ICEM CFD Tutorial

Simple Duct Grid

Scott J. Ormiston
Gavin Joyce

Department of Mechanical Engineering

University of Manitoba
Winnipeg, Manitoba
Canada

V1.01
17 January 2013

1. Introduction

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

1.2 Geometry Nomenclature

The following image shows the basic geometry of the duct.

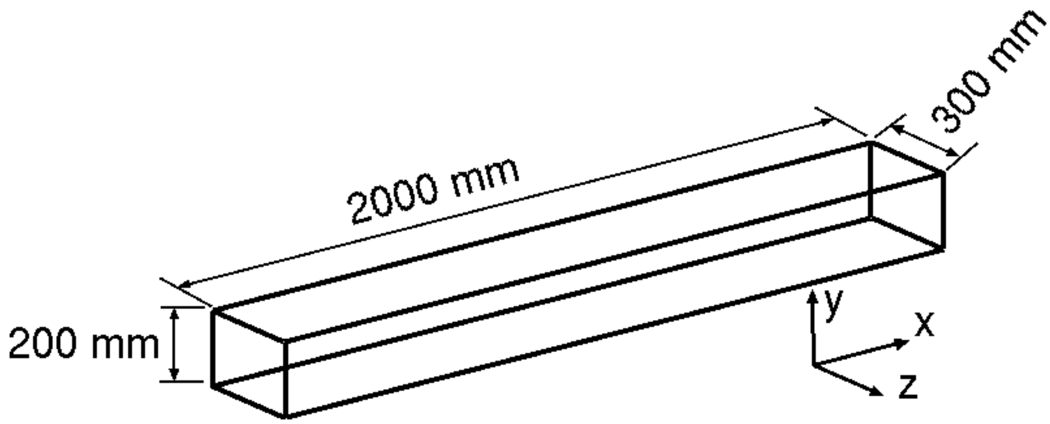


Figure 1: Duct Geometry

1.3 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
4. Associate the created geometry
5. Allocate nodal distribution along the block edges
6. Mesh the block
7. Create the file needed for input in CFX Pre

1.4 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. You can create this new directory while in your root directory or while in any other subdirectory. To do this, you could enter the following commands (`%` is the command prompt):

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

1.5 Assumptions about Running ICEM CFD

It is assumed that you are logged into a VNCviewer session (or equivalent) or using a Linux/Unix workstation. You must have a graphical interface in order to use ICEM, meaning that you cannot run ICEM through the SSH client alone. This tutorial was created for ANSYS ICEM CFD version 14.0.

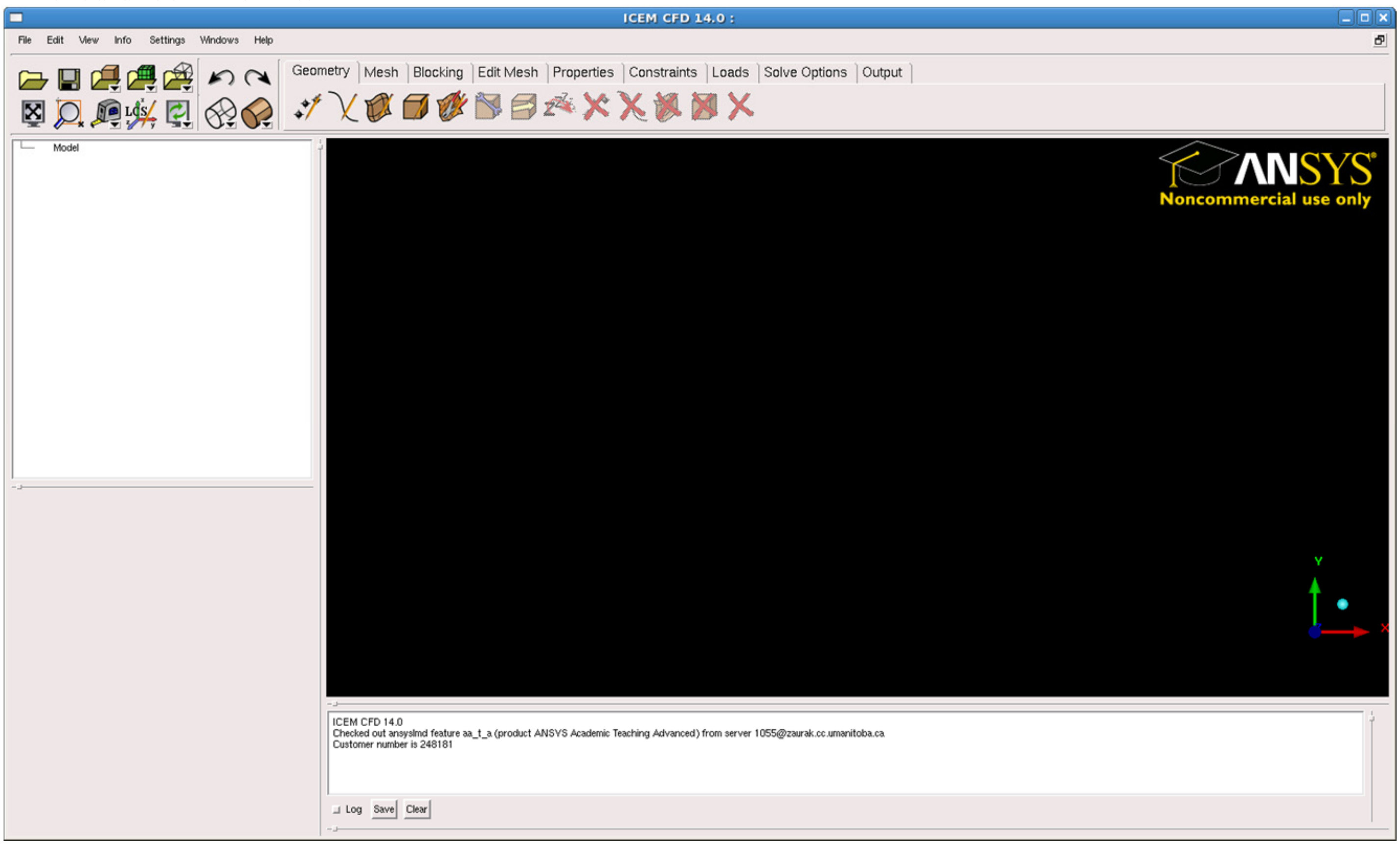
Note that throughout the tutorial, unless directed to use the right or middle mouse buttons explicitly, all commands are to be performed using the left mouse button.

1.6 Starting the Program

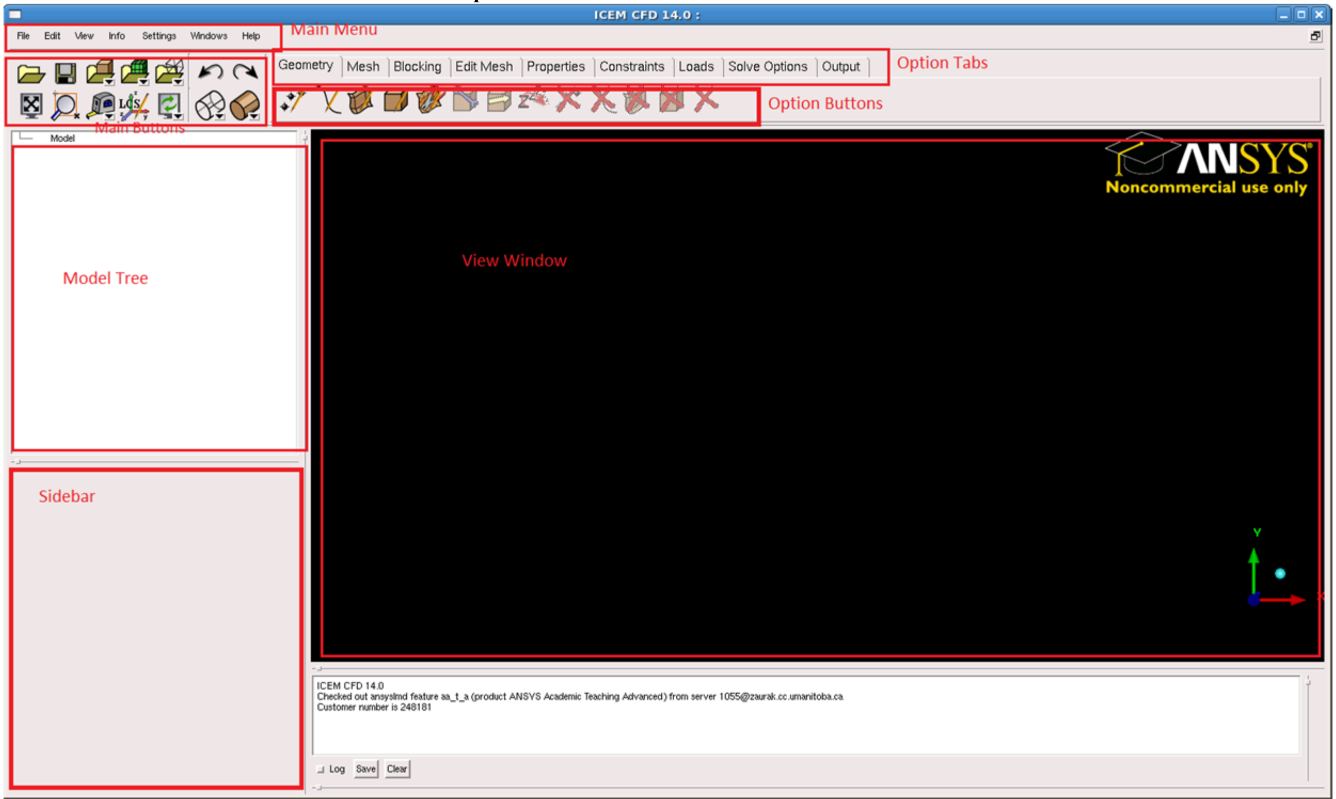
Start ICEM CFD by typing the following command in your terminal:

```
% icemcfd
```

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

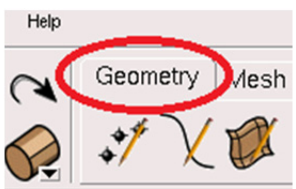


Some general names for different parts of the ICEM window will be used to help identify where certain commands are found. The different parts are named the main menu, main buttons, option tabs, options buttons, sidebar, model tree, and view window. These are all labelled in the following image.



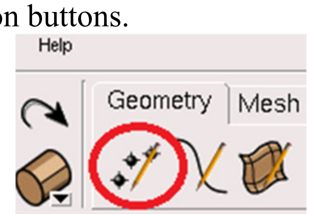
2. Create the Geometry

The geometry will be created in units of millimetres and then scaled to metres at the output step. Begin this phase by selecting "Geometry" from the option tabs.



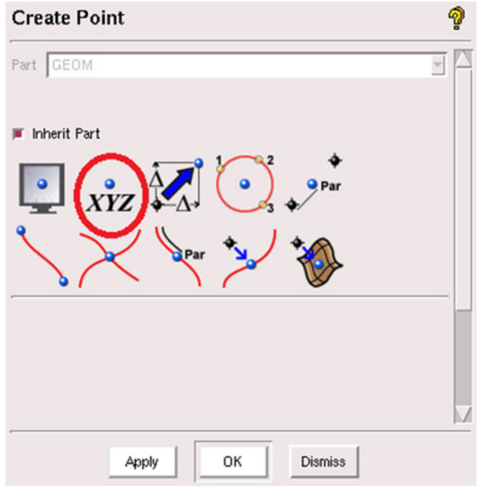
2.1 Create points

- Select 'Create Point' from the option buttons.

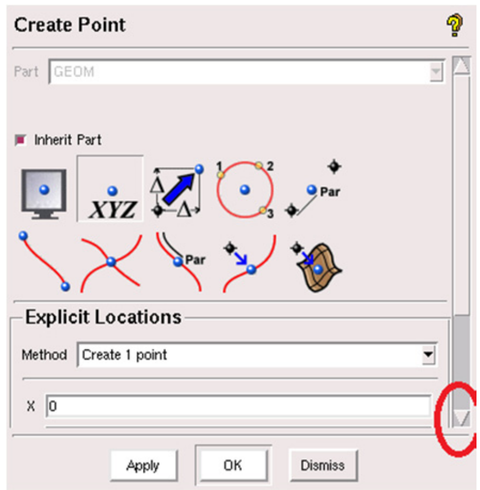


The new sidebar 'Create Point' will be opened.

- Within the sidebar, select the 'Explicit Coordinates' button.



- Scroll down the 'Create Point' sidebar.

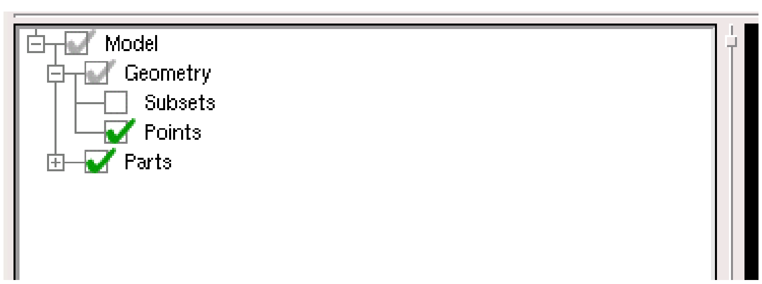


- Leave the default setting of 'Create 1 point' and enter the following coordinates:

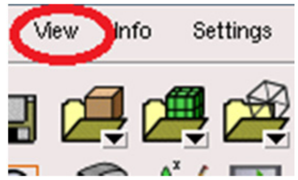
	pnt.00
X	0
Y	0
Z	0

Then click on Apply.

- Let the point be created under the part GEOM. The first point will be called 'pnt.00'.
- Now Geometry and Parts will appear in the model tree. Click on the plus sign to expand the Geometry to get:



- To see the names of the points right click on points and select "Show point names". You should see the point with its label "Pnt.00" in the view window.
- In order to view the remainder of the points as you create them, the following is suggested. Modify the view by selecting View:

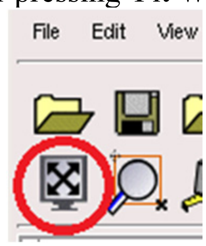


Then, select “Isometric”.

- Now enter the remaining points shown in the table below. Click Apply each time after entering the three coordinate values.

	pnt.01	pnt.02	pnt.03	pnt.04	pnt.05	pnt.06	pnt.07
X	0	0	0	2000	2000	2000	2000
Y	0	200	200	0	0	200	200
Z	300	300	0	0	300	300	0

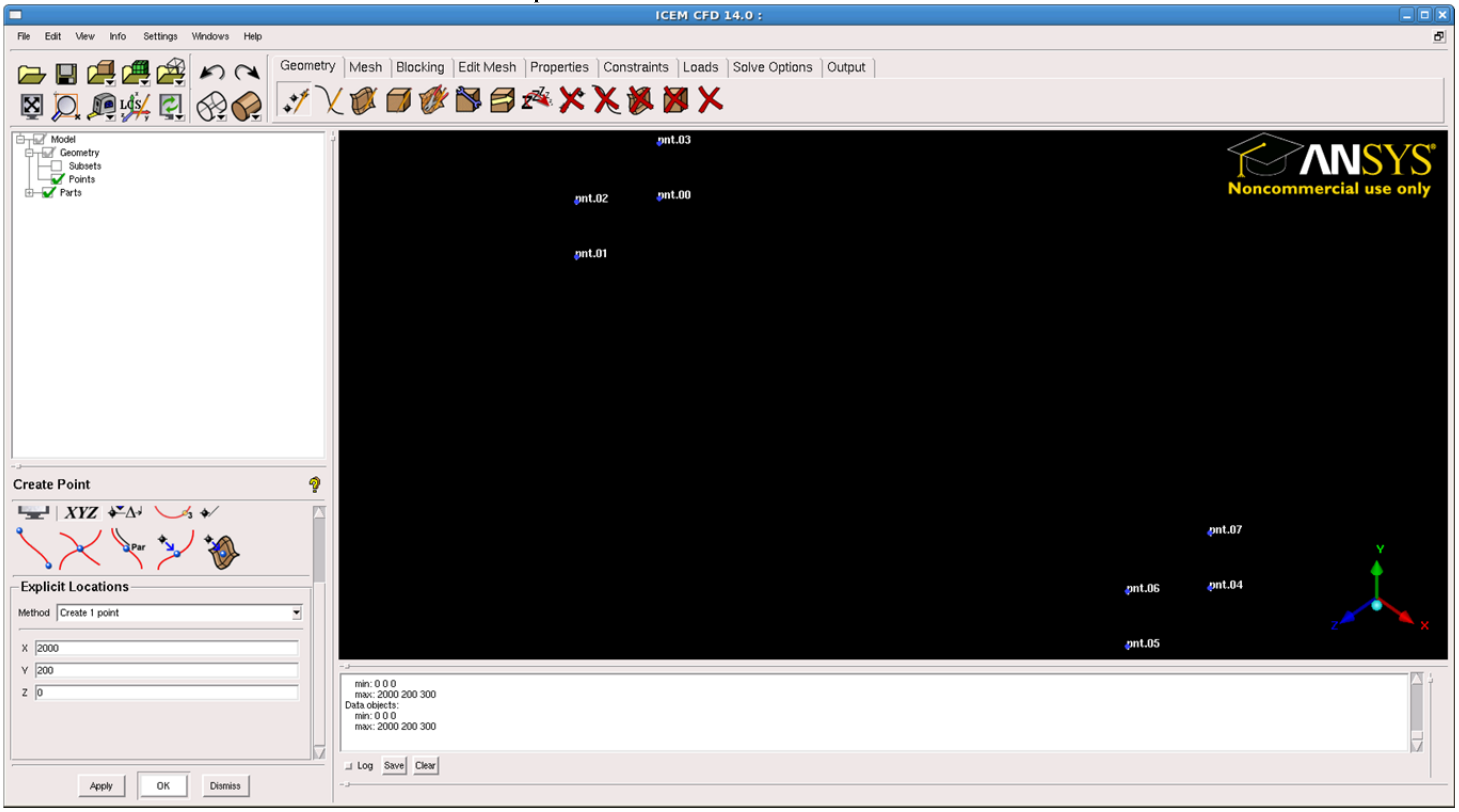
Also, after each point is entered, click on the 'Fit Window' from the main buttons (you may need to left click within the view window after pressing 'Fit Window' to enact the change).



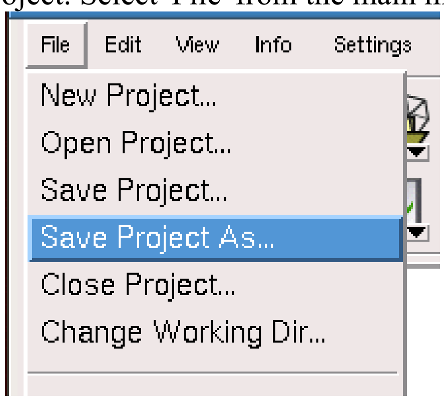
- When all of the points have been entered, click 'Dismiss' in the sidebar. Recall that we have defined the position of these points 1000 times larger than the geometry dictates (2.0 has become 2000) for convenience of working in units that we can think of as mm. We will scale this down by 1000 when we create the CFX readable file (our very last step).
- If you want to change the view of the geometry in the view window 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

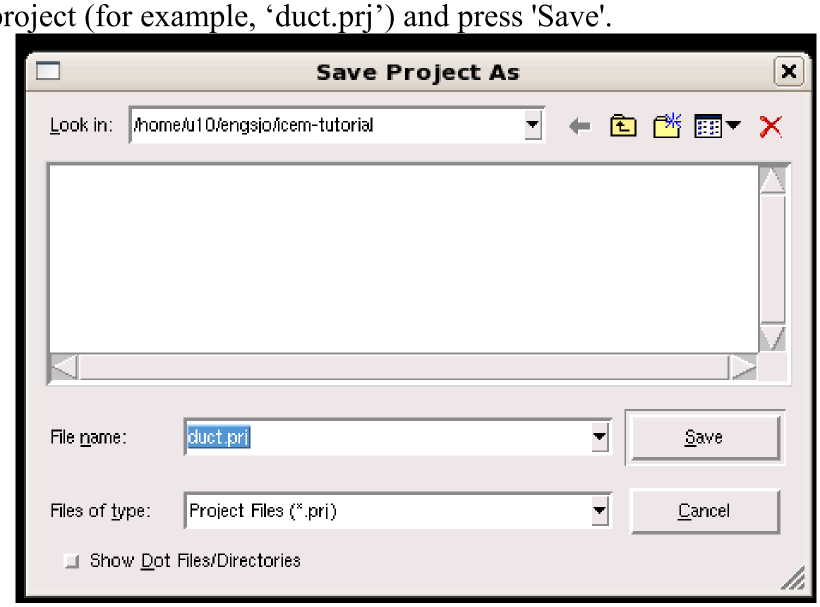
- Familiarize yourself with these controls by experimenting. When you are ready, press ‘Fit Window' from the main buttons and select 'Isometric' from 'View' in the main menu. Your screen should now look like this:



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



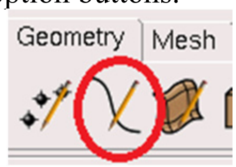
- Move to the directory you created in the setup steps of this tutorial and then choose a meaningful name for this project (for example, 'duct.prj') and press 'Save'.



- In the future, save frequently by clicking on the floppy disk icon ('Save Project') in the main buttons.

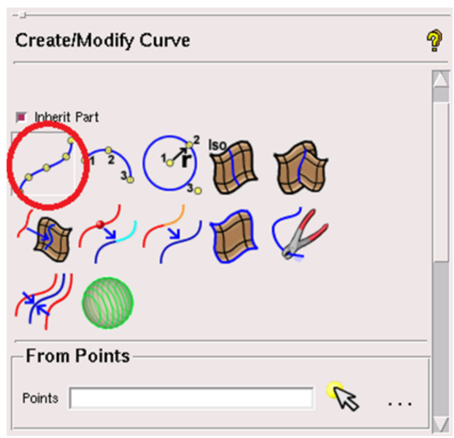
2.2 Creating curves/lines

- Select 'Create/Modify Curve' from the option buttons.

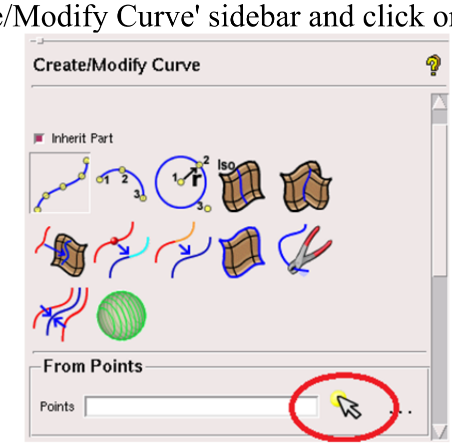


The new sidebar 'Create/Modify Curve' will be opened.

- From within the sidebar select the 'From Points' button. Let the curves be created under part GEOM.

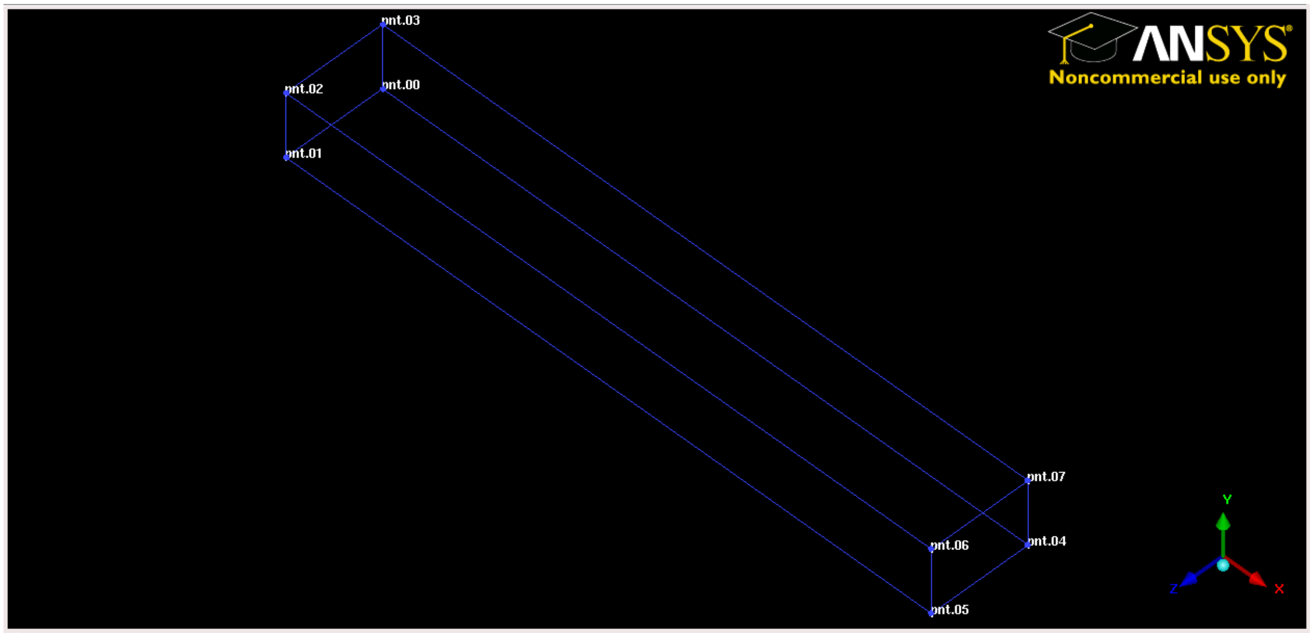


- Scroll down the 'Create/Modify Curve' sidebar and click on the 'Select location(s)' button.



- 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 remaining eleven lines.
- 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

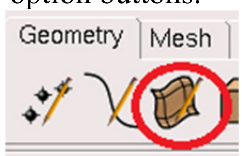
geometry as follows:



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

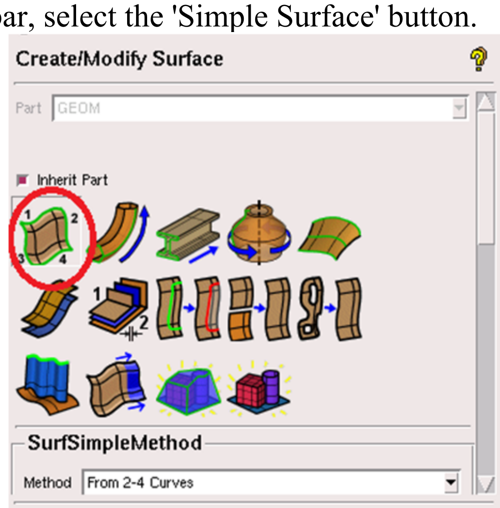
2.3 Creating surfaces

- Select 'Create/Modify Surface' from the option buttons.

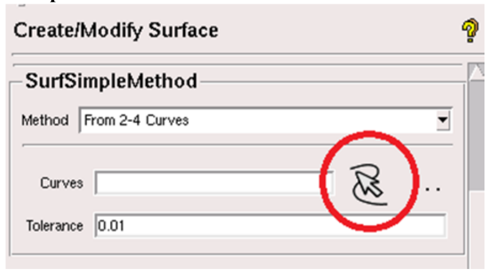


The new sidebar 'Create/Modify Surface' will be opened.

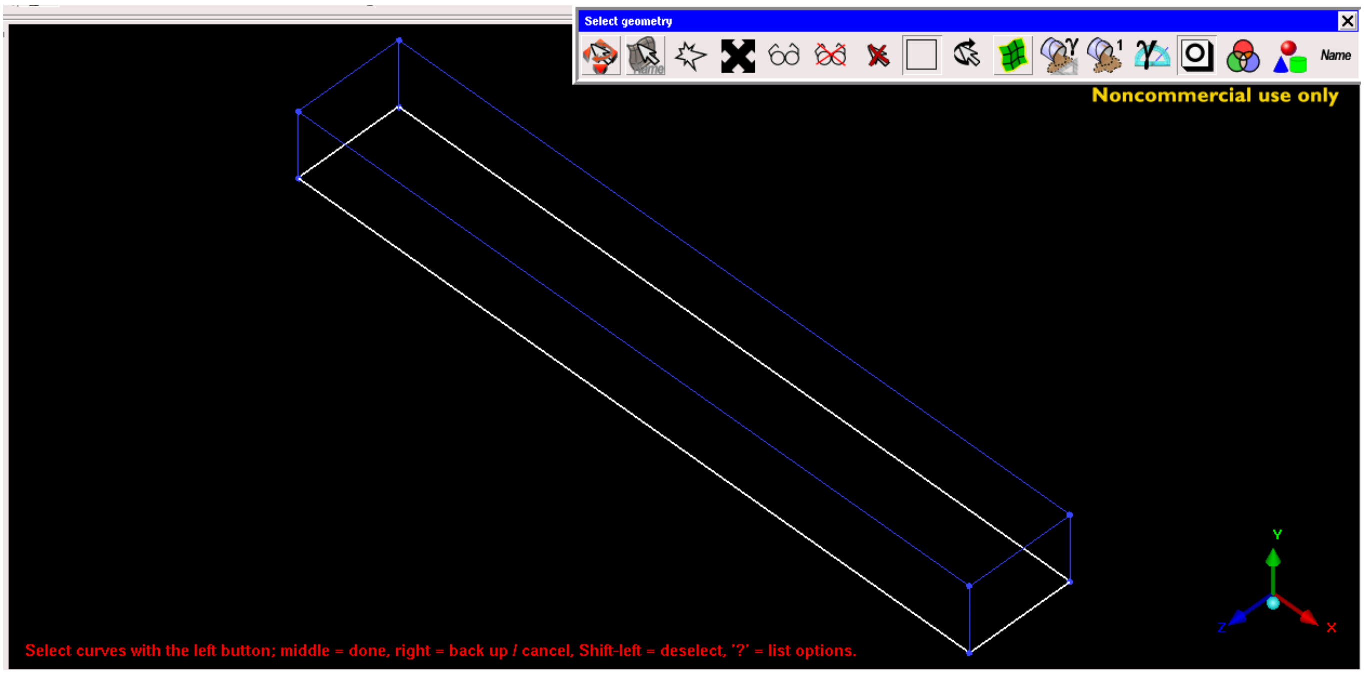
- From within the sidebar, select the 'Simple Surface' button.



- 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.



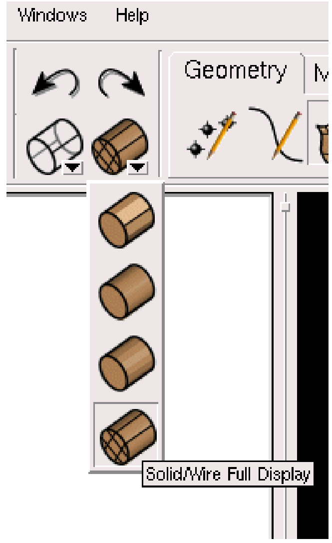
- With the left mouse button, select four curves that make up a face of the duct geometry. For example, the face that is an x-z plane at y=0:



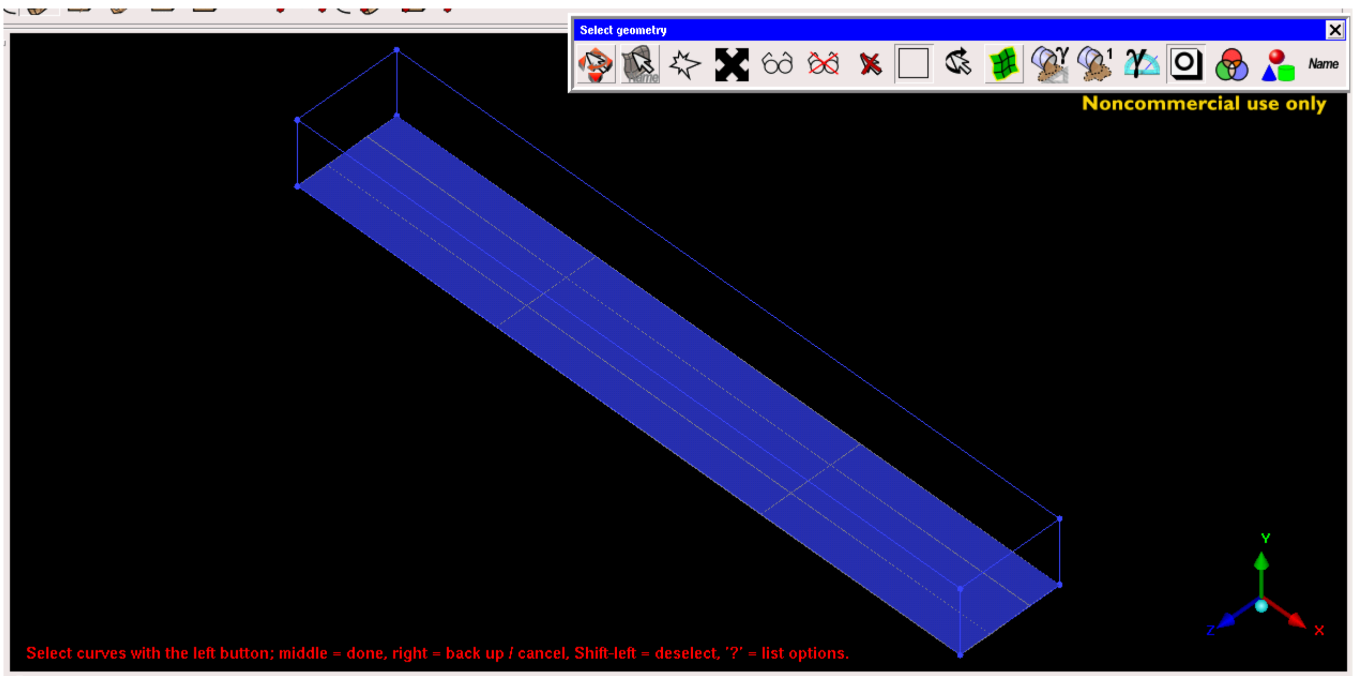
Each of the curves should turn white when clicked. If you find that you have clicked on the wrong curve by mistake, click the right mouse button to unselect it.

Now 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.

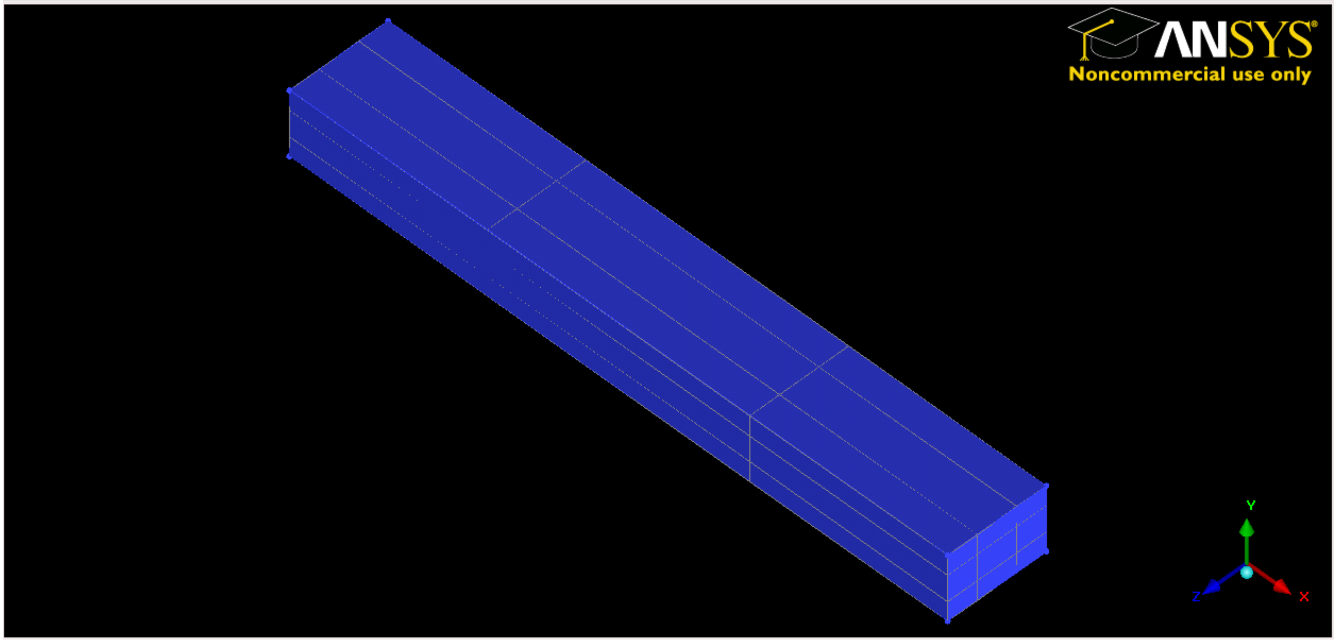
- In order to view the surfaces, click the box beside “Surfaces” in the model tree under “Geometry”. This will show a rough wireframe on the surface that was just created. This presentation of a surface contains extra lines that can be confusing when trying to select other lines for the remaining surfaces. It is suggested that you change the way surfaces are displayed by selecting “Solid/Wire Full Display”:



Your display of the surface that was just created should now look like:



- Repeat the process of selecting the four curves needed for the remaining five surfaces of the duct.
- 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:

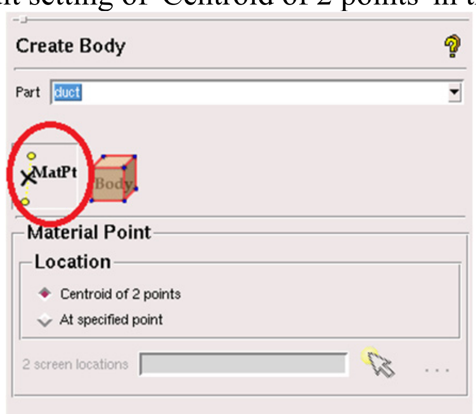


- Save the project.

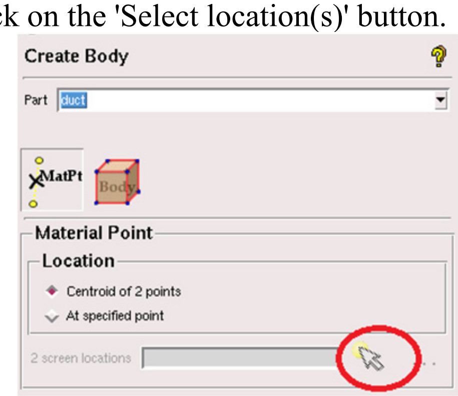
2.4 Defining the volume or 'body'

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

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



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

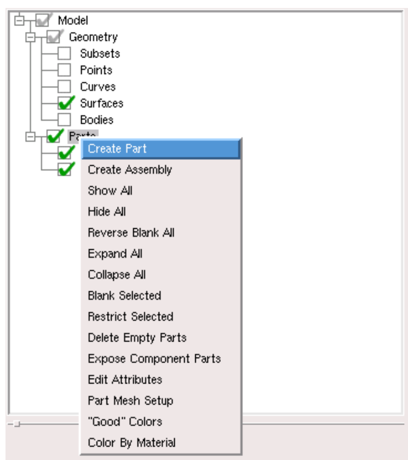


- Now, in order the see the points that are to be selected, turn off (unclick) Surfaces under Geometry in the model tree.

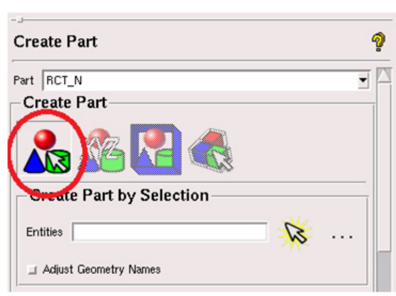
- With the left mouse button select two points that are on opposite corners of the domain. For example, this can be the origin (pnt.00) and the point at x=2000, y=200, and z=300 (pnt.06) in the view window, then press the middle mouse button, and then press 'Apply' in the sidebar. A label "duct" should appear in the middle of the geometry in the 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 on the surfaces again in the model tree.
- Save the project

3. Creating Parts

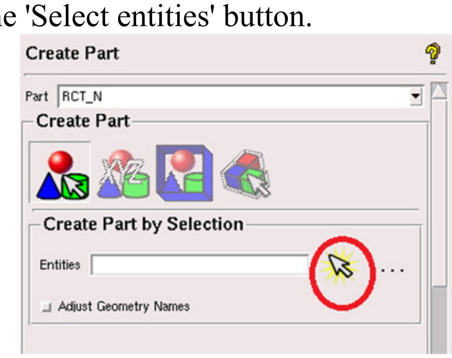
- Now that the full geometry has been developed, it is helpful to assign meaningful names to the different surfaces. These names will be used when applying boundary conditions to surfaces in CFX.
- Because we will be selecting only surfaces to create parts, uncheck Points, Curves, and Bodies under Geometry in the model tree.
- Assigning names to surfaces is done by creating a part for each surface. In the geometry tree **right click** the word 'Parts' and select 'Create Part' from the menu that appears. The new sidebar 'Create Part' will be opened.



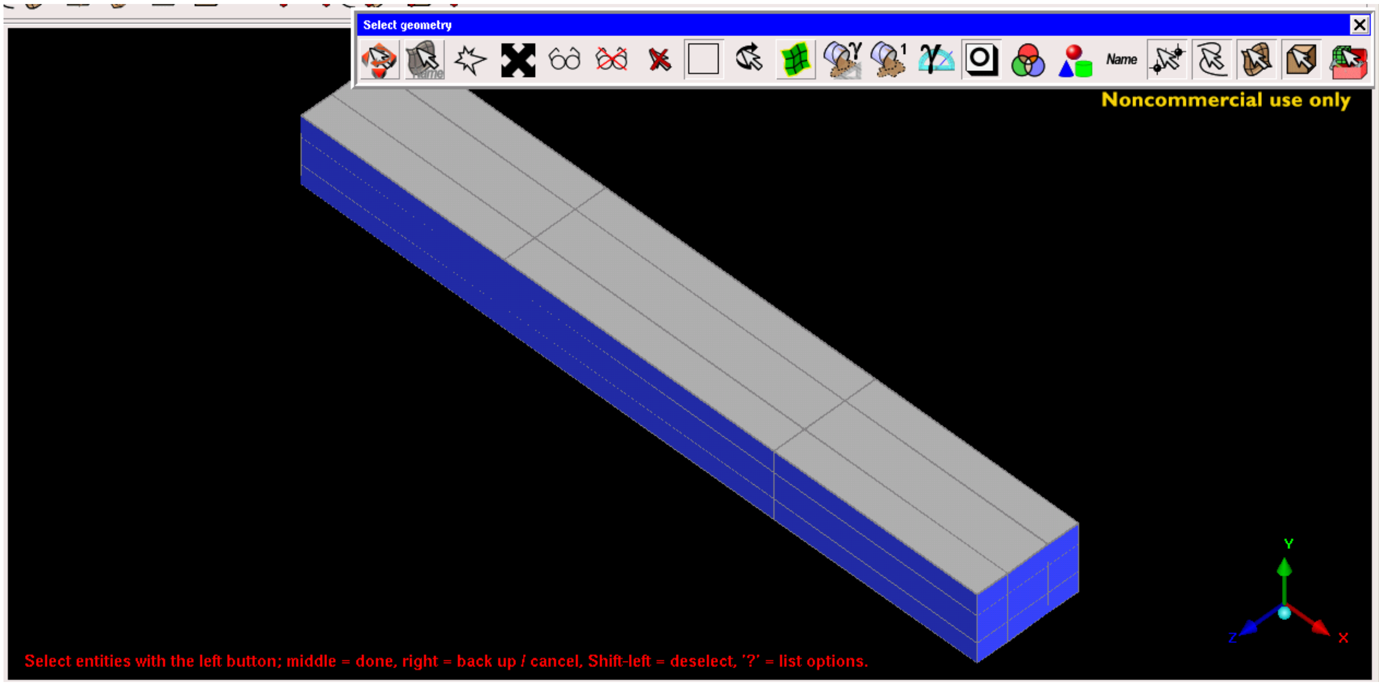
- Within the sidebar, enter the name 'RCT_N' in the 'Part' field and click the 'Create Part by Selection' button.



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



- In the isometric view, select the top surface of the geometry (the x-z plane at y=200) in the view window:

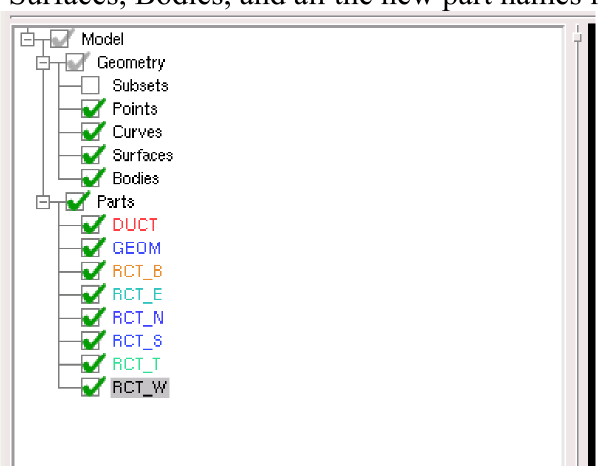


- Press the **middle mouse button**, and then select 'Apply' in the sidebar. The new part 'RCT_N' will now appear in the geometry under 'Parts'. Uncheck RCT_N, so the surface is no longer displayed. Removing RCT_N from the view window will make it easier to select the other surfaces.
- For the remaining five surfaces, follow the procedure above. Remember to uncheck each part as it is created. This will also help to know which surfaces are remaining.

- Assign the following part names to the remaining surfaces:

x-y plane at z=300	RCT_T
x-y plane at z=0	RCT_B
x-z plane at y=0	RCT_S
y-z plane at x=0	RCT_W
y-z plane at x=2000	RCT_E

- Once all of the parts are created, press 'Dismiss' in the sidebar
- Check Points, Curves, Surfaces, Bodies, and all the new part names in the model tree



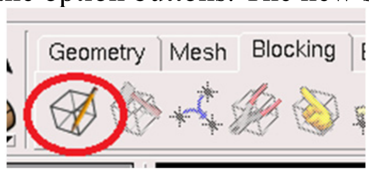
- Save the project.

4. Creating a Block

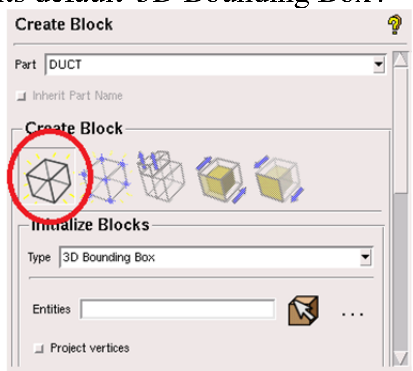
- Select 'Blocking' from the option tabs.

4.1 Create the block

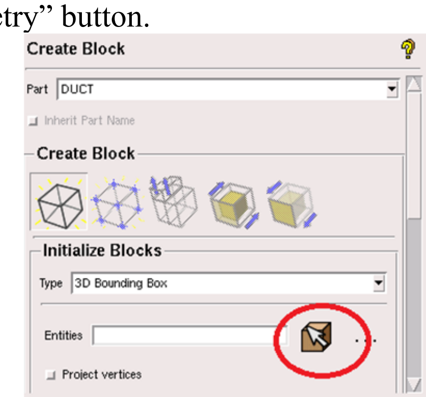
- Select 'Create Block' from the option buttons. The new sidebar 'Create Block' will be opened.



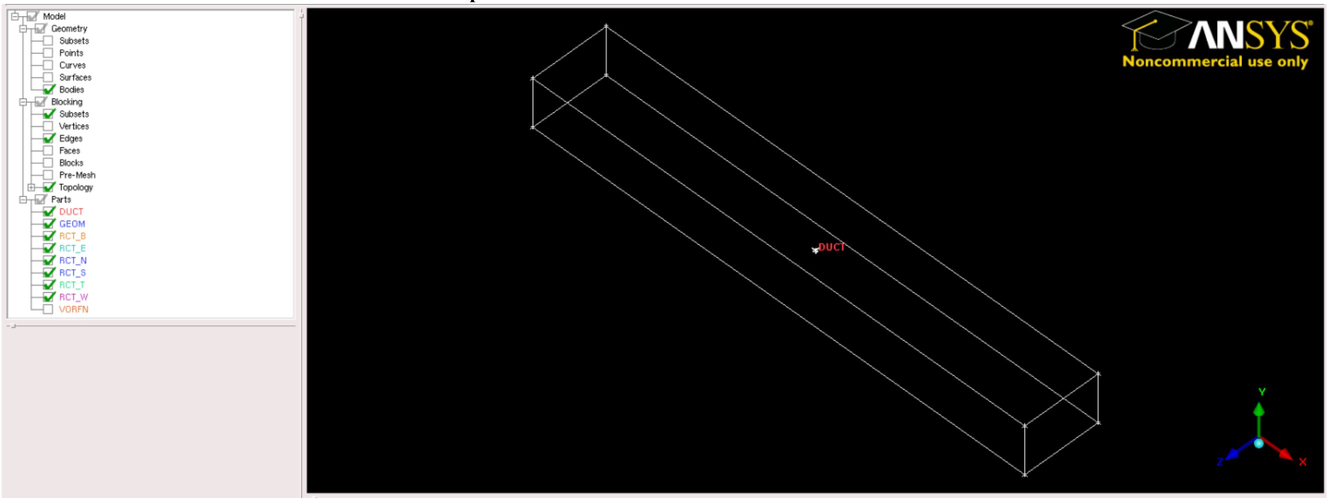
- From within the sidebar select the 'Initialize Blocks' button and set the 'Part' field to ‘DUCT’. Leave the 'Type' field as its default '3D Bounding Box'.



- Reduce the size of the geometry in the view window (by scrolling the mouse scroll button or using the right mouse button). The image must not be near the edges of the view window because you are going to select all of it next.
- Click the “Select geometry” button.



- In the view window, select all of the geometry by 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:



You are seeing the edges of the block. They are coloured white until they are associated with a curve in your geometry.

- Save the project.

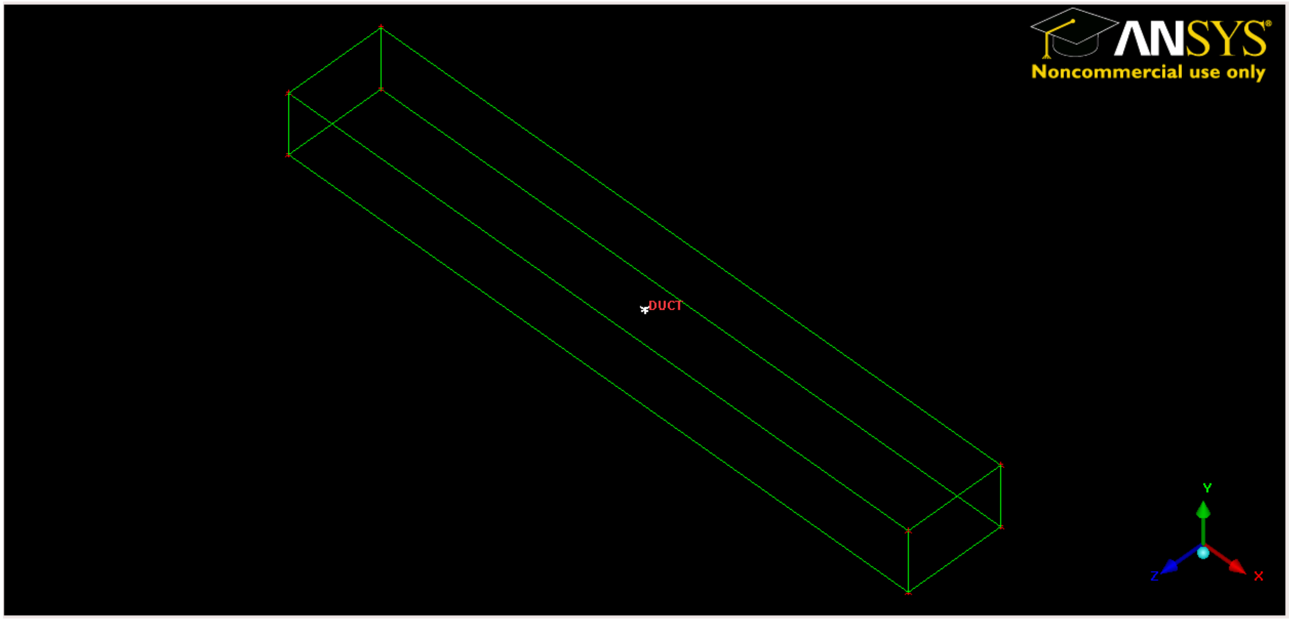
5. Doing the Edge Associations

5.1 Associate the created geometry to the block

- Select 'Associate' from the option buttons. The new sidebar 'Blocking Associations' will be 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:



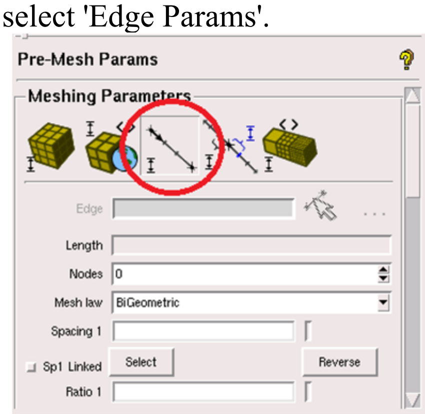
- Note that auto-association can only be used because this is a very simple geometry. For more complicated geometries, manual association must be performed.
- Save the project.

6 Nodal Distributions

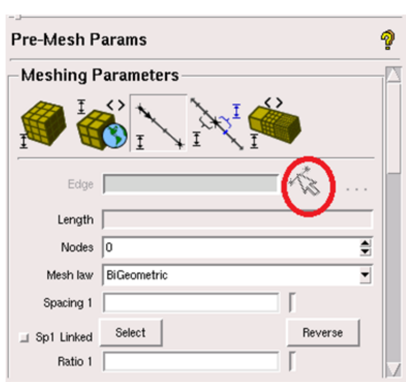
6.1 Allocate nodal distribution along the block edges

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

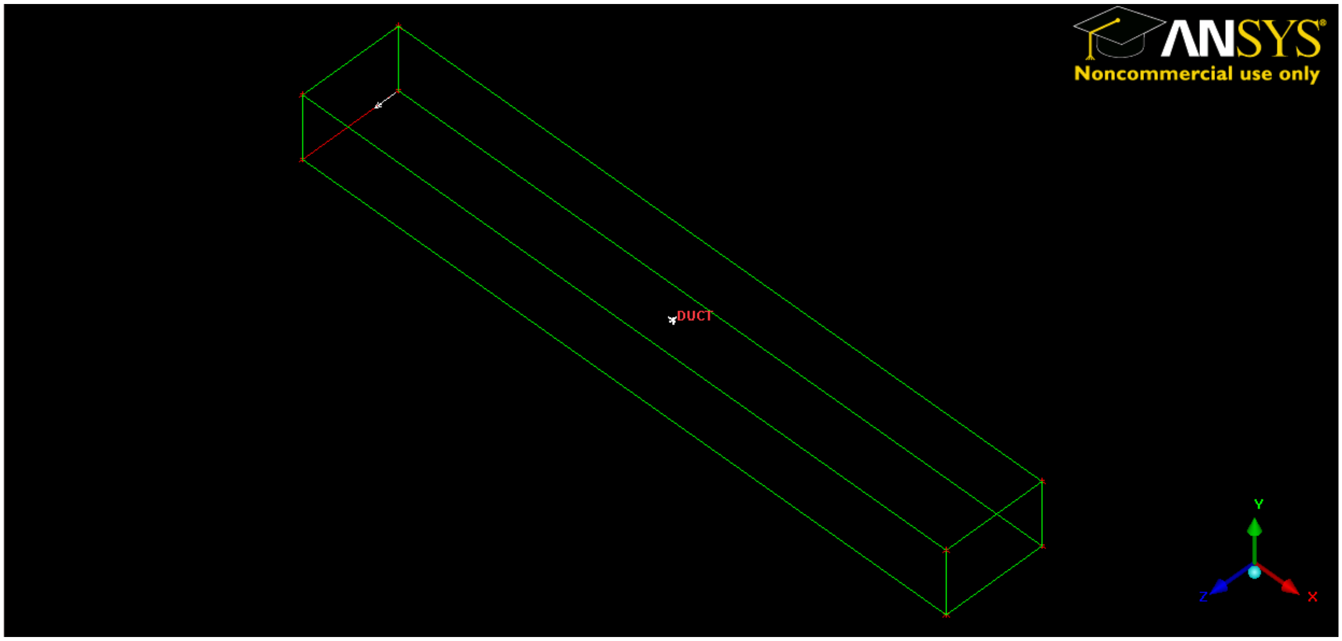
- From within the sidebar select 'Edge Params'.



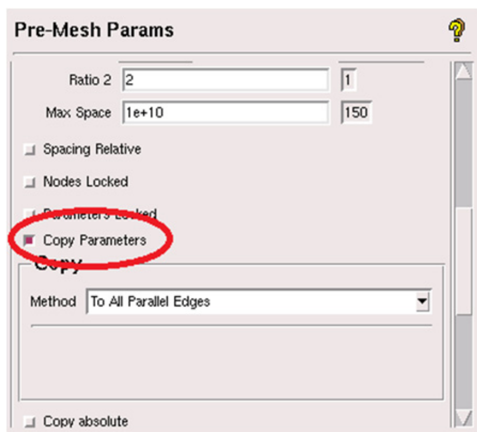
- Click 'Select edge(s)



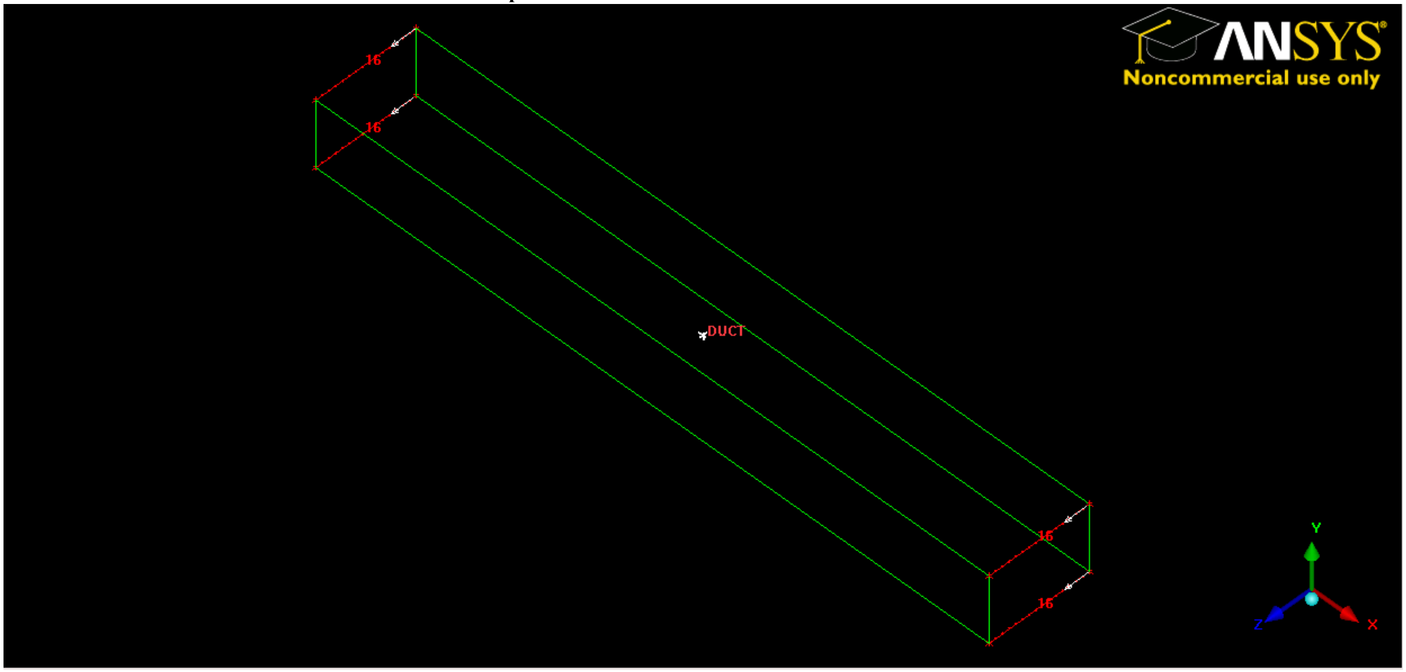
- Select the edge line the goes from (0,0,0) to (0,0,300) (i.e., from pnt.00 to pnt.01) in the view window.



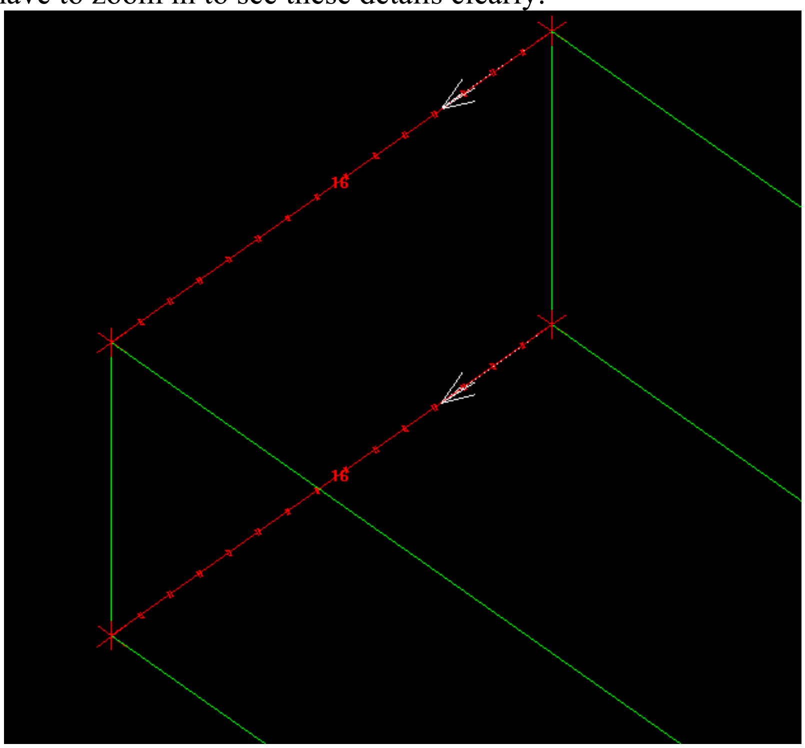
- In the sidebar again, enter 16 in the 'Nodes' field and set the 'Mesh law' field to 'BiGeometric' (note that you may have to reselect the curve and enter the number of nodes a second time).
- Scroll down the sidebar and select 'Copy Parameters' and then set the 'Method' field to 'To All Parallel Edges'.



- Press 'Apply' and you should now see small red ticks along all the lines in the z-direction. There is also the number of nodes printed in red.



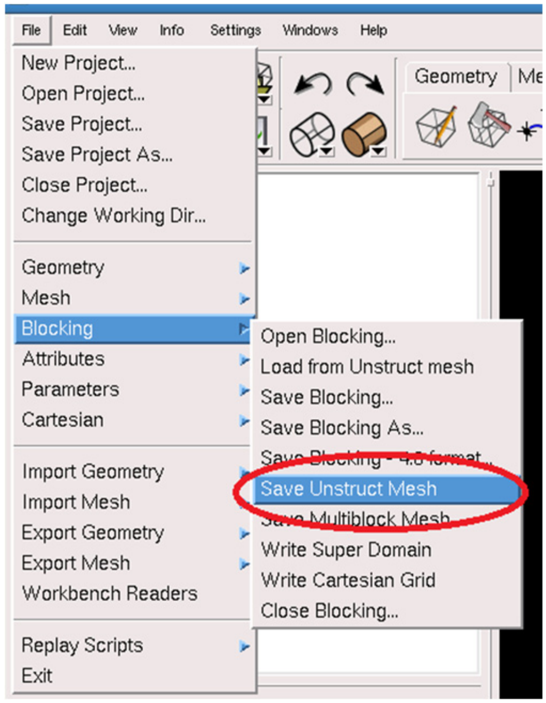
- You will have to zoom in to see these details clearly.



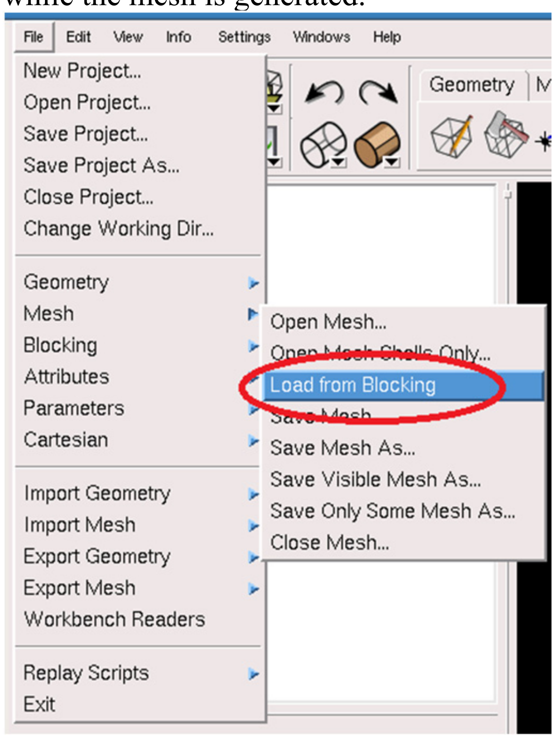
- Scroll back to the top of the sidebar and click 'Select edge(s)' again. This time select the line from (0,0,0) to (0,200,0) in the view window and then enter 11 in the 'Nodes' field of the sidebar. Ensure that the 'Mesh law' is still set to 'BiGeometric', the 'Copy Parameters' button is still highlighted, 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 the line from (0,0,0) to (2000,0,0) in the view window and then enter 41 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.

7 Meshing the Block

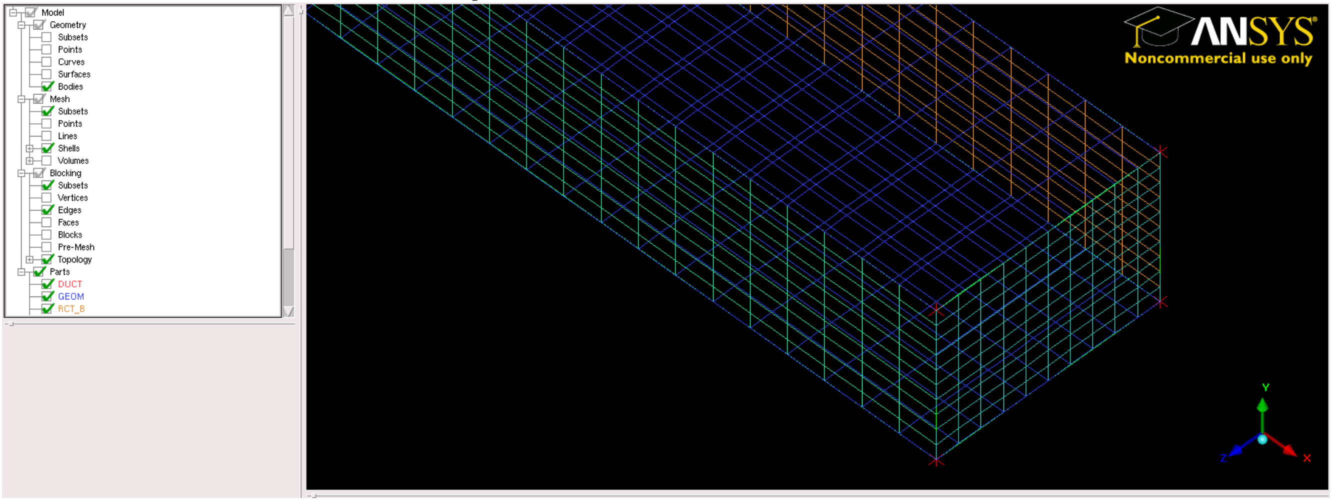
- Select 'File' from the main menu, then select 'Blocking' and 'Save Unstruct Mesh' in the menus that appear.



- Name the file something meaningful like ‘duct’ 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.

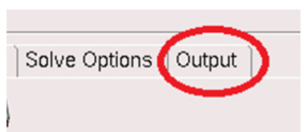


- Save the project.

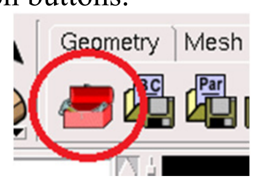
8 Creating the file for CFX

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.

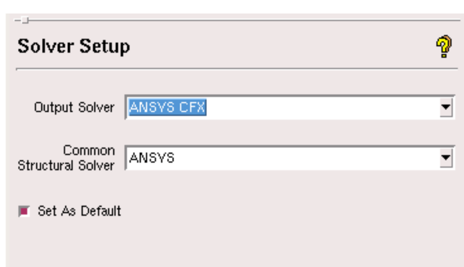


- Select 'Select solver' from the option buttons.

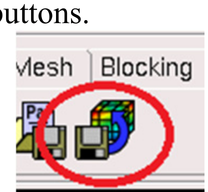


The sidebar 'Solver Setup' will be opened.

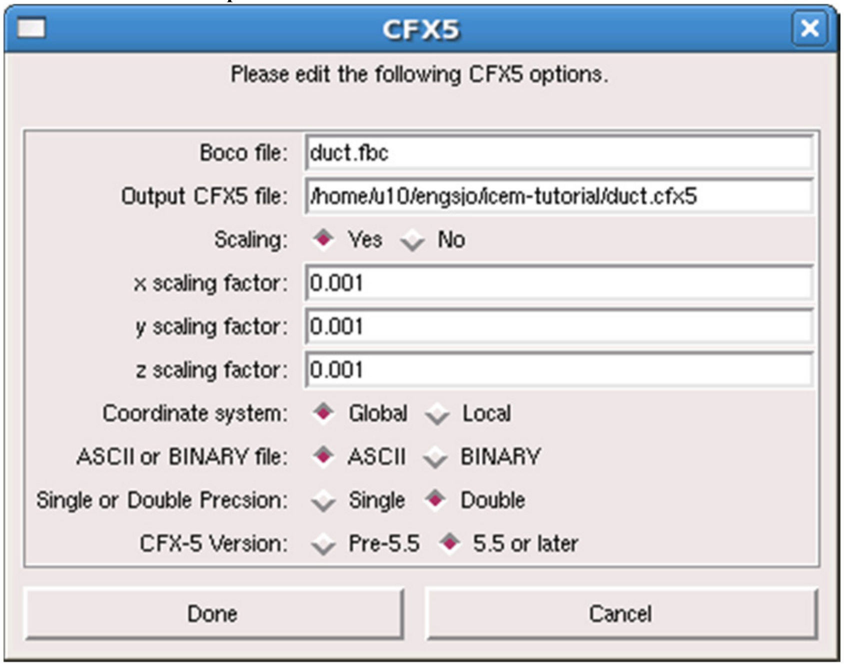
- In the sidebar, set the 'Output Solver' field to 'ANSYS CFX', the 'Common Structural Solver' field to 'ANSYS'.



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



- You will be prompted to save the current project first, 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 created a basic mesh using ANSYS ICEM. The .cfx5 file that you created can be read into CFX-Pre.