

# First Steps - Conjugate Heat Transfer

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

# **Open the SolidWorks Model**

- 1 Copy the **First Steps Electronics Cooling** folder into your working directory and ensure that the files are not read-only since COSMOSFloWorks will save input data to these files. Click **File**, **Open**.
- 2 In the Open dialog box, browse to the Enclosure Assembly.SLDASM assembly located in the First Steps Electronics Cooling folder and click Open (or double-click the assembly). Alternatively, you can drag and drop the Enclosure Assembly.SLDASM file to an empty area of SolidWorks window.

# Preparing the Model COSMOSFloWorks 2004 Tutorial

# **Preparing the Model**

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

The assembly consists of the following components: enclosure, MotherBoard and PCBs, capacitors, power supply, heat sink, chips, fan, screws, fan housing, and lids. You can view these components by clicking on the features in the SolidWorks Feature Manager. In this tutorial we will simulate the fan by specifying a **Fan** boundary condition on the inner face of the inlet lid. The fan has very complex geometry that may cause delays while rebuilding the model. Since it is outside of the enclosure we can exclude it to hasten operations with SolidWorks.

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



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

Now you can start with COSMOSFloWorks.

# **Create a New Material**

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

- 1 Click FloWorks, Tools, Engineering Database.
- 2 In the Database tree select Material, Solids, User D<u>a</u>tabase tree: 🚱 Custom - Visualization Parameters - 🔚 Fan Curves **Defined**. Click **New Item** on the toolbar. 🗄 🚰 Material 🗄 😫 Gases The blank Item Properties tab appears. Double-click the 🗄 🧿 Liquids 👲 Non-Newtonian/Compressible liquids empty cell to set the corresponding property value. 🗄 🖽 Solids 📅 FW Defined 于 User Defined 3 Specify the material properties as follows: 🌠 Porous Media 🗄 🔷 Radiative Surface 🛷 Units Name = Epoxy, **Comment** = Epoxy Resin, 🍻 🖬 📐 🚳 🖏 🌾 😘 < → 🛃 × G. List 🗇 Item Properties 📴 Tables and Curves **Density** = 1120 kg/m3, atabase tree Custom - Visualization Paramete Specific Heat = 1400 J/kgK, Materia Kases
   Cases
   Cares
   Solids
   Solids
   FW Defined Commen Epoxy P Density **Thermal Conductivity** = 0.2 W/mK, 1120 kg/m\*3 1400 J/(kg\*K) 0.2 W/(m\*K) 1000 K Specific heat ductionly **Melting Temperature** = 1000 K. FW Defined Melting temperature Norous Media Badietive Surface Click Save 🔲 4
- You can enter the material properties in any unit system you want by typing the unit name after the value and COSMOSFloWorks will automatically convert the value to metric. You can also enter material properties that are temperature dependent using the **Tables and Curves** tab.

# **Define a Custom Unit System**

COSMOSFloWorks allows you to work with several pre-defined unit systems but often it is more convenient to define your own custom unit system. Both pre-defined and custom unit systems are stored in the Engineering Database. Prior to starting the project Wizard you can create the desired system of units in the Engineering Database. To adjust the selected system of units after passing the Wizard you can use either Engineering Database or click **FloWorks**, **Units**. In this example we will adjust a pre-defined system of units before creating the project.

### Define a Custom Unit System

# COSMOSFIoWorks 2004 Tutorial

- 1 In the **Database tree** select **Units**, **FW Defined**.
- 2 On the List tab select the USA system of units and click Copy
- You can modify only custom entries in the Engineering Database. To adjust the pre-defined material, porous media, unit system or fan curve you must copy it into the corresponding User Defined folder first and then make necessary changes.

glabase tree:	List CP Nem F	Properties	
Control of Addition of Additioner and Addita Additioner and Additioner and Additioner and Additioner and A	Name 9 NMU (mm-p+) 9 FS (mbcs) 1 USA 9 COS (mcg-s) 9 COS (mcg-s) 9 COS (mcg-s) 9 COS (mcg-s)	Comment         Image: Comment of the comment of	
		an Abar I	(Film he al

- 3 In the tree, select the Units, User Defined item and click Paste 📳
- 4 Click the **Item Properties** tab to adjust the USA unit system for this example.
- By scrolling through the different groups in the **Parameter** tree you can see the units selected for all the parameters. Although most of the parameters have convenient units such as ft/s for velocity and CFM (cubic feet per minute) for volume flow rate we will change a couple units that are more convenient for this model. Since the physical size of the model is relatively small it is more convenient to choose inches instead of feet as the length unit.
- 5 For the **Length** entry, click on the right hand side of the units box and select inches.
- 6 Next expand the **Heat** group in the **Parameter** tree.



### Create a COSMOSFIoWorks Project

- Since we are dealing with electronic components it is more convenient to specify power and heat flux in Watts and Watts/m<sup>2</sup> respectively.
- 8 Name the new system of units USA Electronics.

Name Comment ⊇ Main Pressure & stress	USA  USA Units	Decimal Places	
Parameter D Main Pressure & stress	Units	Decimal Places	internet in the second
Main  Pressure & stress	Cime	Ceceniar Proces	1 11 1 444 50 m
Pressure & stress			1.0 One St.*
	lbt5n°2	4	.0001
Velocity	11/5	1	3,2808
Mass	Ib	2	2,2046
Length	in	3	39 3701
Temperature	F	1	-457.87
- Physical time	5	1	1
Geometrical Characteristic			
Loeds&Motion			
E Heat			
- Energy	Btu	4	.0009
<ul> <li>Total heat flow &amp; power</li> </ul>	w	0	1
- Heat flux	W/m*2	0	- 1
<ul> <li>Heat release per unit volume</li> </ul>	Bhu/(h/#1~3)	0	.0966
- Starton number			12
Heat transfer coefficient	IbVi/tVF	4	.0685
Medium Properties			
	Mass Langh Tempetate Physical time Physical tempetate Loads/Alkdon Loads/Alkdon Heart Loads/Alkdon Heart Telesa par una volume Stanton number Heart tensies par una volume Stanton number Heart tensies ceticient Mass Honder Appetas	Mass IB Langh III III III III III III IIII IIII II	Mass         Ib         2           Longh         in         3           Temperature         T         1           Physical trian         1         1           Longh         in         3           Longh         T         1           Longh         in         3           Longh         T         1           Longh         E         1           Longh         Heat         1           Longh         Bu         4           Total heat flow & power         W/n *2         0           Heat flux         W/m *2         0           Stat/on runder         Bu/(1**7)         0           Stat/on runder         4         1           Madum Poperties         4         1

- Click Save 🔚.
- 9 Close the **Engineering Database** by clicking **File**, **Exit**.

# Create a COSMOSFIoWorks Project

- 1 Click FloWorks, Project, Wizard.
- 2 Once inside the Wizard, select Create new in order to create a new configuration and name it Inlet Fan.

		×
To begin the project you must fi project will be connected. You o create a new configuration by g configuration. You can also atte	rst specify the configuratio can either use the current o rving a new name to the cu sch comments to the project	in to which the configuration, or irrent ct
Configuration Cloote new C Use current	Comments:	
Configuration name: Inlet Fan Current configuration. Detoutt	_	تے.
	To begin the project you must it project will be connected. You create a new configuration by a configuration. You can also an Configuration Configuration Configuration Configuration Configuration Configuration Configuration name: Intel Fan Current configuration. Octaut	To begin the project you must first specify the configuration project will be connected. You can either use the current create a new configuration by giving a new name to the co- configuration. You can also ettach comments to the project Configuration © Configuration © Configuration © Use current Cigrifiquation nome: Toter in the configuration: Default Use Configuration Current configuration Default Use Configuration Use Configuration Current configuration C

Click Next.

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

gal fath" mm İn kg/day	Please select a sys selected system of design.	tem of units in which units hes no influenc	the endysis is performed. The se on the unit system used for m	odel
	System	Path	Comment	
<sub>grad</sub> yd cm/s² <sub>mile/h</sub>	CGS (cm-g-s) FPS (th/b-s) IPS (n-lb-s) NIM4 (mm-g-s) SI (m-kg-s) USA USA USA Electronice	FW Defined FW Defined FW Defined FW Defined FW Defined FW Defined User Defined	CUS (cm-g-s) FPS (hth=s) IPS (intb=s) IPS (intb=s) IPS (intb=s) SI (intk-g-s) USA USA	

Wizard - Units

Click Next.

### Create a COSMOSFIoWorks Project

# **COSMOSFIoWorks 2004 Tutorial**

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

	Fluid type C Liquid C Dasi	C Man Newtonian/Compressible Ravid
110	Please cick "Heat transfer in co celculations (e. for coujugate h "carevlational effects" and enteri- are importantel (e.g. natural conve and unrar the "Time Settings" (f) endy/re high-velocity gas fores ( stendy-state and about 1 for time number for%. Cick "Heat transfer in solido one andywisi. Under "Row type", you Laminer Ony or Turbolert Only.	tidd 'to include sold heet conduction extrander simulations). Click the 'Gravitational Settings' If these election for problems', Lick' Time dependent' our problems is unsteady. If you want to which number is greater than about 3 for sands analyses), then click 'High Mach enable surface-to-variace radiation. 'r if no fauld avsists in your heat trensfor can choose to consider the flow as
	Physical teatures F Heat transfer in solids	C Heattrunster in solids only
	Flow type:	F Registion
		1
	Laminar and Turbulent	] I High Mach number flow
	□ Lamnar and Turbulent	Time Settings

Click Next.

- Heat transfer in solids was selected because heat is generated by several electronics component and we are interested to see how the heat is dissipated through the heat sink and other solid parts and then out to the fluid. Therefore we must simulate heat conduction in the solid parts.
- **5** Set the analysis type to **Internal**.
- We want to analyze the flow **through** the structure. This is what we call an internal analysis. The opposite is an external analysis, which is the flow **around** an object. From this dialog box you can also choose to ignore cavities that are not relevant to the flow analysis without having to fill them in using SolidWorks features.

Click Next

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



<b>T</b> <sub>f</sub> - 777 <b>q</b> - 777	Please specify the thermal wall condition applied by detault b all of the model outer wells. This condition allows you to specify heet exchange between the externel flow and the outer model wells.	
$Q - 777$ $Q = \alpha (T_f - T_w)$ $T_w$	- Default outer well conditions - Adjabatic - Heat transfer gueficient - Heat transfer gueficient - Heat generation rate - Heat generation rate - Mail temperature - Wall temperature	
	<back cancel="" help<="" td=""><td></td></back>	

### Create a COSMOSFIoWorks Project

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

Click Next.

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

Choose **Air** as the fluid. You can

either double-click Air or select the

Wizard - Selecting Solid Sub ٠ <Back Next> Cancel Help



Click Next.

10

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

# Create a COSMOSFIoWorks Project

# **COSMOSFIoWorks 2004 Tutorial**

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



-

Gancel Help

Click **Next**.

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

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

Click Next.

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

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



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

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



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

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

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

Solution (Section 1) (Sect

# **Define the Fan**

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

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

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



# **Define the Fan**

# **COSMOSFIoWorks 2004 Tutorial**

- 2 Select the inner face of the **Inlet Lid** part as shown. (To access the inner face, right-click the mouse to cycle through the faces under the cursor until the inner face is highlighted, then click the left mouse button).
- **3** Select **External Inlet Fan** as **Fan type**.
- 4 Click **Browse** to select the fan curve from the **Engineering database**.
- Select the 405 item under the Fan Curves, FW Defined, PAPST, DC-Axial, Series 400, 405 item.
- 6 Click **OK** to return to the **Fan** dialog box.





- 7 On the **Settings** tab expand the **Thermodynamic Parameters** item to check that the **Ambient pressure** is atmospheric pressure.
- 8 Expand the Flow parameters item and select Swirl in the Flow vectors direction list.
- **9** Specify the **Angular velocity** as 100 rad/s and accept the zero **Radial velocity** value.



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

# **Define the Boundary Conditions**

- **10** Go back to the **Definition** tab. Hold down the **Ctrl** key and in the FeatureManager design tree select the **Inlet Coordinate System**.
- **11** Select the **Global Coordinate System** item and press the **Delete** key.
- **12** Select **Y** in the **Reference Axis** list.

an Definition Settings				<b>[</b> 2]
Fan c <u>u</u> rve:	405		Browse	OK Cancel
Ean type:	External Inlet Fan			Help
Inlet (from fan to fluid) Faces to apply: Face <1 Inlet Lid-1@	)Base-Extrude>	Outlet (from fluid to fan)     Fages to apply:		
Coordinate <u>s</u> ystem:		Coordinate system:		
Inlet Coordinate Sys	tem	Inlet Coordinate Syste	em	
Beference axis:	Y	Reference exis:	×	
✓ Create associated go	els			

- **13** Accept to **Create associated goals**.
- It is often convenient to specify an appropriate goal along with the specified condition. For example, if you specify a pressure opening it makes sense to define a mass flow rate surface goal at this opening. COSMOSFloWorks allows you to associate a condition type (boundary condition, fun, heat source or radiative surface) with a goal(s), which will be automatically created along with this condition. You can associate goals with a boundary condition type under the Automatic Goals item of the COSMOSFloWorks Options dialog box.
- **14** Click **OK**. The new **External Inlet Fan1** item appears in the COSMOSFloWorks design tree.

Boundary Conditions

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

# **Define the Boundary Conditions**

A **boundary condition** is required anywhere fluid enters or exits the system excluding openings where a fan is specified. A boundary condition can be set as a Pressure, Mass Flow, Volume Flow or Velocity. You can also use the **Boundary Condition** dialog for specifying an **Ideal Wall** condition that is an adiabatic, frictionless wall or a **Real Wall** 

# **Define the Boundary Conditions**

# **COSMOSFIoWorks 2004 Tutorial**

kert Boundary Condition.

Definition Settings

· Elow openings

C Pressure openings
 ⊂ Wall

Basic set of boundary conditions

Faces to apply boundary condition: Face <20utlet Lid-1@Base Extrude> Face <20utlet Lid-2@Base-Extrude> Face <20utlet Lid-3@Base Extrude>

Component Control

**•**18

🔟 Bounde

]]∃ Fans \_\_\_\_\_\_ 99 Heat Sources

condition to set the wall roughness and/or temperature and/or heat transfer coefficient at the model surfaces. For internal analyses with "Heat transfer in solids" you can also set thermal wall condition on outer model walls by specifying an **Outer Wall** condition.

- 1 In the COSMOSFloWorks design tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**Click **Insert, New Boundary Condition**.
- 2 Select the inner face of all of the outlet lids as shown. (To access the inner face, right-click the mouse to cycle through the faces under the cursor until the inner face is highlighted, then click the left mouse button).
- 3 Select **Pressure openings** and **Static Pressure**.



- 4 Keep the defaults under the **Settings** tab.
- 5 Click **OK**. The new **Static Pressure1** item appears in the COS-MOSFloWorks design tree.



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

# **Define the Heat Source**

- 1 In the COSMOSFloWorks design tree, right-click the Heat Sources icon and select **Insert Volume Source**.
- 🤏 Materia Insert Volume Source.. 🔨 Initial Co 2 Since the inner faces of the outlet lids are still selected, the lids automatically appear in the Components to apply the volume source list. Remove all lids from the list. To remove a component, select it in the list and press the **Delete** key.

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

■ 🤤 Capacitor<1> ->	Volume Source	
Capacitor<2> ->     Gapacitor<3> ->     Capacitor<3> ->     Capacitor<3> ->	Definition Settings	
Heat Sink(1>->	Source type:	
🗉 🍕 Main Chip (1>	Heat Generation Rate	•
표 🍕 Small Chip♥1> -: 표 🍕 Small Chip<2> -:	Components to apply the volume source.	
	Main Chip-1	

Insert Surface Sources...

⊡ M Boundary Conditions

🥦 Heat Sources

⊨...]∃ Fans

Static Pressure1

📲 External Inlet Fan1

- 3 Select the Main Chip from the SolidWorks FeatureManager tree as the component to apply the volume source.
- 4 Select the Source type as Heat Generation Rate.



### Define the Heat Source

. 🈏 Main Chip

# **Define the Heat Source**

- 5 Click the **Settings** tab and enter 5 W in the Heat generation rate box.
- 6 Click OK.

Definition Settings	
	OK Cancel Help
Hget generation reto:	
✓ Create associated goals	

- In the COSMOSFloWorks design tree click-pause-click the 🗄 🦻 Heat Sources 7 new VS Heat Generation Rate 1 item and rename it to Main Chip.
- Volume Heat Sources allows you to specify the heat generation rate (in Watts) or the volumetric heat generation rate (in Watts per volume) or a constant temperature boundary condition for the volume. It is also possible to specify Surface Heat Sources in terms of heat transfer rate (in Watts), heat flux (in Watts per area).
- 1 In the COSMOSFloWorks design tree, right-click the Heat Sources icon and select Insert Volume Source.
- 2 L. Origin ⊨ 🍕 (1) Enclosure <1> In the SolidWorks FeatureManager tree select all Capacitor components.
- 3 Select the **Temperature** in the **Source** type list.
- 4 Click the **Settings** tab and enter 100 °F in the **Temperature** box.
- 5 Click OK.



Design..

Definition Settings

MotherBoard<1:

☑ Create associated goals

### **Define the Material Conditions**

- 6 Click-pause-click the new VS Temperature1 item and rename it to Capacitors.
- 7 Following the same procedure as above, set the following volume heat sources: all chips on PCB (Small Chip) - total heat generation rate of 4 W, Power Supply - temperature of 120 °F
- MotherBoard<1>
   PCB<1>->
   PCB<2>->
   Capacitor<1>-> Capacitor<1>-> Capacitor<2>-> Source type: - Capacitor<3> -> Heat Generation Rate • Heat Sink<1>-> Main Chip<1> Small Chip<1>-> Components to apply the volume source Coordinate system Vol mall Chip-3 mall Chip-4 mall Chip-5 mall Chip-6 mall Chip-7 mall Chip-8 Definition Settings Inlet Lid<1>-> 🔏 Outlet Lid<1> -> Fan Housing<1> (Default) 🔽 Create associati Heat generation rate 4 W | ÷ PCB<1>-> Volume Sc PCB<2>->
   Q
   Capacitor<1>->
   Capacitor<2>-> Definition Settings Source type Capacitor<3>-: Capacitor(3) ->
   Power Supply(1) ->
   Heat Sink(1) ->
   Main Chip(1) -Components to apply the volume source Coordinate syster Small Chip<1>-> Small Chip<7>-> Small Chip<2>-> Small Chip<2>-> Small Chip<3>-> ower Supply Volume Source Definition Settings Small Chip<5> -> & Small Chip(8) -> & Small Chip(6) -> & Small Chip(7) -> & Small Chip(8) -> & Inlet Lid(1) -> Cutlet Lid<1>-> Cutlet Lid<1>-> Fan Housing<1> (De Crew<1> (Default) Crew<2> (Default) 🔽 Create associated Temperature 120 °F ÷ Rename the source applied to the chips to Chips and the 🗄 🦻 Heat Sources

Definition Settings

- 8 source for the power supply to Power Supply.
- 9 Click File, Save.

# **Define the Material Conditions**

- 1 Right-click the Material Conditions icon and select Insert Material Condition.
- 2 In the SolidWorks FeatureManager select MotherBoard, PCB(1), PCB(2) components.
- 3 Select Epoxy from the Selected material list.
- L. Origin Material ( () En 13 Epoxy Components to apply the material condition Capacitor(1>-> Mother PCB-1 PCB-2 Capacitor<2>-> Capacitor (3) -> Power Supply <1>
  Heat Sink <1> ->

🞐 Main Chip 🮐 Capacitors 🮐 Chips 🦻 Power Supply

4 Click OK.

### **Define the Engineering Goals**

- **COSMOSFIoWorks 2004 Tutorial**
- 5 Following the same procedure as above, set the following material conditions: the chips are made of silicon, the heat sink is made of aluminum, and the 4 Lids (Inlet Lid and three Outlet Lids) are made of insulator material. All four lids can be selected in the same material condition definition. Note that two of the outlet lids can be found under derived pattern (DerivedLPattern1) in the SolidWorks FeatureManager. Alternatively you can click on the actual part in the SolidWorks graphics area.
- 6 Change the name of each material condition. The new descriptive names are: PCB - Epoxy, Heat Sink - Aluminum, Chips - Silicon, and Lids - Insulator.



- Material Conditions are used to specify the material type of solid parts in the assembly.
- 7 Click File, Save.

# **Define the Engineering Goals**

# **Specifying Volume Goals**

- 1 Right-click the Goals icon and select Insert Volume Goal.
- 2 Select the **Chips** item in the COSMOS-FloWorks design tree. This selects all components belonging to the **Chips** heat source.



## **Define the Engineering Goals**

- **3** Select **Temperature of Solid** as the goal type and calculate **Maximum value**.
- 4 Accept to Use the goal for convergence control.
- 5 Click OK. The new VG Maximum Temperature of Solid1 item appears in the COSMOS-FloWorks design tree.
- 6 Change the name of the new item to VG Small Chips Max Temperature. You can also change the name of the item using the **Feature Properties** dialog appearing if you right-click the item and select **Properties**.
- 7 Right-click the Goals icon and select Insert Volume Goal.
- 8 Select the Main Chip item in the COSMOS-FloWorks design tree.
- **9** Select **Temperature of Solid** as the goal type and calculate **Maximum value**.
- **10** Click **OK**.
- 11 Rename the new VG Maximum Temperature of Solid1 item to VG Chip Max Temperature.



# **Specifying Surface Goals**

- 1 Right-click the Goals icon and select Insert Surface Goal.
- 2 Since the Main Chip is still selected, all its faces automatically appear in the Faces to apply the surface goal list. Remove all faces from the list. To remove a face, select it in the list and press the Delete key.

- **3** Click the **External Inlet Fan1** item to select the face where it is going to be applied.
- 4 Keep the **Static Pressure** and the **Average Value**.
- 5 Accept to Use the goal for convergence control.
- For the X(Y, Z) Component of Force and X(Y, Z) Component of Torque goals you can select the Coordinate system in which these goals are calculated.
- 6 Click OK and rename the new SG Average Static Pressure1 item to SG Av Inlet Pressure.
- 7 Right-click the Goals icon and select Insert Surface Goal.



- 8 Click the Static Pressure1 item to select the face where it is going to be applied.
- 9 Select Mass Flow Rate as the Goal type.
- 10 Accept to Use the goal for convergence control.
- 11 Click OK and rename the new SG Mass Flow Rate1 item to SG Outlet Mass Flow Rate.

# **Specifying Global Goals**

- 1 Right-click the Goals icon and select Insert **Global Goal**.
- 2 Keep the Static Pressure, Average Value and accept to Use the goal for convergence control.
- 3 Click OK. Rename the new GG Average Static **Pressure1** item to GG Av Pressure.
- 4 Right-click the Goals icon and select Insert **Global Goal**.
- 5 Select the **Temperature of Fluid** as the **Goal** type, keep the Average Value and accept to Use the goal for convergence control.



Delete All

•

Calculate

Average Value

Cancel

	OK	Cancel	Help
Global Goal			2 X
Goal type:		Calculate:	
Temperature of F	uid 💌	Average Value	•
☑ Use the goal fo	r convergence contro	ol	
Coordinate <u>s</u> ystem	1:		

# **Define the Engineering Goals**

Surface G Goal type

Static Pressure

Eaces to apply the s

ndary Condit

Static Pressu

B D B

= DE 104

<u>G</u>oal type

Static Pressure

Use the goal for convergence control



<u>H</u>elp

? ×

•

# **Changing the Geometry Resolution**

# COSMOSFIoWorks 2004 Tutorial

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



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

Click File, Save.

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

# **Changing the Geometry Resolution**

- 1 Click FloWorks, Initial Mesh.
- 2 Select the Manual specification of the minimum gap size check box.

	2		- 12 E	1 E S	0	7	.0
	-		- 5		0	- 13	
-		-)-		1			-
Minim	um gap s	120					
⊽ Ma	nual spec	ification	of the mir	nimum ga	p size		
Mir	imum aa		ars to the	foature c	fimensio		
Minim	um can c	10.	ore to alle	no on o o		8	
0.15	aur Zah s	-					
0.151	n	Ξ					
Minim	um wall th	ickness					
	nual sper	rification	of the mir	aimum wa	ll thickne	9.9	
1000							
T Mr							
Minim	imum wa um <u>w</u> ail th	ickness					
Minim Minim	imum wa um <u>w</u> ali th	ickness					
Minim	imum wo um <u>w</u> ali th	ickness +					
Minim Minim	imum wo um <sub>W</sub> all th anced na	ickness 	nnel refin	ement			
Minim Ad <u>v</u> i	imum wo um <u>w</u> all th anced na	ickness ÷	nnel refin	ement			

**3** Enter 0.15" for the minimum flow passage (i.e. passage between the fins of the heat sink).



### Solution

Bur

<u>C</u>lose

the walls of the steel cabinet. Both the **minimum gap size** and the **minimum wall** thickness are tools that help you to create a model-adaptive mesh resulting in increased accuracy. However the minimum gap size setting is the more powerful one. The fact is that the COSMOSFloWorks mesh is constructed so that the specified Result Resolution level controls the minimum number of mesh cells per **minimum gap size**. And this number is equal to or greater than the number of mesh cells generated per **minimum wall thickness**. That's why even if you have a thin solid feature inside the flow region it is not necessary to specify minimum wall thickness if it is greater than or equal to the minimum gap size. Specifying the minimum wall thickness is necessary if you want to resolve thin walls smaller than the smallest gap.

Click OK.

# Solution

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

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

This is the solution monitor dialog box. Notice for this tutorial that the SG Av Inlet Pressure and GG Av Pressure converged very quickly compared to the other goals. Generally different goals take more or less iterations to converge. The goal-oriented philosophy of COSMOS-FloWorks allows you to get the answers you need in the shortest amount of time. For example, if



New calculation

you were only interested in the pressure drop through the enclosure, COSMOS-FloWorks would have provided the result more quickly then if the solver was allowed to fully converge on all of the parameters.

### Viewing the Goals

### **COSMOSFIoWorks 2004 Tutorial**

Ý

# Viewing the Goals

- Right-click the **Results** icon and select **Load Results** to acti-1 vate the postprocessor.
- 2 Select the project's results (1.fld) file and click **Open**.
- 3 Right-click the Goals icon and select Create.



Load Results

<u>P</u>lot Manager..

Custom Parameters

- 4 Click Add All in the Goals dialog.
- 5 Click OK.

zailable:	Selected:	Goal filter
	GG Av Pressure	All
-	Add SG Av Inlet Pressu SG Outlet Mass FI VG Chip Max Tem VG Small Chips M	r C Plot options P Abscissa:
		Iterations
B	emove All	Templato:
	4 3	goalsxit

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

# Enclosure Assembly.SLDASM [Inlet Fan]

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

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

Goal's progress bar is a qualitative and quantitative characteristic of the goal's convergence process. When COSMOSFloWorks analyzes the goal's convergence, it calculates the goal's dispersion defined as the difference between the goal's maximum and minimum values over the analysis interval reckoned from the last iteration and compares this dispersion with the goal's convergence criterion

### **Flow Trajectories**

dispersion, either specified by you or automatically determined by COSMOSFloWorks as a fraction of the goal's physical parameter dispersion over the computational domain. The percentage of the goal's convergence criterion dispersion to the goal's real dispersion over the analysis interval is shown in the goal's convergence progress bar (when the goal's real dispersion becomes equal or smaller than the goal's convergence criterion dispersion, the progress bar is replaced by word "achieved"). Naturally, if the goal's real dispersion oscillates, the progress bar oscillates also, moreover, when a hard problem is solved, it can noticeably regress, in particular from the "achieved" level. The calculation can finish if the iterations (in travels) required for finishing the calculation have been performed, as well as if the goals' convergence criteria are satisfied before performing the required number of iterations. You can specify other finishing conditions at your discretion.

To analyze the results in more detail let us use the various COSMOSFloWorks post-processing tools. For the visualization of how the fluid flows inside the enclosure the best method is to create flow trajectories.

# Flow Trajectories

- 1 Right-click the Flow Trajectories icon C Inlet Fan Flow Traie Input Data Computational Domai
   Computational Domai
   L
   Global Coordinate Sys
   Component Control Definition Table Settings Particle Excel Animation 👍 Isosurfaces Flow Trajec and select Insert. Direct Both \* F Physical particle 😽 XY Plots undary Conditions Static Pressure1 Start points from External Inlet Fault ٠ 2 In the COSMOSFloWorks design tree Heat Sources Main Chip Chips Capacitors Power Supply select the External Inlet Fan1 item. This Number of tra -200 selects the inner face of the Inlet Lid. Material Conditio
- 3 Set the Number of trajectories to 200.
- Chips Silicon
   Chips Silicon
   Heat Sink Alt
   Lids Insulator
   PCB Epoxy 4
- 4 Keep the **Reference** in the **Start points from** list.
- If **Reference** is selected, then the trajectory start points are taken from this selected face.

- **Flow Trajectories** 
  - **5** Click **View Settings**.
  - 6 In the View Settings dialog box, change the Parameter from Pressure to Velocity.
  - 7 Go to the Flow Trajectories tab and notice that the Use from contours option is selected.
  - This setting defines how trajectories are colored. If Use from contours is selected then the trajectories are colored with the distribution of the parameter specified on the Contours tab (Velocity in our case). If you select Use fixed color then all flow trajectories have the same color that you specify on the Settings tab of the Flow Trajectories dialog box.



- 8 Click **OK** to save the changes and exit the **View Settings** dialog box.
- **9** In the **Flow Trajectories** dialog box click **OK**. The new **Flow Trajectories 1** item appears in the COSMOSFloWorks design tree.

This is the picture you should see.

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



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

Let us see the velocity distribution in more detail.

# **Cut Plots**

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

- 2 Keep the **Front** plane as the section plane.
- 3 Click View Settings.
- 4 Change the Min and Max values to 0 and 10 respectively. The specified integer values produce a palette where it is more easy to determine the value.
- 5 Set the Number of colors to 30.
- 6 Click OK.

🔏 Point Parameters 💫 Goals ? × Solid-OK Porous Contours Contour Apply 🗖 Isojines 🗖 isglines ■ Vectors ⊻iew Settings Cancel Reference -<u>H</u>elp Section plane/face \* ? ×

Isosurfaces Contours	Op Isolines	ions   Vectors	Coordinate S	jystem jectories	ОК
Settings Parameter:		Velocity	•		Apply
<u>M</u> in: 0 ft/s May: 9 0418202	19.ft/o	0 ft/s			<u>O</u> pen
Pajette:		Palette 6			Save As Beset Min/May
Number of colors	:				Cancel
	] ·	30			<u>H</u> elp

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

Cut Plot

Fluid

0 in

□ Vectors





🗄 🙀 Results 🔶 Cut Pl

### **Cut Plots**

### **Cut Plots**

8

Select the **Top** view.

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

- **9** Double-click the palette bar in the upper left corner of the graphics area. The **View Settings** dialog appears.
- **10** Change the **Parameter** from **Velocity** to **Temperature**.
- **11** Change the **Min** and **Max** values to 50 and 120 respectively.

View Settings	? ×
Isosurfaces     Options     Coordinate System       Contours     Isolines     Vectors     Flow Trajectories       Barameter:     Temperature     Image: Coordinate System       Min:     49.3961072 'F     50 'F     Image: Coordinate System       Mag:     120.000005 'F     120 'F     Image: Coordinate System       Palette:     Palette 6     Image: Coordinate System       Mumber of colors:     Image: System System     30	OK Apply Open Save As Beset Min/Max Cancel Help

Velocity [ft/s]

- 12 Click the **Vectors** tab and change the **Arrow size** to 0.2 by typing the value in the box under the slider.
- Notice that you can specify a value that is outside of the slider's range.
- 13 Set the Max value to 1 ft/s.

v Settings		Ŷ
Isosurfaces 0 Contours Isolines	otions Coordinate System Vectors Flow Trajectories	ок
C Use from contours	Use fixed color	Apply
<u>P</u> arameter:	Velocity	<u>0</u> pen
Min: Oft/s	0 ft/s	<u>S</u> ave As
Ma <u>x</u> : 9.04182028 ht/s		Reset Min/Ma
J	Projected Vectors	Cancel
0.2 in		<u>H</u> elp

- By specifying the custom Min and Max values you can control the vector length. The vectors whose velocity exceeds the specified Max value will have the same length as the vectors whose velocity is equal to the Max. Likewise, the vectors whose velocity is less than the specified Min value will have the same length as the vectors whose velocity is equal to the *Min.* We have set 1ft/s to display areas of low velocity.
- 14 Click **OK**.
- 15 Right-click the **Cut Plot 1** item and select **Edit Definition**. 🗄 🐯 Results 🔆 Cut Plots 🔶 Cut Plat 1 Ed 💠 Surface I 👍 Isosurfac 16 Select the **Vectors** check box. Cut Plot 2 X Definition Settings Region Animation OK Fluid Porous 17 Change the **Section position** to Contours Contours Contour Apply □ <u>I</u>solines □ Isoline 🗖 Is<u>o</u>lines -0.2 in. ⊻iew Settings... Vectors 🗖 Vector Cancel 18 Go to the **Settings** tab. Using the Section plane definition Beference <u>H</u>elp slider set the **Vector spacing** to Section plane/face Section position 0.18 in. -0.2 in \* Front 19 Click **OK**.

# **Cut Plots**

# **Surface Plots**

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



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

# **Surface Plots**

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



- 2 Click **Solid** as the **Medium**. Since the **Temperature** is the active parameter, you can display plots in solids; otherwise only the fluid medium would be available.
- Hold down the Ctrl key and select the Heat Sink - Aluminum and Chips - Silicon items in the COSMOS-FloWorks design tree.
- 4 Click **OK**. The creation of the surface plot may take a time because 76 faces need to be colored.



# **Surface Plots**

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

Now you need to hide the model because it overlaps the plots. In additional to using COSMOS-FloWorks menu or COSMOS-



FloWorks design tree it is very convenient to use COSMOSFloWorks Toolbars.

- 6 Click View, Toolbars, COSMOSFloWorks Results, Display.
- Click Display Outlines <sup>1</sup> to display a wireframe representation of the model.
   Then click Display Model Geometry <sup>1</sup> to hide the model.



### **Surface Plots**

### **COSMOSFIoWorks 2004 Tutorial**

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