

ICEM CFD Tutorial

Parallel Plate Channel Grid

Gavin Joyce
Scott J. Ormiston

V2.10
13 October 2010

1. Introduction

This tutorial will guide you through the creation of a simple parallel plate mesh, and how to export that mesh into a CFX readable format.

2. Geometry Nomenclature

The following image shows the basic geometry of the parallel plates. The parallel plates problem is a 2D problem, but is solved in CFX using a 3D mesh. As such, our final geometry looks like a narrow box.

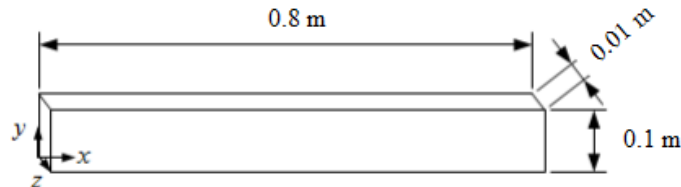


Figure 1: Parallel Plate Channel Geometry

3. Overview

The following is a summary of the general steps in mesh creation:

1. Creation of geometry
 - 1.1 Create points
 - 1.2 Use points to define curves
 - 1.3 Use curves to define surfaces
 - 1.4 Define the volume or 'body'
2. Create parts from surfaces
3. Create a block and associate the created geometry
4. Allocate nodal distribution along the block edges
5. Mesh the block
6. Create the file needed for input in CFX Pre

4. Setup

It is highly recommended that you create a new directory to work in. Several files will be created and it is much easier to find them later if they are all in a new directory. To do this, you could enter the following commands after opening a terminal (% is the command prompt):

```
% cd
% mkdir mech-4822
% cd mech-4822
% mkdir icem-tutorial
% cd icem-tutorial
% pwd
/home/u7/umjohndoe/mech-4822/icem-tutorial
```

Note that the ANSYS set of programs (including CFX and ICEM) are unable to read directory names with space characters in them. This means that the name of the new directory AND the path leading to the new directory MUST NOT have any spaces. The last line of the above set of commands gives the path and current directory names; simply ensure there are no spaces.

5. Assumptions about Running ICEM CFD


It is assumed that you are logged into a VNC session or using a Unix workstation. You must have a graphical interface in order to use ICEM, meaning that you cannot run ICEM through the SSH client alone.

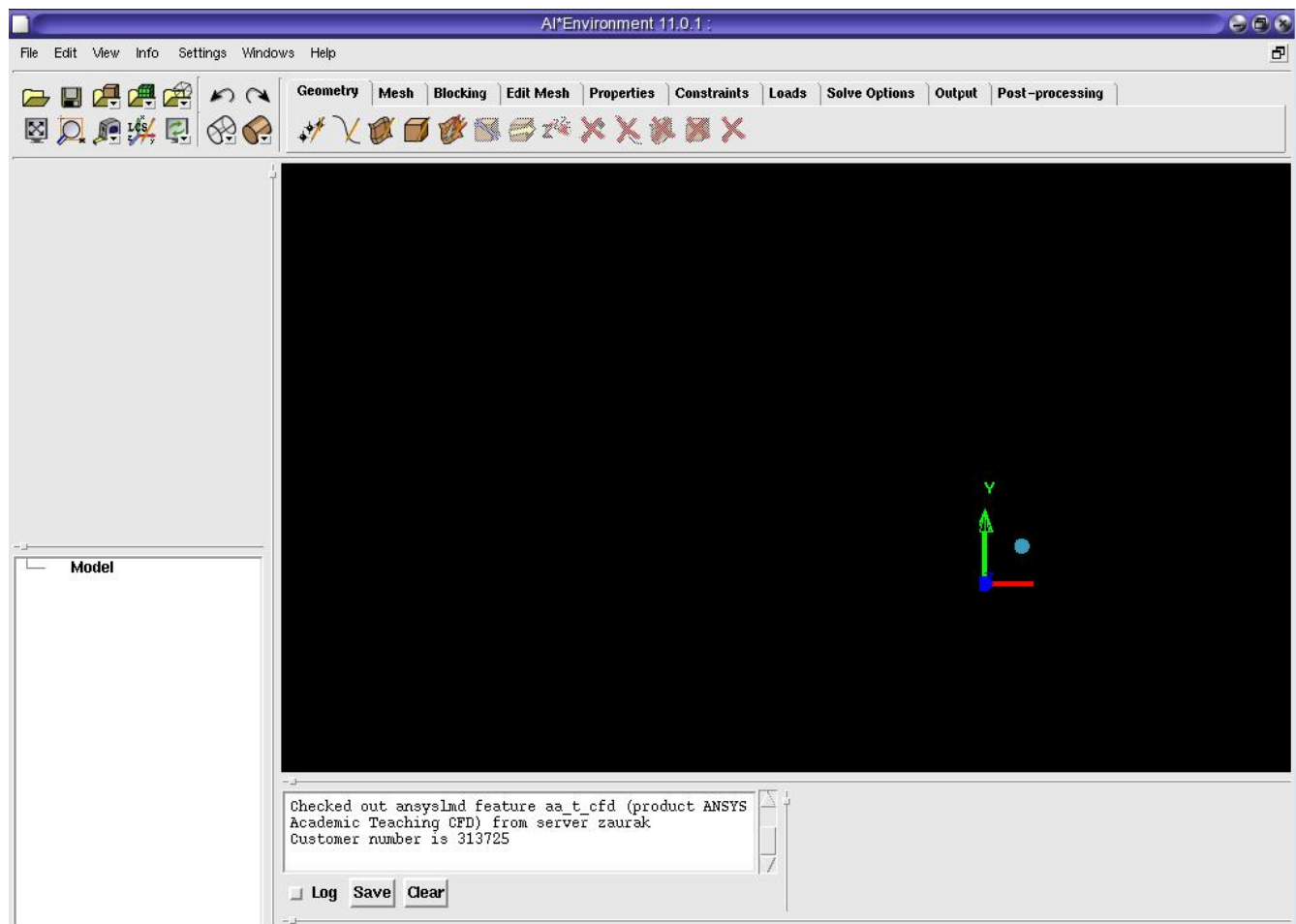
Note that throughout the tutorial, unless directed to use the right or middle mouse buttons explicitly, all commands are to be performed using the left mouse button.

6. Creation of the Mesh

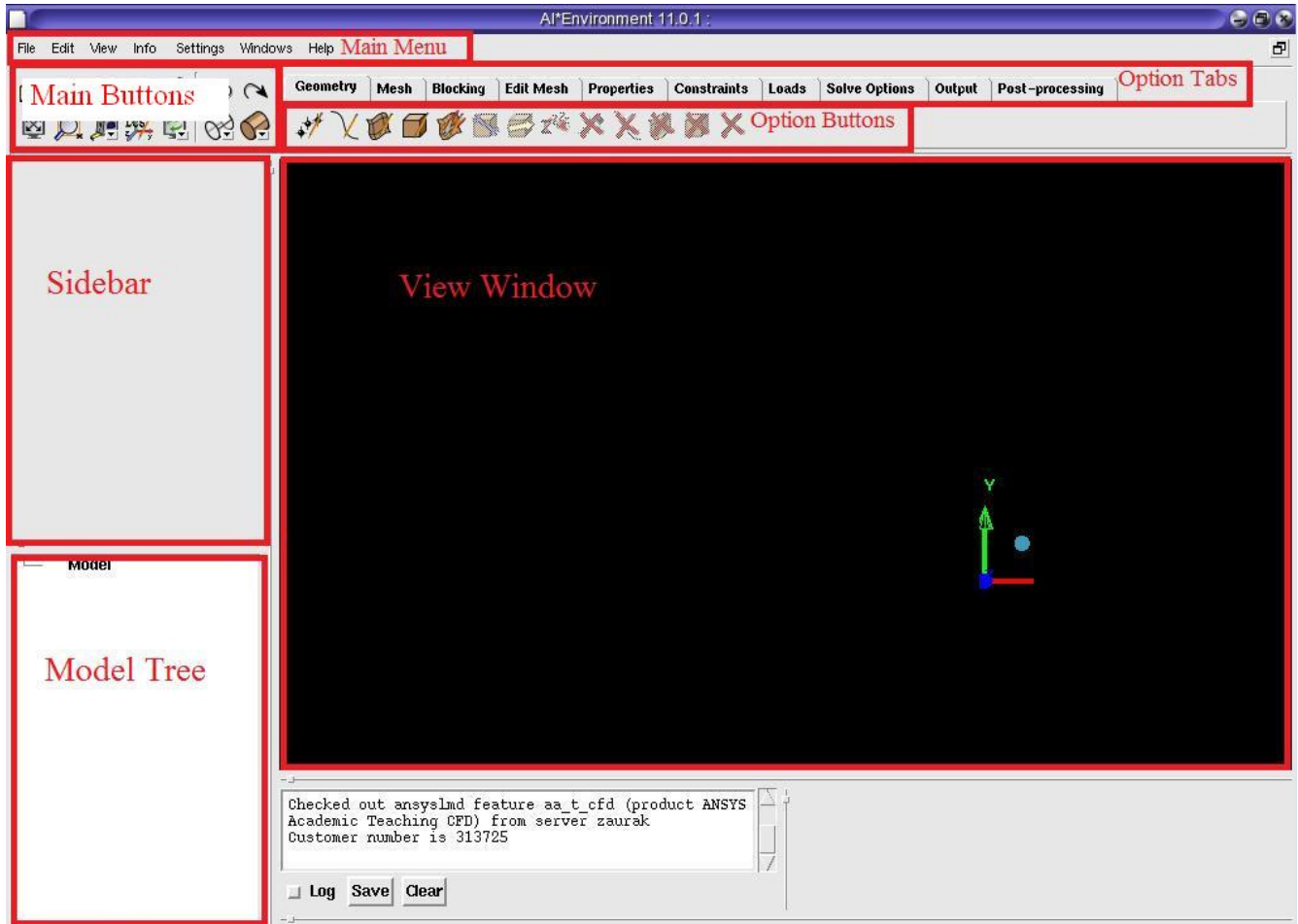
1. Start ICEM CFD by typing the following command in your terminal:

```
% icemcfd
```

'Maximize' the ICEM window by clicking the  button at the top right corner of the window. Your screen should look like this:



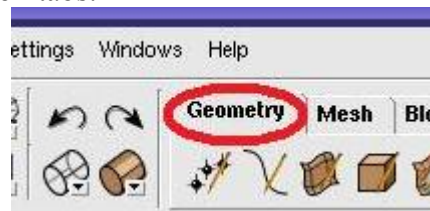
Some general names for different parts of the ICEM window will be used to help identify where certain commands are found. The different parts are named the main menu, main buttons, option tabs, options buttons, sidebar, model tree, and view window. These are all labelled in the following image.



2. Create the geometry

The geometry will be created in units of mm and scaled to m at the output step.

- Select 'Geometry' from the option tabs.

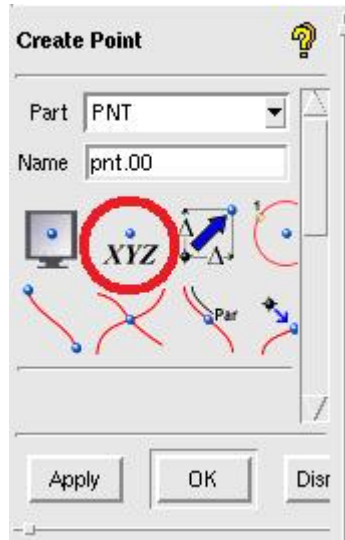


2.1 Create points

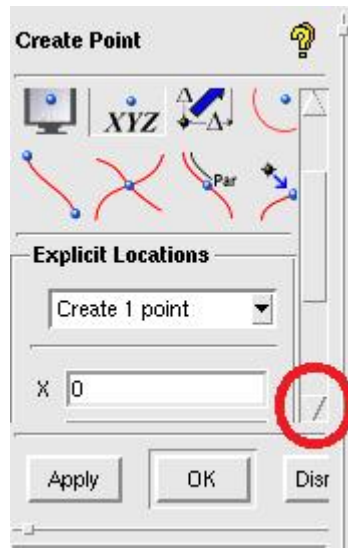
- Select 'Create Point' from the option buttons. The new sidebar 'Create Point' is opened.



- Within the sidebar, select the 'Explicit Coordinates' button and rename the 'Part' as 'PNT'.



- Leave the 'Name' at its default of 'pnt.00' and note that this will be automatically updated as more points are created.
- Scroll down the 'Create Point' sidebar.



- Leave the default setting of 'Create 1 point' and enter the following coordinates:

	pnt.00
X	0
Y	0
Z	0

- Click 'Apply' and the given point will be created. You will see the point in the centre of your screen and you can now observe the 'Name' field is updated. Repeat the above steps for the following 7 points:

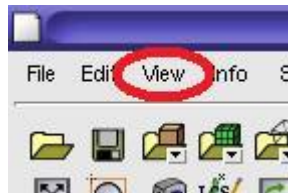
	pnt.01	pnt.02	pnt.03	pnt.04	pnt.05	pnt.06	pnt.07
X	0	0	0	800	800	800	800
Y	0	100	100	0	0	100	100
Z	10	10	0	0	10	10	0

- When all of the points have been entered, click 'Dismiss' in the sidebar. Recall that we have defined the position of these points 1000 times larger than the geometry dictates (0.8 m has become 800 m) for convenience of working in units that we can think of as mm. We will scale this down by 1000 when we create the CFX readable file (our very last step).
- In order to view all of the points, we need to modify the viewable area. To get all of the points on

the screen, select 'Fit Window' from the main buttons (you may need to left click within the view window after pressing 'Fit Window' to enact the change).



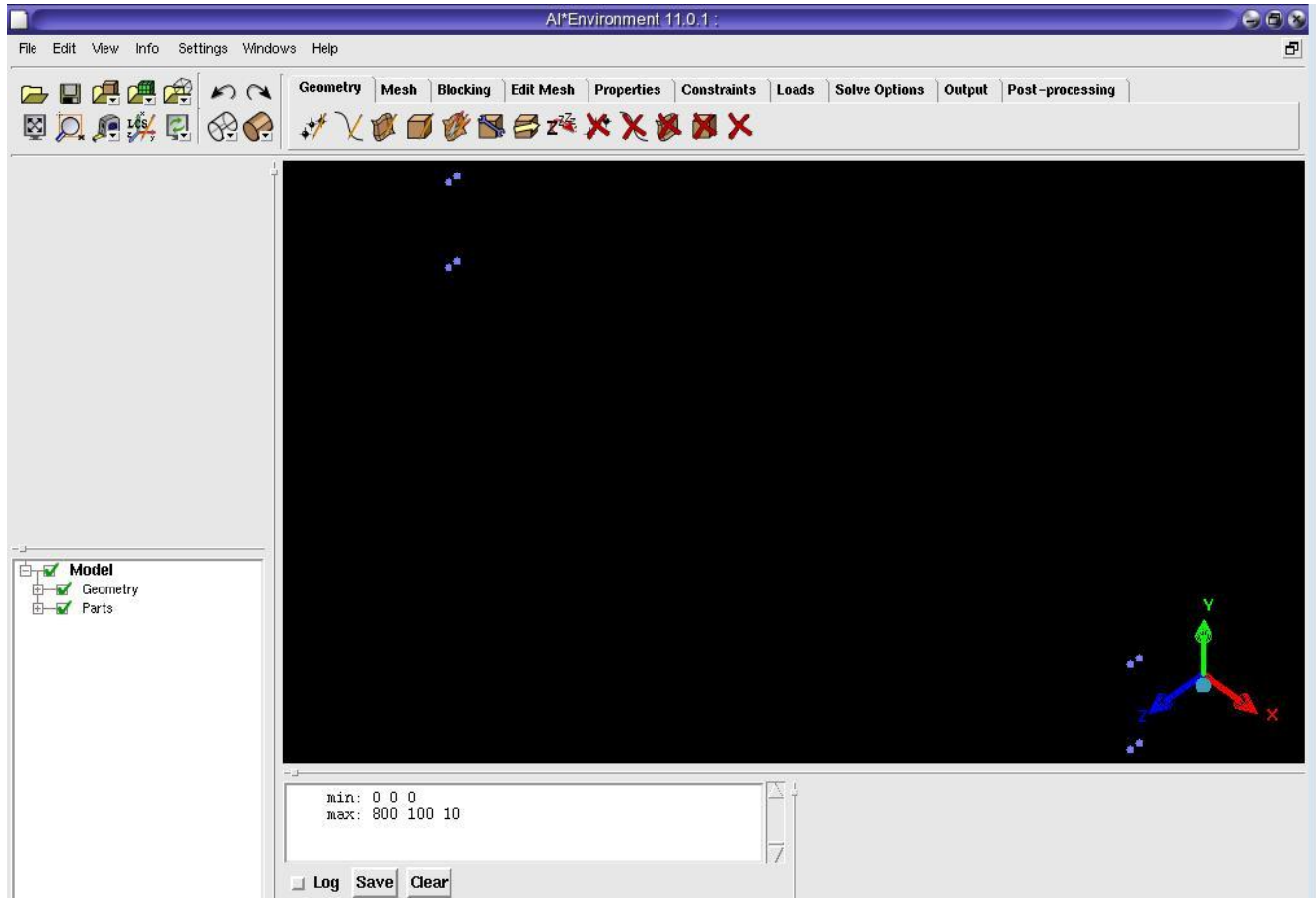
- Now all of the points fit on the screen, but because you are looking directly at the 'XY' plane only four points are visible and the other four are behind them. In order to see all of the points you can use the main menu's 'View' options



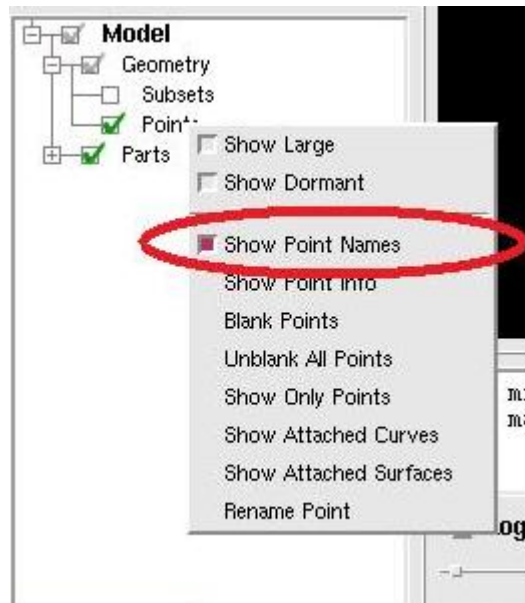
- Alternatively, you can use the following mouse controls:

Mouse Control	Action Performed
Hold down the left mouse button within the view window and drag in any direction	Rotate view
Hold down the right mouse button within the view window and drag left or right	Tilts view
Hold down the right mouse button within the view window and drag up or down	Zooms in or out
Hold down the middle mouse button within the view window and drag in any direction	Pans view
Scroll the mouse wheel up or down within the view window	Zooms in or out

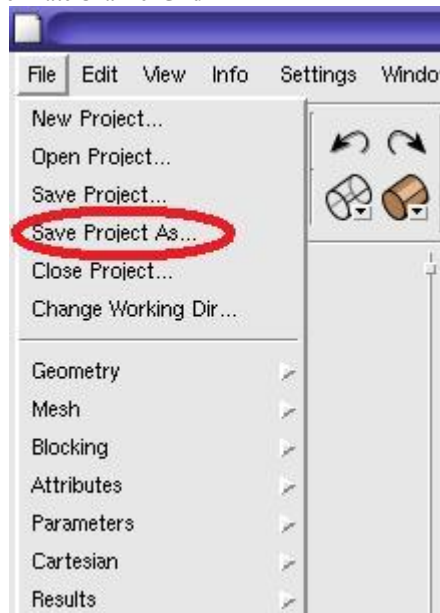
- Familiarize yourself with these controls by experimenting. When you are ready, press 'Fit Window' from the main buttons and select 'Isometric' from 'View' in the main menu. Your screen should now look like this:



- We now wish to display the names of the points for easy reference. To do this, select the '+' next to the word 'Geometry' in the model tree, then right click the word 'Points' and select 'Show Point Names' from the menu that appears. You will now see the names above the points in your view window.



- We will now save the project. Select 'File' from the main menu, and then select 'Save Project As'.

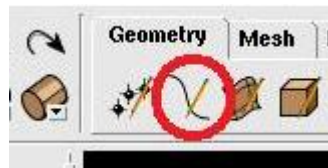


- Move to the directory you created in the setup steps of this tutorial and then choose a meaningful name for this project (for example, 'pplates') and press 'Save'. In the future, save frequently by pressing 'Save Project' in the main buttons.

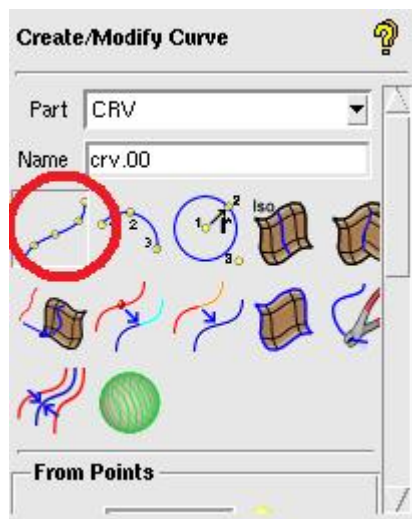


2.2 Creating curves

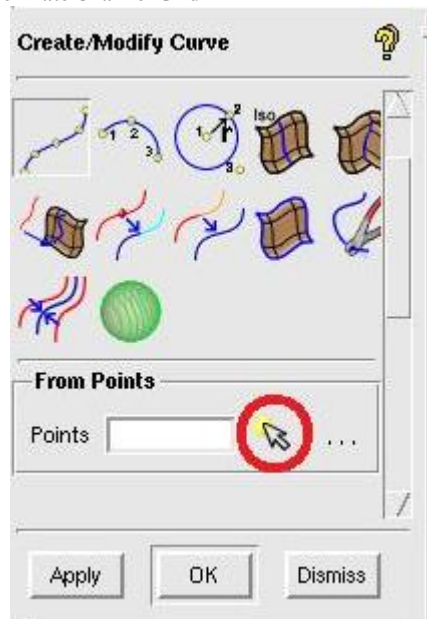
- Select 'Create/Modify Curve' from the option buttons. The new sidebar 'Create/Modify Curve' is opened.



- From within the sidebar select the 'From Points' button and rename the 'Part' as 'CRV'. Leave the 'Name' at its default of 'crv.00' and note, again, that this will be automatically updated as more curves are created.



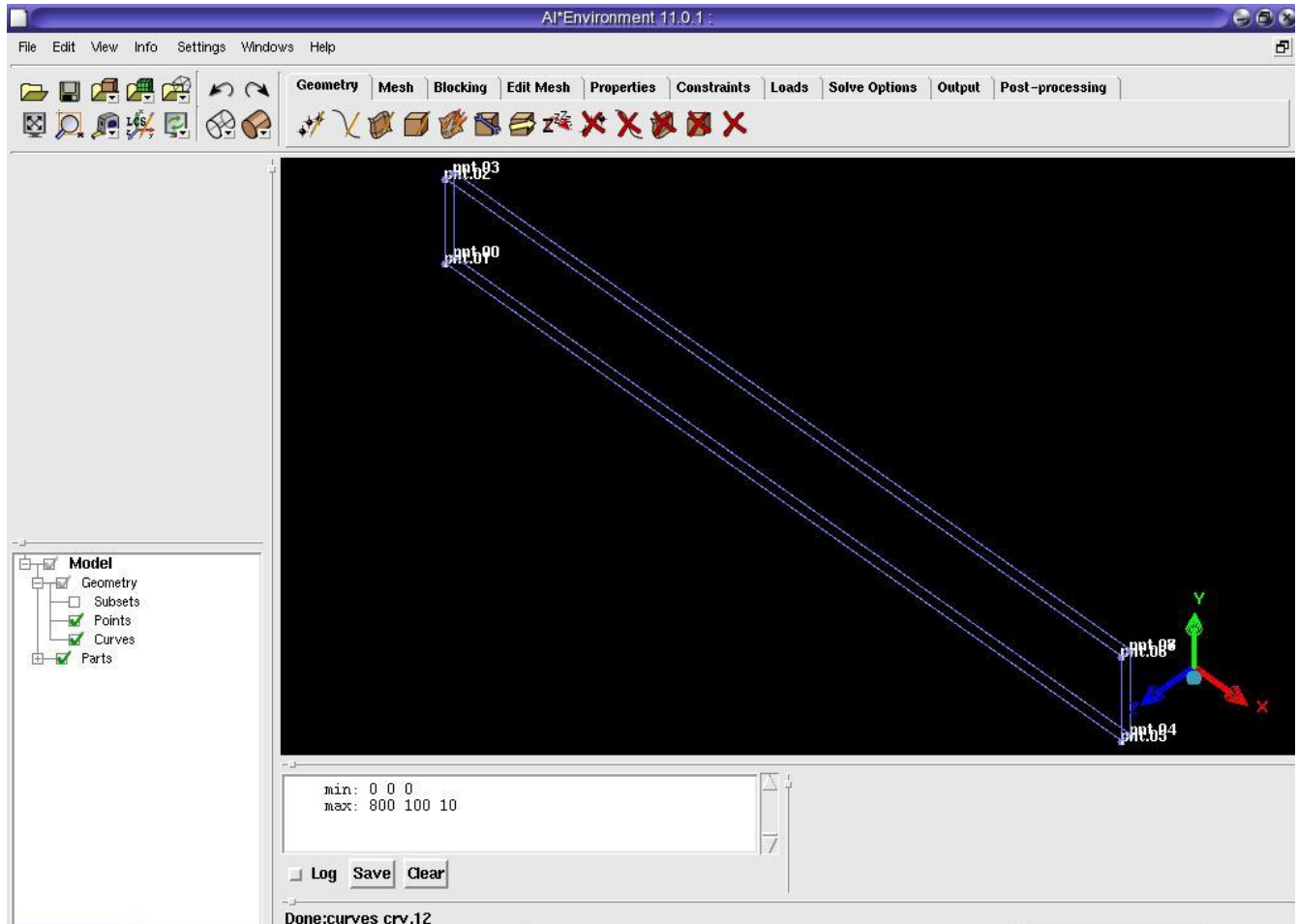
- Scroll down the 'Create/Modify Curve' sidebar and click on the 'Select location(s)' button.



- With the left mouse button select points pnt.00 and pnt.01 in the view window, and then click the **middle mouse button**. Note that it may be difficult to select the points from the current view because the points are very close together, but you cannot pan or zoom with the mouse normally because you are in selection mode. In order to leave the selection mode press **F9** on your keyboard, and then use the mouse controls normally. Once you have adjusted the current view simply press **F9** on your keyboard again to return to selection mode.
- Repeat this process for the following pairs of points:

crv.01	pnt.01, pnt.02
crv.02	pnt.02, pnt.03
crv.03	pnt.03, pnt.00
crv.04	pnt.00, pnt.04
crv.05	pnt.01, pnt.05
crv.06	pnt.02, pnt.06
crv.07	pnt.03, pnt.07
crv.08	pnt.04, pnt.05
crv.09	pnt.05, pnt.06
crv.10	pnt.06, pnt.07
crv.11	pnt.07, pnt.04

- Once all of the curves have been created click 'Dismiss' in the sidebar and select 'Fit Window' from the main buttons and 'Isometric' from 'View' in the main menu to display the current geometry as follows:



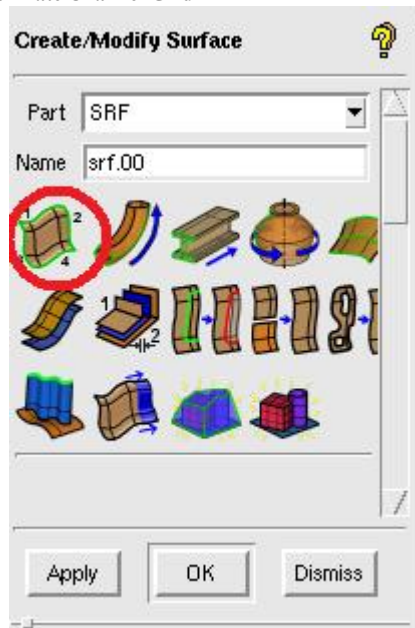
- You can now turn off point names and turn on curve names in the model tree by right-clicking the word 'Points' and selecting 'Show Point Names' from the menu that appears, and then right-clicking the word 'Curves' and selecting 'Show Point Names' from the menu that appears.
- Save the project.

2.3 Creating surfaces

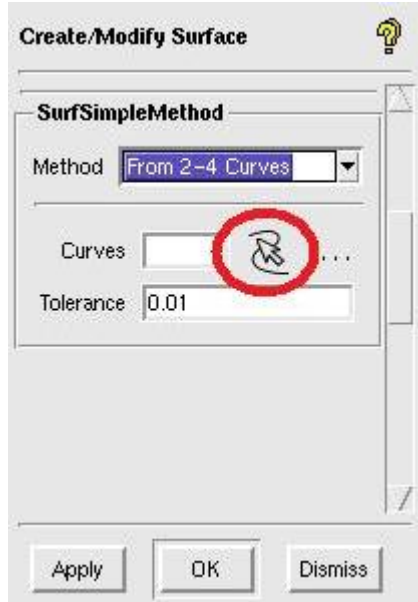
- Select 'Create/Modify Surface' from the option buttons. The new sidebar 'Create/Modify Surface' is opened.



- From within the sidebar, select the 'Simple Surface' button and rename the 'Part' as 'SRF'. Leave the 'Name' at its default of 'srf.00'.



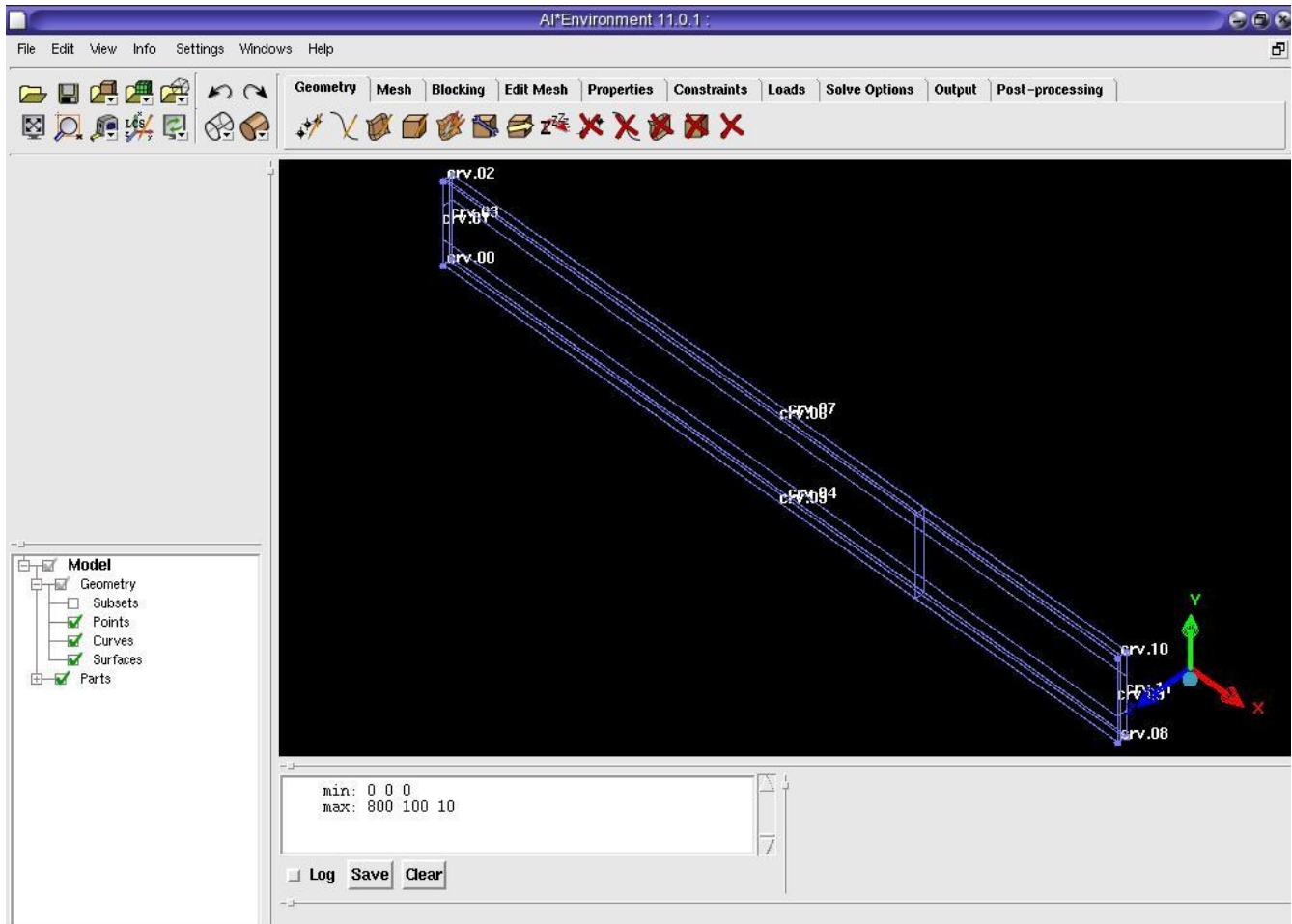
- Scroll down the 'Create/Modify Surface' sidebar and click on the 'Select curve(s)' button. Leave the 'Method' and 'Tolerance' at their default values of 'From 2-4 Curves' and 0.01, respectively.



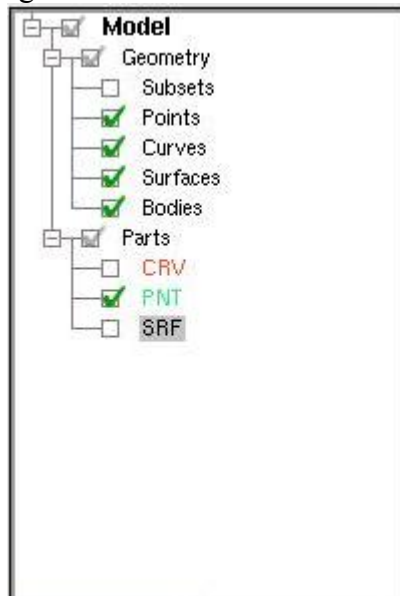
- With the left mouse button select curves crv.00, crv.01, crv.02, and crv.03 in the view window, then click the middle mouse button, and then press 'Apply' in the sidebar (to create surfaces you must press the middle mouse button **and** then the apply button, whereas for curves you only needed to press the middle mouse button). As with the creation of curves you can modify your view while in selection mode by pressing **F9** on your keyboard, using the mouse-view controls normally, and then pressing **F9** again.
- Repeat this process for the following groups of curves:

srf.01	crv.00, crv.04, crv.05, crv.08
srf.02	crv.01, crv.05, crv.06, crv.09
srf.03	crv.02, crv.06, crv.07, crv.10
srf.04	crv.03, crv.04, crv.07, crv.11
srf.05	crv.08, crv.09, crv.10, crv.11

- Once all of the surfaces have been created, click 'Dismiss' in the sidebar and select 'Fit Window' from the main buttons and 'Isometric' from 'View' in the main menu to display the current geometry as follows:



- You can now use the model tree to turn off the curve names and turn on the POINT names again. Notice that there are several point names over top of one another so that the names are illegible. This is because we have several parts: PNT, CRV, and SRF that all have the same vertices. To resolve this, in the geometry tree select the '+' next to the word 'Parts' and then deselect the parts 'CRV' and 'SRF' by clicking on the check marks next to them.



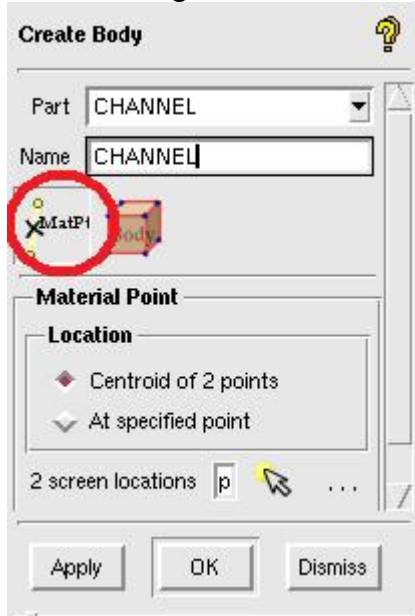
- Save the project.

2.4 Defining the volume or 'body'

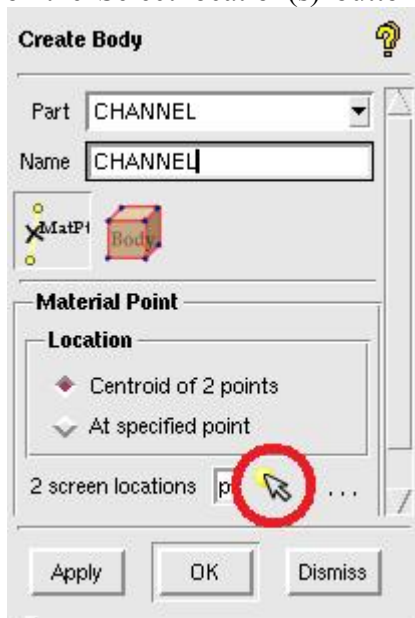
- Select 'Create Body' from the option buttons. The new sidebar 'Create Body' is opened.



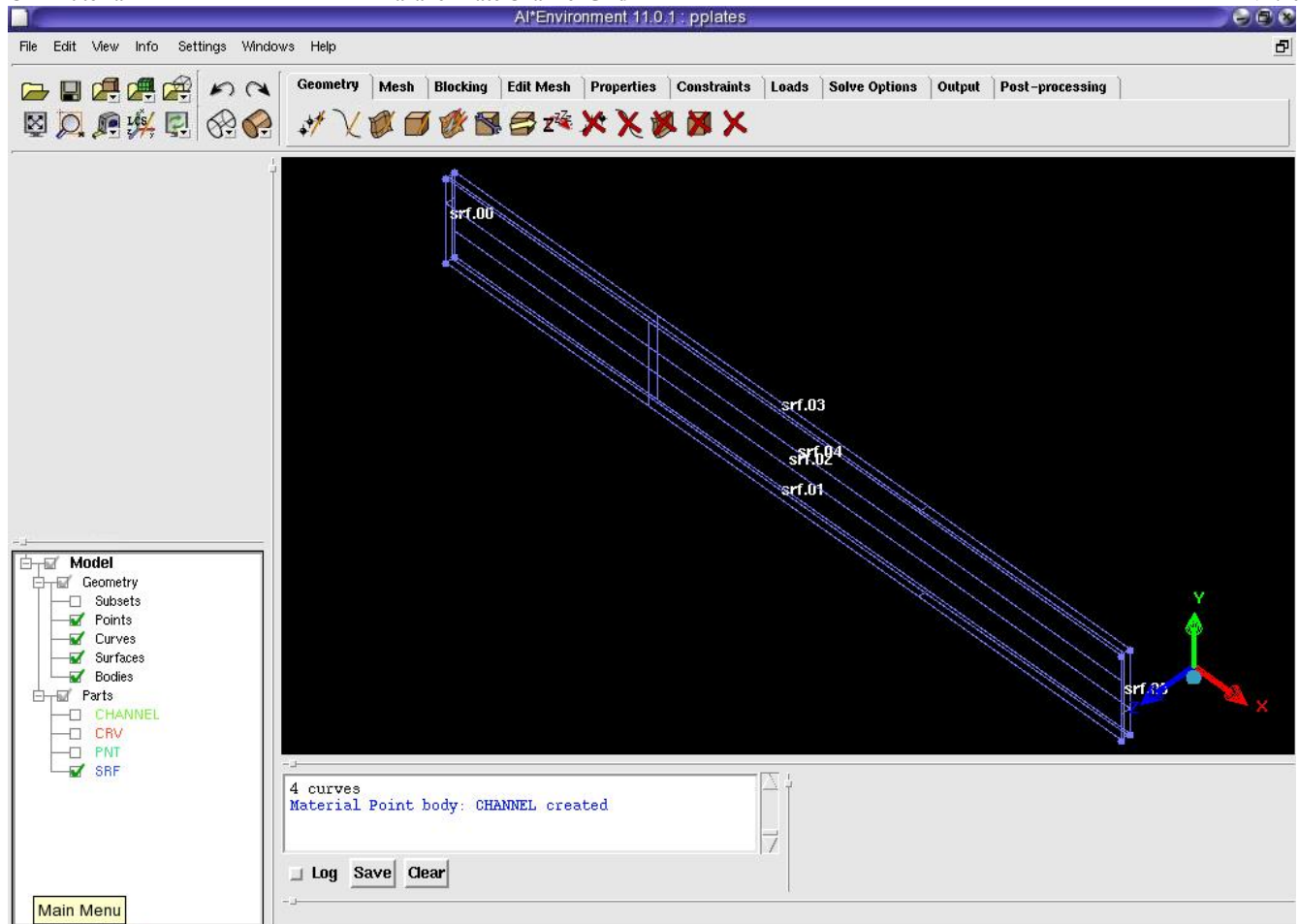
- From within the sidebar select the 'Material Point' button and fill in both the 'Part' and 'Name' as 'CHANNEL'. Leave the default setting of 'Centroid of 2 points' in the 'Location' choice.



- Within the sidebar, click on the 'Select location(s)' button.



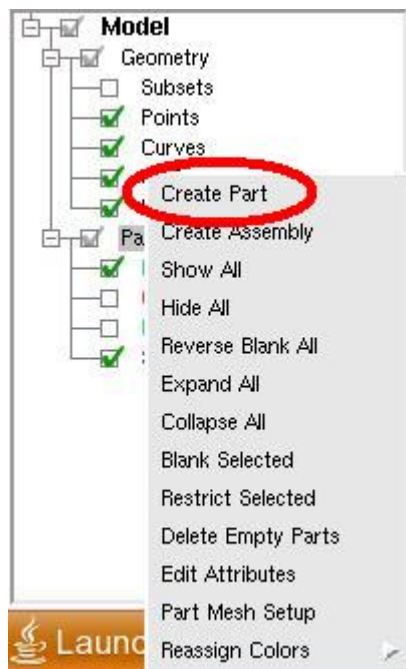
- With the left mouse button select the points pnt.00 and pnt.06 in the view window, then press the middle mouse button, and then press 'Apply' in the sidebar. An asterisk and the word 'CHANNEL' should appear in your view window.
- Press the 'Dismiss' button in the sidebar. Note that the way we defined this body it is in the centre of the domain and that any two, opposite vertices could have been chosen with the same final result. Also, the 'By Topology' option in the 'Create Body' sidebar could have been chosen and either the entire model, or all of the surfaces used to define the body in the centre.
- You may now turn off the point names, turn on the surface names, and make only the 'Parts' SRF visible in the geometry tree. When you use the 'Fit Window' and 'Isometric' view options you should have the following:



- Save the project

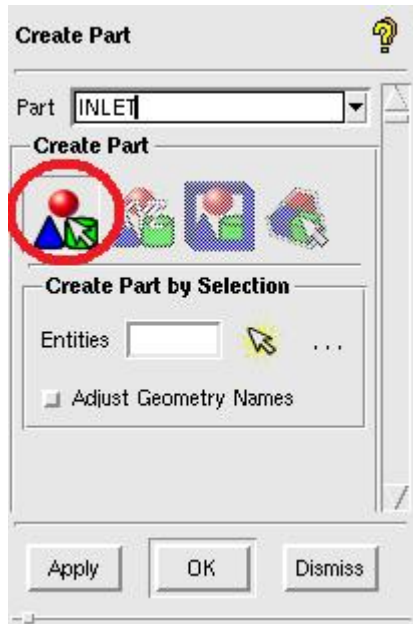
3. Create parts from surfaces

- Now that the full geometry is developed, it is helpful to assign meaningful names to the different surfaces for future reference. To do this we will create new parts. In the geometry tree right click the word 'Parts' and select 'Create Part' from the menu that appears. The new sidebar 'Create Part' is opened.

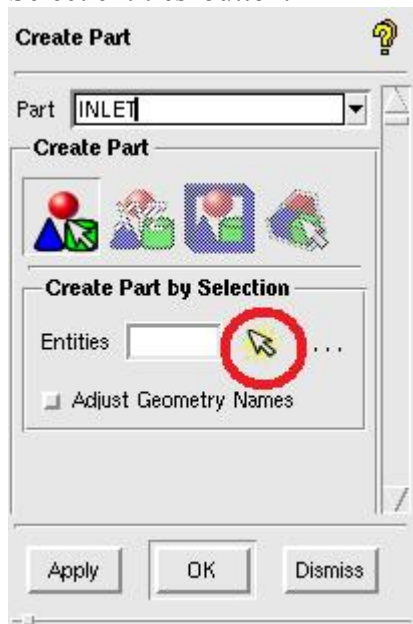


- Within the sidebar, enter the name 'INLET' in the 'Part' field and click the 'Create Part by Selection'

button.



- Within the sidebar, click the 'Select entities' button.



- Select 'srf.00' in the view window, press the middle mouse button, and then select 'Apply' in the sidebar. The new part 'INLET' will appear in the geometry under 'Parts'. Repeat this process for the other five surfaces with the following names:

srf.01	BOTTOM
srf.02	SYMMETRY1
srf.03	TOP
srf.04	SYMMETRY2
srf.05	OUTLET

- Once all of the parts are created, press 'Dismiss' in the sidebar. You may now turn off surface names in the model tree.
- Save the project.

4. Blocking

- Select 'Blocking' from the option tabs.

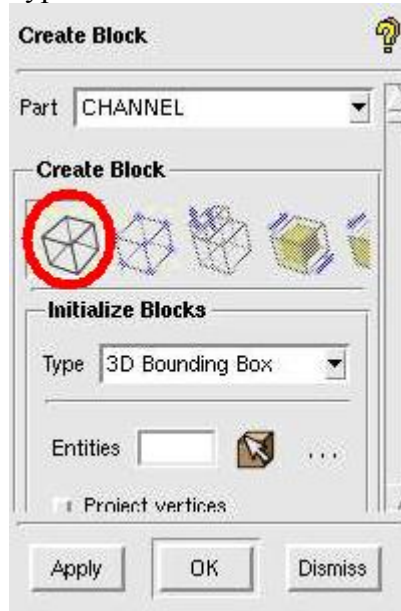


4.1 Create the block

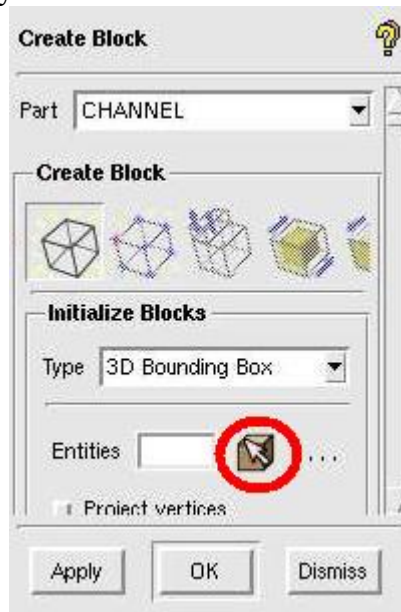
- Select 'Create Block' from the option buttons. The new sidebar 'Create Block' is opened.



- From within the sidebar select the 'Initialize Blocks' button and set the 'Part' field to 'CHANNEL'. Leave the 'Type' field as its default '3D Bounding Box'.

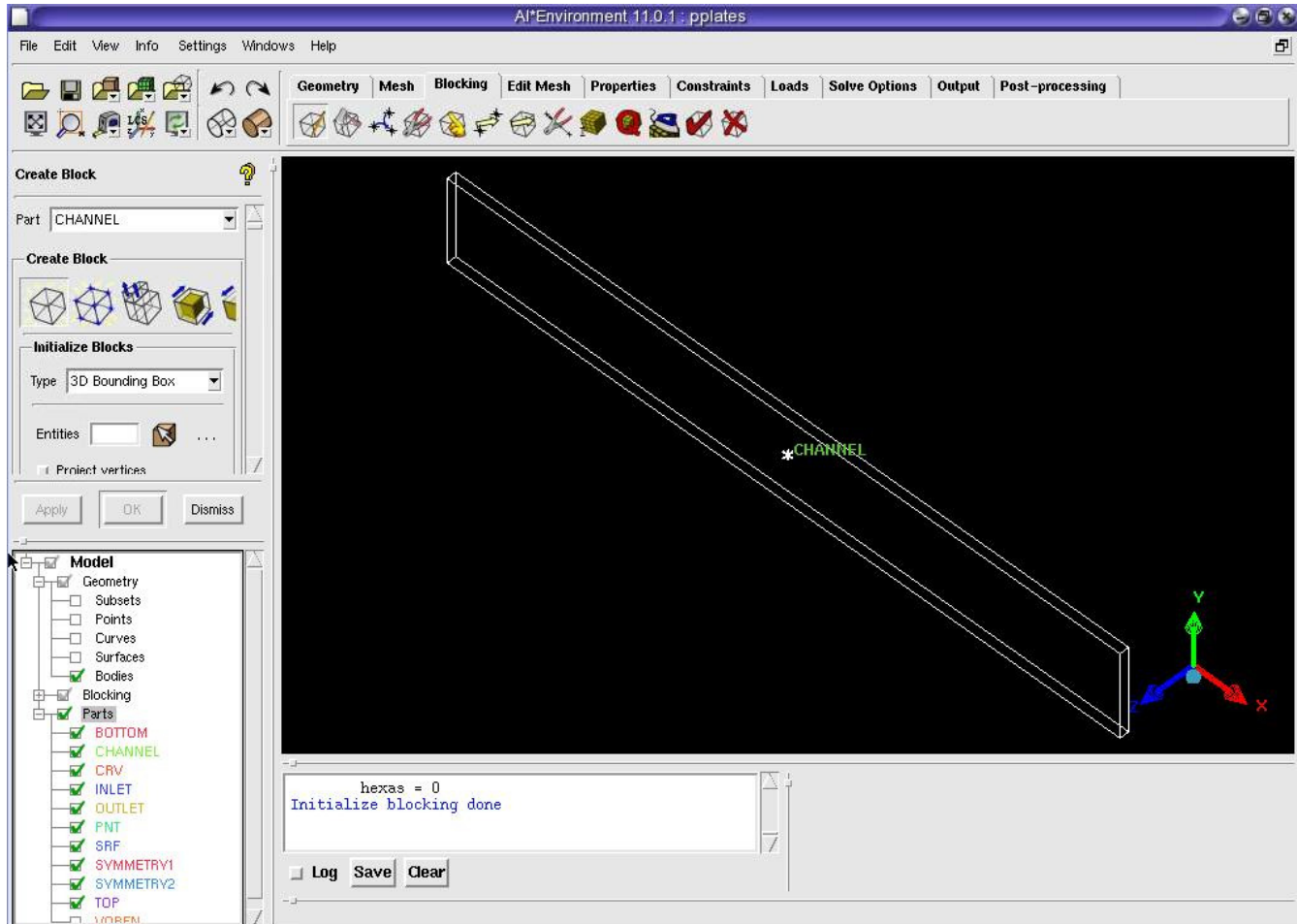


- Click the 'Select geometry' button.



- In the view window, select all six of the surfaces with the left mouse button by either selecting each one individually or left clicking, holding, and dragging a box around the entire geometry. Press the middle mouse button and then press 'Apply' in the sidebar. Press 'Dismiss' in the sidebar. In the model tree, you should now have a new branch named 'Blocking'. Also in the model tree, under 'Parts', you should have something named 'VORFN'.

- In the model tree, under 'Geometry', turn off 'Points', 'Curves', and 'Surfaces' so that only 'Bodies' has a check mark next to it. You should see the following after pressing 'Fit Window' and choosing 'Isometric' from 'View' in the main menu:



- Save the project.

4.2 Associate the created geometry to the block

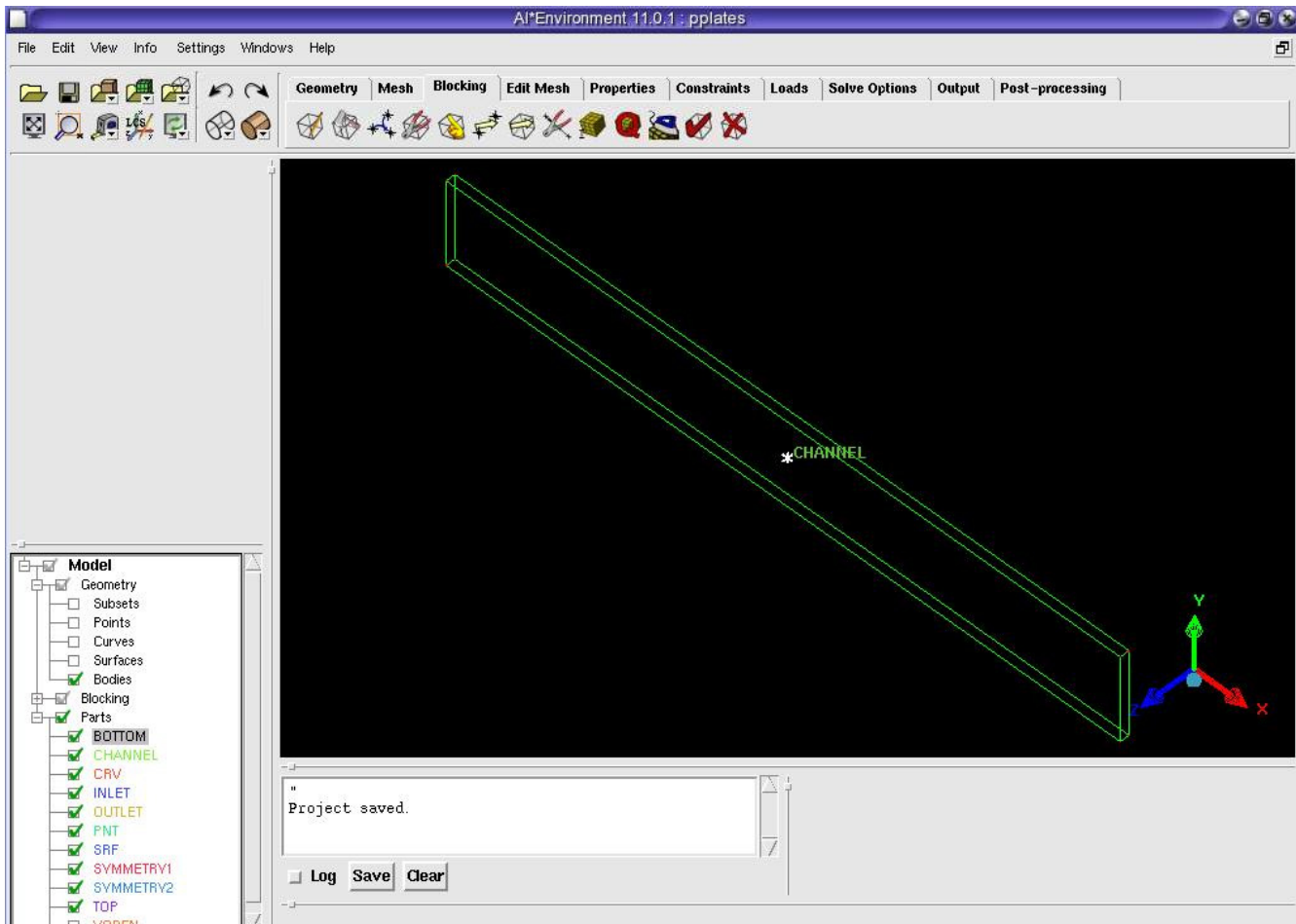
- Select 'Associate' from the option buttons. The new sidebar 'Blocking Associations' is opened.



- From within the sidebar select 'Auto Association'!



- Leave the default 'Snap Project Vertices' and then press 'Apply' and 'Dismiss' in the sidebar. You should see the white lines turn green:



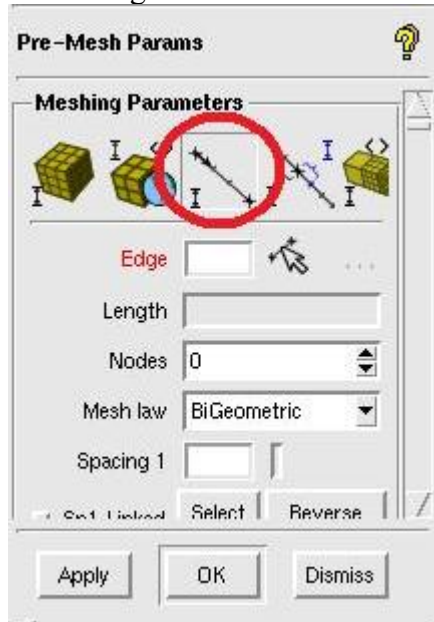
- Note that auto-association can only be used because this is a very simple geometry. For more complicated geometries, manual association must be performed. At this point, turn on 'Curves' and also turn curve names on once again in the geometry tree.
- Save the project.

4.3 Allocate nodal distribution along the block edges

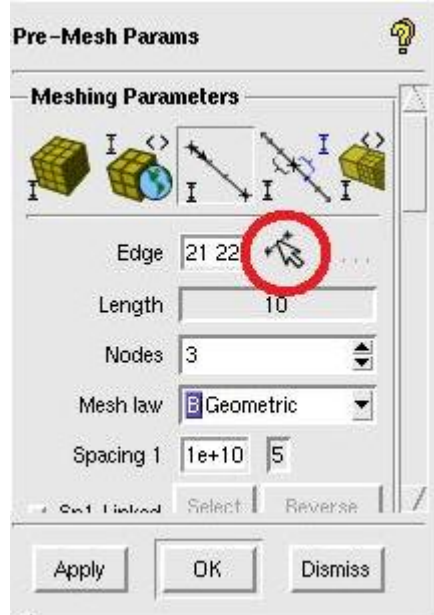
- Select 'Pre-Mesh Params' from the option buttons. The new sidebar 'Pre-Mesh Params' is opened.



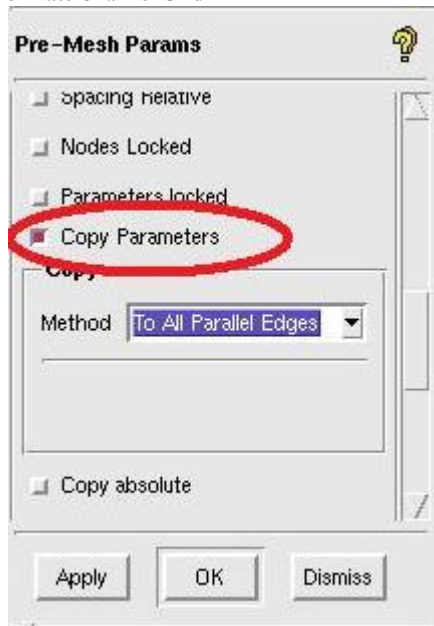
- From within the sidebar select 'Edge Params'.



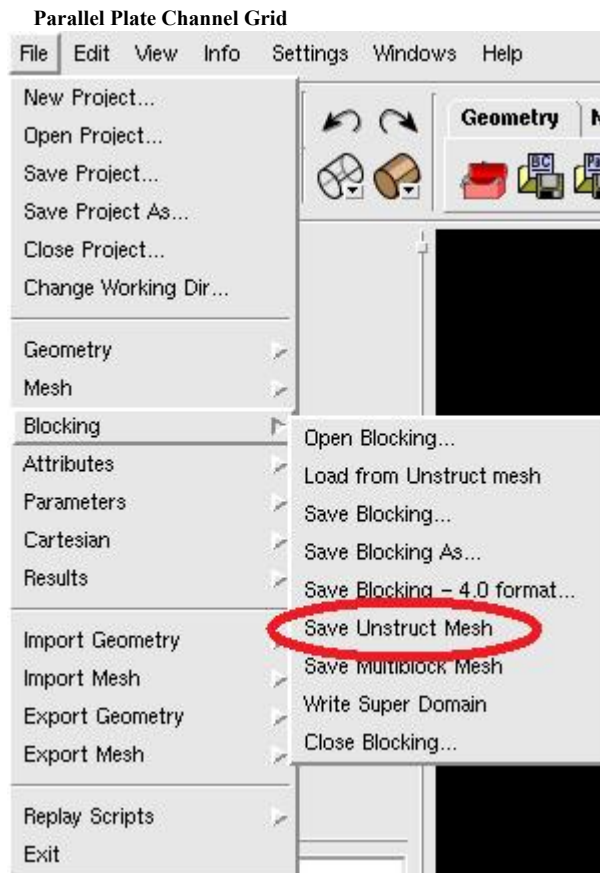
- Click 'Select edge(s)' and then select crv.00 in the view window.



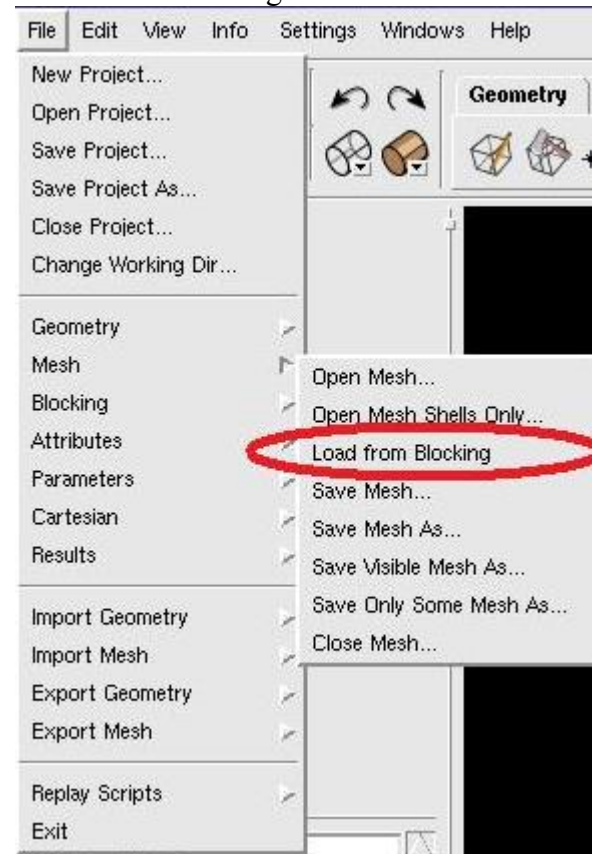
- In the sidebar again, enter 3 in the 'Nodes' field and set the 'Mesh law' field to 'BiGeometric' (note that you may have to reselect crv.00 and enter the number of nodes a second time).
- Scroll down the sidebar and select 'Copy Parameters' and then set the 'Method' field to 'To All Parallel Edges'.



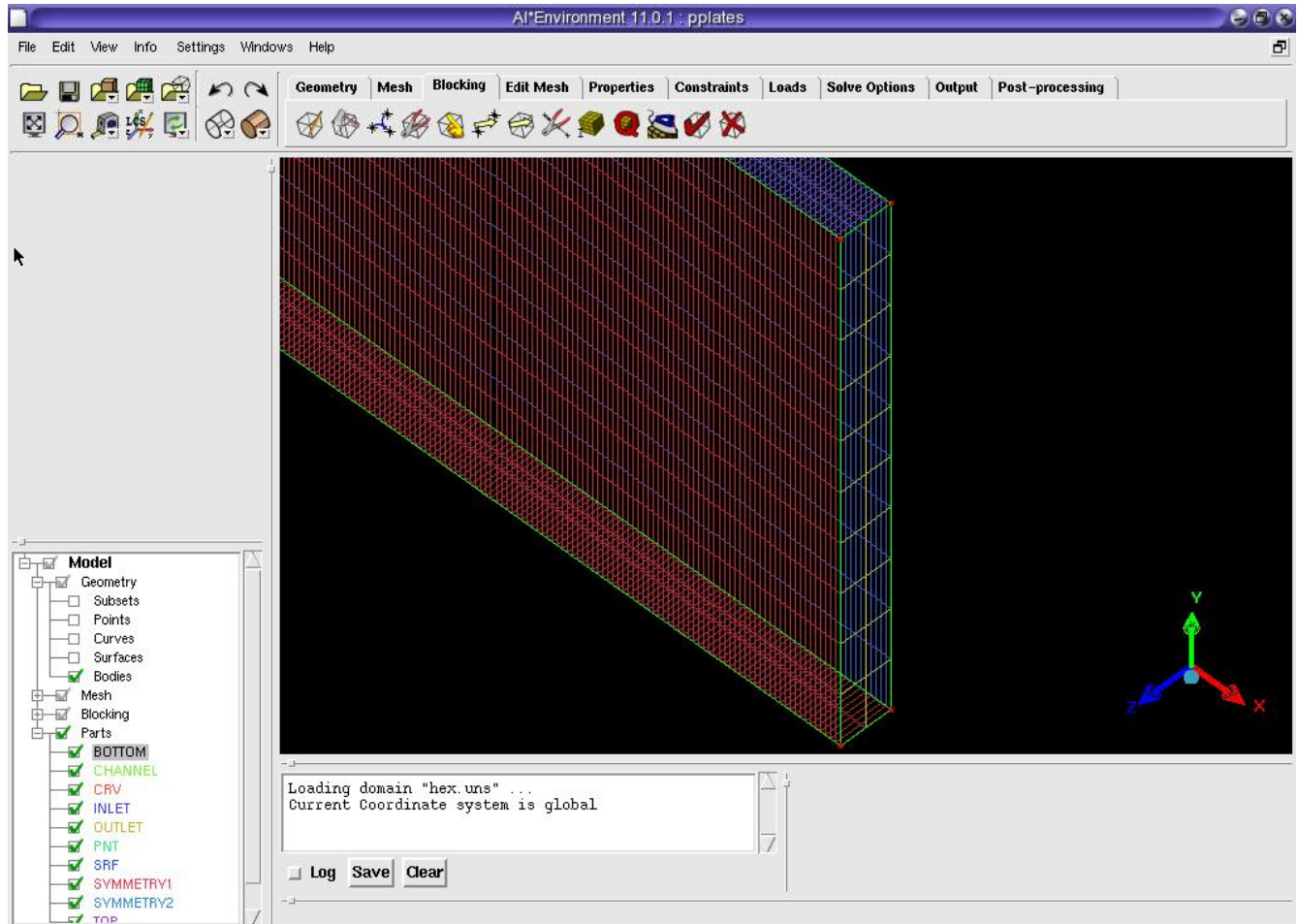
- Press 'Apply' and you should now see small red ticks along all the lines in the z-direction (you will have to zoom in to see them clearly) and in red the number of nodes along each line.
 - Scroll back to the top of the sidebar and click 'Select edge(s)' again. This time select crv.01 in the view window and then enter 11 in the 'Nodes' field of the sidebar. Ensure that the 'Mesh law' is still set to 'BiGeometric', the 'Copy Parameters' button is still highlighted, and the 'Method' is still set to 'To All Parallel Edges'. Again, press 'Apply' and ticks will appear on all the lines in the y-direction.
 - Scroll back to the top of the sidebar and click 'Select edge(s)' again. This time select crv.04 in the view window and then enter 801 in the 'Nodes' field of the sidebar. Ensure that the 'Mesh law' is still set to 'BiGeometric', the 'Copy Parameters' button is still highlighted, and the 'Method' is still set to 'To All Parallel Edges'. Again, press 'Apply' and ticks will appear on all the lines in the x-direction. Press 'Dismiss' in the sidebar.
 - Save the project.
5. Create the mesh
- Select 'File' from the main menu, then select 'Blocking' and 'Save Unstruct Mesh' in the menus that appear.



- Name the file something meaningful like 'pplates' and press save.
- Select 'File' from the main menu, then select 'Mesh' and 'Load from Blocking' in the menus that appear. Wait a moment while the mesh is generated.



- You can now see your mesh in the view window. When you zoom in there will be a grid around the geometry.



- Save the project.

6. Write the mesh to an input file for CFX Pre

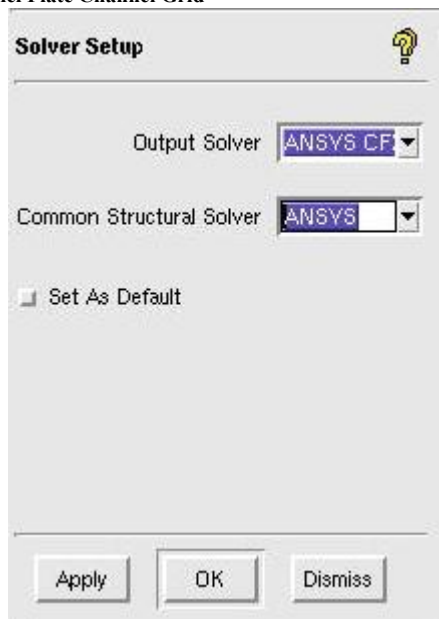
- The final step is to create a CFX readable file from the ICEM mesh. To do this, select 'Output' from the option tabs.



- Select 'Set solver' from the option buttons. The sidebar 'Solver Setup' is opened.



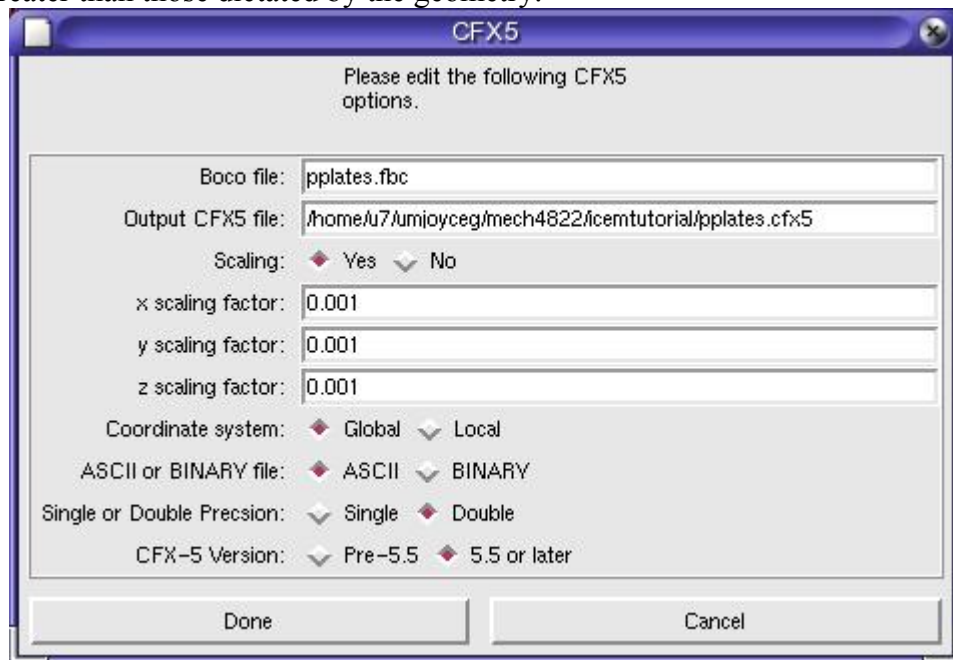
- In the sidebar, set the 'Output Solver' field to 'ANSYS CFX', the 'Common Structural Solver' field to 'ANSYS'.



- Click 'Apply' and 'Dismiss'.
- Select 'Write input' from the option buttons.



- You will be prompted to save the current project, select 'Yes'
- A new window with the title 'CFX5' will appear. Highlight 'Yes' in the 'Scaling' field and set the x-, y-, and z-scaling factors to 0.001. Note the scaling is necessary because we input dimensions 1000 times greater than those dictated by the geometry.



- Click done.

CONGRATULATIONS! You have just defined a basic mesh using ANSYS ICEM. The .cfx5 file that you created can be read into CFX-Pre.