type and set its value to 2000 W.

Specify surface goals of the maximum, average, and minimum temperatures at the **Heated Sphere** surface and the blackbody surface of **Screen** .
In addition, specify the volume goal of the **Heated Sphere** average temperature. (In all cases you should select **Temperature of Solid** as the goal parameter). You can rename the goals as shown to make it easier to monitor them during the calculation.

Goals
   SG Max T of Heated Sphere
   SG Average T of Heated Sphere
   SG Min T of Heated Sphere
   SG Max T of Screen
   SG Min T of Screen
   SG Average T of Screen
   VG Average T of Heated Sphere

Save the model and run the calculation.

If you take a look at the goals convergence, you can see that the sphere's temperature at the start of the calculation is high. This happens because the initial sphere's temperature (293.2 K) is too low to take away by radiation the heat produced by the 2000 W heat source . To illustrate this better, in cases number 2 and 3 we will increase the initial temperature of the heated sphere to 1000 K, thus providing the greater amount of heat being lost by the sphere starting from the very beginning of the calculation.

Case 2: All reflector surfaces are blackbody

In contrast to the Case 1, in this case the reflector inner surface is blackbody and all other surfaces of the reflector are also blackbody.

Create a new **Case 2** project by cloning the current **Case 1** project.
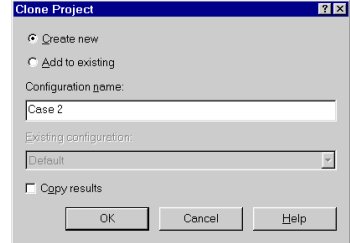
Changing the Radiative Surface Condition

1 Delete the **Whitebody Wall** condition.

2 Right-click the **Blackbody Walls** item and select **Edit Definition**.

3 Click the **Reflector** item in the Flyout FeatureManager Design tree in order to select all its surfaces. The surfaces are added to the list.

4 Click **OK** ✓.
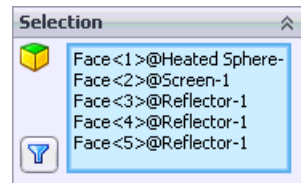
Goals Specification

Specify the additional surface goals of the maximum, average, and minimum temperature of solid for the **Reflector** inner and outer surfaces.

Specifying Initial Condition in Solid

Specify the initial temperature of the heated sphere of 1000 K using **Initial Condition**.

Save the model.
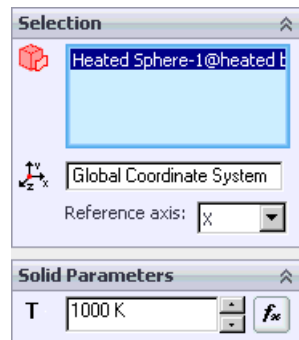
Case 3: The reflector is removed

In contrast to **Case 1** and **Case 2**, the reflector is removed in **Case 3**.

Create a new **Case 3** project by cloning the current **Case 2** project.

1   Edit definition of the **Blackbody Walls** condition: under **Selection** remove from the list all faces belonging to **Reflector**. To delete a face from the list of **Faces to Apply the Radiative Surface**, select the face and press the **Delete** key.

2   Delete the surface goals related to reflector.

3   Disable the **Reflector** component in the **Component Control** dialog box.

Goals
- SG Max T of Heated Sphere
- SG Average T of Heated Sphere
- SG Min T of Heated Sphere
- SG Max T of Screen
- SG Min T of Screen
- SG Average T of Screen
- VG Average T of Heated Sphere

Using **Batch Run**, calculate the cases 2 and 3.

Results

In Case 1, due to the radiation returned by the reflector, the ball surface facing the reflector is hotter than the ball surface facing the screen (see pictures below). Therefore, the screen temperature in Case 1 is higher than in the other cases.

In Case 2, radiation coming from the ball to the reflector heats up the reflector and heat is radiated from the reflector outer surface to ambient, therefore being lost from the system. Since less heat is returned to the ball by the radiation from the reflector , the ball temperature is lower, although it is distributed over the ball in the same manner as in Case 1. Less heat is also coming from the reflector to the screen. As a result, the screen temperature is lower than in Case 1.

Since the reflector is removed in Case 3, there is no noticeable heat radiated back to the ball. The ball temperature is lower than in Case 2 and mostly uniform (the non-uniformity is lower than 1 K). Since in the abscence of reflector the screen is only exposed to the radiation from the side of the ball facing the screen, the screen temperature is the lowest among all the cases.
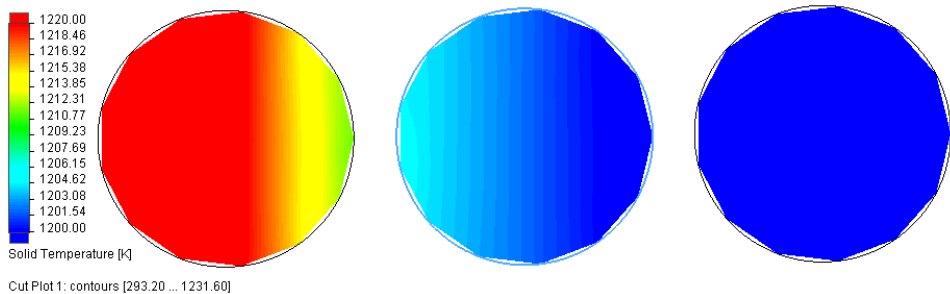
The ball temperature distribution (front plane cross-section) in CASE 1 (left), CASE 2 (center) and CASE 3 (right) in the range from 1200 to 1220 K (the reflector is arranged at the left).
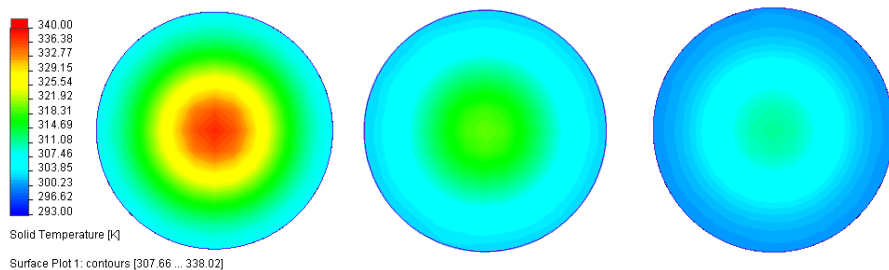
The screen temperature distribution (surface plot of solid temperature) in CASE 1 (left), CASE 2 (center) and CASE 3 (right) in the range from 295 to 340 K.

Parameter		Case 1	Case 2	Case 3
The ball's temperature, K	Maximum	1232.52	1205.16	1193.81
	Average	1221.79	1202.11	1193.68
	Minimum	1211.35	1199.04	1193.59
The screen's temperature, K	Maximum	338.09	320.01	311.40
	Average	317.09	308.26	303.58
	Minimum	307.65	302.80	299.90

Rotating Impeller

Problem Statement

Let us consider the air flow through a centrifugal pump having a rotating impeller (see below).[1] This pump has a stationary axial inlet (an eye), a pipe section of 92 mm radius with a central body of circular arc contour, which turns the flow by 90° from the axial direction. At the inlet's exit the radial air flow is sucked by a rotating impeller, which has seven untwisted constant-thickness backswept blades with wedge-shape leading and trailing edges. Each blade is cambered from 65° at the impeller inlet of 120 mm radius to 70° at the impeller exit of 210 mm radius, both with respect to the radial direction. These blades are confined between the impeller shrouding disks rotating with the same (as the blades) angular velocity of 2000 rpm. Downstream of the impeller the air enters a stationary (non-rotating) radial diffuser.

To complete the problem statement, let us specify the following inlet and outlet boundary conditions: inlet air of 0.3 m^3/s volume flow rate having uniform velocity profile with vectors parallel to the pump's axis; at the radial-directed outlet a static pressure of 1 atm is specified.

1.This example can be run in Flow Simulation PE only.

$\Omega = 2000$ rpm

Outlet Static Pressure

Inlet Volume Flow
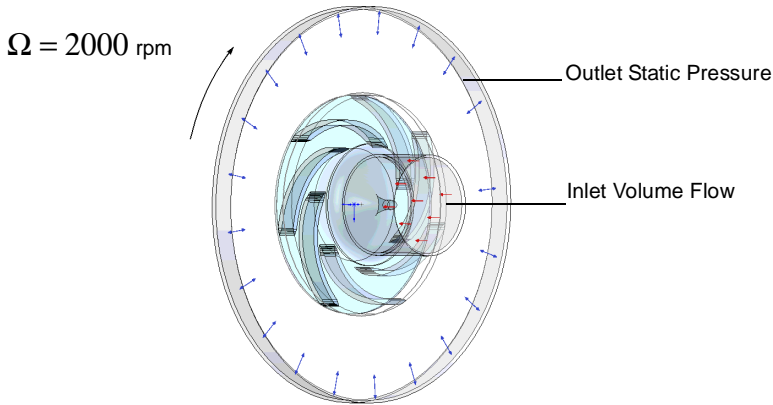
The centrifugal pump with a rotating impeller.

SolidWorks Model Configuration

Copy the **Tutorial Advanced5 - Rotating Impeller** folder into your working directory. Open the **Pump.SLDASM** assembly.
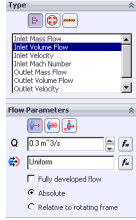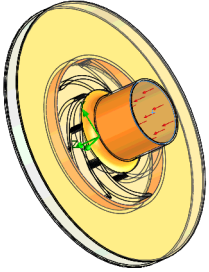
Project Definition

Using the **Wizard** create a new project as follows:

Project name	*Use current: Impeller Efficiency*
Unit system	*SI*
Analysis type	*Internal; Exclude cavities without flow conditions*
Physical features	*Rotation: Type - Global rotating, Rotation axis - Z axis of Global Coordinate system, Angular velocity=2000 RPM (209.43951 rad/s)*
Default fluid	*Air*
Wall Conditions	*Adiabatic wall, default smooth walls*
Initial Conditions	*Default conditions*
Result and Geometry Resolution	*Set the Result resolution level to 4; Minimum gap size = 0.04 m, minimum wall thickness = 0.01 m, other options are default*
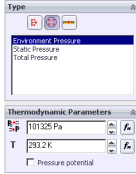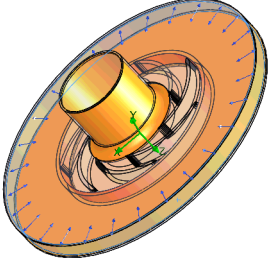
Boundary Conditions

Specify the boundary conditions for inlet and outlet flows as shown in the tables below:

Type	**Inlet Volume Flow**	
Name	**Inlet Volume Flow 1**	
Faces to apply	the inner face of the **Inlet Lid**	
Parameters: Value of 0.3 m^3/s with the Uniform profile in the absolute frame of reference (the **Absolute** option is selected).		

📖*Relative to rotating frame. When the **Relative to rotating frame** option is selected, the specified velocity (Mach number) is assumed to be relative to the rotating reference frame (Vr):*

$$V_{specified} = V_r = V_{abs} - \omega \times r$$

Here, r is the distance from the rotation axis and ω is the angular velocity of the rotating frame. The mass or volume flow rate specified in the rotating reference frame (the Relative to rotating frame option is selected) will be the same in the absolute (non-rotating) frame of reference if the tangential velocity component is perpendicular to the opening's normal, thus not influencing the mass (volume) flow rate value, e.g. when the opening's normal coincides with the rotation axis.

Type	**Environment Pressure**	
Name	**Environment Pressure 1**	
Faces to apply	the inner face of the **Outlet Lid**	
Parameters: Default values (101325 Pa and 293.2 °C) in the absolute frame of reference (the **Pressure potentia**l option is disabled).		

📖*Pressure potential*. *If you enable a rotating reference frame, you can select the* **Pressure potential** *check box. When the Pressure potential check box is selected, the specified static pressure is assumed to be equal to the rotating frame pressure (P_r) and may be calculated using following parameters: absolute pressure, density, angular velocity and radius:*

$$P_{specified} = P_r = P_{abs} - \frac{1}{2}\rho\omega^2 \cdot r^2$$

When the **Pressure potential** *check box is unchecked, the specified static pressure is assumed to be a pressure in terms of the absolute frame of reference (P_{abs}).*

When you specify a rotating reference frame, it is assumed that all model walls are rotated with the reference frame's angular velocity unless you set a specific wall to be stationary. To specify a non-rotating wall, the Stator moving wall boundary condition can be applied to this wall. Specifying the stator boundary condition is the same as specifying the zero velocity of this wall in the absolute (non-rotating) frame of reference. Note that stator face must be axisymmetric with respect to the rotation axis.

Specifying Stationary Walls

We will specify the stator condition at the corresponding walls of the pump's cover.

1   In the FeatureManager Design tree, select the **Cover** component.

2   Click **Flow Simulation**, **Insert**, **Boundary Condition**.

3   Click **Filter Faces** 🔽 and select **Remove outer faces** 🟧 and **Keep outer faces and fluid-contacting faces** 🟩 options. Click **Filter**.

4   Click **Wall** 〰️ and keep the default **Real Wall** condition type.
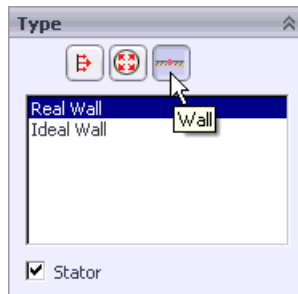
5   Select **Stator**.

6   Click **OK** ✅ and rename the new `Real Wall 1` condition to `Stator Walls`.

Impeller's Efficiency

Engineers dealing with pump equipment are interested in the pump efficiency. For the pump under consideration the efficiency (η) can be calculated in the following way (F.M.White "Fluid Mechanics", 3rd edition, 1994):

$$\eta = \left| \frac{\left(P_{outlet} - P_{inlet} \right) \cdot Q}{\Omega \cdot M} \right|$$

where P_{inlet} is the static pressure at the pump's inlet, P_{outlet} is the bulk-average static pressures at the impeller's outlet (Pa), Q is the volume flow rate (m^3/s), Ω is the impeller rotation angular velocity (rad/s), and M is the impeller torque (N·m). To obtain P_{outlet}, an auxiliary **Measure** component was placed where the flow exits the impeller.

The **Measure** component is only used for the pressure measurement (the corresponding goal will be specified at the inner face of the **Measure** thin ring), thus it should be disabled in the **Component Control** dialog box.

1 Click **Flow Simulation**, **Component Control**.

2 Select the **Measure** item and click **Disable**.

3 Click **OK** to close the dialog.
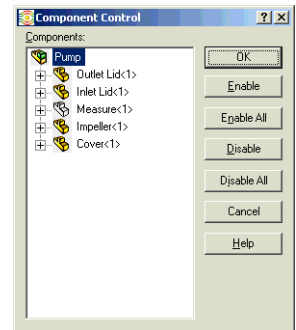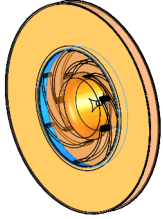
Specifying Project Goals

First, since the pressure and volume flow rate boundary condition are specified, it makes sense to set the mass flow rate surface goal at the pump's inlet and outlet to inspect the mass balance as an additional criterion for converging the calculation.

GOAL TYPE	GOAL PARAMETER	FACE
Surface Goal	Mass Flow Rate	The inner face of the **Inlet Lid**
Surface Goal	Mass Flow Rate	The inner face of the **Outlet Lid**

Next, specify the goals that are necessary for calculating the impeller's efficiency:

GOAL TYPE	GOAL PARAMETER	FACE
Surface Goal	Av Static Pressure	The inner face of the **Inlet Lid**

Surface Goal	Bulk Av Static Pressure	The inner face of the **Measure** ring at the impeller's outlet.	
Surface Goal	Z - Component of Torque	All impeller faces in contact with air (see details below).	

To avoid manual selecting of all impeller's faces in contact with air (more than 150) we will use the **Filter Faces** feature.

1   Select the **Impeller** component by clicking on it in the graphic area or in the FeatureManager Design tree.

2   Click **Flow Simulation**, **Insert**, **Surface Goals**. All impeller faces (including those we do not actually need) appear in the **Faces to Apply the Surface Goal** list.

3   Click **Filter Faces** and select **Remove outer faces** and **Keep outer faces and fluid-contacting faces** options.

📖 *When several options are selected in the faces filter, the filter options to exclude certain faces are combined with the use of logical AND, so that the combination of* **Remove outer faces** *and* **Keep outer faces and fluid-contacting faces** *leads to the removal of all faces but those in contact with fluid.*

4   Click **Filter**.

Rename the created goals as shown below:
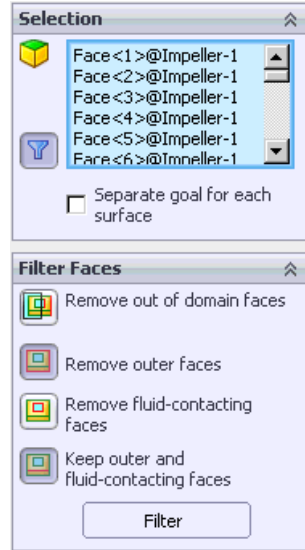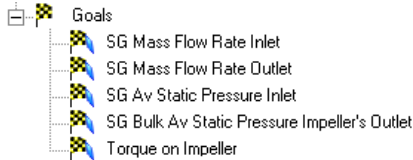
Finally, specify the following Equation goals:

GOAL NAME	FORMULA	DIMENSIONALITY
Pressure Drop	`{SG Av Static Pressure Inlet}-{SG Bulk Av Static Pressure Impeller's Outlet}`	Pressure & stress
Efficiency	`{Pressure Drop}*{Inlet Volume Flow 1:Volume flow rate normal to face:3.000e-001}/209.44/{Torque on Impeller}`	No units

To add inlet volume flow value to the equation goal's expression, click the
Inlet Volume Flow 1 item in the Analysis tree and then click **Volume flow rate normal to face** in the **Parameter list**.

SG Bulk Av Static Pressure Impeller's Outlet
Torque on Impeller
Pressure Drop
Efficiency
Results
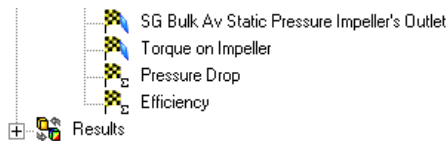
Save the model and run the calculation.

Results

The velocity vectors and static pressure distribution are shown below. To display vectors in the rotating reference frame, select the **Velocity RRF** parameter under the **Vectors** tab of the **Cut Plot** definition window.
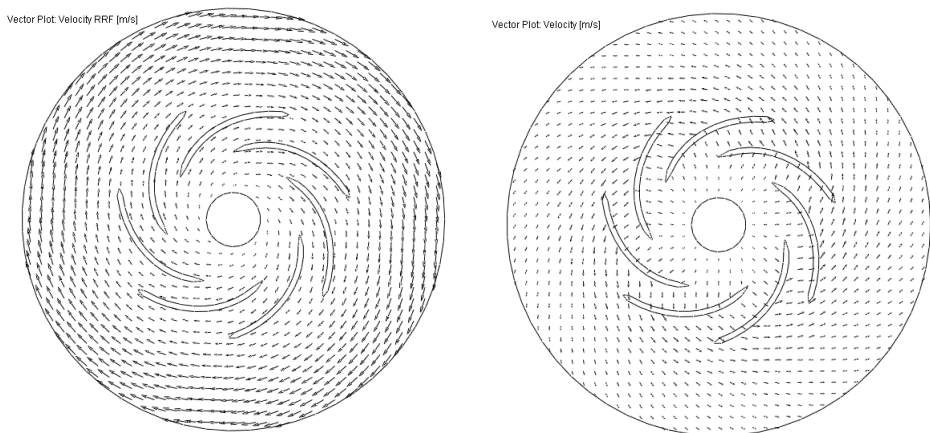
The flow velocity vectors in the frame rotating with the impeller (left) and in the stationary frame (right) at the impeller flow passage midsection (Front plane, position *Z = - 0.02 m*, vector spacing = 0.02 m, arrow size = 0.03 m).
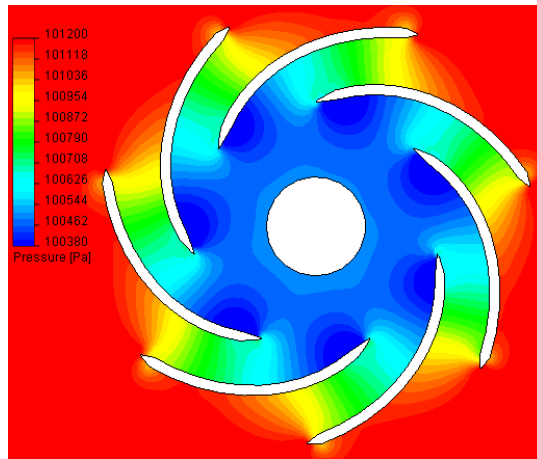
The flow static pressure at the impeller flow passage midsection.
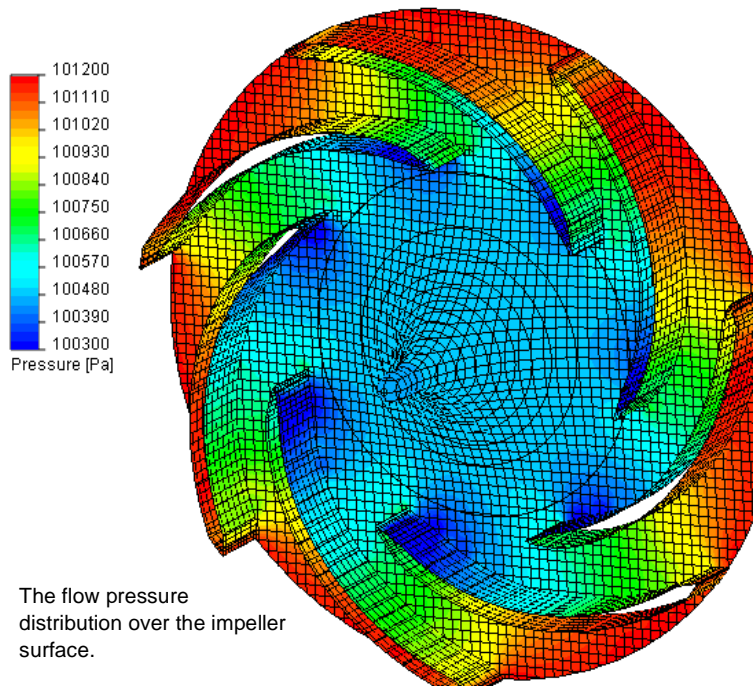
The flow pressure distribution over the impeller surface.

For the impeller under consideration the obtained efficiency is 0.79.

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value
Efficiency	[ ]	0.787039615	0.786371	0.784334	0.787117

CPU Cooler

Problem Statement

Let us consider a CPU cooler consisting of a copper core and an aluminum heat sink with 62 fins. An eight-blade propeller generates a constant flow of air through the heat sink. The CPU is mounted on a socket installed on a PCB. Heat produced by the CPU is transferred through the core to the heat sink and then released into the air flow.

To calculate the problem using Flow Simulation, it is convenient to use the concept of local rotating regions. In order to simplify the problem statement, we do not consider the thermal interface layer between the processor and the cooler. Also, we neglect the thermal conduction through the processor socket and PCB. A quantitative measure of the cooler efficiency is the thermal characterization parameter $\Psi_{CA} = (T_C - T_A)/P_D$, where T_c is the temperature of the CPU cover, T_A is the surrounding air temperature, and P_D is the thermal design power (TDP) of the CPU.
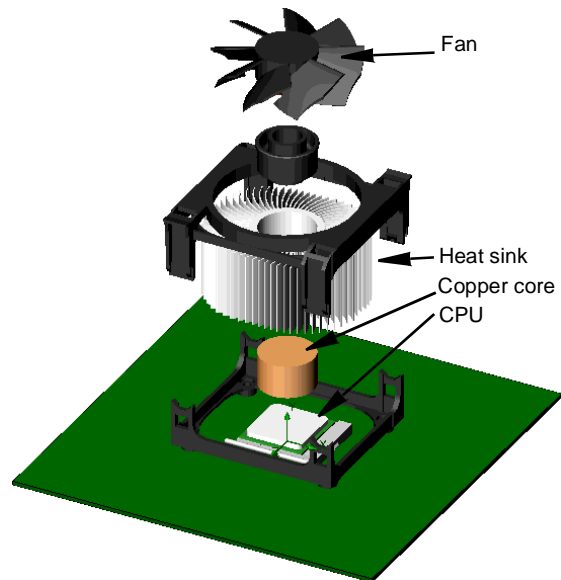
An exploded view of the CPU cooler assembly.

SolidWorks Model Configuration

Copy the **Tutorial Advanced 6 - CPU Cooler** folder into your working directory. Open the **CPU Cooler.SLDASM** assembly.

Project Definition

Using the **Wizard** create a new project as follows:

Project Configuration	*Use current*
Unit system	*SI*
Analysis type	*External; Exclude cavities without flow conditions; Exclude internal space*
Physical features	*Heat conduction in solids;* *Rotation: Type - Local region(s)*
Default fluid	*Gases / Air*
Default solid	*Glasses and Minerals / Insulator*
Wall Conditions	*Default smooth walls*
Initial and Ambient Conditions	*Thermodynamic parameters: Temperature=38°C;* *Solid parameters: Initial solid temperature=38°C;* *other conditions are default*
Result and Geometry Resolution	*Set the Result resolution level to 5; Minimum gap size = 0.001 m, other options are default*

Computational Domain

Specify the computational domain size as follows:

X max = 0.095 m	Y max = 0.1123 m	Z max = 0.095 m
X min = -0.095 m	Y min = 0.0005 m	Z min = -0.095 m

Rotating Region

The **Rotating region** is used to calculate flow through rotating components of model (fans, impellers, mixers, etc.) surrounded by non-rotating bodies and components, when a global rotating reference frame cannot be employed. For example, local rotating regions can be used in analysis of the fluid flow in the model including several components rotating over different axes and/or at different speeds or if the computational domain has a non-axisymmetrical (with respect to a rotating component) outer solid/fluid interface. Each rotating solid component is surrounded by an axisymmetrical rotating region which has its own coordinate system rotating together with the component.

A rotating region is defined by an additional component of the model. This additional component must meet the following requirements:

- the rotating component must be fully enclosed by it,
- it must be axisymmetric (with respect to the rotating component's rotation axis),
- its boundaries with other fluid and solid regions must be axisymmetrical too, since the boundaries are sliced into rings of equal width and the flow parameters' values transferred as boundary conditions from the adjacent fluid regions are circumferentially averaged over each of these rings,
- the components defining different rotating regions must not intersect.

Specify the rotating region as follows:

1 Click **Flow Simulation**, **Insert**, **Rotating Region**.

2 In the flyout FeatureManager design tree select **Rotation Region** component. Note that the **Disable solid components** check box is automatically selected to treat the Rotating Region as a fluid region.

&A component to apply a rotating region must be a body of revolution whose axis of revolution is coincident with the rotation axis. This component must be disabled in the **Component Control**. When specifying the rotating region, make sure that its boundaries do not coincide with the boundaries of other surrounding solid components, because the mesh will not resolve this region. However these components may intersect in some way, but in this case the surrounding components must be also symmetrical relatively to the axis of revolution. Since the flow on the boundary of the rotating region must be axisymmetrical as well, we must provide a reasonable gap between the rotating region boundary and the outer edges of the propeller blades in order to minimize the influence of local non-axisymmetrical perturbations. Due to the same reason, it is preferable to put the rotating region boundary inside the solid bodies whenever possible, rather than putting them in the narrow flow passages. Also, the supposed direction of the flow at the rotating region boundary should be taken into account when defining the shape of the rotating region. You should choose such shape of the rotating region that the flow direction will be as much perpendicular to the rotating region boundary as possible. The picture below provides an additional insight into how the rotating region shape was adapted to the actual geometry of the CPU cooler in this tutorial example (the rotation region boundary is denoted by red).

These gaps are necessary for flow to be more axisymmetrical at the rotating region boundary

By placing the rotating region boundary within a solid instead of putting it into a narrow channel between the fan and the attach clip we avoid the additional mesh refinement and the negative effects of the non-axisymmetrical flow in this narrow channel
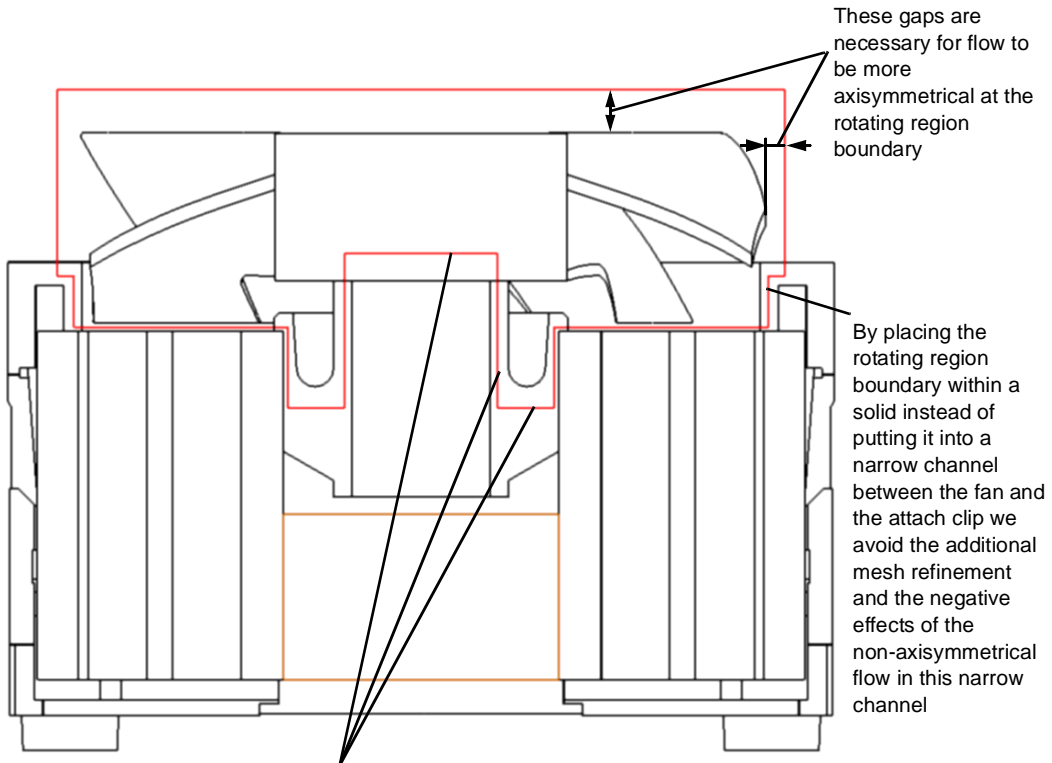
Here the rotating region boundary is placed within a solid to avoid unnecessary and non-realistic calculation of a swirled flow within the closed cavity, which may yield inaccurate results

3 Under **Parameter**, set the **Angular Velocity** to -4400 RPM.

📖 *During the definition of a rotation region, heavy green arrows denoting the rotation axis and the positive direction of rotation speed can be seen in the graphics area. Since we want to define the rotation in the direction opposite to the arrow, we specify negative value of the angular velocity.*
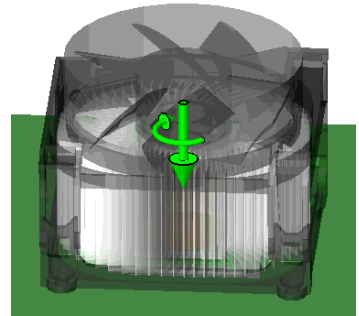
4 Click **OK** .

When you specify a rotating region, it is assumed that all model walls within this region rotate with the region's angular velocity unless you set a specific wall to be stationary. To specify a non-rotating wall, the Stator real wall boundary condition should be applied to the wall. Specifying the stator boundary condition is the same as specifying the zero velocity of this wall in the absolute (non-rotating) frame of reference. Note that the stator face (or a part of the face that is located inside the rotating region in the case when the given face intersects with the rotating region boundary) must be axisymmetric with respect to the rotation axis.

Specifying Stationary Walls

We will specify the stator condition at the appropriate walls of the fan attach and the attachment clip. To easily select the necessary faces, hide the **Fan** and **Rotation Region** components.

1 Click **Flow Simulation**, **Insert**, **Boundary Condition**.

2 Under **Type**, select **Wall** . Keep the default **Real Wall** condition type and select **Stator** option.
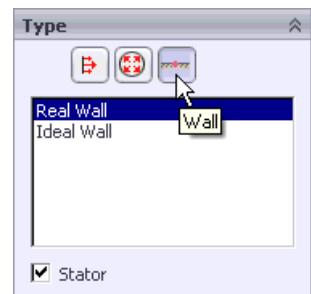
3   Select the two inner circular side faces and two top faces of **Attach Clip** as shown.

4   In the flyout FeatureManager Design Tree select the **Fan Attach** component.

📖 *The **Fan Attach** component has a relatively complex shape with fine features, so it is preferable to select the whole component and then use the faces filter, rather than selecting manually each face we need.*
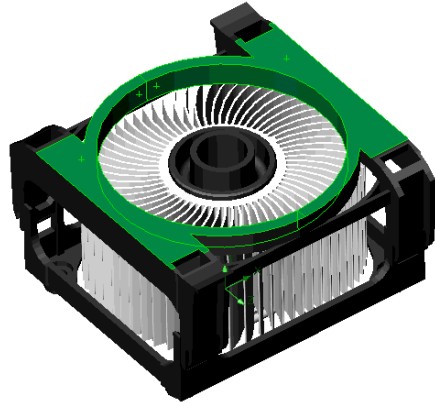
5   Click **Filter Faces** 🔽. Select **Remove outer faces** 🟩 and **Keep outer faces and fluid-contacting faces** 🟩, then click **Filter**.

📖 *Since we have specified the **Exclude internal space** option in the **Wizard**, the faces in contact with the cavity between the **Fan Attach** and the **Copper Core** are considered outer faces. Therefore we need to select the **Remove outer faces** option in **Filter Faces** in order to exclude them.*

6   Click **OK** ✅.

Solid Materials

Specify the solid materials for the project as follows:

a) the **CPU** and the **Heat Sink** are made of aluminum (Pre-Defined/Metals);
b) the **Copper Core**, naturally, is made of copper (Pre-Defined/Metals);
c) all other parts are made of default Insulator.

Heat Source

Define the volume source with the heat generation rate of 75 W in the **CPU** component.

Initial Mesh Settings

To resolve the complex geometry of the fan and heat sink better, let us define six additional control planes and specify the proper **Ratio**s for the intervals between them to make the mesh denser in the central region containing the complex geometry and coarser near the computational domain's boundaries.

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

2   Clear the **Automatic settings** check box.

3 Click **Reset** and then in the **Automatic Initial Mesh** window click **OK**.

4 On the **Basic Mesh** tab, under **Control intervals** select the **0 m** value (either as a **Max** of **X1** interval or as a **Min** of **X2** interval) and click **Delete Plane**.
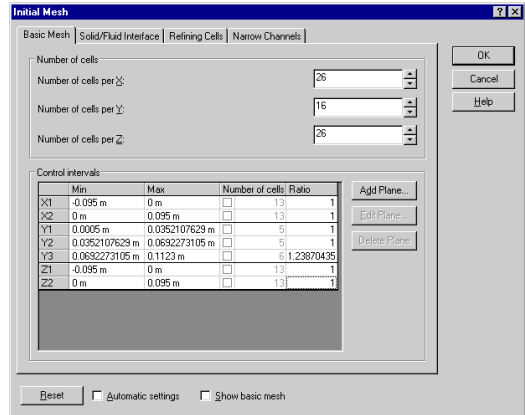
5 Click **Add plane**. In the **Create Control Planes** window make sure that **Creating mode** is set to **Click on screen** and **Parallel to** is set to **YZ**, click anywhere in the graphic area and enter manually -0.05 as a new value for **X**. Click **OK** to return to the **Initial Mesh** window.

6 Following the same procedure, add one more plane at X = 0.05.
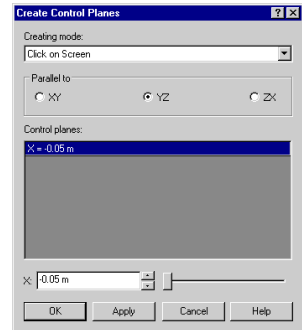
& *By default, Flow Simulation creates six control planes on the computational domain boundaries and a number of planes inside it. We now want to tune the set of control planes to our needs by removing the default planes inside the computational domain and adding new ones.*

7 Click the **Ratio** cell of the **X1** interval and enter the value of 2. In the same manner enter the values 1 and -2 for the intervals **X2** and **X3**.

& ***Ratio** is the ratio of cell sizes on the given interval. The cell sizes are changed gradually along the selected direction so that the proportion between the first and the last cells of this interval is close (but not necessarily equal) to the entered value of the **Ratio**. Negative values of the ratio correspond to the reverse order of cell size increase.*

8 Delete the existing inner control planes perpendicular to Y and add new planes at Y = 0.042 m and Y = 0.047 m. Specify the **Ratio** values for **Y1**, **Y2** and **Y3** intervals as 1.5, 1 and -1.4, respectively.

9 Delete the existing inner control plane perpendicular to Z and add new planes at Z = -0.05 m and Z = 0.05 m. Specify the **Ratio** values for **Z1**, **Z2** and **Z3** intervals as 2, 1 and -2, respectively.

10 Check that the **Numbers of cells per X**, **Y** and **Z** are 26, 12 and 26, respectively. If the numbers are different, please correct them manually.

11 To avoid the unnecessary mesh refinement at the edges of the heatsink fins, go to the **Solid/fluid Interface** tab and set **Small solid features refinement level** to 3, **Tolerance refinement level** to 2, and **Tolerance refinement criterion** to 0.001 m, while leaving other options default.
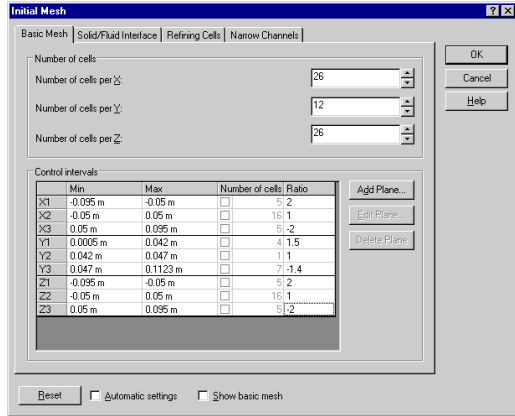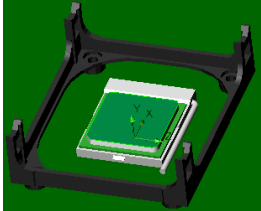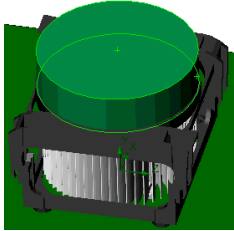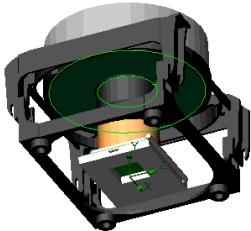
12 Go to the **Narrow Channels** tab and set **Characteristic number of cells across a narrow channel** to 4 and **Narrow channels refinement level** to 1, leaving default values for other options. This will prevent the unnecessary mesh refinement in the narrow channels between heatsink fins.

13 Click **OK**.
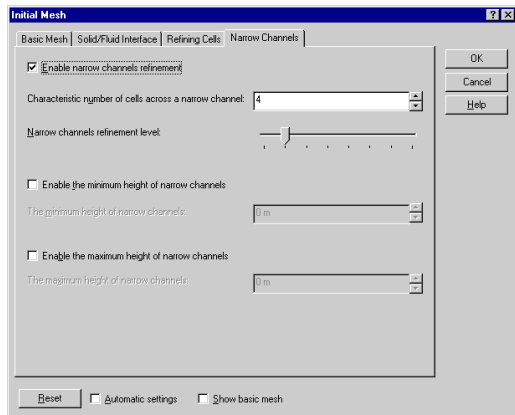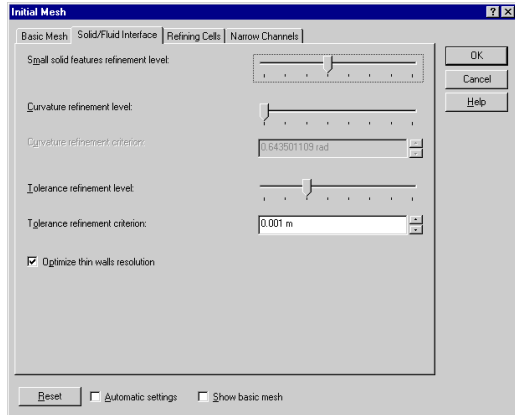
Specifying Project Goals

Specify surface goals for maximum temperature on the CPU cover and mass flow rate for the flows entering the rotating region and exiting from it. To select the necessary faces, you will probably need to hide temporarily some components of the assembly.

GOAL TYPE	GOAL VALUE	FACE	
Surface Goal	Max Temperature of Solid	Top face of the CPU cover. To set this goal you may need to hide the **Heat Sink** and **Copper Core** components.	
Surface Goal	Mass Flow Rate	Top and side surfaces of the **Rotation Region** component.	
Surface Goal	Mass Flow Rate	Bottom face of the **Rotation Region** component. To set this goal you may need to hide the **PCB** component.	
Equation goal	({SG Mass Flow Rate 1}+{SG Mass Flow Rate 2})/{SG Mass Flow Rate 1}	The disbalance of the inlet and outlet mass flow rates. We are using the "+" operand since the inlet and outlet mass flow rate values have opposite signs. Select **No units** for **Dimensionality.**	

To calculate the thermal characterization parameter we will need the temperature of the center of the CPU cover. To get more accurate value of the parameter we will specify a separate point goal.

1   Click Flow Simulation, **Insert**, **Point Goals.**

2   Click **Point Coordinates** $\boxed{{}^{x}{}_{Y_{Z}}}$.

3   Enter the coordinates of the point: **X** = 0 m, **Y** = 0.009675 m, **Z** = 0 m.

4   Click **Add Point** [icon].

5   In the **Parameter** table, select the **Value** check box in the **Temperature of solid** row.

6   Click **OK** [icon].

PG Temperature of Solid 1
SG Max Temperature of Solid 1
SG Mass Flow Rate 1
SG Mass Flow Rate 2
Equation Goal 1

Save the model and run the calculation.
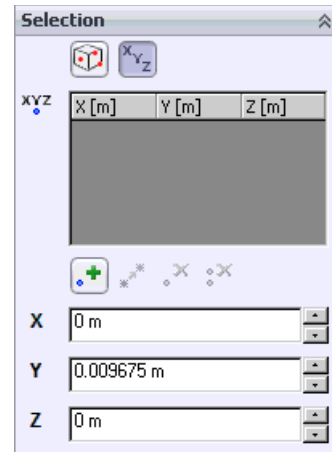
Results

Use the goal plot tool to obtain the value of the temperature of the center of the CPU cover. Now we can calculate the thermal characterization parameter of the heat sink: $\Psi_{CA} = (T_C - T_A)/P_D = (329.9\text{-}311.15)/75 = 0.25$ °C/W. The second most important characteristic of the CPU Cooler is the velocity of the flow above PCB. We can assess the value of this parameter as well as the distribution of the temperature by looking at the cut plots made in the **Front** and **Right** planes (see below).
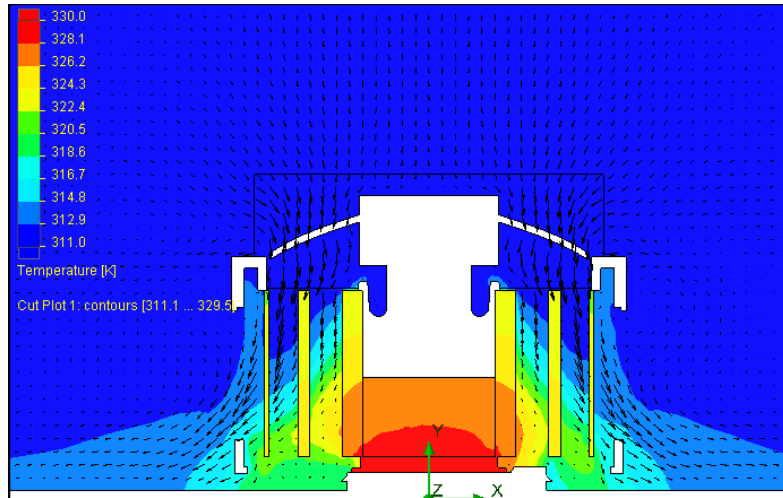
Temperature field and velocity vectors distribution (Front plane, no offset, vector
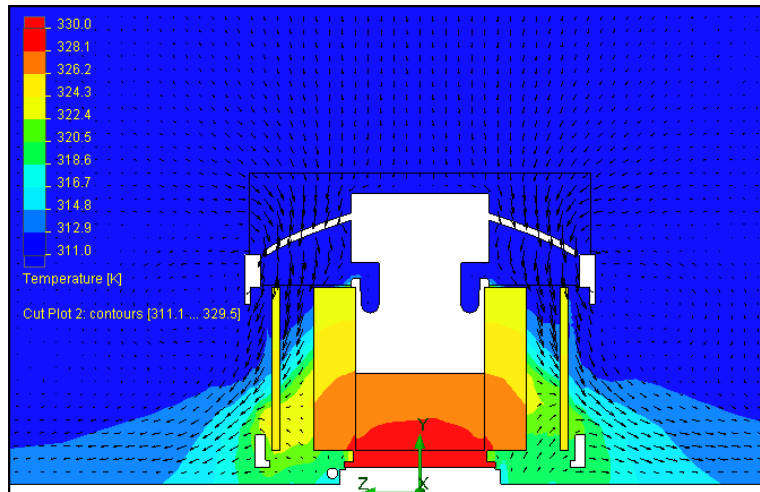
Temperature field and velocity vectors distribution (Right plane, no offset, vector
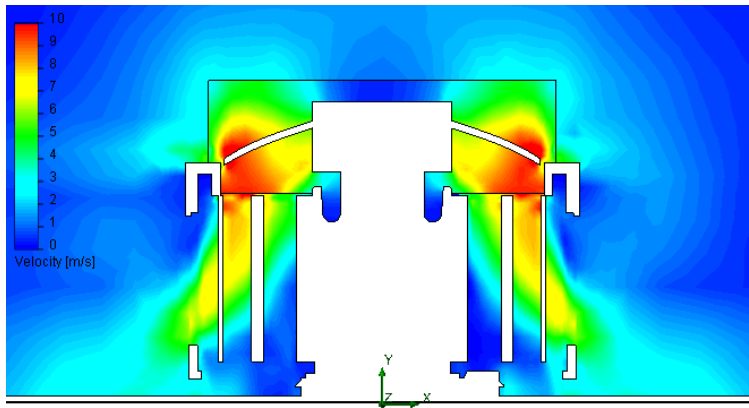
Velocity distribution as a contour plot (Front plane, no offset).
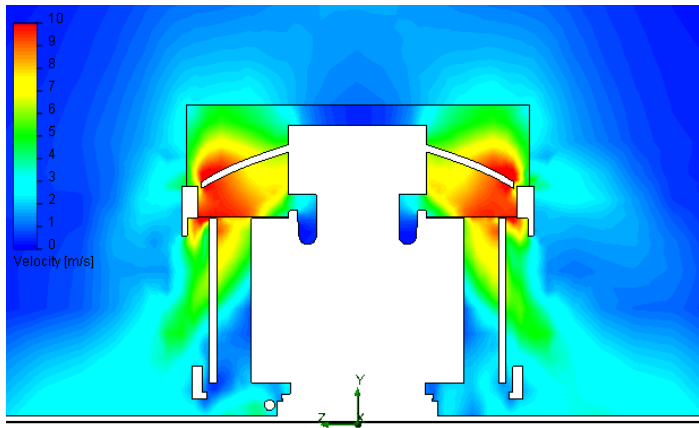
Velocity distribution as a contour plot (Right plane, no offset).

14

Electronic Components

Some of the features used in this tutorial are available for the Electronics module users only.

Problem Statement

This tutorial demonstrates the capabilities of Flow Simulation to simulate cooling of electronic components in an embedded industrial computer by using various features implemented in the **Electronics** module. Here we consider a single board computer with a case, which contains, among other components, CPU, chipset (Northbridge and Southbridge), heat sink with two heat pipes, PCI and ISA slots for a PC104 expansion board, SODIMM slot with memory installed and peripheral connectors.

Air at room temperature enters the case through the vents located at the side and bottom panels and exits through the vents located at the back panel, where an exhaust fan is installed. The resulting flow inside the case removes the heat produced by electronic components (CPU, Northbridge, Southbridge and DDR RAM chips). The heat pipes also transfer the heat produced by CPU and Northbridge to the heat sink, which dissipates it into the air. In the considered model, this heat sink is placed near the exhaust fan.

The objective of the simulation is to ensure that under these conditions, electronic components operate at moderate temperatures. In the table below, you can see the typical values of maximum operating temperatures of the electronic components under consideration.

Electronic component	Maximum operating temperature
CPU	85 °C
Northbridge	80 °C
Southbridge	100 °C
DDR RAM chip	85 °C

Heatsink

Exhaust Fan

CPU Heat Pipe

Back Vents

CPU

Chipset Heat Pipe

Side Vents

Northbridge

Peripheral
Connectors

Southbridge

EPIC PC Mainboard

SODIMM Memory Module

Bottom Vents
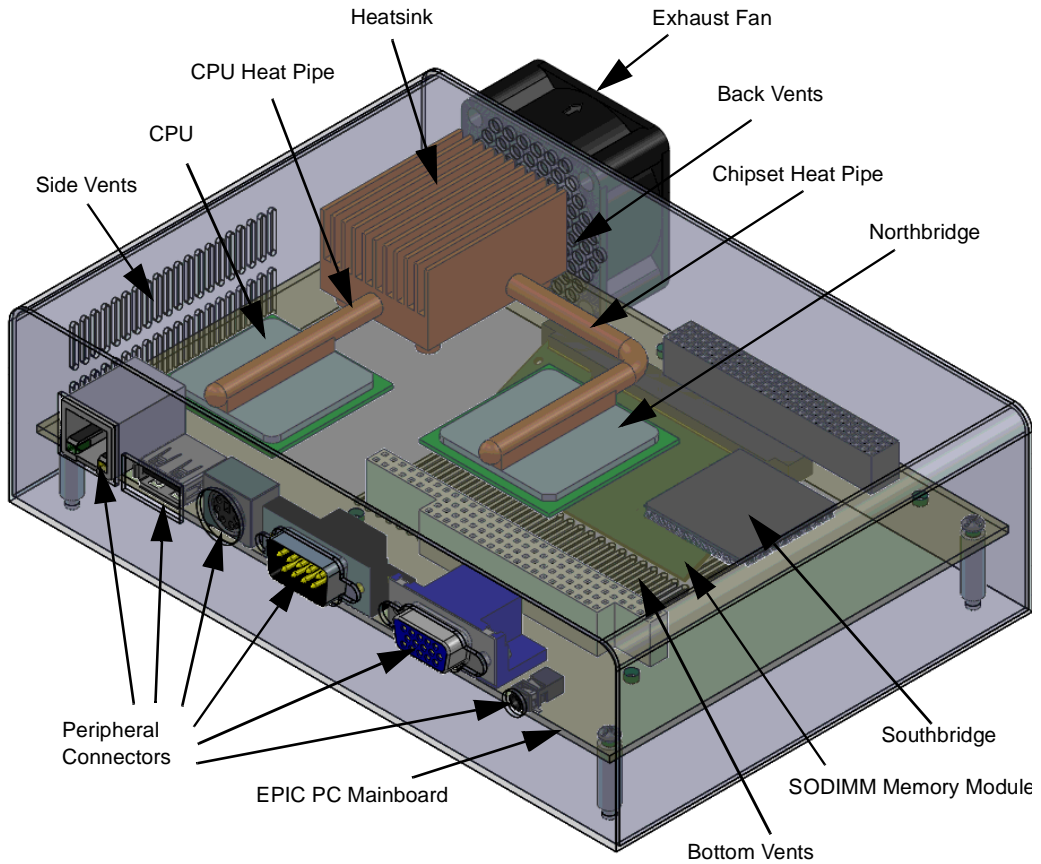
Model Configuration

Copy the **Tutorial for Electronics Module 1 - Electronic Components** folder into your
working directory. Open the **EPIC PC.SLDASM** assembly. Look at the **Default**
configuration. This is the original model geometry in accordance with the problem
statement. After studying this model, switch to the **Simulation Model**.

CPU Heat Pipe   Heat sink

CPU Two-Resistor Model

Side Inlet

Outlet Lid

Chipset Heat Pipe

Northbridge
Two-Resistor Mode

EPIC PC Mainboard

Bottom Inlet Lid

Southbridge
Two- Resistor Model

SODIMM Memory Module
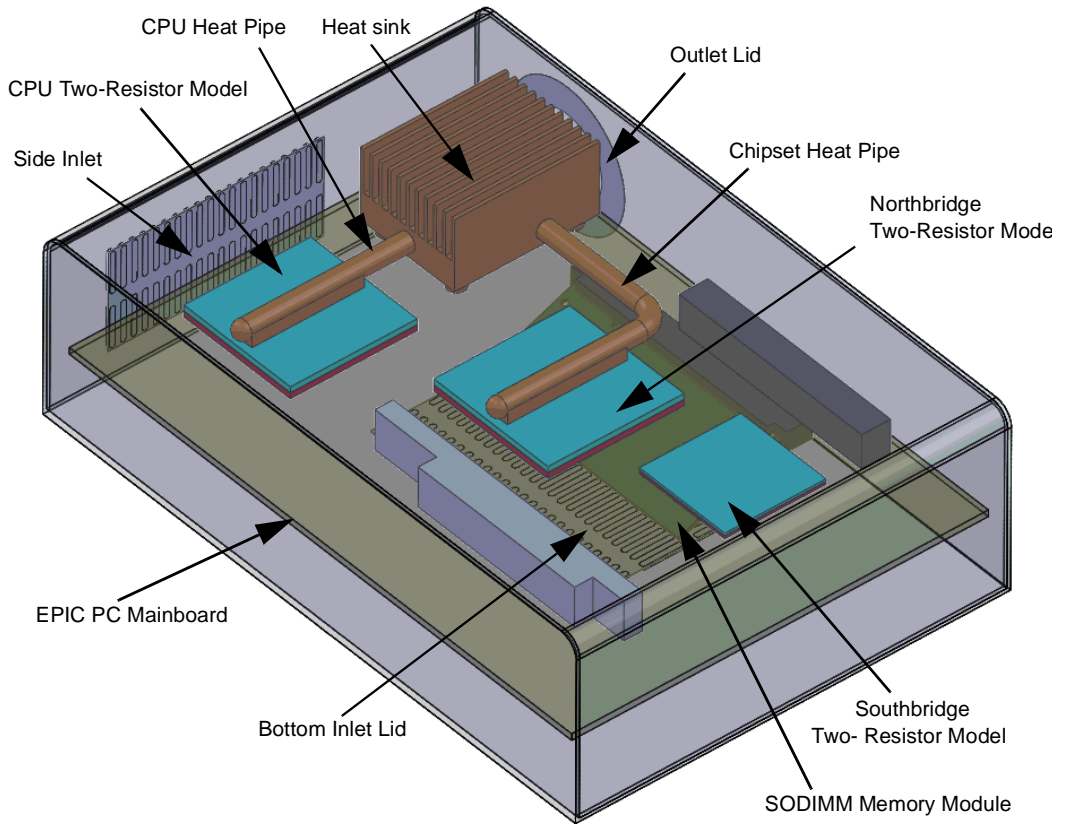
Simulation model

To simplify the problem for this tutorial and, therefore, to save your computer resources, we neglect some particular components and features, which do not affect the flow and heat exchange much. These include holes in PCI and ISA slots, screws and peripheral connectors. The model geometry of exhaust fan is also excluded from the simulation and is replaced by an appropriate boundary condition. In the simulation, we consider CPU, Northbridge, Southbridge and DDR RAM chips as Two-Resistor simplified thermal models, each consisting of two parallelepiped components.

To set the boundary conditions for the inlet and outlet flows, we close the vents by placing a single lid on the inner side of each panel. Thus, we neglect some phenomena, which occur in the flow entering and exiting the case thought the vents. However, we take into account the value of the pressure loss coefficient reflecting the resistance to the flow in accordance with the specific shape and arrangement of the vent holes.

In the **Simulation model** configuration you can see that the vents on the back panel are suppressed . This is done in order to define the exhaust fan boundary condition correctly. If you examine the original model geometry, you will see that the exhaust fan is placed close to the vents on the back panel, and there is no air flow through some of them. Actually, the air flow exits the case through a ring-shaped array of the vent holes (see the picture), so in the **Simulation Model** configuration we place a lid to close only these vent holes without considering other vent holes on the back panel at all. As resolving of each vent hole can be rather time-consuming and they are not the part of the flow simulation anyway, we suppress them. Instead, we specify an External Outlet Fan boundary condition on the inner surface of the ring-shaped lid. In addition, on the same lid we specify the Perforated Plate condition to define the pressure loss due to the resistance of the vent holes to the flow.
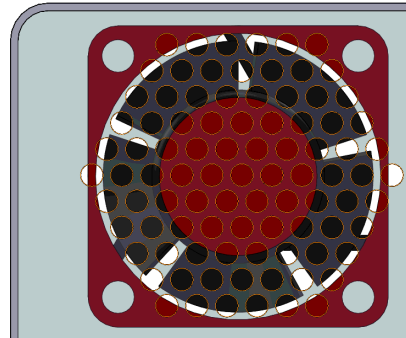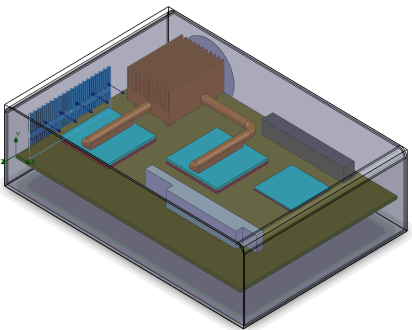
Project Definition
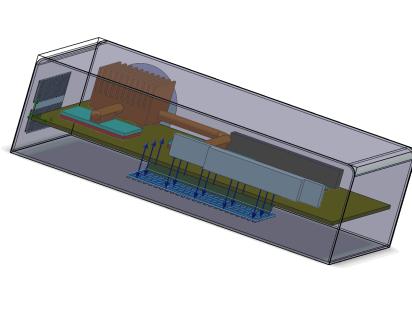
Using the **Wizard** create a new project as follows:
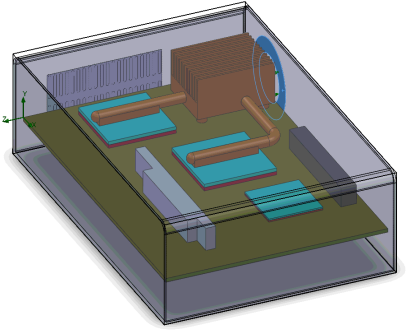
Project Configuration	*Use current*
Unit system	*SI (with Temperature units changed to °C)*
Analysis type	*Internal, Exclude cavities without flow conditions*
Physical features	*Heat conduction in solids,* *Gravity (Y component of -9.81 m/s^2)*
Default fluid	*Gases / Air*
Default solid	*Alloys / Steel (Mild)*
Wall Conditions	*Default outer wall thermal condition: Heat transfer coefficient of 5.5 W/m^2/K*
Initial Conditions	*Default conditions*
Result and Geometry Resolution	*Result resolution level of 3; other options are default*

Boundary Conditions

Specify the boundary conditions for inlet and outlet flows as shown in the tables below:

Type	**Environment Pressure**
Name	**Environment Pressure 1**
Faces to apply	the inner face of the **Inlet Lid**
Parameters: Default values (101325 Pa and 20.5 °C)	

Type	**Environment Pressure**
Name	**Environment Pressure 2**
Faces to apply	the inner face of the **Inlet Lid 2**
Parameters: Default values (101325 Pa and 20.5 °C)	

Type	**External Outlet Fan**
Name	**External Outlet Fan 1**
Faces to apply	the inner face of the **Outlet Lid**
Model: Pre-Defined\Axial\Papst\Papst 412 Parameters: Default value (101325 Pa)	

Specifying Perforated Plates

The **Perforated Plate** feature is used for simulating inlet and outlet flows through thin planar walls with multiple openings without having to create an individual lid for each opening. Instead, the **Perforated Plate** condition is applied together with a boundary condition for a surface of a single lid, which closes multiple openings, and defines the additional resistance of these openings to the flow. It can be useful, for example, when you simulate a flow entering or leaving the model through a series of small openings, which can require some additional mesh refinement if resolved directly. In this simulation, we use **Perforated Plates** to take into account the resistance of inlet and outlet vents in the computer case to the flow .

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

2   In the **Engineering Database**, under **Perforated Plates, User Defined**, create two items with the following parameters:

Property	Value
Name	Tutorial rectangular holes
Comments	
Hole shape	Rectangular
Height	0.01 m
Width	0.0015 m
Coverage	Pitch
X - pitch	0.003 m
Y - pitch	0.015 m

Property	Value
Name	Tutorial round holes
Comments	
Hole shape	Round
Diameter	0.003 m
Coverage	Checkerboard Distance
Distance between centers	0.004 m

📖 *You can specify the **Hole shape** as **Rectangular**, **Round**, **Regular Polygon** or **Complex**. To define the holes arrangement (for non-**Complex** holes), in the **Coverage** you can select either **Pitch** or **Checkerboard distance** (for non-**Rectangular** holes). Depending on the selected option, you can specify the size of a single hole and either the distance between two adjacent holes in two mutually perpendicular directions (**X - Pitch** and **Y - Pitch**) or the **Distance between centers**. The specified values are used to calculate **Free area ratio**, which denotes the ratio of the holes total area to the total*
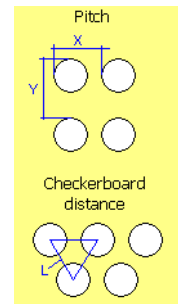
*area of the perforated plate. The automatically calculated **Free area ratio** value appears at the bottom of the table. Alternatively, you can select the **Free area ratio** option in the **Coverage**, and specify this value directly.*

3 Save and exit the **Engineering Database**.

4 In the **Analysis tree**, select the **Environment Pressure 1** boundary condition.

5 Click **Flow Simulation**, **Insert, Perforated Plate**.

6 Under **Perforated Plate** select the created **Tutorial rectangular holes**.

7 Click **OK** ✅ . The new **Perforated Plate 1** item, which corresponds to the side vents of the case, appears in the **Analysis tree**.

8 For the bottom vents, select the **Environment Pressure 2** boundary condition, and repeat steps 5-7.

9 For the back panel vents, select the **External Outlet Fan 1** condition, and repeat steps 5-7, selecting the **Tutorial round holes** item under **User Defined**.
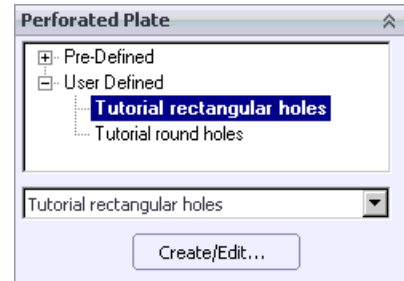
📖 *The **Perforated Plate** feature is used in simulation to define additional parameters for the already specified **Environment Pressure** or **Fan** conditions. It doesn't make any changes in the model geometry itself. So, when you delete the boundary condition or fan from your project, the corresponding **Perforated Plate** (if specified) becomes useless.*

Specifying Two-Resistor Components

The two-resistor model is widely used to estimate the temperature of chips and other small electronic packages. It considers each package consisting of two parallelepiped components (Case and Junction) with identical contact areas, made of material with high heat conductivity (see the picture below). The Junction-to-Case (θ_{JC}) and Junction-to-Board (θ_{JB}) thermal resistances are modeled as infinitely thin plates. The heat conduction through the package is calculated using the values of these resistances.
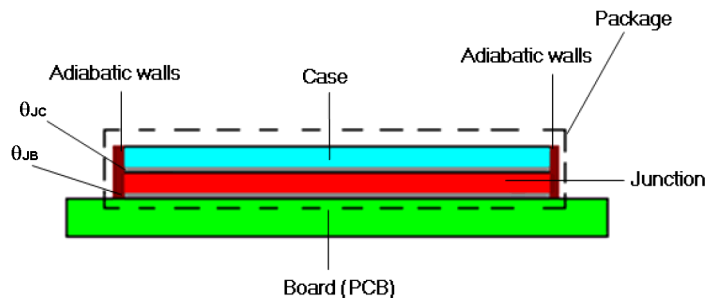
An extensive set of pre-defined two-resistor components is provided in the **Engineering Database**. Each item corresponds to a specific package type.

1   Click **Flow Simulation**, **Insert, Two-Resistor Component**.

2   Select the **CPU 2R Case** component as **Case Body** and **CPU 2R Junction** as **Junction Body**.

3   Under **Component** select the **PBGAFC_35x35mm_2R** item.

4   In the **Source**, enter the value of **Heat Generation Rate** Q equal to `12 W`.

5   Click **OK** . The new **Two-Resistor Component 1** item, which corresponds to CPU, appears in the **Analysis tree**.

6   Rename the created item to **CPU**. We will use this name for selecting this item to specify **Goals**.

7   In the same way specify the **Chipset – Northbridge** and **Chipset – Southbridge** items with the following parameters:
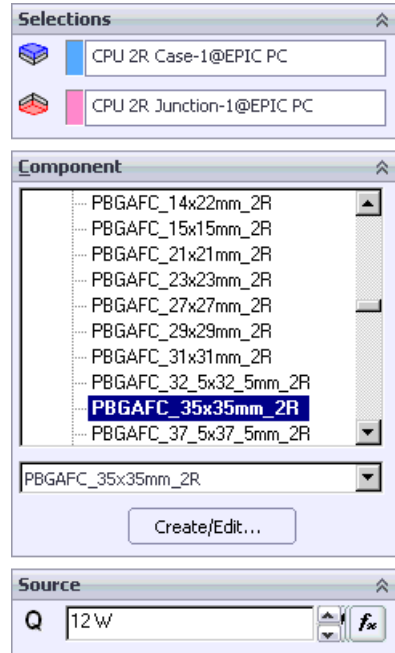
Name	Chipset - Northbridge
Case Body	Northbridge 2R Case
Junction Body	Northbridge 2R Junction
Component	PBGAFC_37_5x37_5mm_2R
Heat Generation Rate	4.3 W

Name	Chipset - Southbridge
Case Body	Southbridge 2R Case
Junction Body	Southbridge 2R Junction
Component	LQFP_256_28x28mm_2R
Heat Generation Rate	2.5 W

8 For each of the four considered DDR RAM chips, specify the same way **RAM chip N** item (with **N** being the chip number) by selecting its corresponding Case and Junction parts under the **SODIMM** assembly:

Name	RAM chip N
Case Body	**RAM Chip 2R Case**
Junction Body	**RAM Chip 2R Junction**
Component	**TSOP_C_10_16x22_22_2R**
Heat Generation Rate	`1 W`

 *If you specify some package as **Two-Resistor Component** in the project, make sure that its dimensions in the **Engineering Database** totally match (or are very close to) the dimensions of the package model geometry (its Case and Junction components). If the dimensions do not match, you must either make changes in the model geometry or select a different **Two-Resistor Component** in the **Engineering Database**.*

Specifying Heat Pipes

The **Heat Pipe** feature is used for modeling heat transfer from the hotter surface to the colder surface through a heat pipe (considered as solid body made of high heat-conducting material).

1 Click **Flow Simulation**, **Insert, Heat Pipe**.

2 Select **CPU Heat Pipe** as **Components to Apply Heat Pipe** .

3 Select the face of the **CPU Heat Pipe** component contacting with the top face of CPU as **Heat In Faces** .

4 Select the face of the **CPU Heat Pipe** contacting with the inner face of heat sink as **Heat Out Faces** .
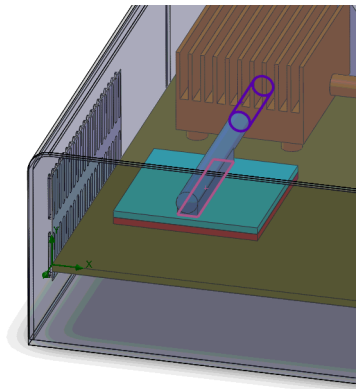
5 Type the **Effective Thermal Resistance** value of
0.3 °C/W. This value models the real efficiency
of heat pipe.

6 Click **OK** ✔ . The new **Heat Pipe 1** item, which
corresponds to the CPU heat pipe, appears in the
Analysis tree.

7 In the same way specify the other heat pipe using
the **Northbridge Heat Pipe** component with the
same value of **Effective Thermal Resistance**.
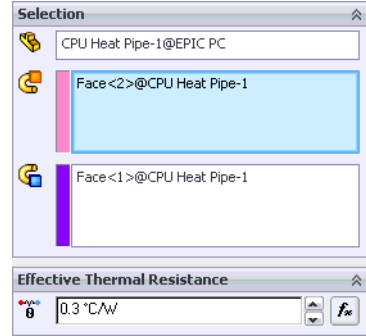
Specifying Contact Resistances

The **Contact Resistance** feature is used for specifying the value of thermal contact
resistance on a face of a solid contacting fluid or another solid. It can be defined by a
specific thermal resistance value or by thickness and thermal properties of the contact
layer material. Taking into account the thermal contact resistance helps to estimate, for
example, such phenomenon as temperature drop at the contact surface. Here we use this
feature to specify thermal interface material attaching heat pipes to CPU and Northbridge
and to specify thermal contact resistance between the surfaces of heat pipes and the
surrounding air.

With the **Library** license, an extensive set of pre-defined thermal interface materials from
several vendors is provided in the **Engineering Database**.

1 Click **Flow Simulation**, **Insert, Contact
Resistance**.

2 Select the faces of the **CPU Heat Pipe** and
Northbridge Heat Pipe components contacting
with the top faces of CPU and Northbridge
correspondingly. We selected these faces earlier as
Heat In Faces to specify the heat pipes.

3 Under **Thermal Resistance,** select **Bond-Ply 660
@ 10 psi** (**Pre-Defined\Interface
Materials\Bergquist\Bond-Ply\Bond-Ply 660 @
10 psi**).

4 Click **OK** ✔ . The new **Contact Resistance 1**
item appears in the **Analysis tree**.

5 Repeat step 1, then hold down the **Ctrl** key and click the **CPU Heat Pipe** and
Northbridge Heat Pipe components in the flyout FeatureManager design tree. All the
faces of both these components appear in the **Faces to Apply the Contact Resistance**
🟨 list.
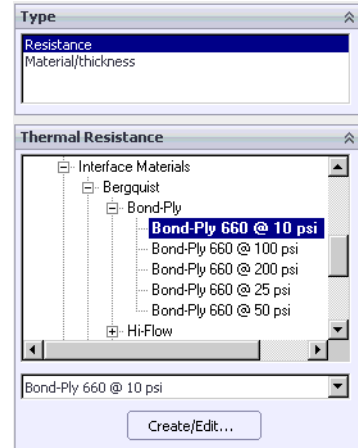
6 Click **Filter Faces** 🔽. Select **Keep outer and fluid-contacting faces** 🔲, and click **Filter**.

7 Under **Thermal Resistance**, expand the **Pre-Defined** list, and select **Infinite resistance**. We use **Infinite resistance** here to reflect the qualitative difference between the intensity of heat transfer inside and outside the considered heat pipes.

8 Click **OK** ✅.

Specifying Printed Circuit Board

The **Printed Circuit Board** feature is used for modeling PCBs as flat solid bodies with anisotropic thermal conductivity, which is calculated from the specified structure of interleaving conductor and dielectric layers. You can define such material in the **Engineering Database** by specifying the properties of conductor and dielectric materials and the structure of layers. We use this feature to specify the material for SODIMM board, which consists of six layers of conductor (Copper) and five layers of dielectric (FR4)

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

2 In the **Engineering Database**, under **Printed Circuit Boards, User Defined**, create a new item with the following parameters:

Property	Value
Name	4s2p PCB
Comments	...
Type	Layer Definition
Dielectric material density	1200 kg/m^3
Dielectric material specific heat	880 J/(kg*K)
Dielectric material conductivity	0.3 W/(m*K)
Conductor material density	8960 kg/m^3
Conductor material specific heat	385 J/(kg*K)
Conductor material conductivity	401 W/(m*K)
PCB total thickness	0.001 m
Conducting layers	(Table) ...

📖 *As you specify the parameters, at the bottom of this table you can see the calculated properties of the equivalent material used in the simulation .*

3   In the **Conducting Layers** table, click the **...** button to switch to the **Tables and Curves** tab. Type the following values to specify the structure of conducting layers:

Layer Thickness	Percentage Cover
3.3e-005 m	20 %
6.6e-005 m	80 %
3.3e-005 m	20 %
3.3e-005 m	20 %
6.6e-005 m	80 %
3.3e-005 m	20 %

As you specify the layers structure, you can see the graphical representation of this structure at the right .

4   Save and exit the **Engineering Database**.

5   Click **Flow Simulation**, **Insert, Printed Circuit Board**.

6   Select **SODIMM PCB** in the graphic area.

7   Under **Printed Circuit Board** select the created **4s2p PCB** item.

8   Click **OK** .
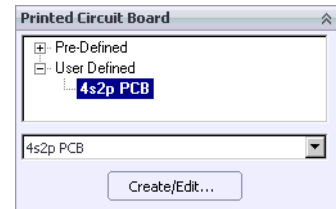
Specifying Solid Materials

For the **EPICPCB** component, we specify a non-isotropic material (**Pre-Defined\Non-isotropic\PCB 8-layers**) with Axisymmetrical/Biaxial conductivity. In this type of conductivity the thermal properties of the material are the same for two directions and differ for the third direction specified by an axis or direction.

To specify the axis for the **EPICPCB**, under **Anisotropy**, set **Axis** to **Y**.
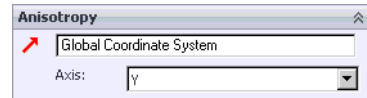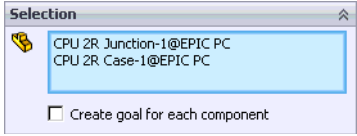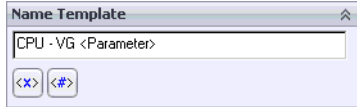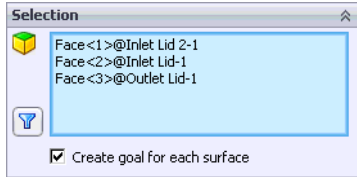
For other components, specify the **Solid Material** as following:

Pre-Defined\Metals\Copper	Heatsink
Pre-Defined\IC Packages\Typical Connector	PC104 PCI Connector, PC104 ISA Connector, SODIMM Connector

To exclude the **Inlet Lid**, **Inlet Lid 2** and **Outlet Lid** from the heat conduction analysis, specify them as insulators (**Pre-Defined\Glasses and Minerals\Insulator**).

Project Goals

1 In the **Analysis tree**, select the two-resistor component **CPU**.

2 Click **Flow Simulation**, **Insert, Volume Goals**.

3 In the **Components to Apply the Volume Goal**
list, make sure that both Case (**CPU 2R Case**) and
Junction (**CPU 2R Junction**) components are
added.

4 Under **Parameter** select both **Max** and **Av Temperature of Solid**.

5 Edit the **Name Template** to: CPU - VG
<Parameter>.

6 Click **OK** ✓ .

7 Repeat the same steps separately for each heat source: **Chipset - Northbridge**,
Chipset – Southbridge, **RAM Chip 1**, **2**, **3**, **4** (select all these four RAM chips at
once) and the **Heatsink**. Edit the **Name Template** in a similar way.

8 When finished, in the **Analysis tree** select all
specified boundary conditions (**Environment
Pressure 1**, **Environment Pressure 2** and
External Outlet Fan 1), holding down the **Ctrl**
key.

9 Click **Flow Simulation**, **Insert, Surface Goals**.

10 Select the **Separate goal for each surface** option to create a separate goal for each of
the selected surfaces.

11 In the **Parameter**, select **Mass Flow Rate**.

12 Click **OK** ✓ .

Setting Initial Mesh

1 To adjust the initial mesh settings, click **Flow
Simulation**, **Initial Mesh**.

2 Switch off the automatic settings by clearing the
Automatic settings checkbox. Go to the **Basic Mesh**
tab.

3 Click **Add Plane**.

4 In the **Create Control Planes** dialog box make sure
that **Creating mode** is set to **Click on Screen**.
In the **Parallel to**, click **ZX**.
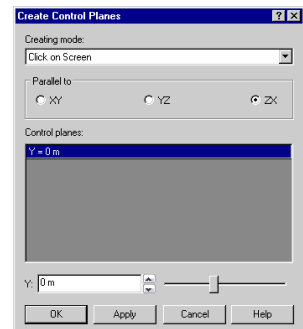
5 In the graphic area, click anywhere, and then type 0 for the **Y**.

6 Click **OK** to return to the **Initial Mesh** window.

7 Click the **Ratio** cell of the **Y1** interval and enter the value of 2.

8 In the same manner enter the **Ratio** value of -2 for the **Y2** interval.

9 Specify the **Number of cells per X, Y and Z** of 40, 15, 30 respectively.

10 Go to the **Solid/Fluid Interface** tab and set **Small solid features refinement level** to 1.

11 Click **OK** to save initial mesh settings.
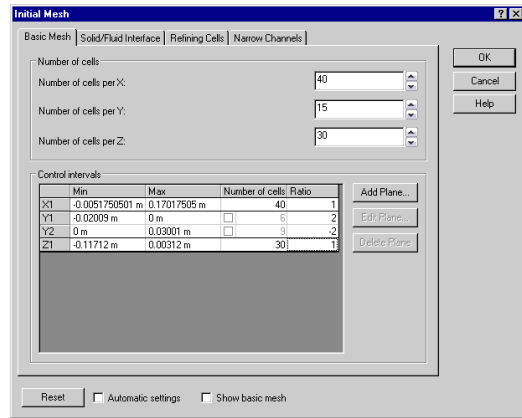
Setting Local Initial Mesh

It is also convenient to specify the **Local Initial Mesh** to obtain more accurate solution in the regions of interest.

1 In the **Analysis tree**, select all created Two-Resistor components (**CPU**, **Chipset – Northbridge**, **Chipset – Southbridge**, **RAM Chip 1**-**4**).

2 Click **Flow Simulation**, **Insert, Local Initial Mesh**.

3 Clear the **Automatic settings** checkbox. Go to **Refining Cells** tab.

4 Select **Refine partial cells** and **Refine solid cells**.

5 Set both **Level of refining solid cells** and **Level of refining partial cells** to 2.

6 Click **OK** to save **Local Initial Mesh** settings.

7 Create another **Local Initial Mesh** for the **Heatsink**. After clearing the **Automatic settings** checkbox, go to the **Narrow Channels** tab, and set **Characteristic number of cells across a narrow channel** to 4 and **Narrow channels refinement level** to 2.

Save the model and run the calculation.

Results

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]
CPU - VG Av Temperature of Solid	[°C]	73.65348951	73.59633002	73.41673831	73.68854902	100
CPU - VG Max Temperature of Solid	[°C]	74.66360454	74.60485091	74.42299428	74.69307706	100
Northbridge - VG Av Temperature of Solid	[°C]	51.14491396	51.1066735	51.05912715	51.14491396	100
Northbridge - VG Max Temperature of Solid	[°C]	51.44302312	51.40441032	51.35484934	51.44302312	100
Southbridge - VG Av Temperature of Solid	[°C]	80.91964681	81.03053274	80.91964681	81.11365414	100
Southbridge - VG Max Temperature of Solid	[°C]	82.79001636	82.88950708	82.79001636	82.966598	100
RAM Chips - VG Av Temperature of Solid	[°C]	66.49844912	66.52817986	66.42244119	66.70378852	100
RAM Chips - VG Max Temperature of Solid	[°C]	70.11750432	70.15496185	70.00624722	70.36198185	100
Heat Sink - VG Av Temperature of Solid	[°C]	42.31832529	42.25710885	42.22057223	42.31832529	100
Heat Sink - VG Max Temperature of Solid	[°C]	42.69643848	42.63542913	42.59875423	42.69643848	100

In accordance with the obtained results, we can say that electronic components operate at moderate temperatures, and there is no need to introduce any additional design features in order to improve the efficiency of heat exchange inside the considered case.
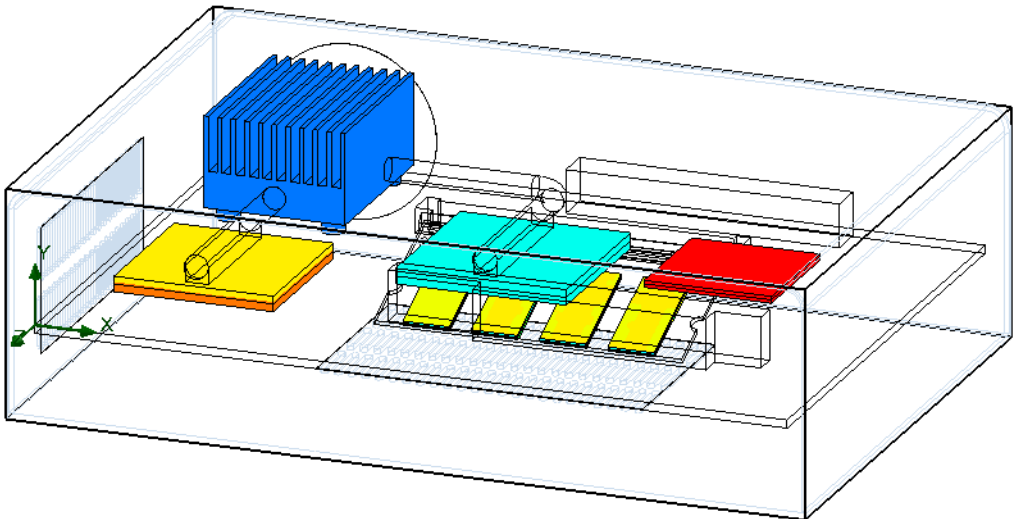
Oil Catch Can

Problem Statement

Here we consider the motion of motor oil droplets in the air flow inside the oil catch can installed in the car. The presence of the droplets in this flow is caused by the rotating crankshaft that churns up oil inside the crankcase. As oil catch can traps these droplets, it eliminates the possibility of oil suction into the engine and its subsequent combustion with fuel and oxidizer (air) that produces a lot of smoke in the exhaust.

For this tutorial we consider the geometry of oil catch can shown on the picture right. The dividing wall is placed so that most of the droplets entering through the inlet nipple along with the air flow collide to it. Once the collision occurs, the oil droplet adheres to the wall and then trickles down. However some particular smaller-sized droplets may evade collision with the wall due to their small inertia and escape the can through the outlet nipple.

The objective of the simulation is to estimate the probability of trapping oil droplets in the oil catch can considering the following droplet sizes: 8, 13 and 18 μm. Quantitatively, we can calculate this probability value for each individual droplet size with the following expression: $P = m_{outlet} / m_{inlet}$,

where m_{inlet}, m_{outlet} is the mass flow rate of oil droplets in the inlet and in the outlet correspondingly. The value of m_{inlet} is set to equal to 0.5% of air mass flow rate.

We assume that oil droplets do not influence the air flow because of their small size and mass ($\sim 10^{-13}$ kg). Therefore, we also neglect the impact of oil accumulation on the flow inside the oil catch can.
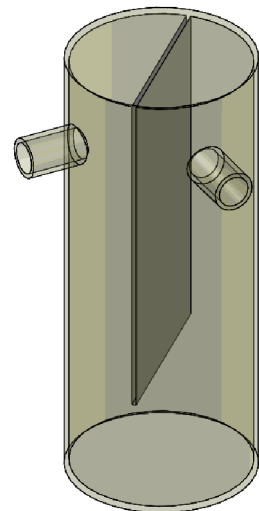
SolidWorks Model Configuration

Copy the **Tutorial Advanced 7 - Oil Catch Can** folder into your working directory. Open the assembly.
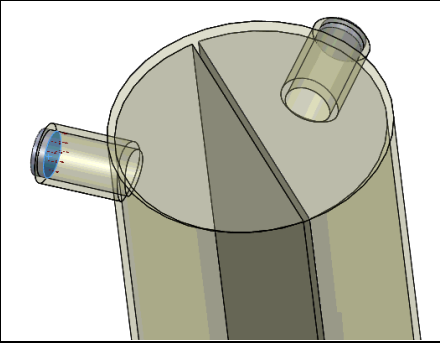
Project Definition
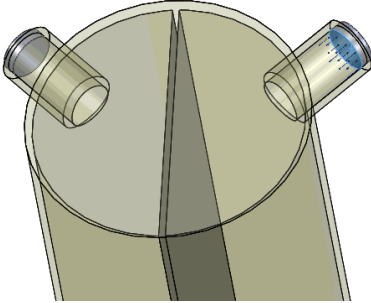
Using the **Wizard** create a new project as follows:

Project Configuration	*Oil particles*
Unit system	*SI*
Analysis type	*Internal*
Default fluid	*Gases / Air*
Wall Conditions	*Adiabatic wall, default smooth walls*
Initial Conditions	*Default conditions*
Result and Geometry Resolution	*Set the Result resolution level to 3; other options are default*

Boundary Conditions

Specify the boundary conditions for inlet and outlet flows as shown in the tables below:

Type	**Inlet Volume Flow**	
Name	**Inlet Air Volume Flow**	
Faces to apply	the inner face of the **Inlet Lid**	
Parameters: Volume Flow Rate Normal To Face with a value of 100 l/min (0.00167 m^3/s)		

Type	**Static Pressure**
Name	**Outlet Static Pressure**
Faces to apply	the inner face of the **Outlet Lid**
Parameters: Default values (101325 Pa and 293,2 K)	

Project Goals

1 In the **Analysis tree**, select the **Inlet Air Volume Flow** boundary condition

2 Click **Flow Simulation**, **Insert, Surface Goals**.

3 Under **Parameter**, select **Av Static Pressure**.

4 Click **OK** ✔. This goal will be an intermediate one to calculate pressure drop through the oil catch can.

5 Click **Flow Simulation**, **Insert, Equation Goal**.

6 In the **Analysis tree**, select the created **SG Av Static Pressure 1** goal. It will appear in the **Expression** box.

7 Click the minus "-" button in the calculator.

8 In the **Analysis tree**, select the **Outlet Static Pressure** boundary condition.

9 In the **Parameter list**, select **Static Pressure**.

10 Select **Pressure & stress** for **Dimensionality.**

11 Click **OK**.

12 Rename the created equation goal to **Pressure Drop**.

13 Click **Flow Simulation**, **Insert, Global Goals**.

14 Parameter select **Y - Component of Torque**, and click **OK** ✔.
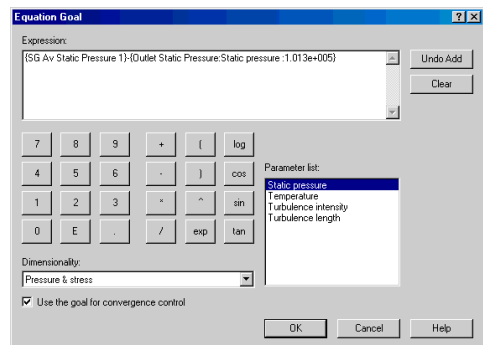
Setting Solution Adaptive Mesh Refinement

With the specified **Result Resolution** value of 3, it may be not sufficient to resolve accurately the regions with large velocity gradients and swirls, which are obviously present here. When analyzing the particles, this may also lead to incorrect predictions of particle trajectories. So, to improve the accuracy of the solution in those regions, it is convenient to perform additional (adaptive) mesh refinement during the calculation.

1 Click , **Calculation Control Options**. Go to the **Refinement** tab.

2 Set the **Refinement** value to **level = 1**.

3 Expand the **Adaptive Refinement in Fluid** item, and click the **Use global parameter variation**.

4 Make sure that the value of the **Refinement Strategy** item is set to **Tabular Refinement**.

5 To edit the table of refinements, first expand the **Refinement Strategy** item and make sure that the value of **Units** is set to **Travels**. Then, click the **...** button in the **Table of refinements** field.

6 In the opened window, click **Add Row**. A single blank row will appear.

7 Enter the value of 2 in the created row. This means that mesh refinement will occur during the calculation when the value of travels reaches 2.

8 Click **OK**. Go to the **Finish** tab.

9 Under the **Finish Conditions**, make sure that the **Minimum refinement number** is set **On**. Edit its value to 1.

10 Set **Off** the **Maximum travels**.

11 Click **OK**.
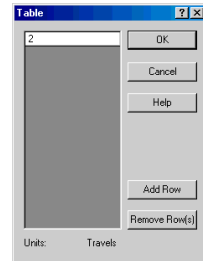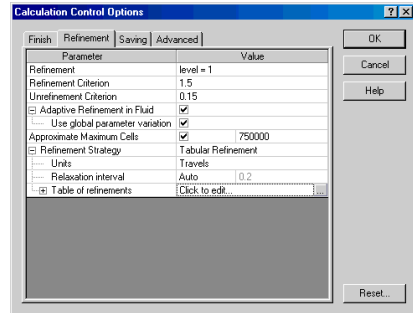
Save the model and run the calculation. During the calculation you can preview the velocity field in the **Front Plane** or other plane and see how mesh refinement improves the final solution.

Specifying Motor Oil Material

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

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

3 Click **New Item** [ ] in the toolbar. The blank **Item Properties** tab appears. Double-click the empty cell to set the corresponding property value.

4 Specify the material properties as shown in the table below:

Name	Tutorial Motor Oil
Density	900 kg/m^3
Dynamic viscosity	0.01 Pa*s
Specific heat	1900 J/(kg*K)
Thermal conductivity	0.2 W/(m*K)

Studying the Motion of Oil Droplets

1 In the Analysis tree, right-click the **Particle Studies** icon and select **Wizard**.

2 Keep the default name for the Particle Study and click **Next** 🔁.

3 Click the **Inlet Air Volume Flow** boundary condition, so that the corresponding face appears under the **Starting Points**.

4 Set the **Number of points** [🔁#] to 200.

5 Under the **Particle Properties**, set the **Diameter** equal to 8e-06 m and change the **Material** to the created **Tutorial Motor Oil** (**Materials, Liquids, User Defined**).

6 Change the **Mass flow rate** value to 1e-05 kg/s. This value is obtained once we take the 0.5% of inlet air mass flow rate (the product of volume flow rate and density) in accordance with the problem statement.

📖*The value set for the **Number of points** reflects the number of different possible trajectories of the considered particles used for tracing. Obviously the larger this value is, the more accurate information about possible particle trajectories can be obtained. As a result, you can obtain a more detailed picture of the particles' distribution in the considered domain and, if necessary, calculate their mass flow rate in the outlet with a higher precision.*
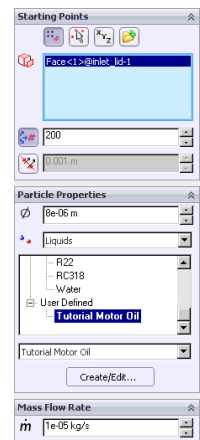
7 Click **Next** 🔁.

8 Under the **Default Wall Condition**, make sure that **Absorption** is selected. Click **Next** ⊕.
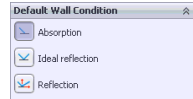
📖 *For the **Particle Study**, there are three types of boundary conditions that can be assigned to the walls:* **Ideal reflection, Absorption** *and* **Reflection**. *The first two indicate perfectly elastic and inelastic collision respectively. In the third type, you have to specify the restitution coefficients that define the ratios of normal and tangential (to the wall) velocity components after and before the collision.*

9 Under **Physical Features**, select **Gravity**. Click **Next** ⊕.
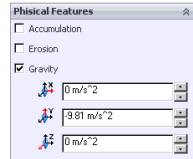
10 Under **Default Appearance**, set **Draw Trajectories as** ✦ **Lines**.

11 Click **OK** ✔. A new **Particle Study 1** item with one sub-item (**Injection 1**) appear in the **Analysis tree**.
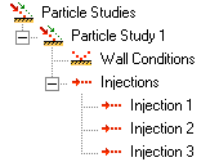
12 Right-click the created **Injection 1** item and select **Clone**. Create this way **Injection 2** and **Injection 3** items.For the **Injection 2** and **Injection 3**, edit the **Diameter** to `1.3e-05 m` and `1.8e-05 m` respectively.

13 Right-click the **Particle Study 1** item and select **Run**.

Results

You can see the trajectories of each droplet size (injection), by right-clicking on the **Injection** of interest and selecting **Show**. The resulting trajectories colored by the **Velocity** parameter are presented below.

8µm droplets   13µm droplets   18µm droplets
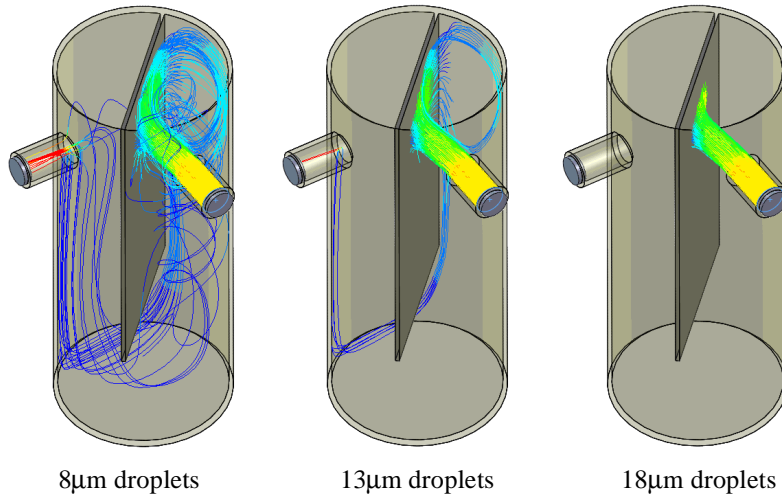
For each particular droplet size, we can obtain the precise amount of particles flown out of the Oil Catch Can by evaluating the integral parameter **Number of Particles** on the outlet face using the **Surface Parameters** feature.

With these values, we can conclude that the probability of trapping the 18µm droplets is 100%; 13µm is about 97%; 8µm is about 90%.

16

150W Halogen Floodlight

This feature is available for the HVAC module users only.

Problem Statement

This tutorial demonstrates the capability of Flow Simulation to simulate heat transfer by convection and radiation, including the radiation absorption in semi-transparent solids and the radiation spectrum. It is shown how to define a project, specify the radiation properties of semi-transparent solid materials, radiation conditions and calculation goals.

Here we consider a halogen floodlight with an aluminum housing, which contains a quartz glass front window, a silicone gasket, an aluminum internal reflector, a ceramic lampholder and a 150W linear halogen lamp.

The linear halogen lamp consists of a quartz glass bulb, a straight-line tungsten filament, molybdenum pinch pins and ceramic base sockets. The lamp is filled with argon at 2 atm and 293.2 K. The lamp operates in typical indoor conditions at the room temperature (~20 °C) and without any forced cooling.

Components of the floodlight and the halogen lamp are shown at the figures below.

Case   Cover

Lamp

Lampholder   Front Window

Reflector

Gasket
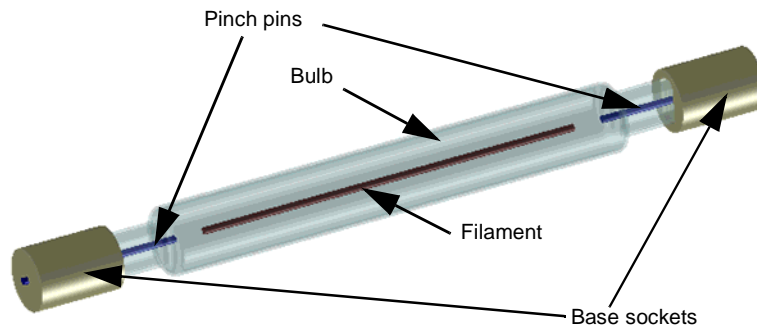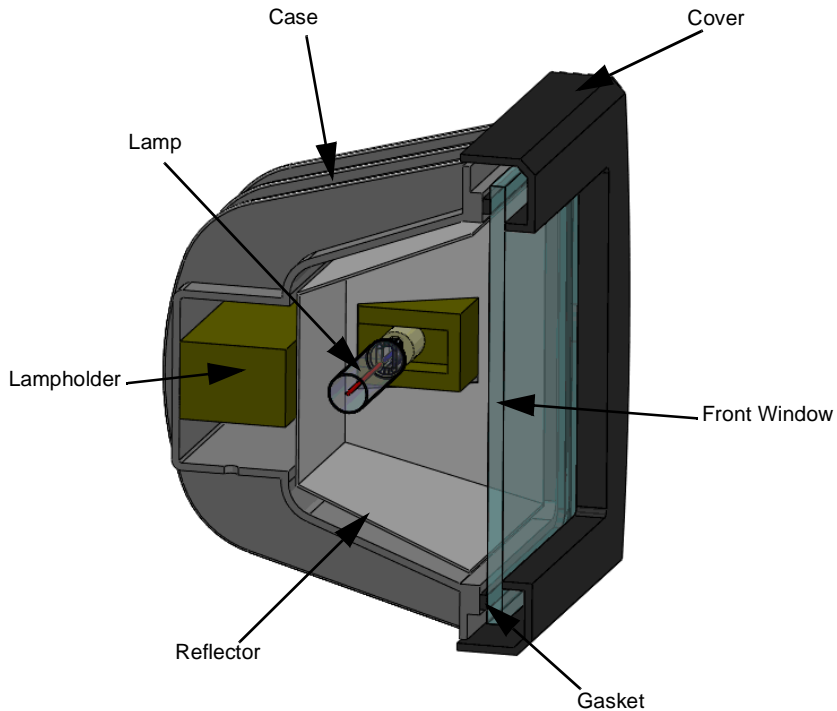
In the table below, you can see the typical values of the maximum allowable operating temperatures for some of these components. The objective of the simulation is to ensure that the pinch pins, the lamp bulb and the front glass are not overheated.

Component	Maximum permissible temperature
Pinch pins	350 °C
Bulb glass	900 °C

Model Configuration

Copy the **Tutorial for HVAC Module 1 - Halogen Floodlight** folder into your working directory. Open the **Floodlight.SLDASM** assembly.

Project Definition

Using the **Wizard** create a new project as follows:

Project Configuration	*Use current*
Unit system	*SI*
Analysis type	*External*

Physical features	*Heat conduction in solids,*
	Radiation:
	    Environment radiation (Environment
	    temperature of 293.2K),
	    Absorption in solids,
	    Spectrum (Number of bands is 2, Bands edge 1
	    at 2500nm)
	Gravity (Y component of -9.81 m/s^2)

Default fluid	*Gases / Air (Default fluid)* *Gases / Argon (clear the* **Default fluid** *check box)*
Default solid	*Metals / Aluminum*
Wall Conditions	*Default wall radiative surface of Pre-Defined /* *Real Surfaces / Aluminum, commercial sheet* *Roughness is default*
Initial Conditions	*Default conditions*
Result and Geometry Resolution	*Result resolution level of 4*

  📖 *Only one semi-transparent solid material, the quartz glass, is used in this device. Its absorption properties are specified as dependent on the wavelength with an abrupt change in absorption at 2500 nm. The UV radiation from the tungsten filament is neglegible at 2900K. Thus, a two-bands spectrum with the bands edge at 2500 nm allows to simulate the radiation absorption in the glass components of the lamp accurately enough.*

Computational Domain

Specify the computational domain size as follows:

X max = 0.15 m	Y max = 0.2 m	Z max = 0.15 m
X min = 0 m	Y min = -0.12 m	Z min = -0.15 m

Specify the **Symmetry** 🔷 condition at **X min** 🔲ₓ

Specifying Fluid Subdomain

Halogen lamps are filled with an inert gas and some small amount of halogen (iodine or bromine). For the purposes of this simulation we can consider the lamp as filled with an inert gas only. The gas in a halogen lamp is at the pressure several times higher than atmospheric. We use fluid subdomain to define both the gas filling the lamp and its pressure.

1   Click **Flow Simulation**, **Insert**, **Fluid Subdomain**.

2   Select the inner cylindrical surface of the **Lamp\Lamp - Bulb** component. Immediately the fluid subdomain you are going to create is displayed in the graphics area as a body of blue color.
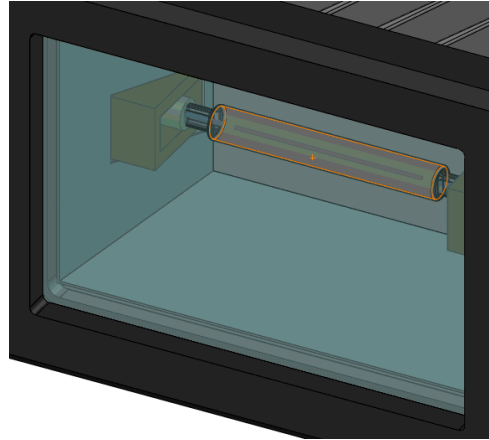

3   Under **Fluids** make sure that **Gases/ Real Gases/Steam** is selected in the **Fluid type** list and clear the **Air (Gases)** check box in the list of fluids below, so that only **Argon** remains selected.

4   Under **Thermodynamic parameters** in the **Pressure** **P** box type 2 atm**.**

5   Click **OK** ✔.
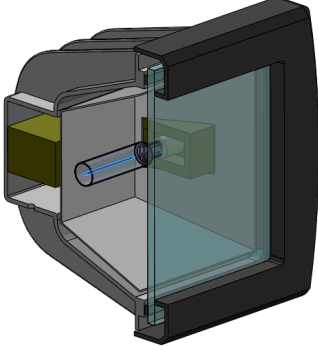
Specifying Heat and Radiation Conditions

There are several ways to define a heat source in . The surface area of the cylindrical straight-line filament can substantially differ from the actual surface area of the coil. If you specify a heat source by its power, this difference must be considered. To avoid discrepancy between the actual and specified radiation heat transfer you can:

a)   define a heat source with the temperature specified,

b)   then define a radiation source with the power specified.
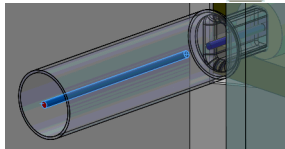
To do this, we define a **Volume Heat Source** with the temperature of 2900K. The value of the convective heat transfer rate is determined as a **Surface Goal** and the **Radiation Source** power is defined as 150 Watt minus the convective heat transfer rate. And finally, the absorption of radiation by the filament must be excluded from the calculation, so the filament surface must be defined as a whitebody surface.

Specify the volume heat source as shown in the table below:

Type	**Volume Heat Source**	
Name	**2900 K**	
Component to apply	**Lamp\Lamp - Wire**	
Parameter: Temperature: 2900 K		

 📖*The true temperature of the filament can be estimated from its color temperature. The typical values of the filament color temperature are specified by the lamp manufacturer. For the filament temperature of about 3000 K the color temperature of tungsten is 2-3 % higher than its equivalent true temperature.*

Specify the goal necessary for calculating the convective heat transfer rate:

GOAL TYPE	GOAL PARAMETER	FACE
Surface Goal	Heat Transfer Rate	The faces of the **Lamp\Lamp - Wire** component located within the computational domain. Select the **Lamp\Lamp - Wire** component in the Flyout FeatureManager Design tree and use **Filter** with the option **Remove out of domain faces** 🔲.

Defining Radiation Source

Follow the steps below to specify the radiation source:

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

2 In the Flyout FeatureManager Design tree select the **Lamp\Lamp - Wire** component. All the filament faces are selected as **Faces to Apply the Radiation Source** 📦 . However, the side face is out of the computational domain and must be excluded.
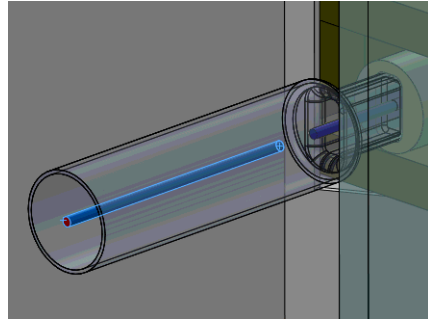
3 Click **Filter Faces** 🔽 . Select **Remove out of domain faces** 📭 , and click **Filter**.
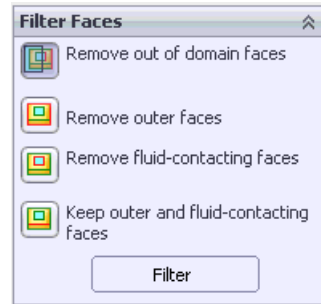
4 Select **Diffusive** ☀ as the **Type**.

5 Under **Power** select **Power** 🔼 and then click **Dependency** f_x .
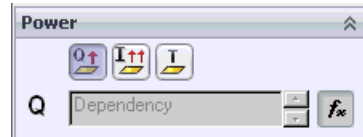
6 In the **Dependency Type** list select **Formula Definition**. In the **Formula** box, type the formula for the total heat power emitted by the source. To add a goal to the formula, select the goal in the **Dependent on goal** list and click **goal** in the input panel. The resulting expression must be the following:
150/2-{SG Heat Transfer Rate 1}
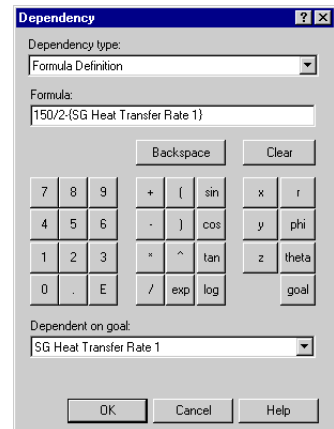
📖 *We used a **Volume Heat Source** to define the heat transferred from the filament by the convection. To specify the remaining heat power, transferred by the radiation, the calculated heat transfer rate of the volume source must be subtracted from the total heat power.*

7 Click **OK** to return to the **Radiation Source** dialog.

8 Under **Spectrum** select **Blackbody Spectrum** and

enter 2900 K in the **Blackbody Temperature** $T_{\blacktriangle}$
box.

9 Click **OK** ✔. The new **Diffusive Radiation source 1** item appears in the **Analysis tree**.

## Defining Radiative Surfaces

Follow the steps below to specify the radiative surfaces:

1 Click **Flow Simulation**, **Insert**, **Radiative Surface**.

2 Under **Type**, expand the list of **Pre-Defined** radiative surfaces and select **Whitebody wall**.

3 In the Flow Simulation Analysis Tree select the **Diffusive Radiation source 1** item.

4 Click **OK** ✔. Rename the new **Radiative Surface 1** item to
Radiative Surface Filament.
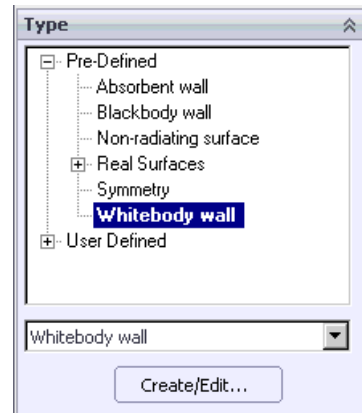
Click anywhere in the graphic area to clear the selection.

5 Click **Flow Simulation**, **Insert**, **Radiative Surface**.

6 Under **Type**, click **Create/Edit**.

7 In the **Engineering Database**, under **Radiative Surfaces, User Defined**, create a new item and change its **Name** to Tutorial Aluminum, polished.

8 Change the parameters of the surface as shown below:

Radiative surface type	Wall
Specularity coefficient	0.8
☐ Emissivity	Specific for thermal and solar radiation
┈┈ Emissivity coefficient	0.1
┈┈ Solar absorptance	0.1

9 Save the created radiative surface and exit the **Engineering Database**.

10 Under **Type**, expand the list of **User-Defined** radiative surfaces and select Tutorial Aluminum, polished.

11 Select the inner faces of **Reflector** located (at least, partially) within the computational domain.

12 Click **OK** ✔ . Change the name of the new radiative surface to `Radiative Surface Reflector`.
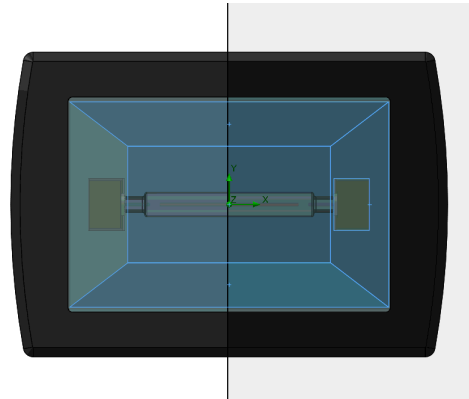
Specifying Solid Materials

For the opaque components, specify the **Solid Material** as following:

Lamp\Lamp - Wire	Pre-Defined\Metals\Tungsten
Lamp\Lamp - Pinch<1>	Pre-Defined\Metals\Molybdenum
Lamp\Lamp - Base<1> Holder	Pre-Defined\Ceramics\Alumina (96%)
Seal	Pre-Defined\Glasses and Minerals\Glass Lid Seal

Specifying Bodies and Materials Transparency

Assign the **Quartz glass** material to the bulb and the glass cover and specify these components as semi-transparent to radiation.

1 Click **Flow Simulation**, **Insert**, **Solid Material**

2 In the Flyout FeatureManager Design Tree select the **Glass** and **Lamp\Lamp - Bulb** components.
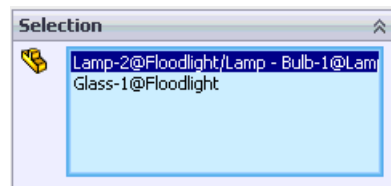
3 Under **Solid** expand the list of **Pre-Defined** solid materials and select **Quartz Glass** under **Glasses and Minerals**.
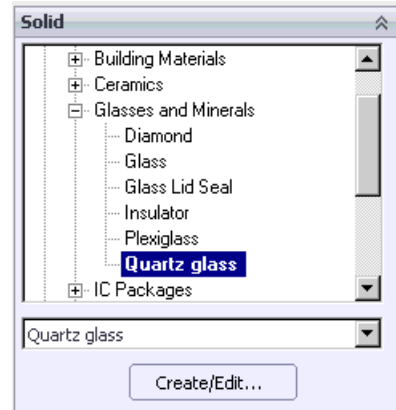
4 Under **Radiation Transparency** select **Absorptive** .

📖 *Absorptive body is semi-transparent, it means that it absorbs the heat radiation within its volume. This option is available only if the absorption coefficient is specified in the solid material definition in the Engineering Database and the **Absorption in solids** check box is selected under **Radiation** in the **Wizard** or **General Settings**. The **Absorption Coefficient** and **Refractive Index** values are specified in the Engineering Database and are provided here just for reference.*
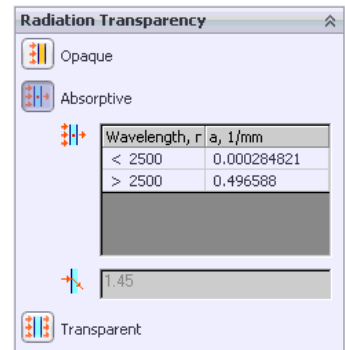
5 Click **OK** ✔. Flow Simulation now treats this solid material and all solid bodies it is assigned to as semi-transparent to the thermal radiation.

Specifying Goals

Specify surface goals of the maximum and average temperatures at the outer surface of the **Glass** component.
In addition, specify volume goals of the **Glass**, **Lamp\Lamp - Bulb** and **Lamp\Lamp - Pinch<1>** maximum and average temperatures. (you must select **Temperature of Solid** as the goal parameter). You can rename the goals as shown to make it easier to monitor them during the calculation.
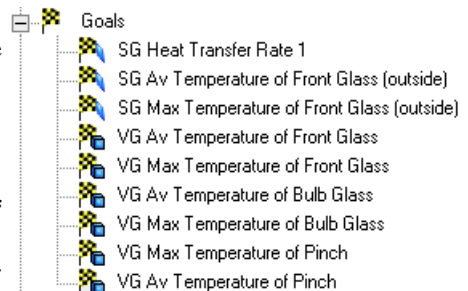
Setting Local Initial Mesh

It makes sense to adjust the computational mesh to better resolve the semi-transparent solid bodies and the fine filament. The most convenient way to do this is to specify **Local Initial Mesh** - it allows us to obtain more accurate solution in these specific regions without creating an excessively fine mesh in other regions.

1   In the FeatureManager Design Tree select the filament and bulb components of the halogen lamp (**Lamp\Lamp - Wire, Lamp\Lamp - Bulb**).

2   Click **Flow Simulation**, **Insert, Local Initial Mesh**.

3   Clear the **Automatic settings** check box. Go to the **Refining Cells** tab.

4   Select **Refine solid cells** and set **Level of refining solid cells** to 5.

5   Click **OK** to save local initial mesh settings.

6   Create another **Local Initial Mesh** for the **Glass** component. After clearing the **Automatic settings** check box, go to the **Refining Cells** tab, select **Refine solid cells** and set **Level of refining solid cells** to 3.

Checking Calculation Control Options

1   Click **Flow Simulation, Calculation Control Options**.

2   Switch to the **Advanced** tab.

3   Under **Radiation** make sure that the value of **Discretization level** is set to 3. This value is appropriate for the given conditions and allows to obtain an acceptable accuracy in the case of compact radiation sources.

    📖*The **Discretization level** controls the discretization of the whole directional domain into equal solid angles or directions. The higher the discretization level, the better the accuracy, but the more CPU time and memory resources are required for the calculation.*

4   Click **OK**.

Save the model and run the calculation.
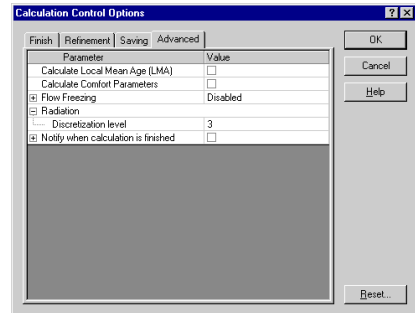
Results

In accordance with the obtained results, we can say that the glass cover and the lamp bulb operate at permissible temperatures.

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]
SG Heat Transfer Rate 1	[W]	8.721428463	8.727580369	8.717889859	8.737026601	100
SG Av Temperature of Fron	[K]	412.384966	411.1346174	409.3971595	412.384966	100
SG Max Temperature of Fr	[K]	465.428311	463.569023	460.9968718	465.428311	100
VG Av Temperature of Fron	[K]	414.2004285	412.9358877	411.1827422	414.2004285	100
VG Max Temperature of Fr	[K]	468.8906105	467.0358562	464.4303868	468.8906105	100
VG Av Temperature of Bulb	[K]	716.2479984	715.196375	713.7749115	716.2745334	100
VG Max Temperature of Bu	[K]	912.6794703	912.009111	910.9425318	912.7472728	100
VG Av Temperature of Pinc	[K]	489.5586685	488.7493109	487.0186254	490.6339404	100
VG Max Temperature of Pi	[K]	512.3509967	511.2211429	509.6762805	512.7591063	100
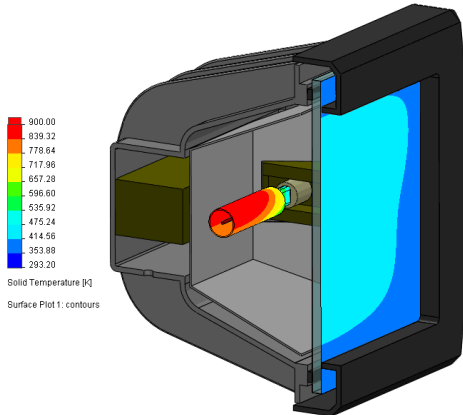
The glass temperature distribution (surface plot of solid temperature) in the range from 293 to 900 K.
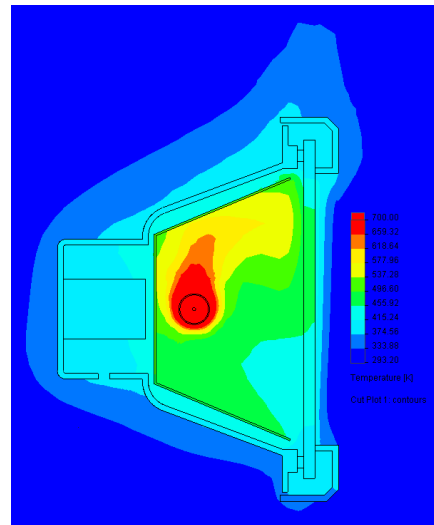
The temperature distribution in the symmetry plane (cut plot of temperature) in the range from 293 to 700 K.

Hospital Room

Some of the features used in this tutorial are available for the HVAC module users only.

Problem Statement

This tutorial demonstrates the capability of Flow Simulation to predict the performance of a building ventilation system and to estimate air quality and general thermal sensation by calculating comfort criteria. It is shown how to define a project, i.e. specify the heat sources, boundary conditions and calculation goals, and how to obtain values of comfort criteria.

Here we consider a hospital isolation room and estimate the ventilation system effectiveness with respect to the contaminant removal and thermal satisfaction of people in the room. A typical patient room includes standard features such as a patient bed, exhausts, lightening, equipment. The overhead ventilation system contains an overhead ceiling supply diffuser, the ceiling and the washroom exhausts. The contaminant source is assumed to be the patient breathing. The heat sources are lights, a medical equipment, a TV, a patient and a caregiver.

The ventilation system and the patient room features are shown at the figure below.
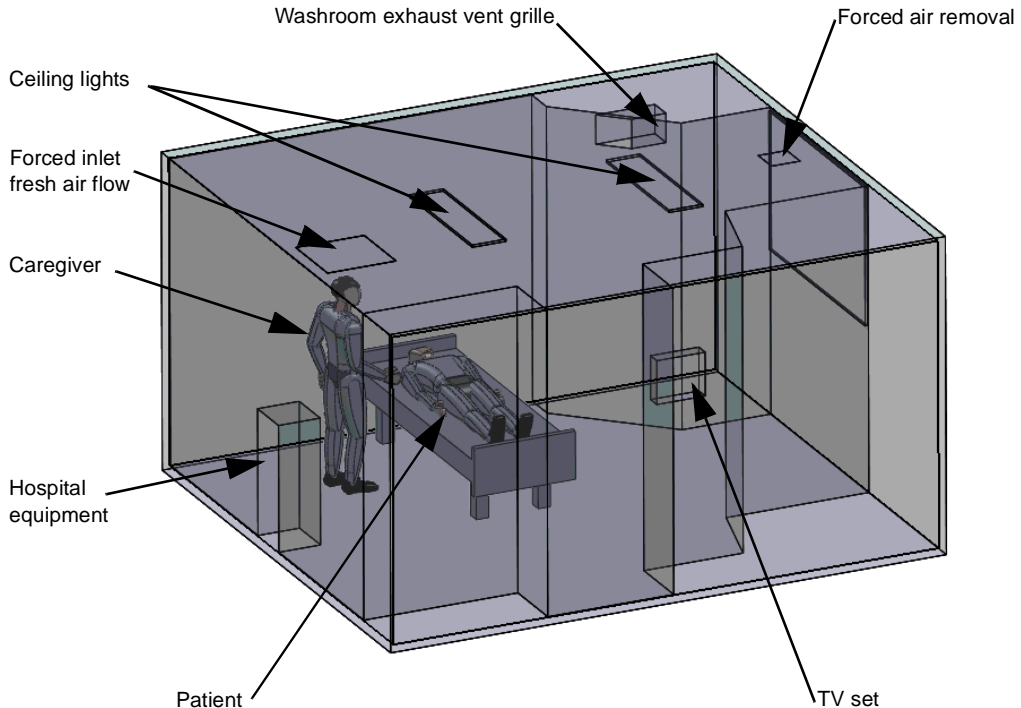
The following parameters are used to estimate the ventilation system effectiveness with respect to contaminant removal: Contaminant Removal Effectiveness (CRE) and Local Air Quality Index (LAQI).

The following parameters are used to estimate the ventilation system effectiveness with respect to thermal satisfaction of people: Air Diffusion Performance Index (ADPI), Predicted Mean Vote (PMV) and Predicted Percent Dissatisfied (PPD).

Model Configuration

Copy the **Tutorial for HVAC Module 2 - Hospital Room** folder into your working directory. Open the **Hospital room.SLDASM** assembly.
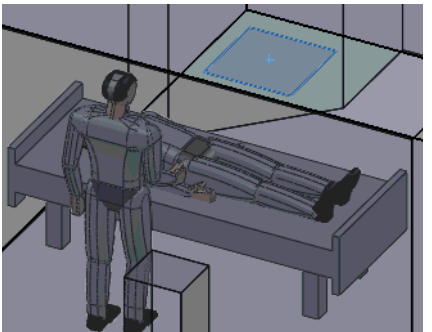
Project Definition
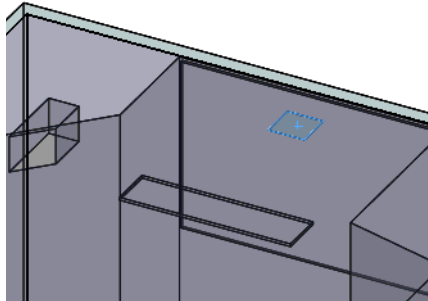
Using the **Wizard** create a new project as follows:
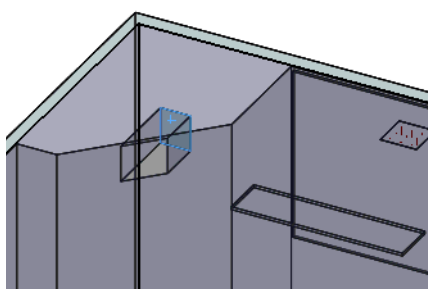
Project Configuration	*Use current*
Unit system	*SI* *Units for Temperature: °C (Celsius)*
Analysis type	*Internal* *Exclude cavities without flow conditions*
Physical features	*Gravity (Y component of -9.81 m/s^2)*
Default fluid	*Gases / Air (Default fluid)* *Gases / Expired Air (user defined)* *Click* **New** *and in the Engineering Database create a new item named* Expired Air *by copy-pasting the pre-defined* **Air,** *available under* **Mateials\Gases\Pre-Defined***, to the* **Mateials\Gases\User Defined** *folder*
Wall Conditions	*Default*
Initial Conditions	*Thermodynamic parameters:* *Temperature of 19.5°C* *Concentration:* *Mass fraction of Air is 1* *Mass fraction of Expired air is 0*
Result and Geometry Resolution	*Result resolution level of 3*
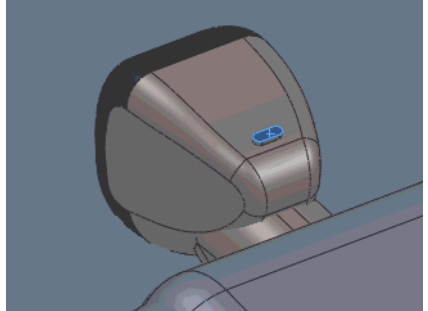
Boundary Conditions

Specify the inlet and outlet boundary conditions as shown in the tables below:

Type	**Inlet Volume Flow**	
Name	**Inlet Volume Flow 1**	
Faces to apply	the inner face of the **Room** component, as shown	
Forced inlet fresh air flow Parameters: **Volume Flow Rate Normal To Face** of 4.8 m^3/min		

Type	**Outlet Volume Flow**	
Name	**Outlet Volume Flow 1**	
Faces to apply	the inner face of the **Room** component, as shown	
Forced air removal Parameters: **Volume Flow Rate Normal To Face** of `2.6 m^3/min`		

Type	**Environment Pressure**	
Name	**Environment Pressure 1**	
Faces to apply	the inner face of the **Room** component, as shown	
Washroom exhaust vent grille **Thermodynamic Parameters**: Default values (101325 Pa and 19.5°C)		

Type	**Inlet Volume Flow**	
Name	**Inlet Volume Flow 2**	
Faces to apply	a face of the **Patient** component, representing the patient's mouth, as shown	
Contaminated expired air Parameters: **Volume Flow Rate Normal To Face** of `12 l/min` **Substance Concentrations**: **Mass fraction** of **Air** is 0 **Mass fraction** of **Expired Air** is 1		

Specifying Heat Sources

There are several heat sources in the hospital room: ceiling lights, a TV set and hospital equipment. The caregiver and the patient are the sources of heat also. The amount of heat produced by a human body depends on the kind of activity the person is involved in. A patient laying on the bed produces significantly less heat than a caregiver, whose work requires physical activity and concentration.

Since we do not consider heat conduction in solids in this simulation, we use surface heat sources with the fixed heat transfer rate.
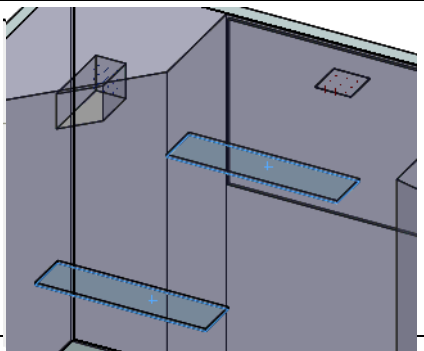
1 Click **Flow Simulation**, **Insert, Surface Source**.

2 In the Flyout FeatureManager Design tree select the **Patient** component. All faces of the component are selected as **Faces to Apply the Surface Source** 🟩. However, the **Patient** component is partially submerged into the bed and some faces are not in contact with the fluid, so we need to remove such faces with filter.

3 Click **Filter Faces** 🔽 . Select **Keep outer faces and fluid-contacting faces** 🟦 and click **Filter**.

4 Manually remove the face representing the patient mouth by selecting it in the graphics area. We need to exclude this face since there is a boundary condition already specified at it.

5 Under **Parameter** specify **Heat Transfer Rate** of 81 W .

6 Click **OK** ✅ .
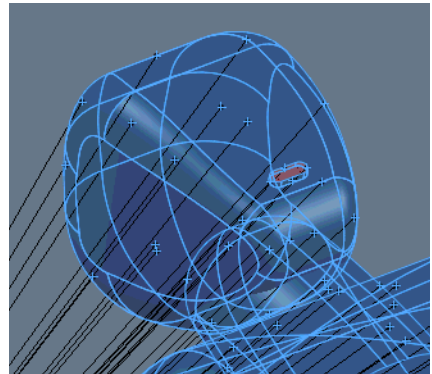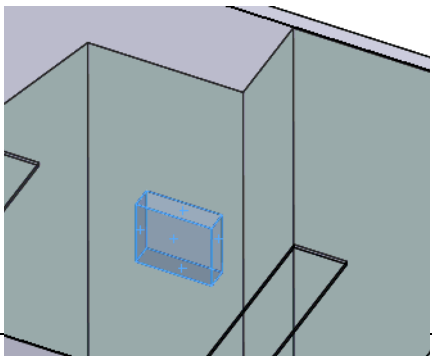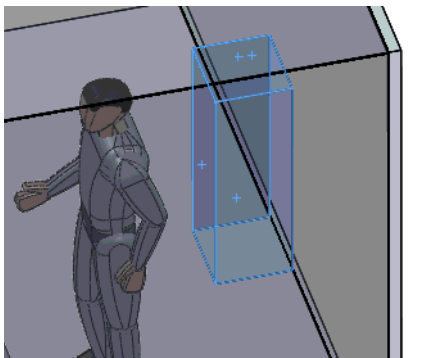
In the same way specify the surface heat source of 144 W at all faces of the **Caregiver** component. Use the **Filter Faces** tool to only select faces which are in contact with the fluid. Since there is no boundary condition specified at the **Caregiver** component, you do not need to manually exclude any faces.

Other sources of heat are not represented by separate components, but by cuts and extrudes made on the **Room** component. Use the tables below as a reference to specify the remaining heat sources:

Type	**Surface Heat Source**
Name	**SS Heat Transfer Rate 3**
Faces to apply	both inner faces of the **Room** component representing the ceiling lights
Parameters: **Heat Transfer Rate** of 120 W	

Type	**Surface Heat Source**
Name	**SS Heat Transfer Rate 4**
Faces to apply	inner faces of the **Room** component representing the TV set
Parameters: **Heat Transfer Rate** of 50 W	

Type	**Surface Heat Source**
Name	**SS Heat Transfer Rate 5**
Faces to apply	inner faces of the **Room** component representing the hospital equipment
Parameters: **Heat Transfer Rate** of 50 W	

Specifying Calculation Control Options

By default, calculation of comfort parameters is disabled in Flow Simulation to save the CPU time and memory resources. Besides comfort parameters, Flow Simulation is capable of calculating Local Mean Age (LMA) and Local Air Change Index(LACI) parameters:

- LMA is the average time for fluid to travel from the selected inlet opening to the point considering both the velocity and diffusion.

- LACI (Local Air Change Index) is the ratio of the *V*/*Q* value, where *V* is the computational domain fluid volume and *Q* is the volume flow rate of the fluid entering this volume, to the average time τ for the fluid to travel from the selected inlet opening to the point considering both the velocity and diffusion.

Calculation of comfort parameters, LMA and LACI can be enabled in the **Calculation Control Options** dialog.

1 Click **Flow Simulation, Calculation Control Options**.

2 Switch to the **Advanced** tab.

3 Select the **Calculate Local Mean Age (LMA)** and **Calculate Comfort Parameters** check boxes.

☞ *Selecting* **Calculate Local Mean Age (LMA)** *check box enables calculation of LMA, Dimensionless LMA and LACI.*

4 Click **OK**.
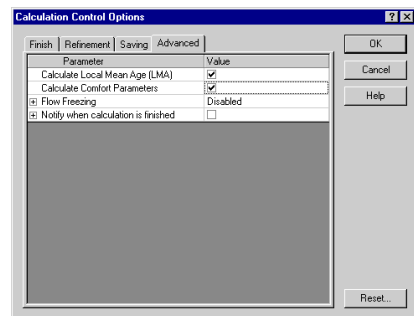
Specifying Goals

Specify global goals of **Av Mean Radiant Temperature**, **Av Operative Temperature**, **Av Velocity** and **Av Volume Fraction of Expired Air**.

☞ *You can use* **Mean Radiant Temperature** *and* **Operative Temperature** *as the goal parameters only after you enable calculation of comfort parameters in the* **Calculation Control Options** *dialog.*
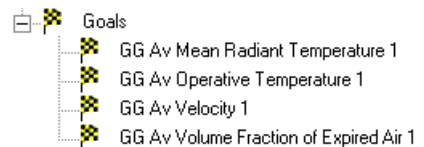
Adjusting Initial Mesh

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

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

You can see that Flow Simulation determined the **Minimum gap size** value as 0.012 m, which is equal to the width of the face representing the patient's mouth (12 mm).
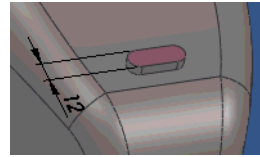
3   Select the **Manual specification of the minimum gap wall thickness** check box.

You can see that the value of **Minimum wall thickness**, determined by Flow Simulation, is too small, which can lead to unnecessary mesh refinement and increased amount of memory required for calculation.

To avoid this, specify the value of **Minimum wall thickness** equal to **Minimum gap size** of `0.012 m.`
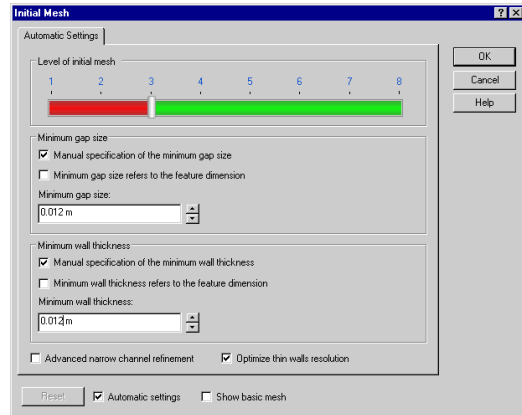
4   Click **OK** to save the initial mesh settings and close the dialog.

Setting Local Initial Mesh

To better resolve the complex geometry of the **Caregiver** and **Patient** components and more accurately account the heat produced by the heat sources specified at these components, we employ the local initial mesh refinement.

1   In the FeatureManager Design Tree select the **Caregiver** and **Patient** components.

2   Click **Flow Simulation**, **Insert, Local Initial Mesh**.

3   Clear the **Automatic settings** check box. Go to the **Solid/Fluid Interface** tab.

4   Set **Small solid features refinement level** to 4.

5   Go to the **Refining Cells** tab. Select **Refine fluid cells** and set **Level of refining fluid cells** to 2.

6   Click **OK** to save local mesh settings.

Run the calculation.

Results

Overview of Comfort Parameters

It is a common practice to assess the performance of a ventilation system by some standard criteria, named comfort parameters. With Flow Simulation you can simulate various environments and get the values of comfort parameters, determining whether the air quality and temperature are safe and comfortable for people working or living in these environments. Later we will use Flow Simulation results processing tools to see and analyze the values of comfort parameters obtained in the calculation.

The following two parameters are used to assess the ventilation system effectiveness in contaminated air removing:

- **Contaminant Removal Effectiveness (CRE)**. This parameter is an index that provides information on the effectiveness of a ventilation system in removing contaminated air from the whole space. For a perfect mixing system CRE = 1. Values above 1 are good, values below 1 are poor.

- **Local Air Quality Index (LAQI)** is an index that provides information on the effectiveness of a ventilation system in removing contaminated air from a local point.

The following several parameters are used to estimate the ventilation system effectiveness with respect to the thermal satisfaction of people in the ventilated area:

- **Mean Radiant Temperature (MRT)** is the uniform surface temperature of an imaginary black enclosure in which an occupant would exchange the same amount of radiant heat as in the actual non-uniform space.

- **Operative Temperature** is the uniform temperature of an imaginary black enclosure, in which an occupant would exchange the same amount of heat by radiation plus convection as in the actual non-uniform environment.

- **Draft Temperature** is the difference in temperature between any point in the occupied zone and the control condition. *"Draft"* is defined as any localized feeling of coolness or warmth of any portion of the body due to both air movement and air temperature, with humidity and radiation considered constant.

- **Air Diffusion Performance Index (ADPI)** is the percentage of the space in which the air speed is less than 0.35 m/s and the Draft Temperature falls between -1.7 °C and 1.1 °C.

  📖*Note: If Draft Temperature or ADPI is calculated as Volume Parameters, the reference space or zone is the specified volume region. In all other cases the whole computational domain is considered.*

- **Predicted Mean Vote (PMV)** is an index that predicts the mean value of the votes of a large group of persons on the 7-point thermal sensation scale, based on the heat

balance of the human body. Thermal balance is obtained when the internal heat production in the body is equal to the loss of heat to the environment.

| cold	cool	slightly cool	neutral	slightly warm	warm	hot
-3	-2	-1	0	+1	+2	+3

- **Predicted Percent Dissatisfied (PPD)** is an index that provides information on thermal discomfort or thermal dissatisfaction by predicting the percentage of people likely to feel too warm or too cool in a given environment.

Obtaining CRE Value

You can see the calculated value of the Contaminant Removal Effectiveness (CRE) in the calculation results summary.

In the Analysis tree, right-click the **Results** icon and select **Summary**.

You can see the **CRE of Expired Air** value at the bottom of the **Results Summary** page, in the **Comfort Parameters** section. The value of **CRE of Expired Air** is higher than 1, which means that the ventilation system is reasonably effective in removing the contaminated air.
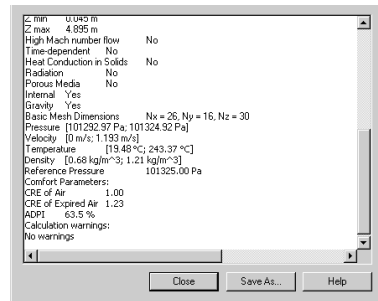
Volume Parameters

We can obtain the values of thermal satisfaction parameters with the **Volume Parameters** results processing feature. However, we need to define the volume, in which the parameters will be calculated. In our case, this volume is the entire fluid region within the computational domain. We can easily create a component representing the entire fluid region with the **Check Geometry** tool.

1 Click **Flow Simulation**, **Tools, Check Geometry**.

2 Select the **Create fluid body assembly** check box to create a new assembly including all fluid regions of the model as solid components. The fluid body assembly is stored in the **Directory for temporary geometry**, specified in the **Flow Simulation Options** dialog, available by clicking **Tools**, **Options**, **Third Party**, **Flow Simulation Options**.

3 Click **Check** to create the fluid body assembly. When the operation is completed, close the **Check Geometry** dialog.

4 Save the Part1 component from the newly created assembly as **Fluid Volume.SLDPRT** and add it to the project assembly.

5 Click **Flow Simulation**, **Component Control** and **Disable** the newly added **Fluid Volume** component. Click **OK**.
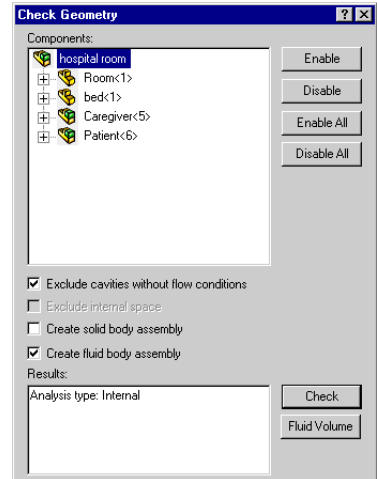
We also need to check the values of reference parameters: metabolic rate, external work, closing thermal resistance and relative humidity, used to calculate comfort parameters such as PMV and PPD. These reference parameters define the approximate heat power produced by a human body depending on the activity and health condition, insulating properties of the closing and humidity of the air.

1 Click **Flow Simulation**, **Results**, **Default Reference Parameters**.

2 Specify **Metabolic rate** of 100 W/m^2. Keep the other values default.

 The closing thermal resistance of 0.11 $K \cdot m^2/W$ corresponds to a light working ensemble: light underwear, cotton work shirt with long sleeves, work trousers, woolen socks and shoes. The definition of clothing insulation relates to heat transfer from the whole body and, thus, also includes the uncovered parts of the body, such as head and hands.
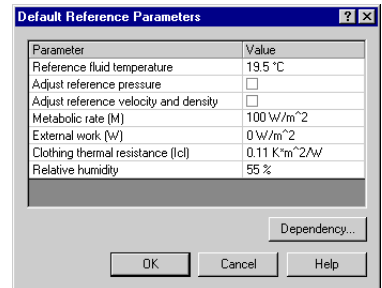
 *The relative humidity of 55% is typical for indoor conditions. If the relative humidity is considered in the analysis (the **Humidity** option is selected in the **General Settings**), the actual calculated value of the relative humidity is used as the reference parameter.*

3 Click **OK**.

Now we can use the **Volume Parameters** feature to see the values of comfort parameters.

1 In the Flow Simulation Analysis tree right-click the **Volume Parameters** icon and select **Insert**.

2 In the Flyout FeatureManager Design tree select the **Fluid Volume** component.

3 Under **Parameters** click **More Parameters**. The **Display Parameters** dialog appears.

4 Expand the **Local** item and select the following parameters:

- **Mean Radiant Temperature**,
- **Operative Temperature**,
- **PMV**,
- **PPD**,
- **Draft Temperature**
- **LAQI of Air**,
- **LAQI of Expired Air**.

5 Click **OK** to close the **Display Parameters** dialog.

6 In the **Volume Parameters** dialog make sure that the selected parameters are also selected as the **Parameters to Evaluate** under **Parameters**. Additionally select the **ADPI** parameter.

7 Click **Export to Excel**. A spreadsheet with the selected parameters values appears.

Parameter	Average	Bulk Average	Volume [m^3]
Mean Radiant Temperature [°C]	23.9847308	23.9811646	41.8581029
Operative Temperature [°C]	23.4965288	23.4930785	41.8581029
PMV [ ]	0.721211205	0.720537619	41.8581029
PPD [%]	17.0303015	17.0086269	41.8581029
Draft Temperature [K]	0.721984015	0.718661076	41.8581029

Parameter	Value
ADPI [%]	63.4301412

Cut Plots and Isosurfaces

To see how the quality of air with respect to the contained contaminant changes through the room, we create a cut plot by the **LAQI of Expired Air** parameter at the distance of 1 m from the floor - i.e. slightly above the level of the patient's head. The higher the value, the less the concentration of the contaminant and better it is removed.
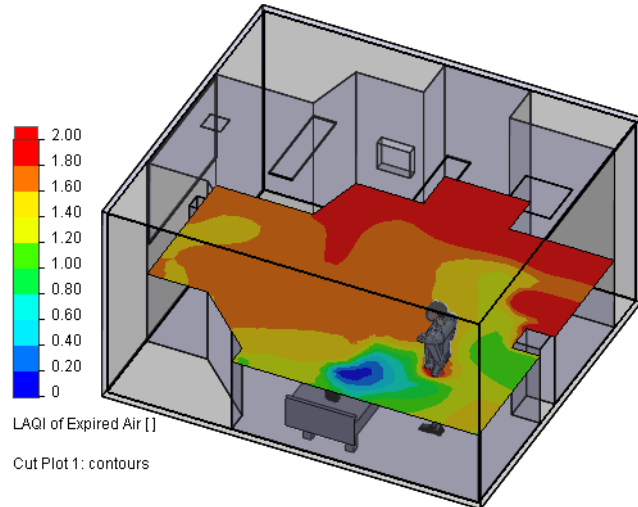
The isosurfaces of **PMV** at 0, 0.25, 0.5, 0.75 and 1 allows us to estimate the level of thermal comfort through the room - from 0 (normal) to +1 (slightly warm).