

Fluid Mechanics Tutorial in SDRC Ideas

ENGG*2230 - Winter 2006

Preface

This tutorial has been created to assist you in conducting your assigned task. The tutorial runs through the steps associated with fluid modeling a *sudden contraction/sudden expansion* and will be the focus of the tutorial session. In addition to working with this tutorial you are encouraged to execute the tutorials offered within I-DEAS. Background Tutorials in ENGG*2100 and ENGG*2120 will have introduced you to many of these tutorials – although they are not necessary to complete the assignment.

- Design, Part Modeling - Fundamentals: Tutorials 1-23
- Simulation - Simulation Projects: Tutorials 1-31

Specific tutorials that will assist you in fluid modeling are:

- **Simulation - ESC Projects: Tutorial 1**
- You can find those tutorials inside the I-DEAS on the network. To do this, Go to Help (from one of the pull down menu)-->Help Library-->Tutorials.

Procedure Overview (details later)

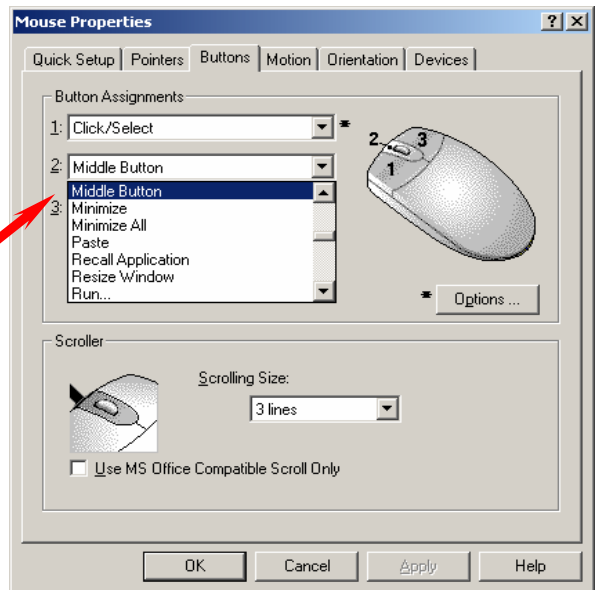
1. Draw your object. (A *Venturi* has been built and stored in the library for your use.)
2. Mesh your object.
 - a. Define a null shell (a 2D, zero thickness shell used at inlets and outlets)
 - b. Meshing is a step in which you define the total solid volume as the summation of a large number of 3D, finite elements. The larger the number of elements, the smaller each element is and the closer the numerical technique simulates the partial differential equations.
3. Assign boundary conditions.
 - a. No different than analytically solving differential equations. The initial and boundary conditions affect the solution and thus must be specified.
 - b. Boundary conditions are fans, vents, screens, blockages and surfaces. In this assignment you will use fans and vents.
4. Solve the model.
 - a. Select flow solving only.
 - b. Choose mixing length or turbulent solution as appropriate.
5. Post-Processing of the results.

- a. That is, convert them into a form that aids interpretation (often graphical).
6. Think about the results.
 - a. Do you believe them? Think about your fundamental fluid mechanics in relation to your results. You may realize that a smaller mesh size was necessary or that different boundary conditions should have been applied. You may decide that the model isn't well suited to your application.
7. Interpret your results.
 - a. To make a decision. To write a report. To modify a design.

******Important**** Before starting this tutorial it is highly recommended that you perform the online tutorial under Simulation.../4. ESC Projects/1. Cooling a Simple Electronics System**

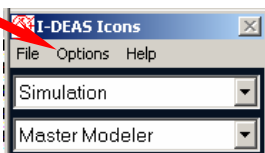
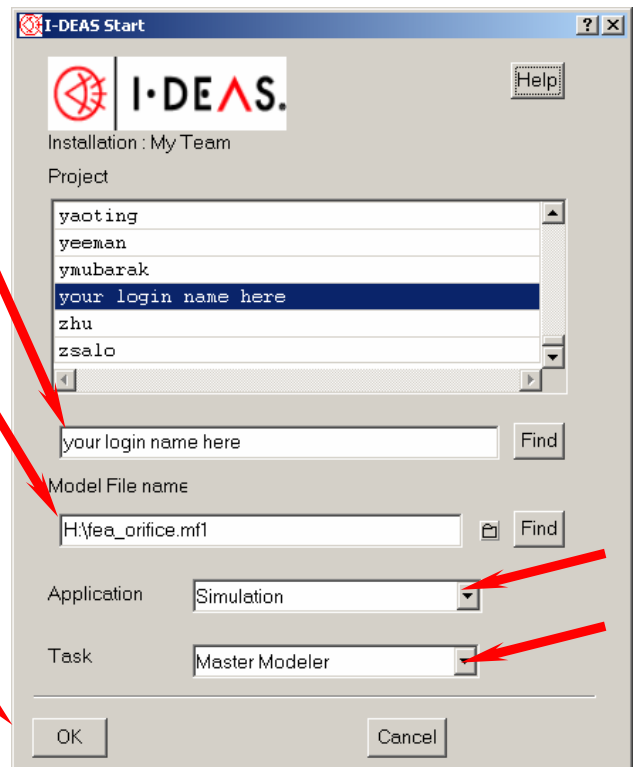
Before you start I-DEAS
Changing your mouse for I-DEAS

- Log on to one of computers in the School network
- Double click on the mouse icon on the bottom right of your screen
 - Within the *Mouse Properties* menu click on the Buttons tab and select the **Middle Button** option



Procedure Details
Entering I-DEAS

- Log on to one of computers in the School network
- Windows start menu to I-DEAS 11, I-DEAS Open GL
 - The I-DEAS Start Screen
 - **Project** ... “your login name”
 - **Model File name** ... H:\Venturi_FEA.mfl (this will save your model file in your H: drive)
 - **Application** ... choose “Simulation”
 - **Task** ... start in “Master Modeler”
 - <OK>
 - Drawing Screen
 - Click on **Options**
 - Change **Units** to **mm (newton)**



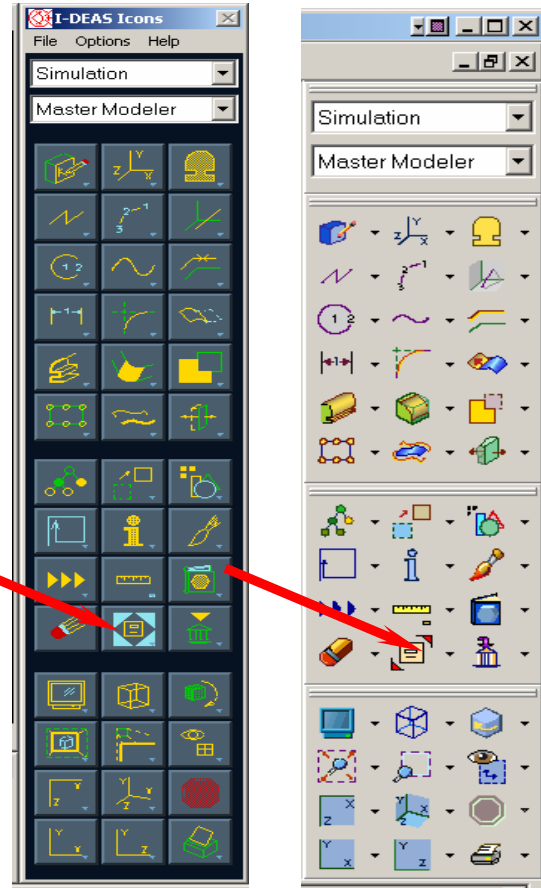
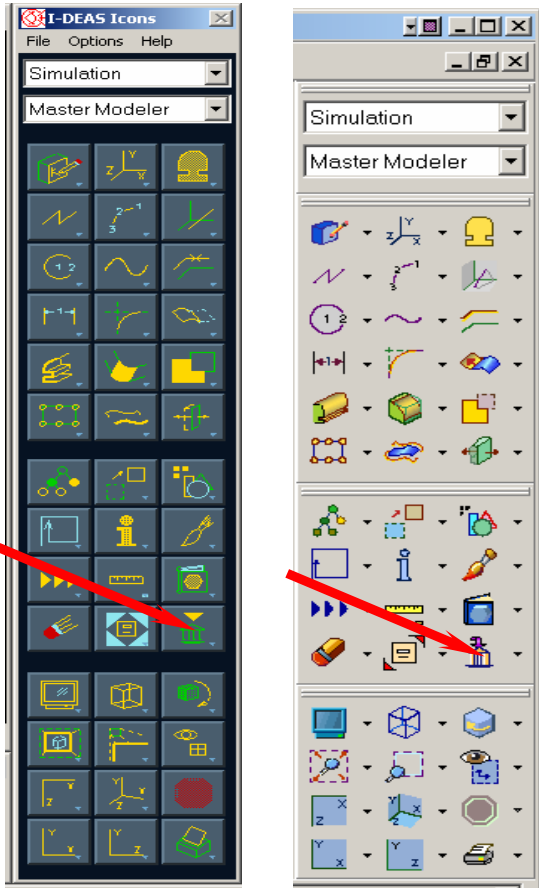
Getting the Object

A project named *ENGG2230-W06* has been created. Within *ENGG2230-W06* is a library named **Tutorial Parts** and within this library is a part named **Venturi**. The pipe diameter $D=50\text{mm}$. The entrance contraction ratio will be held constant at 4:1. The expansion will be held constant at 16:1. You will probably have to change the throat diameter d (this should be the only dimension you will have to change).

Take a **copy** of this part from the library to complete your assignment. After retrieving the part from the library it can be found in your bin. You must then **get** the part from your bin to begin working on it. **(if you are having troubles using the Libraries review the online tutorial under Design Part Modeling/1. Fundamentals/8. Using Libraries with Full I-DEAS Data Management)**

Retrieve the Part from Library

Get the Part from Your Bin



Old Panel

New Panel

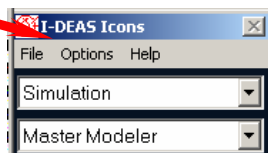
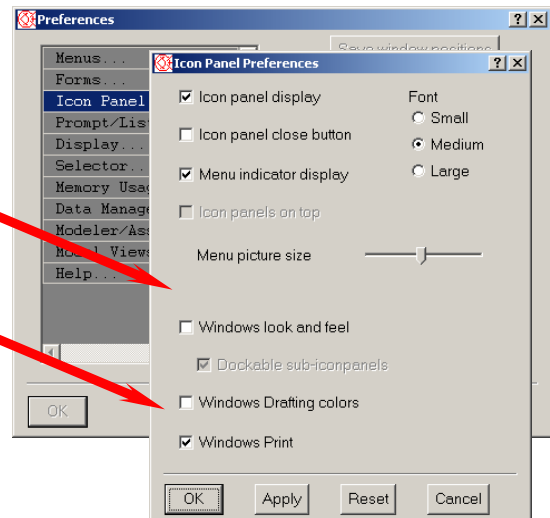
Old Panel

New Panel

******Important****** If your Panel looks like the New Panel please use the instructions below to change your view to the Old Panel view. (this will help you with the tutorial)

Old Panel view: go to Options/Preferences/Icon Panel and toggle off **Windows look and feel** and toggle on **Windows Print** if it is not toggled on already.

Close I-DEAS and restart it.



Editing the Venturi

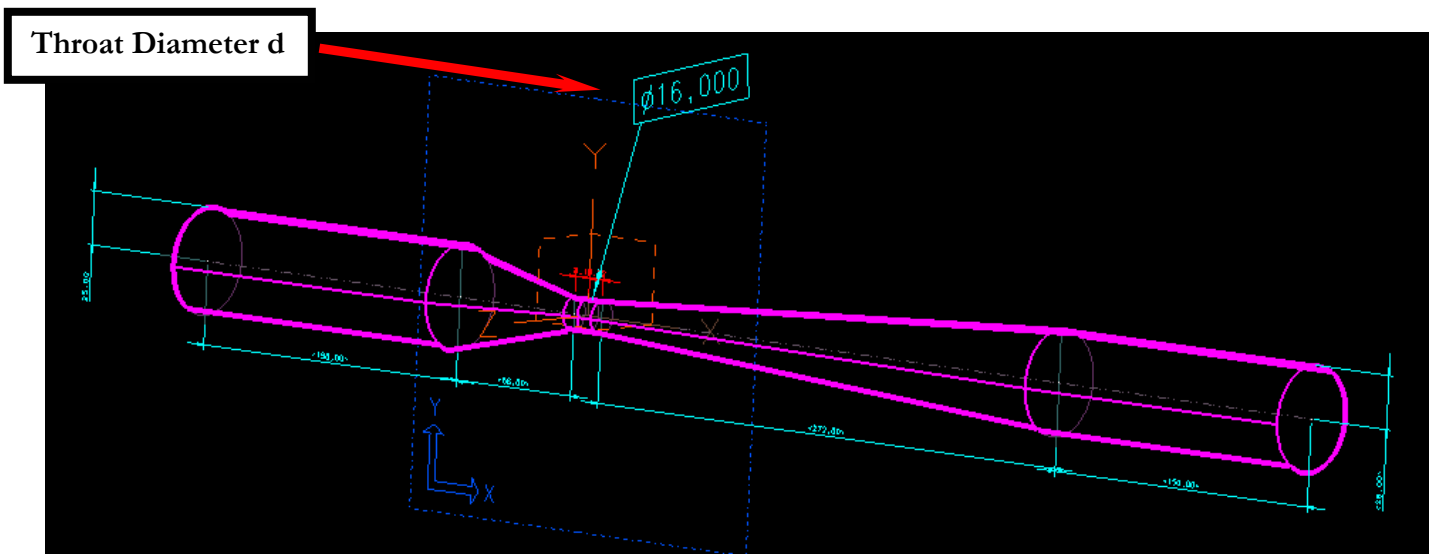
Note: Use F2 (zoom) to get whole part on screen and use F1 move part on the screen.

- Select the **Modify** icon.
- Select the Venturi on the screen by clicking on it
- Click <enter> the middle mouse button.
- A new menu will appear, choose: **Show Dimensions** and change the **Throat diameter d** visible on the screen to **16mm**, then **update** your part (if you are having difficulties with editing your part review the online tutorial under **Design Part Modeling/1. Fundamentals/17. Modifying Features**)
- **Or** Choose: Dimension Values (The *Dimensions* Table should appear in the window.)

Dimensions Table

Name	Expression	Displayed Value
Throat_Diameter_d	16	
AlongVecDist	8.125	
AgainstVecDist	8.125	
TwistAngle	0	
Draft1	0	
Upper_Case_D	25	
Upstream_Extension	Upper_Case_D*6	150 [mm]
I.1	From Equation	68 [mm]

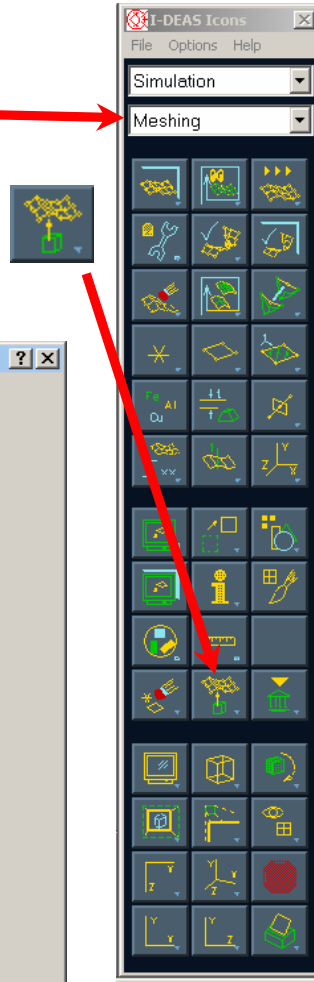
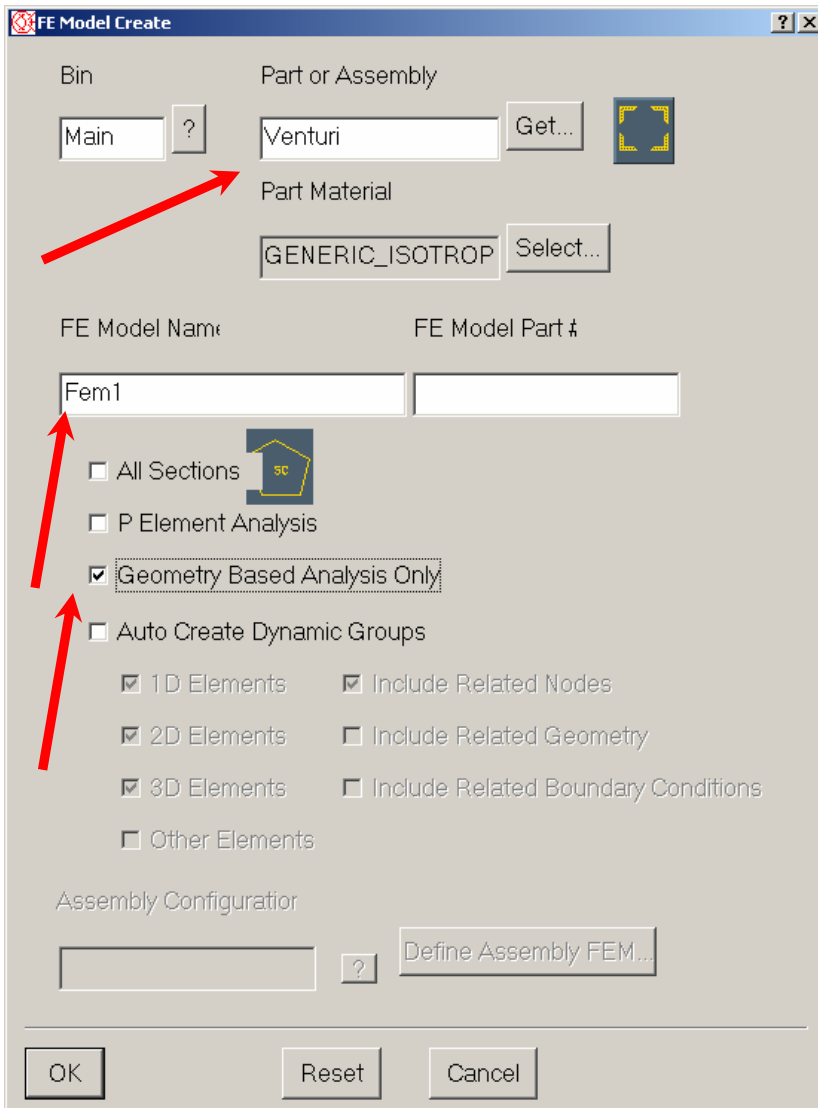
- Click <OK> once you have modified the throat diameter (This may be a good stage to save your model. Remember you can return to your last save with CTRL-Z)



Meshing

- Switch Task to **Meshing**
Note Change in Icon Panel


- Create an FEM (Finite Element Model) click on the **FE Model Create** icon
FEM Create Form



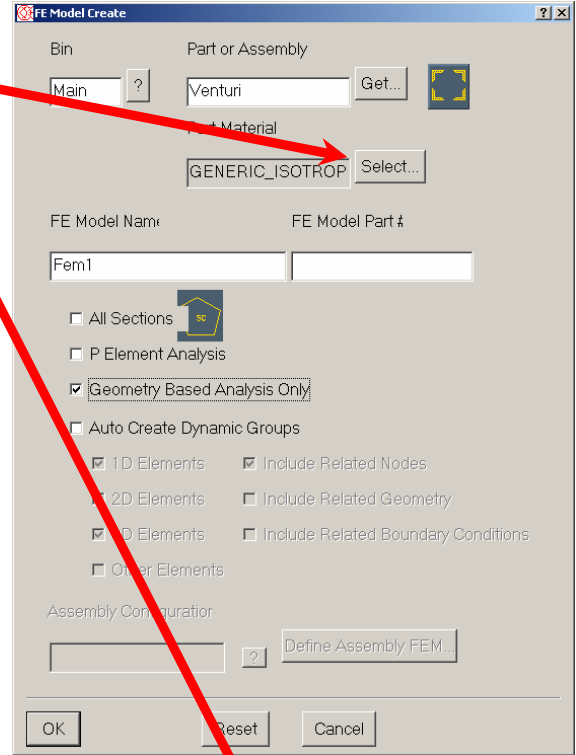
*****Important*****
(if you are having difficulties creating a part material, perform the online tutorial under **Simulation.../4. ESC Projects/1. Cooling a Simple Electronics System**)

- Your part name should appear in the **Part or Assembly** window in the *FE Model Create* form.
- **FE Model Name** defaults to **FEM1**, you may choose to enter your a name that might have more meaning for you.
- Toggle on the **Geometry Based Analysis Only** option

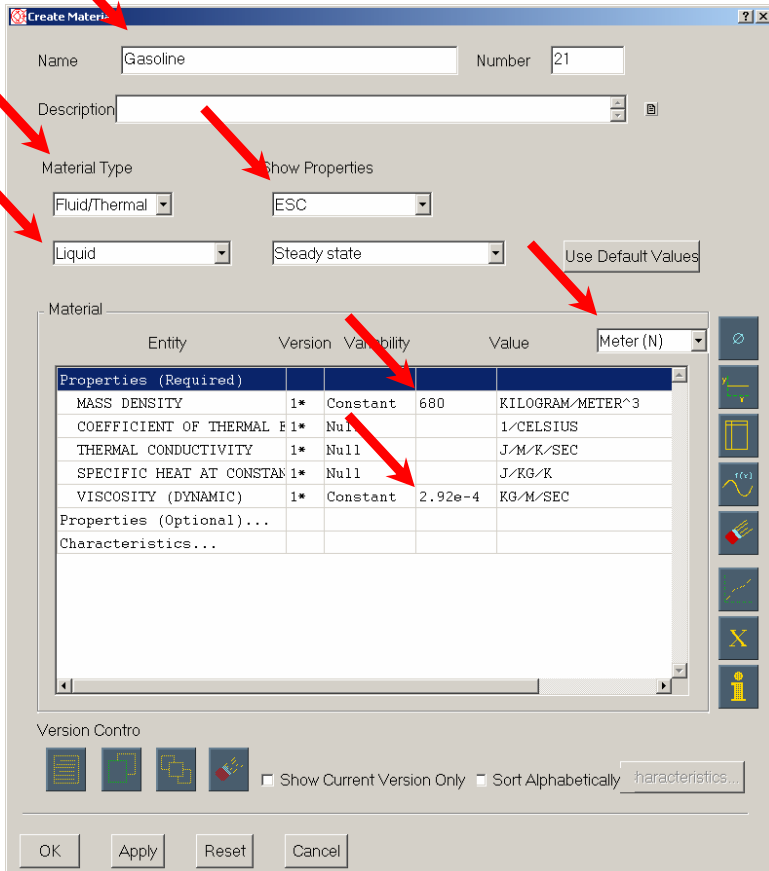
Meshing Cont.

- To change the **Part Material** click the Select... button
- In the **Materials** form click on the **Create** icon 
- Within the **Create Materials** form, enter the **Name: Gasoline**, change the **Material Type** to **Fluid/Thermal** and **Liquid**, change the **Show Properties** to **ESC**.
- Change the units to **Meter (N)** within the form
- In order to conduct a fluid mechanics analysis with I-DEAS a material must have the **Mass Density** and **absolute (Dynamic) Viscosity** μ defined (watch units)
- Make the **MASS DENSITY** 680 KG/M^3
- Make the **Dynamic Viscosity** 2.92e-4 KG/M/SEC, enter <OK>

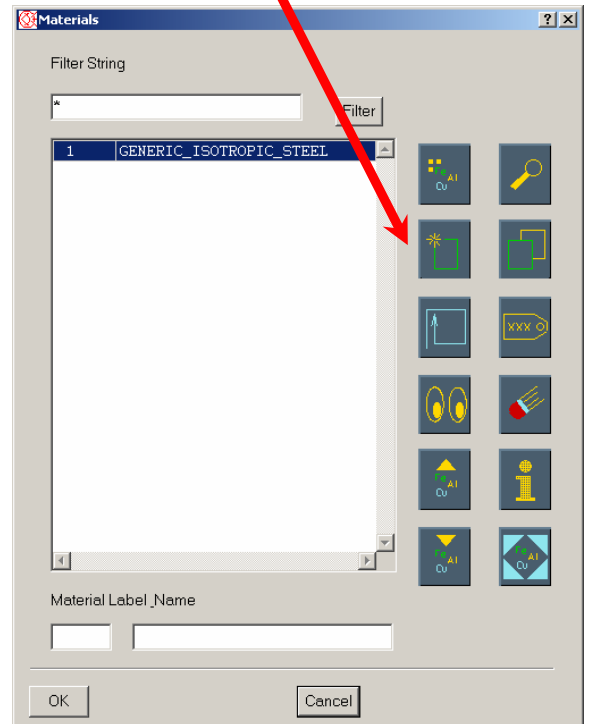
FEM Create form



Create Materials form



Materials form



Meshing Cont.

- Click on the **Material Type Filter** icon within the *Materials* form
- Unclick the **Finite Element Modeling** box
- Toggle on the **Fluid/Thermal** option
- Select **LIQUID** then click <OK>
- Select (highlight) your newly created material: **Gasoline**
- Click <OK> to the *Materials* form and the *FE Model Create* form.
- A window will appear with a warning. Click <Dismiss>.



Materials form

Material Type Filter Form


Materials

Materials

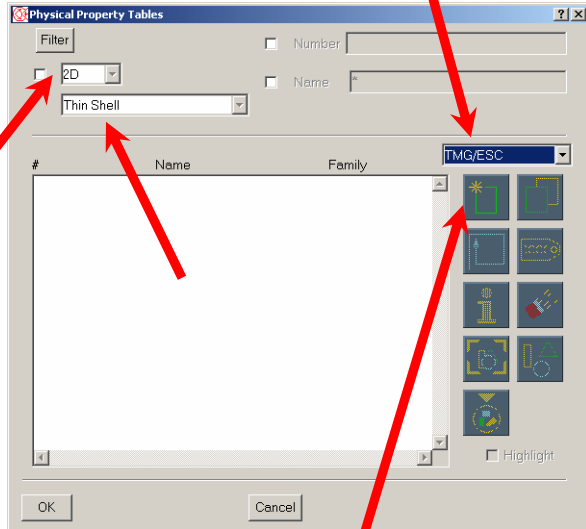
*****Important*****
(if you are having difficulties creating a part material, perform the online tutorial under Simulation.../4. ESC Projects/1. Cooling a Simple Electronics System)


****Select <OK> until you are done with the forms
Dismiss the errors and warnings**

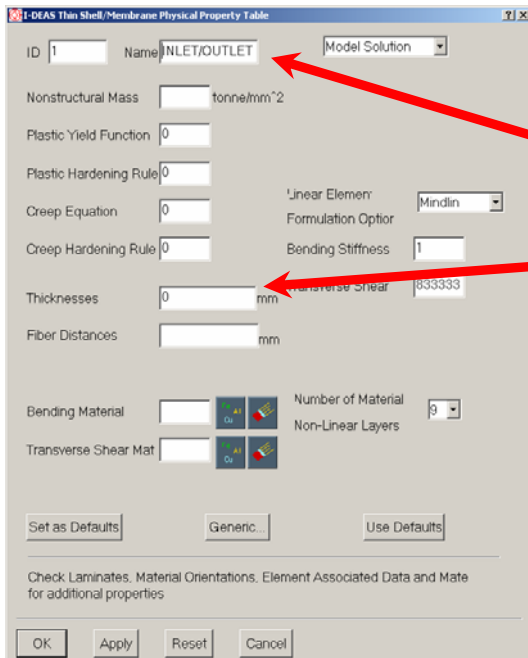
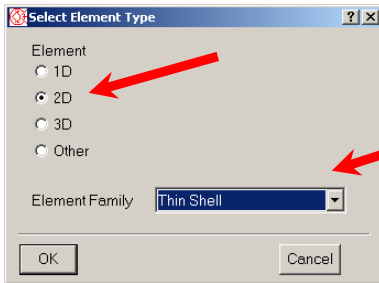
Establishing a Null Shell

- Choose the **Physical Property Icon** 
- Within the *Physical Properties Table* form:
- Select....
2D Thin Shell for TMG/ESC

Physical Property Table



- Select the **New Table icon** 
- Make sure the Element Type is 2D and Thin Shell




*****Important*****
 (if you are having difficulties creating a Null Shell, perform the online tutorial under Simulation.../4. ESC Projects/1. Cooling a Simple Electronics System)

In the *I-DEAS Thin Shell/Membrane Physical Property Table*:

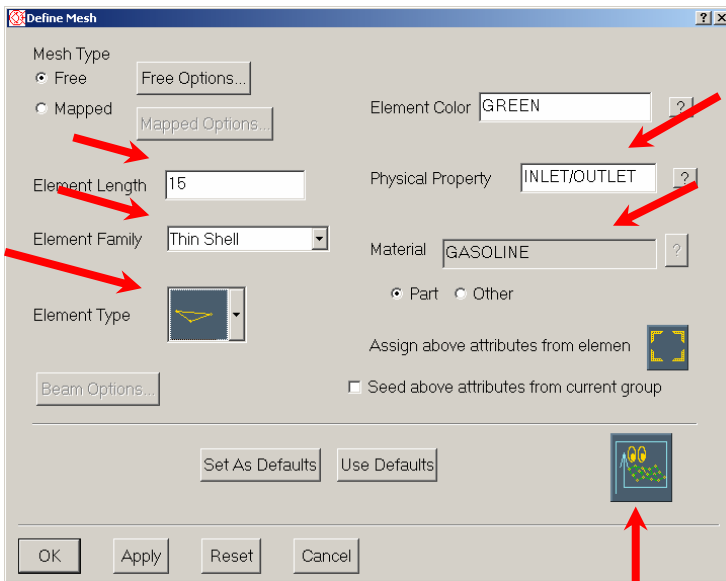
- Change name to “Inlet/Outlet”
- Thickness set to 0mm.
(This will serve as the fan and vent location on the 2D surfaces of your system)
- Select <OK> and <OK> again to get out of *Physical Property Table*


Shell Meshing the Inlet/Outlet surfaces

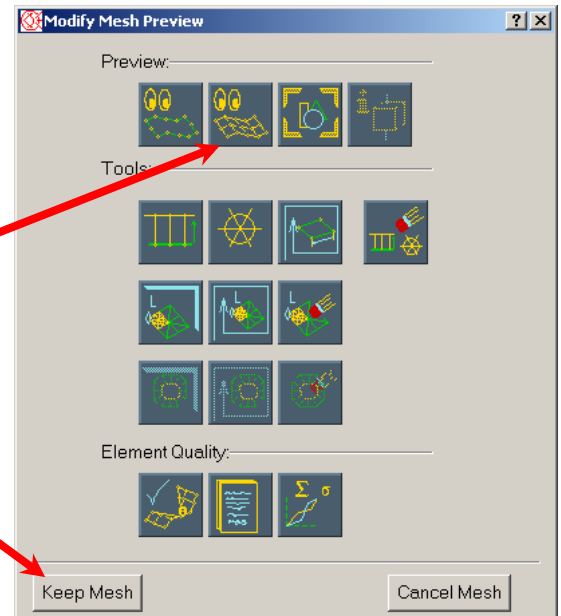
- Choose the **Define Shell Mesh** icon 
- Holding down your shift key on the keyboard select the two surfaces that correspond to the location for your fan and for your vent (i.e. your inlet and outlet)
- Then select <enter> your middle mouse button
- **Within the Define Mesh form:**
 - Use the default (make 15mm the default value you will use for your assignment)
Element Length
 - Set the **Element Family** to **Thin Shell** and set the **Element Type** to **3pt linear triangle**
 - Make sure the **Physical Property** is set to **INLET/OUTLET** and the **Material** is **Gasoline**





*****Important*****
 (if you are having difficulties creating a Shell Mesh, perform the online tutorial under Simulation.../4. ESC Projects/1. Cooling a Simple Electronics System)

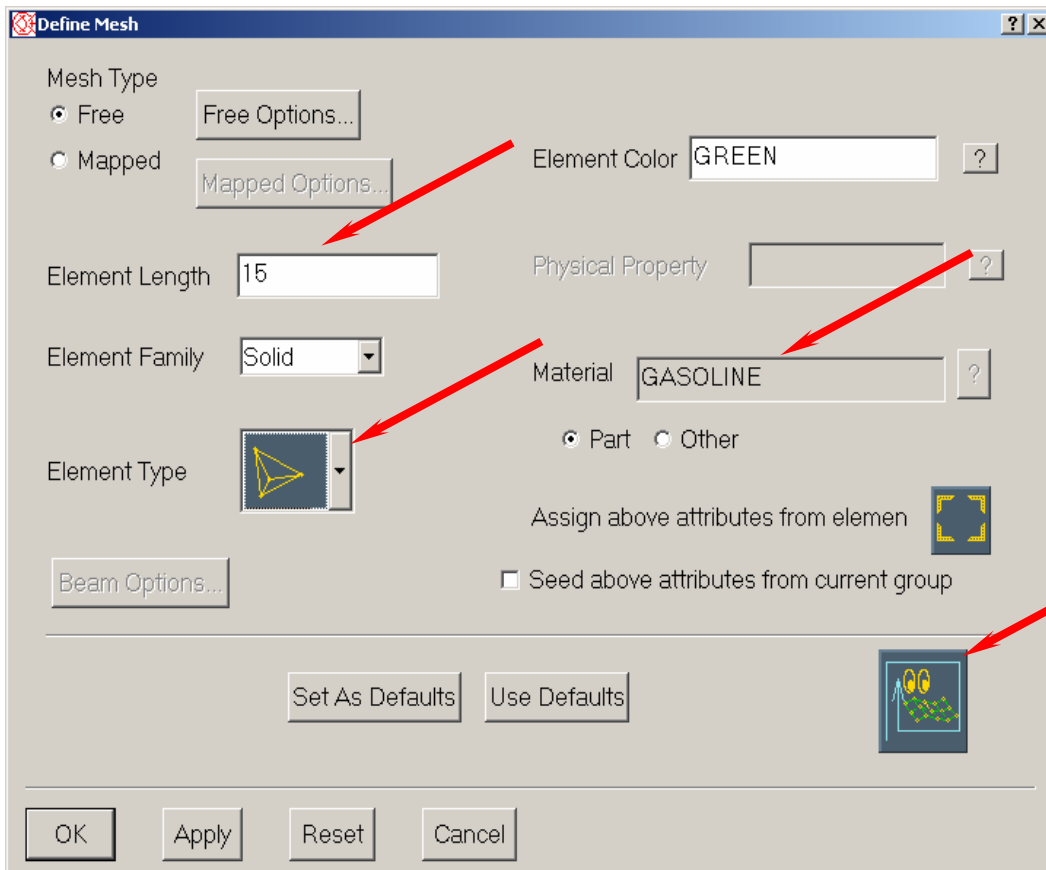
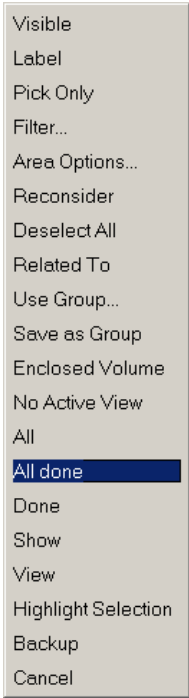


- Select the **Modify Mesh Preview** icon (Googly eyes) preview mesh by selecting the **Mesh** icon within the *Modify Mesh Preview* form 
- **Keep Mesh** - after previewing, you may choose to keep the define mesh or edit it by canceling and returning to the *Define Mesh* form
- Page 11 shows what the thin shell mesh will look like



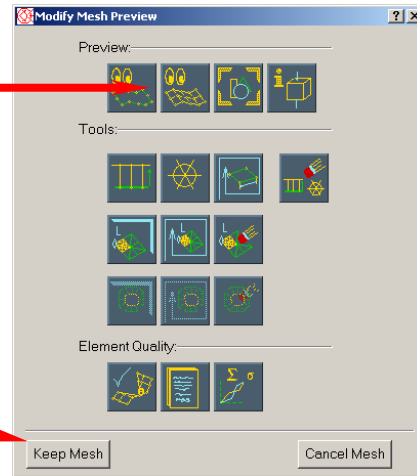
Solid Meshing

- Choose **Define Solid Mesh** icon 
- The I-DEAS Prompt window (at the bottom of the screen) will ask you what Volumes do you want to pick
- Move the cursor over the view port and hold the third mouse button down and choose **All Done** (this will automatically choose all the volumes on the screen)
- The *Define Mesh* form will now appear
- Make sure the material is set to **Gasoline**, set the element length to **15mm** then select the **Modify Mesh Preview** button 

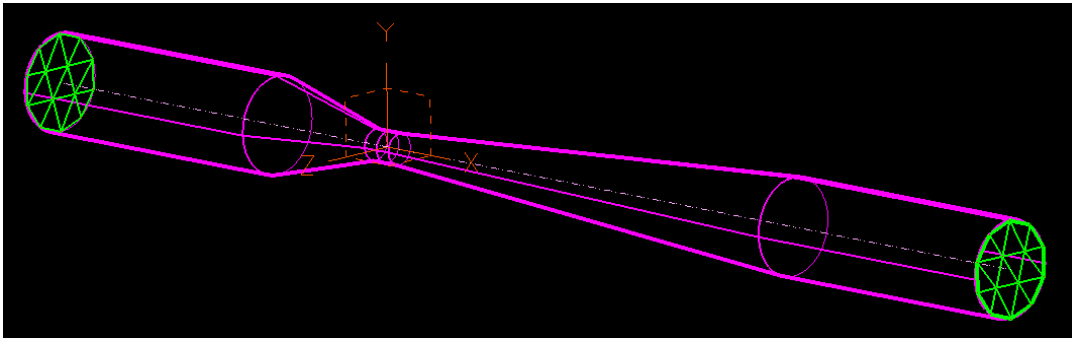


Solid Meshing Continued,

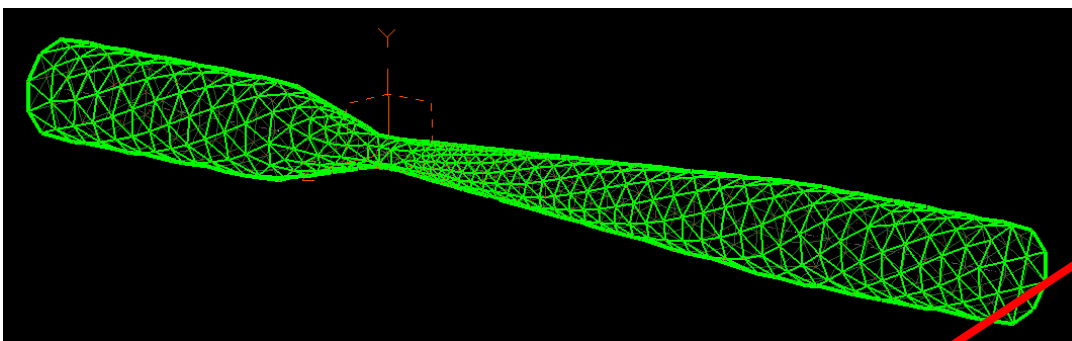
- Select the Mesh Preview icon, the number of elements and nodes will now appear at the top right of your screen. (You may want to record these numbers)
- After the software has successfully meshed the volume select **Keep Mesh**





Inlet/Outlet Thin Shell Mesh




Solid Mesh

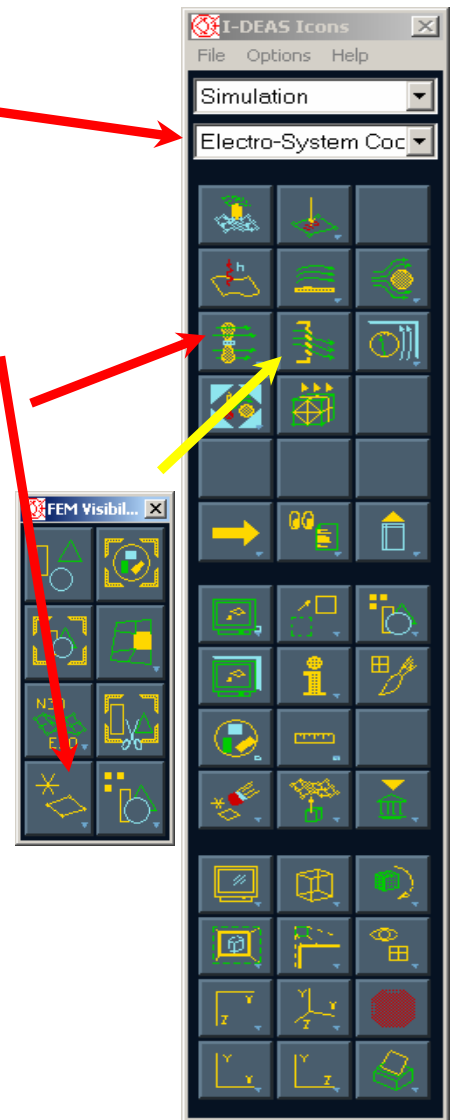


- In order to better visualize your model you may want to turn off the visibility of the mesh. You can do this by using the FEM Visibility Control Panel. Select the Opens FEM Visibility Control Panel icon 
- Within the *FEM Visibility Control* Panel the **Toggles on and off display of all FEM entities** icon will turn the visibility of FEM entities. 



Boundary Conditions

- Switch Task to **Electro-System Cooling** (Again note that the icon panel changes)
- In order to make the boundary conditions for inlet and outlet flow you must be able to select the models surfaces easily. The mesh may make selecting the surfaces difficult so you may want to hide the mesh
- Refer to the procedure on the previous page and hide the FEM entities
- Select the **Fan** icon and indicate done, select the fan surface (inlet end of your pipe) 
- Within the **Fan-Create** form Choose **Fan Type Inlet** so that pressures in your system are generally positive. Name the fan: **1.0m/s inlet**
- Select **Flow Parameter** and change to **Velocity**
- In the drop down units selection change the units to: **m/s** and make the magnitude **1**, click **<OK>**. (you have to calculate a velocity from your Reynolds number for your assignment)



Fan -Create Form

Name: 1.0m/s Inlet

Fan Type: Inlet

Flow Direction: As Shown

Flow Parameter: Velocity

Constant: 1 m/s


Vent -Create Form


Name: Vent 1

Vent to Ambient

Vent to Temperature: -6.10352E-006 [C]

and Pressure (absolute): 101.351 [mN/mm^2]

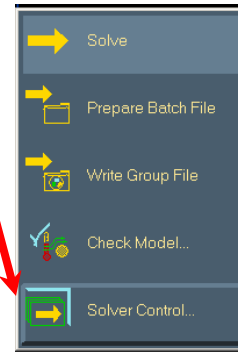
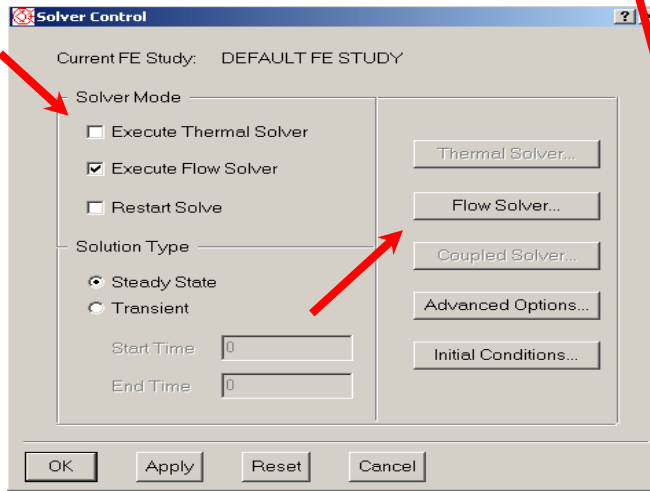
- Select **Vent** Icon (yellow arrow on the panel shown above) 
- Select the vent surface (one end or the other of your pipe)
- Indicate Done
- Within the **Vent-Create** form toggle **Vent to Ambient** on
- Click **<OK>**

(Default **Ambient conditions** are defined with the icon to the right of the Vent icon) 

Solver Control

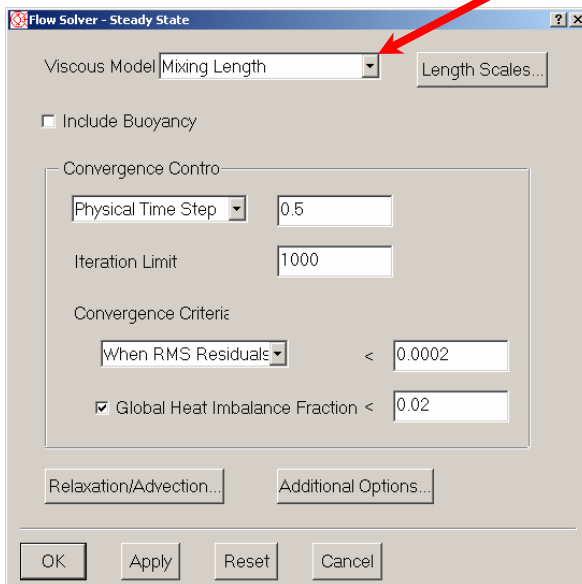
- Go to the **Solve** icon, hold it down and underneath is the **Solver Control...** icon
- Within the *Solver Control* form Toggle off **Execute Thermal Solver**


Solver Control Form



Flow Solver – Steady State Form

- Click on the **Flow Solver...** button
- Choose **Mixing Length** Viscous model
- **<OK>**, **<OK>**



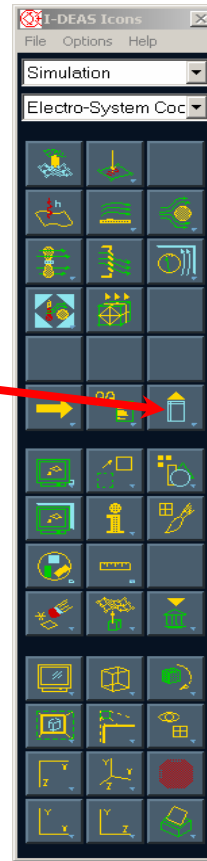
- **Save your model file**
- Select the **Solve** icon 
- The solve could take as long as 5 minutes

The software closes down all graphic displays/icons to devote resources to solving the FEM. You can follow the convergence by clicking on the **Flow** button within the *Solution Monitor* form. You should read and check all of the messages following completion. This may take several minutes to finish even for the default values.

When the I-DEAS Graphics screen returns then you need to GET RESULT

*****Under Options change the Units to Meter (Newton) this way your results for pressure will be in Pascals (Pa)

*****Important*****
 (if you are having difficulties solving your model or are having difficulties with the Post Processing of your results, perform the online tutorial under Simulation.../4. ESC Projects/1. Cooling a Simple Electronics System)



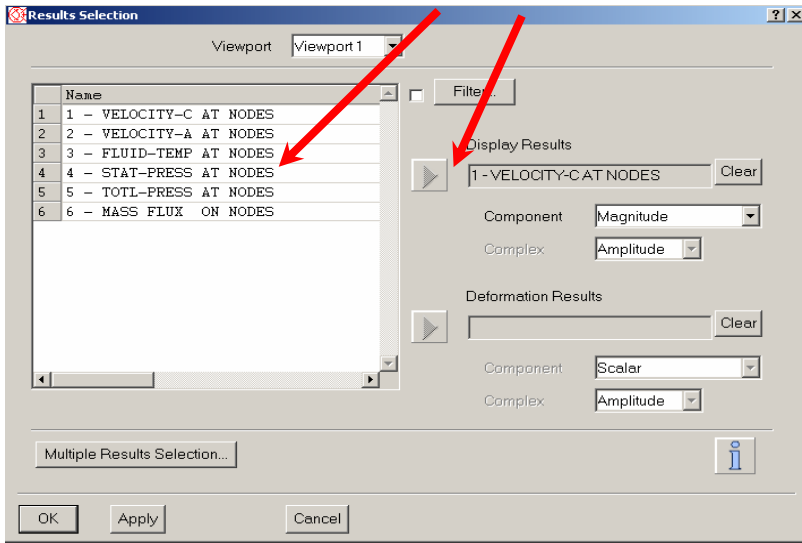
Change task to **Post Processing**

- Select the Select which results to display icon

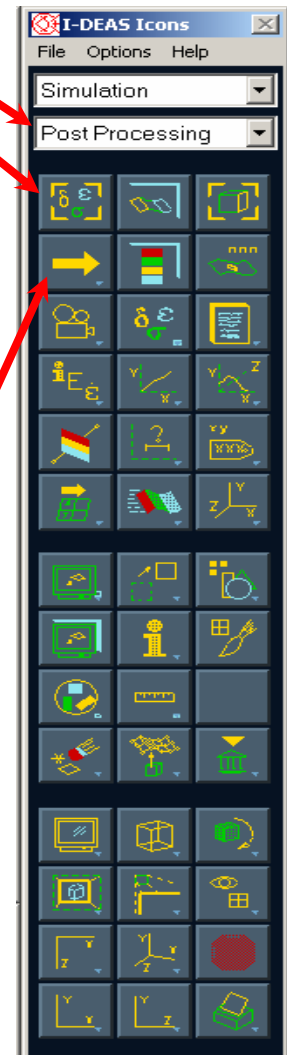


Results Selection form

- Velocities and pressures are of obvious interest but there are other choices worth exploring
- Select the **STAT-PRESS AT NODES** and use the Toggle arrow (little yellow arrow head) to move it into the **Display Results window**, <OK>





- Select the **Executes a Results Display icon** hold your third mouse button down, this will display a drop down menu. Select **All Done** (this will results associated with all elements)



Adjust display to suit interests

(Lots of different options.....Explore)

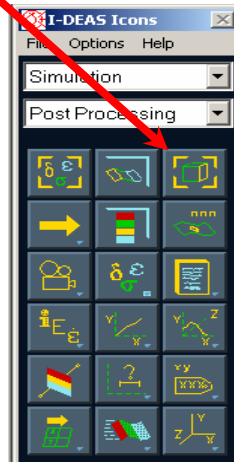
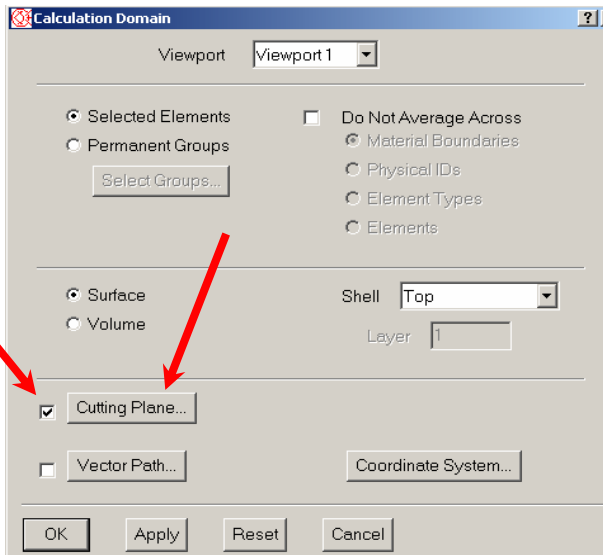
- To clear away the FEM results displayed select the **Wire Frame** view icon 
- Next we are going to create a cutting plane 
- Choose the **Calculation Domain** Icon



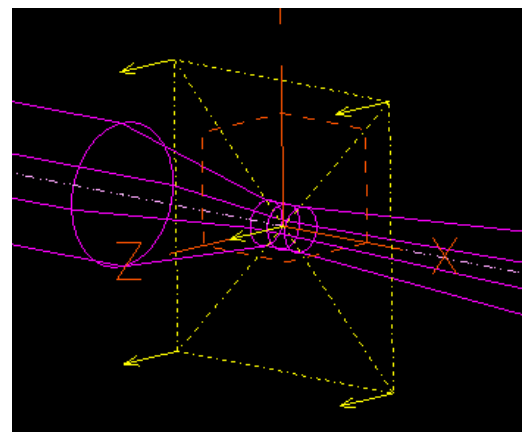
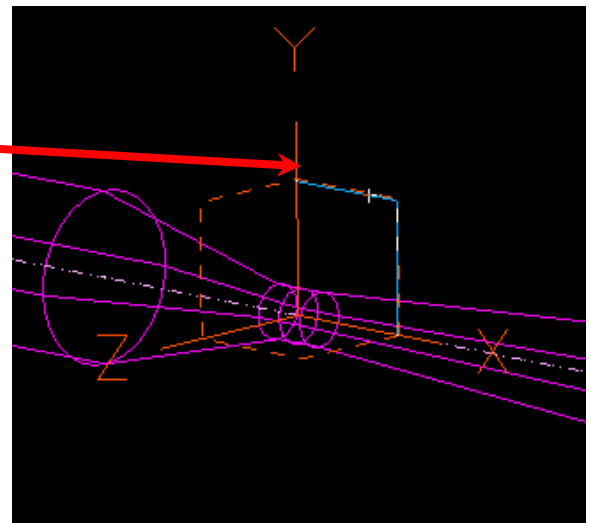
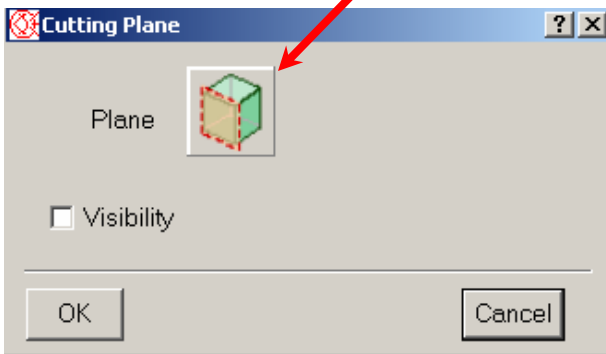
*****Important*****
(if you are having difficulties with the Post Processing of your results, perform the online tutorial under Simulation.../4. ESC Projects/1. Cooling a Simple Electronics System)

In the Calculation Domain form

- Toggle on **Cutting Plane**
- Select the **Cutting Plane...** button

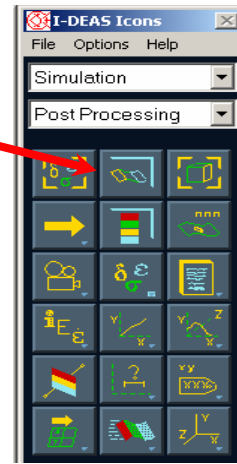
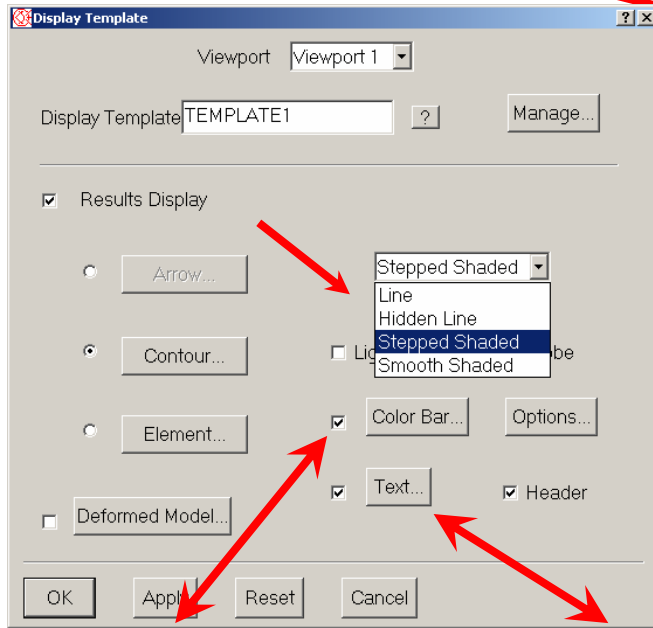


- Select the **Select Plane** button within the **Cutting Plane** form, in the viewport select the **XY** plane of your parts coordinate system
- Select **<OK>** twice

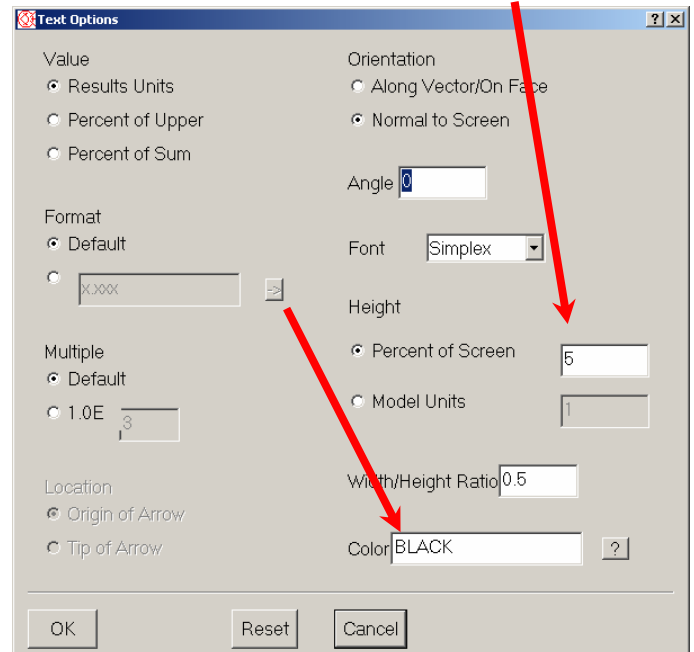
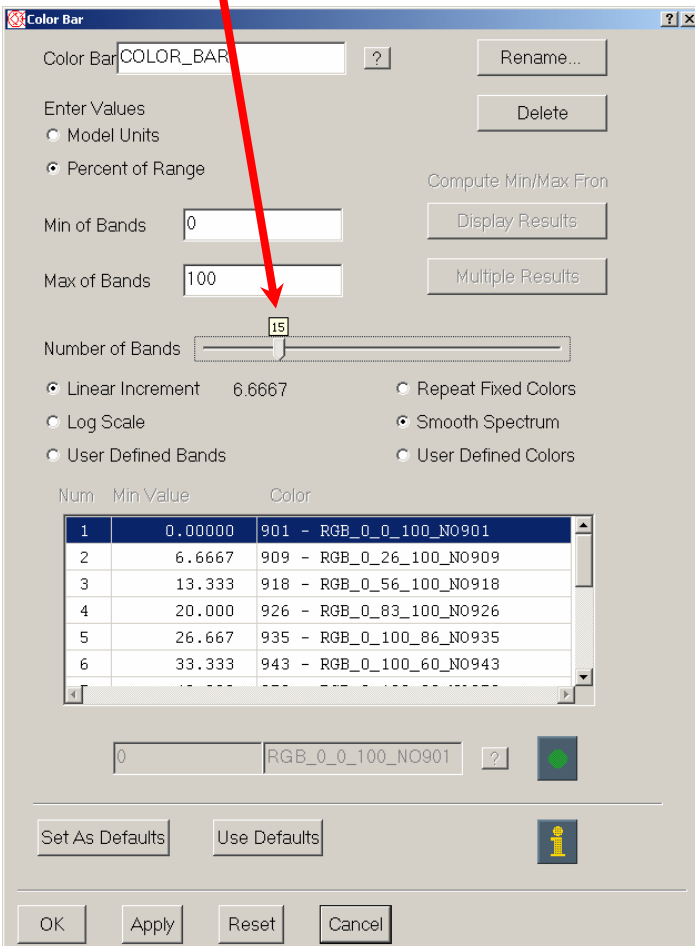


Adjust display to suit interests Cont.

- Select the **How results will be displayed** icon 

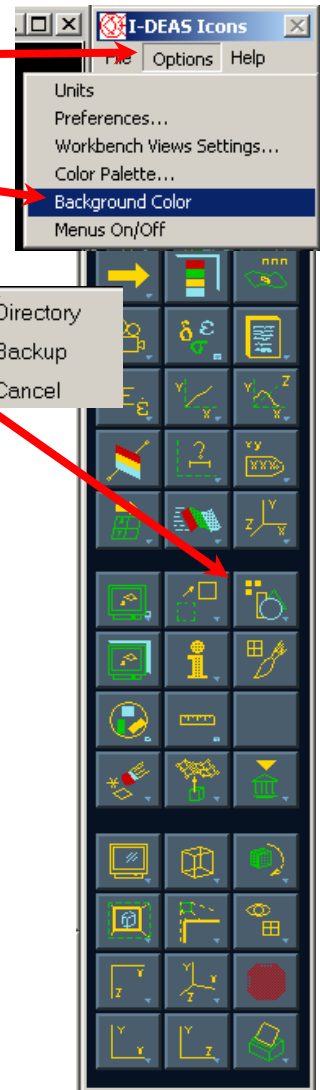
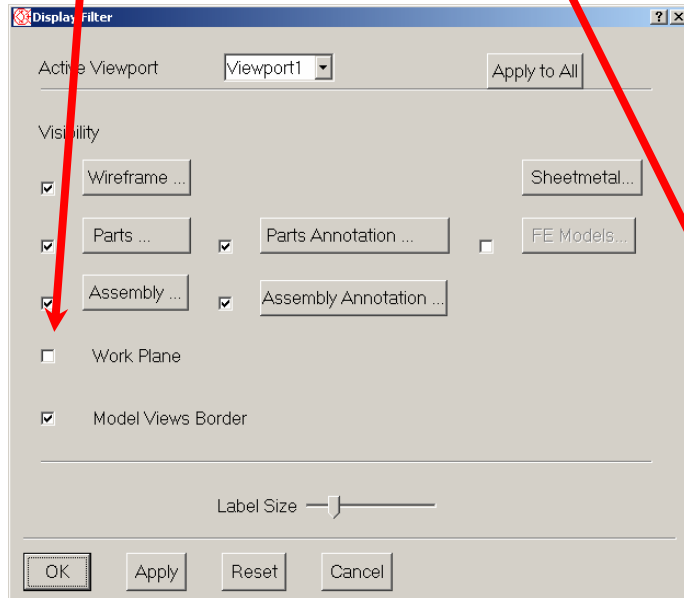


- Instead of Smooth Shaded, make the choice Stepped Shaded
- Toggle on the Color Bar and Text boxes
- Select the Color Bar button, change the number of band to 15 (below to the left) **<OK>**
- Select the Text button, change the Height Percent of Screen to 5 and make the color Black (blow to the right) **<OK>**
- <OK>**



Adjust display to suit interests Cont.

- Under Options select **Background Color**
- Choose **Directory**
- Choose **White**
- Select the **Controls Display of Entities** icon
- Within the Display Filter form deselect **Work Plane**
- **<OK>**



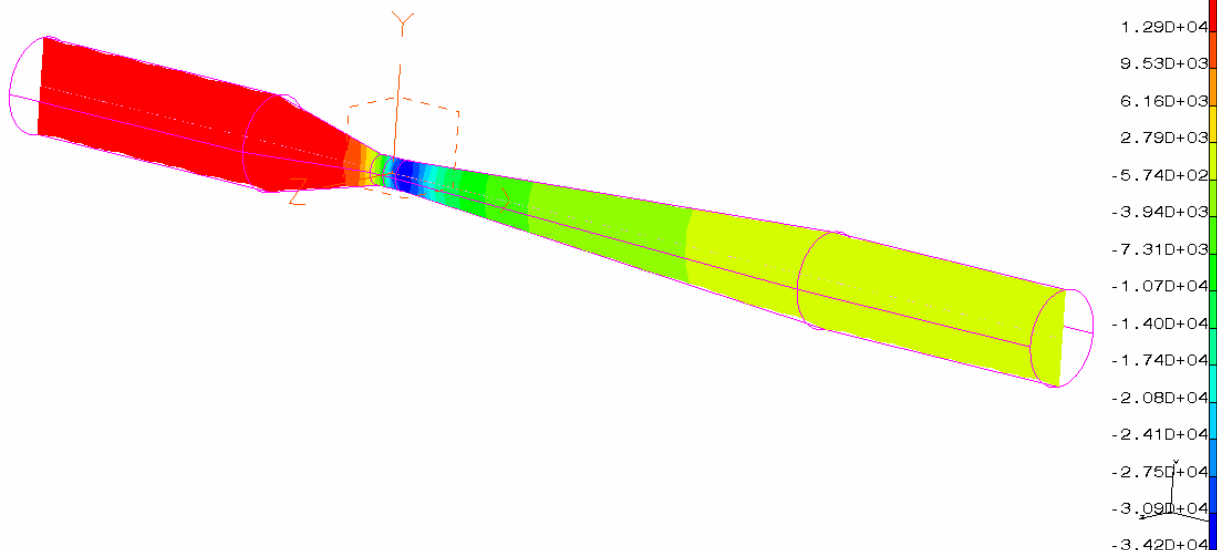
- Select the **Executes a Results Display** icon and hold your third mouse button down, this will display a drop down menu. Select **All Done** (this will results associated with all elements)

STAT-PRESS / AT NODES

RESULTS: 3-STAT-PRESS AT NODES

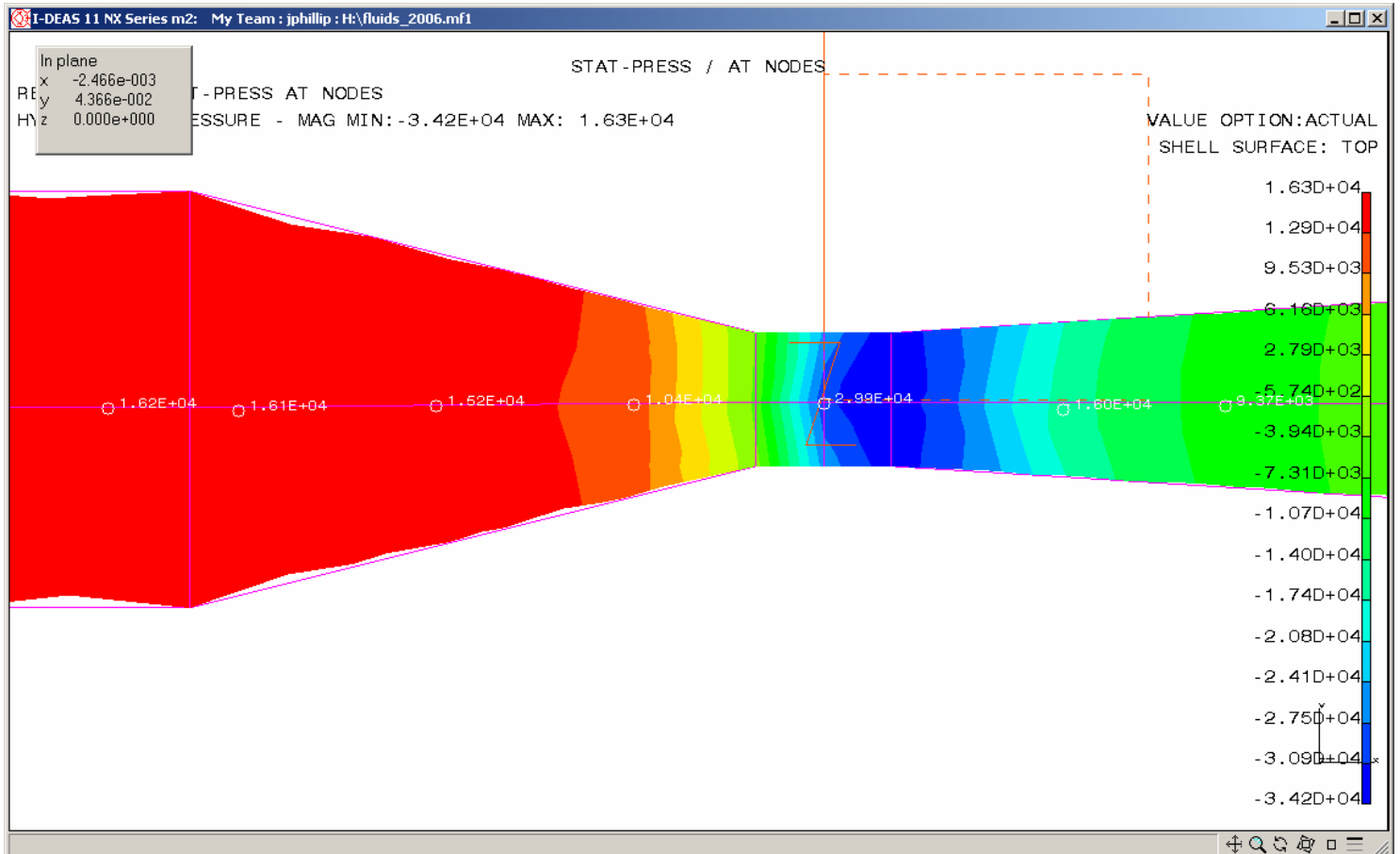
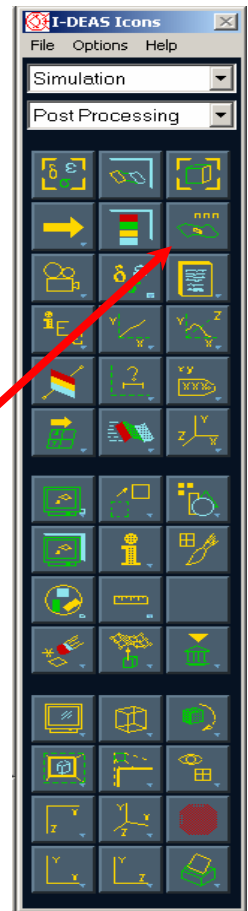
HYDROSTATIC PRESSURE - MAG MIN:-3.42E+04 MAX: 1.63E+04

VALUE OPTION:ACTUAL
SHELL SURFACE: TOP



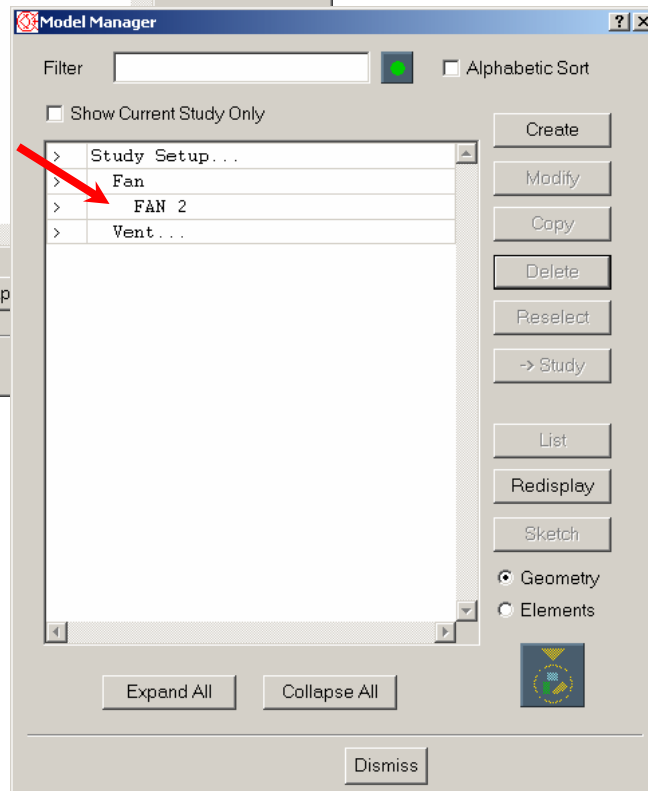
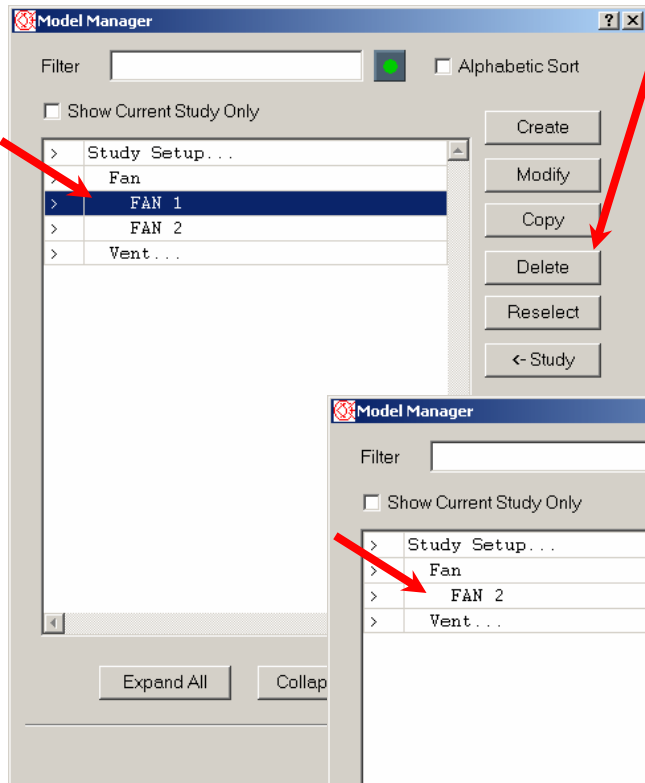
Collecting Results using the Probe

- You can use the Probe icon to generate a LIST of results from your model
- You can observe the values on the screen or by looking at the **LIST** region (bottom left). ******It's better to use the LIST region because it is hard to see negative values on the screen******
- Select the **Front View** icon and zoom in on your model
- Select the **Probe** icon and the position the mouse on the model on your screen. Where you click on your model the probe will display a value on the screen and in the **LIST** region
- You want to collect values upstream and downstream. It is up to you to judge where to take these measurements.



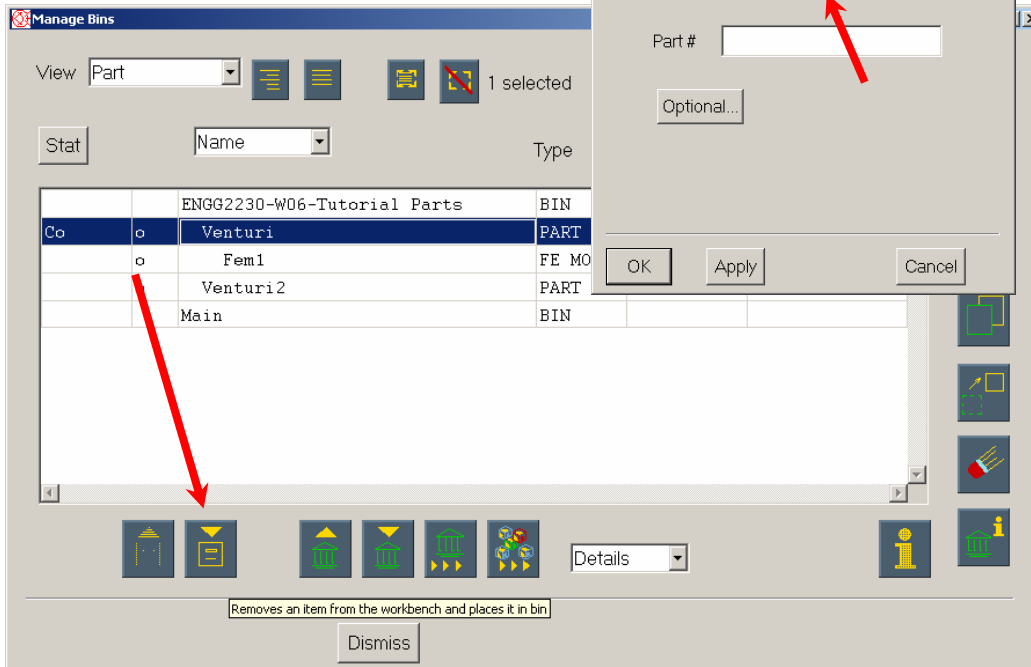
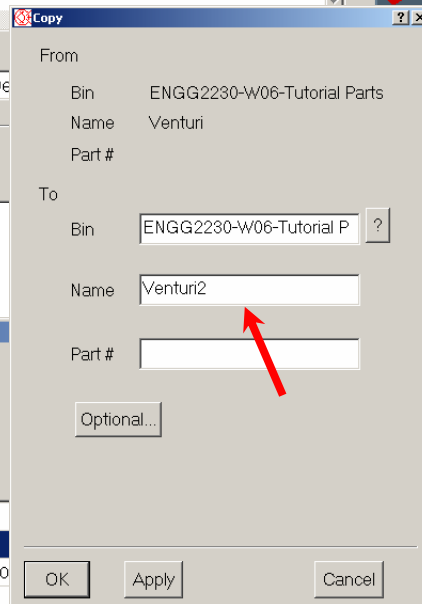
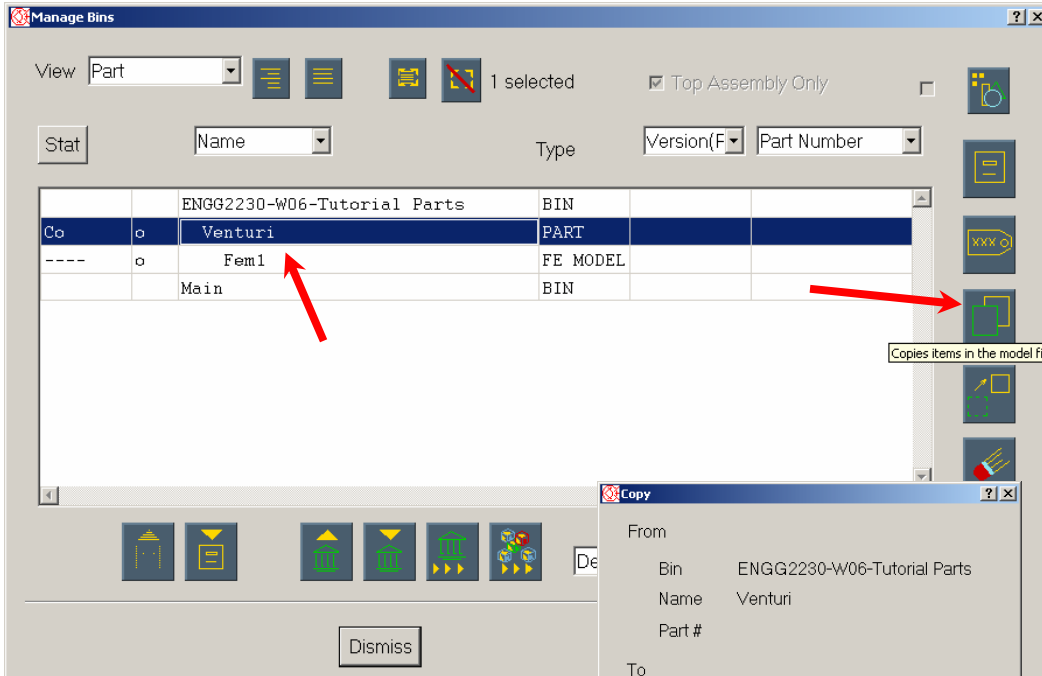
Performing another FE Study (adjusting the inlet velocity)

- When you need to change your inlet velocities you need to generate a new fan and make it current
- Make a new fan at the inlet with a different velocity, name the new fan FAN2 (Procedure shown on page 12)
- Select the **Model Manager** icon and in the *Model Manager* form highlight FAN1 and delete it with the **Delete** button (you might have to double click on Fan to see the underlying FAN. This makes FAN2 active for your model. <Dismiss>
- Run your model again. ****Make sure to look at the right results****



Performing another FE Study (generating a new mesh)

- To generate a new mesh go to the **Manage Bins** icon available in the **Master Modeler** task
- Highlight the Venturi model and then select the **copy** icon within the **Manage Bins** form, name the new model Venturi2 within the **Copy** form
- Put away the Venturi model using the **Put Away** icon in the **Manage Bins** form. This will leave the Venturi2 model on the screen
- Now create a new mesh using the Venturi2 model
- ******Remember to turn your element visibility back on (page 11)******



Common Problems

Your model will not finish running. There may be a problem with the mesh size. Change the size of the mesh by 0.1 and try again.

You keep getting the same results (when you don't think you should be). Make sure you import your new results before going to the Post Processing task.

You are starting I-DEAS and trying to open your model file and a warning comes up saying the model file is locked. What is happening is I-DEAS might have crashed when you shut it down last and therefore created a lock file (*.lck). Go into your H drive (where you are saving your model files) and delete the *.lck file.

Make sure you have enough room on your H drive to run your models. They could be as large as 50MB.

Sometimes it's hard to track down what you might have done wrong. It's a good idea to start over again and run your model a second time. You will usually find your error this way.