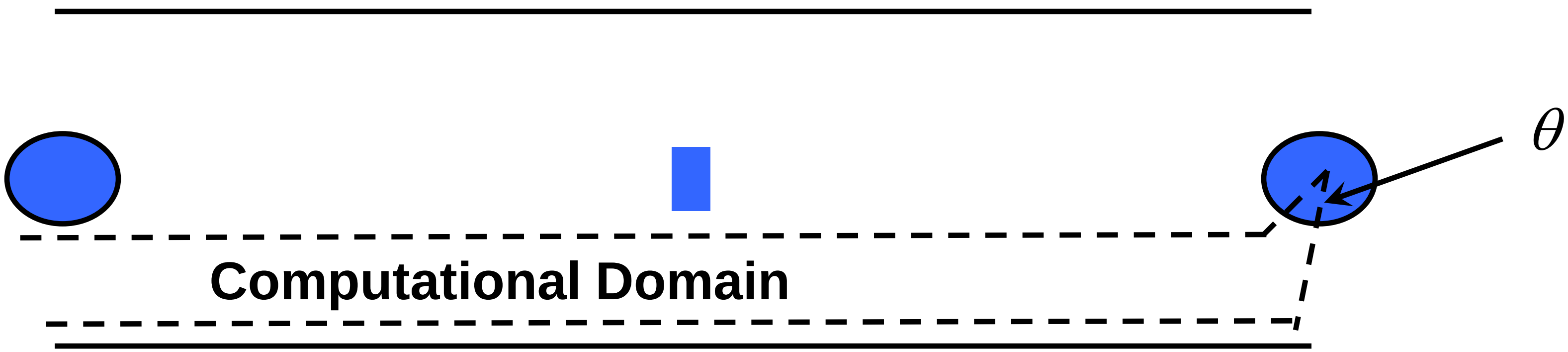
2-D Internal Flow

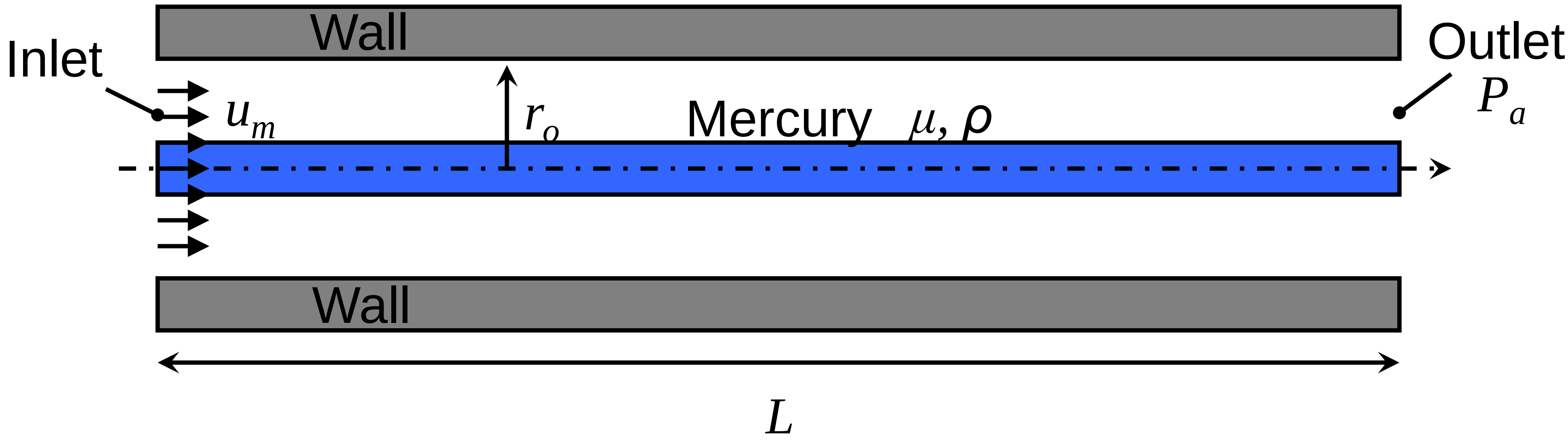
Axisymmetry

- Axisymmetric problems in CFX require a wedge-shaped computational domain.
- The angle defining this section is arbitrary. However, it must be small, relative to the full geometry and large enough for the mesher to resolve. (i.e. less than 30º, greater than 5º).



- Wedge-shaped elements are then created, and the continuity and momentum equations are solved in polar coordinates.

Geometry: Pipe



- Axisymmetric, laminar, internal flow through pipe.

Parameters

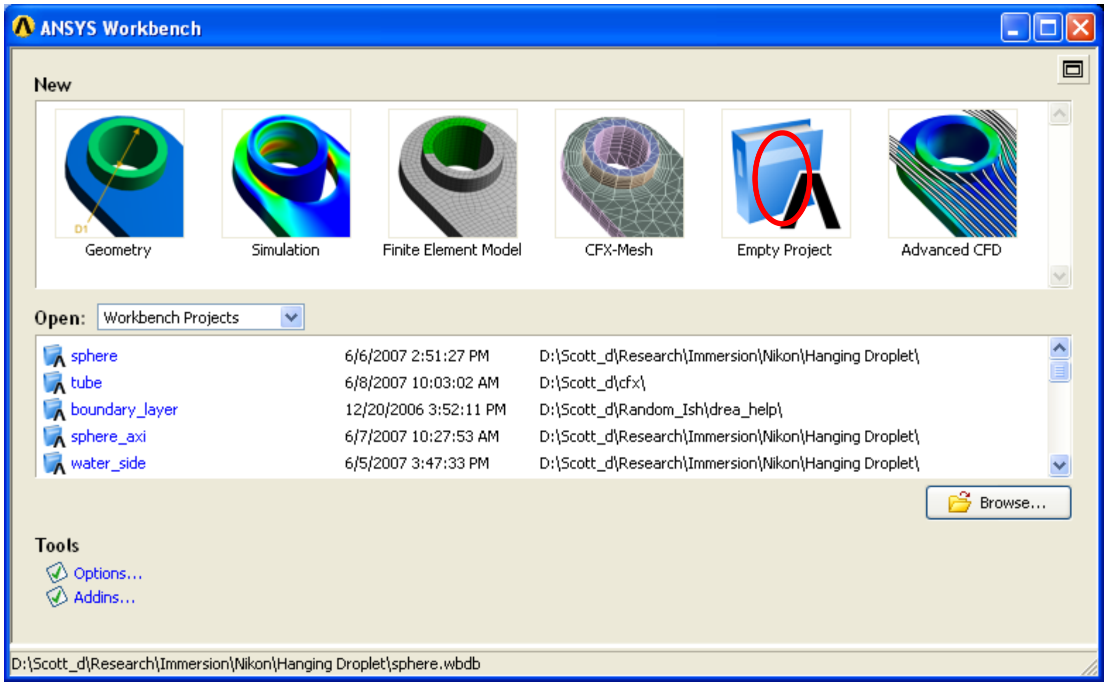
Parameter	Symbol	Value
radius	r_o	5 mm
length	L	65 mm
mean velocity	u_m	1.0 mm/s
viscosity	μ	1.552E-3 kg/m-s
density	ρ	13546 kg/m^3
atmospheric pressure	P_a	1 atm

Workbench

1. Create new directory

- called **Pipe**

2. Open ANSYS Workbench
3. Create Empty Project



4. Save file in new directory: **pipe.wbdb**

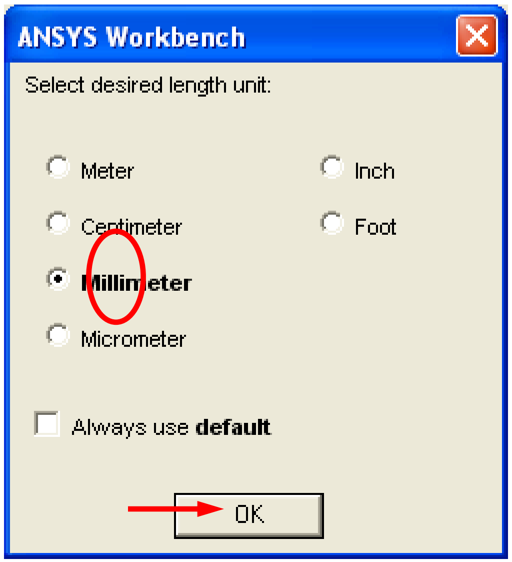
- ***File - Save***

5. Open new geometry in DesignModeler

- On left side of page click on ***New Geometry***

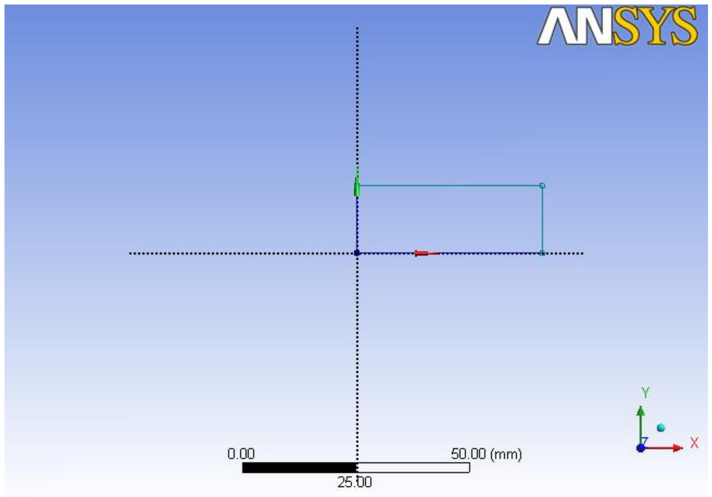
6. Select unit of length

- Select Millimeter, ***OK***



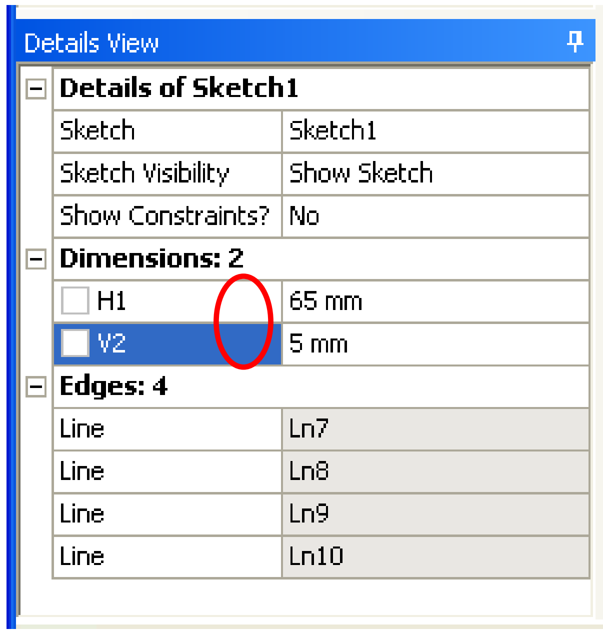
7. Create rectangle for pipe flow

- Highlight ***XYPlane*** in ***Tree Outline***.
- ***Toolbar – Look At Face/Plane/Sketch***
- ***Toolbar – New Sketch***
- ***Tree Outline – Sketching – Draw – Rectangle***
- Select global origin as first point by left-clicking on it when ***P*** is displayed.
- Select second point by left-clicking anywhere in first quadrant.



8. Dimension rectangle

- ***Tree Outline – Sketching – Dimensions – General***
- Left-click and hold line at $y = 0$. Drag dimension marker to easily visible location and release left mouse button to drop.
- Repeat for line at $x = 0$.
- In ***Details View***, update dimensions such that horizontal line is **65 mm** and vertical line is **5 mm**.

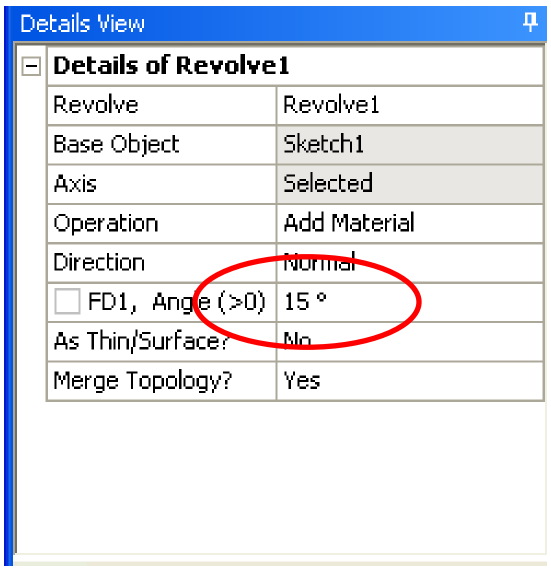


9. Revolve sketch

- In ***Tree Outline,*** select ***Sketch1***.
- From ***Toolbar***, select ***Revolve*** 

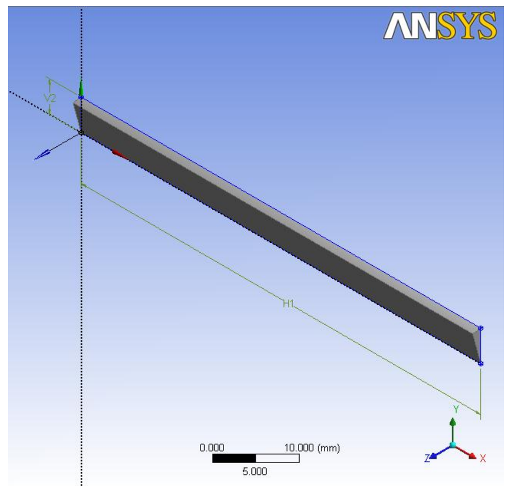
 icon.
- In ***Details View*** for ***Axis***, left-click ***X – Axis*** in Viewer. Click ***Apply***.
- In ***Details View*** for ***FD1, Angle (>0)***, enter **15**.
- From ***Toolbar***, select ***Generate*** 

 icon.



Details View

Details of Revolve1	
Revolve	Revolve1
Base Object	Sketch1
Axis	Selected
Operation	Add Material
Direction	Normal
FD1, Angle (>0)	15 °
As Thin/Surface?	No
Merge Topology?	Yes



10. Save Project

- On ***Project Page***, select ***File – Save All***. When prompted, save DesignModeler file as **pipe.agdb**.

11. Open CFX-Mesh

- On ***Project Page*** with ***pipe.agdb*** highlighted, select ***New Mesh*** 

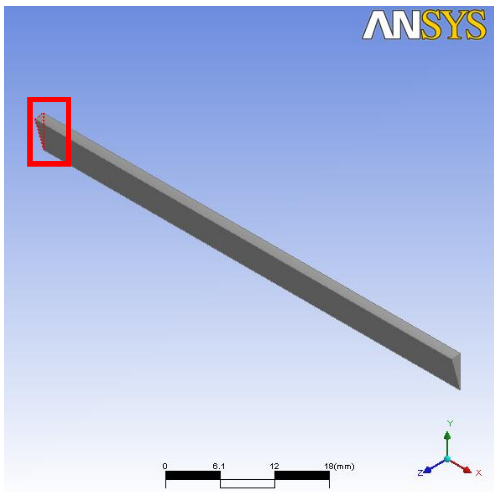
icon from left pane.

12. Save Mesh File

- Once CFX-Mesh opens, select ***File – Save***.
- Save file as **pipe.cmdb**.
- Note that it should now appear on ***Project Page*** as ***Model***.

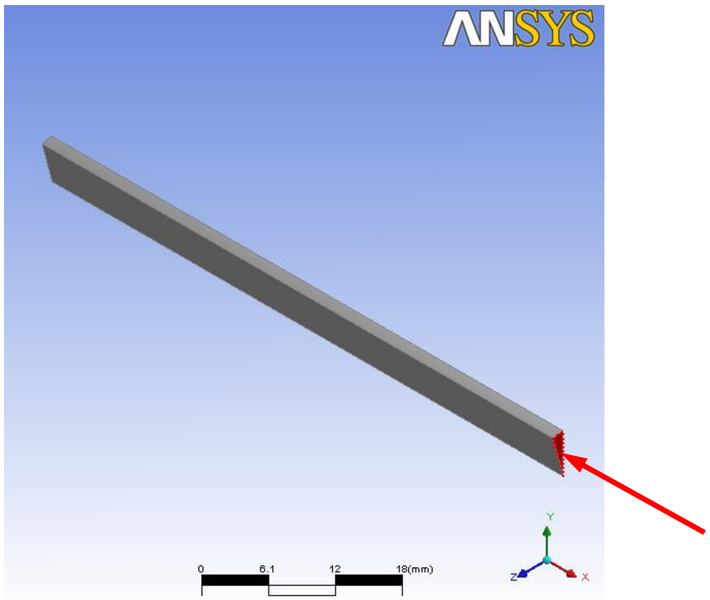
13. Create Inlet Region

- Place geometry in isometric view by selecting blue ***Iso*** ball on triad in ***Viewer***.
- In ***Tree View***, right-click on ***Regions***. ***Insert – Composite 2D Region***.
- When ***Composite 2D Region 1*** first appears in ***Tree View***, you may enter a new name. Enter ***LetIn***.
- In ***Viewer***, select surface with lowest x-coordinate by dragging a box around it. In ***Details View***, select ***Apply***.



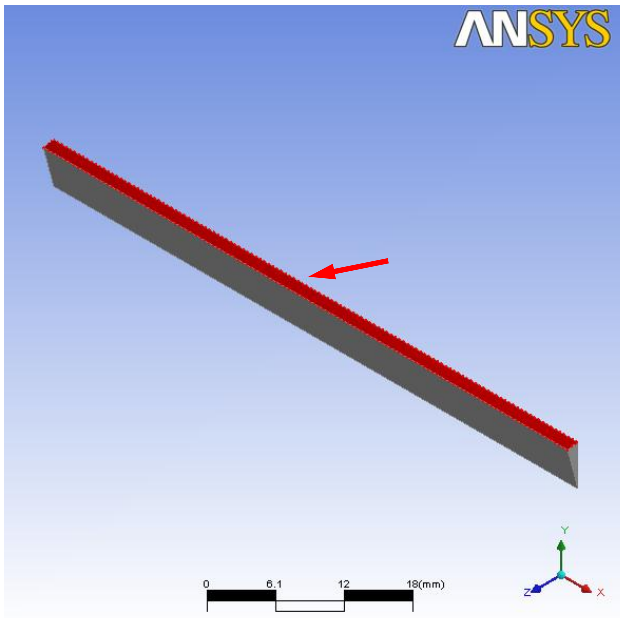
14. Create Outlet Region

- In ***Tree View***, right-click on ***Regions***. ***Insert – Composite 2D Region***.
- When ***Composite 2D Region 1*** first appears in ***Tree View***, you may enter a new name. Enter ***LetOut***.
- Using left mouse button in Viewer, select surface with largest x – coordinate. In ***Details View***, select ***Apply***.



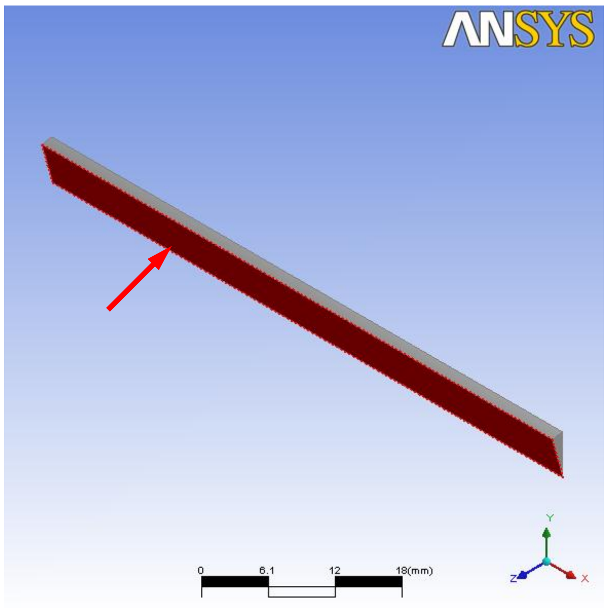
15. Create Outer Wall Region

- In ***Tree View***, right-click on ***Regions***. ***Insert – Composite 2D Region***.
- When ***Composite 2D Region 1*** first appears in ***Tree View***, you may enter a new name. Enter ***Wall***.
- Using left mouse button in Viewer, select surface with largest y – coordinate. In ***Details View***, select ***Apply***.



16. Create First Symmetry Region

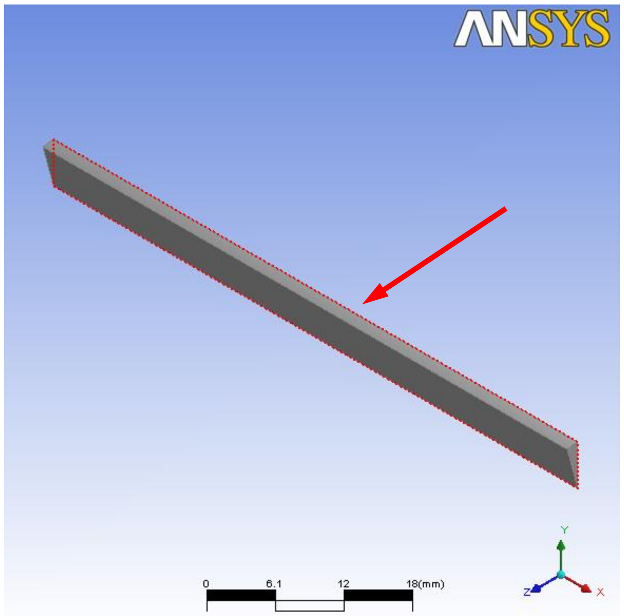
- In ***Tree View***, right-click on ***Regions***. ***Insert – Composite 2D Region***.
- When ***Composite 2D Region 1*** first appears in ***Tree View***, you may enter a new name. Enter ***Sym1.***
- Using left mouse button in Viewer, select surface with largest z – coordinate. In ***Details View***, select ***Apply***.



17. Create Second Symmetry Region

- In ***Tree View***, right-click on ***Regions***. ***Insert – Composite 2D Region***.
- When ***Composite 2D Region 1*** first appears in ***Tree View***, you may enter a new name. Enter ***Sym2.***
- Using Face Selector Helper, select surface with lowest z – coordinate. In ***Details View***, select ***Apply***.

File - Save

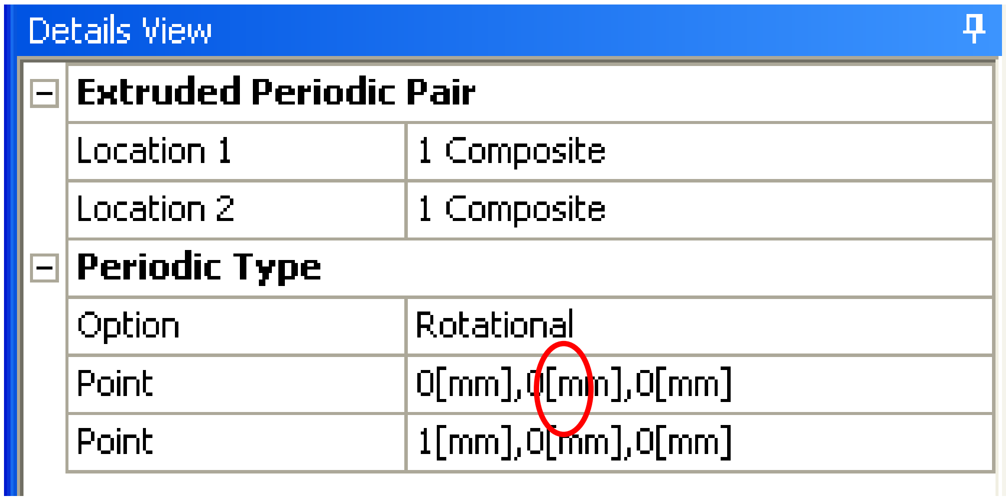


18. Change Mesh to 2D

- In ***Tree View*** under ***Mesh***, select ***Options***.
- ***Details View – Meshing Strategy*** – ***Option***. Select **Extruded 2D Mesh**.
- ***Details View – 2D Extrusion Option – Number of Layers***, enter **1**.
- In ***Tree View*** under ***Mesh***, select ***Extruded Periodic Pair***.
- ***Details View – Periodic Type – Option***. Select **Rotational**.
- ***Details View – Periodic Type*** . Right-click on first ***Point*** and select ***Edit***. Set first point to **0, 0, 0**. Hit Enter.
- ***Details View – Periodic Type*** . Right-click on second ***Point*** and select ***Edit***. Set second point to **1, 0, 0**. Hit Enter.

18. Change Mesh to 2D (cont.)

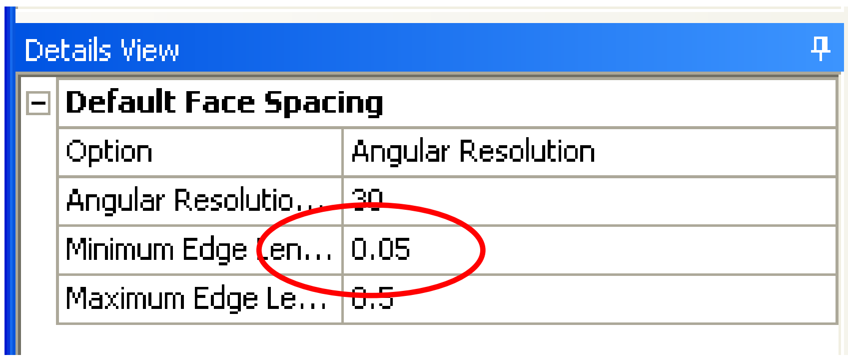
- ***Details View – Extruded Periodic Pair – Location 1***. Select **Sym1** from ***Tree View***. Click ***Apply***.
- ***Details View – Extruded Periodic Pair – Location 2***. Select **Sym2** from ***Tree View***. Click ***Apply***.



19. Update Face Spacing

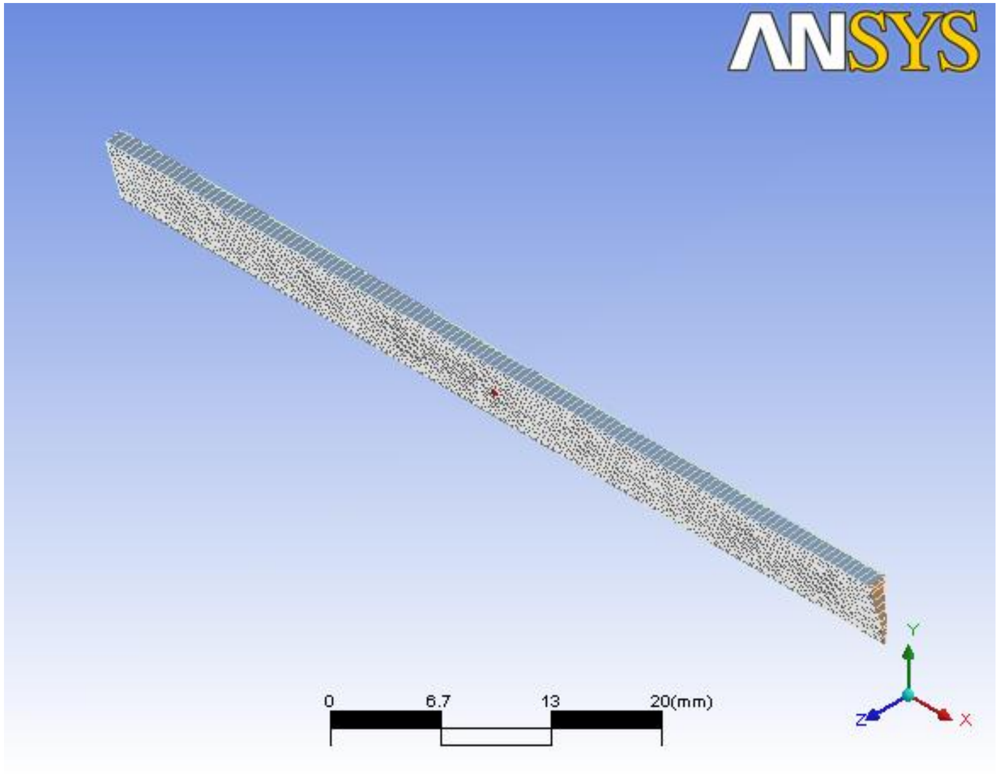
- In ***Tree View*** under ***Mesh – Spacing***, select ***Default Face Spacing***.
- In ***Details View***, set ***Minimum Edge Length*** to **0.05 mm**.
- In ***Details View***, set ***Maximum Edge Length*** to **0.5 mm**.
- Note that Maximum Edge Length was set such that there would be approximately 10 elements through the radius.

$$\text{Maximum Edge Length} = r_o / 10$$



20. Create Preview

- ***Toolbar – Generate Surface Meshes***



File - Save

21. Create Mesh

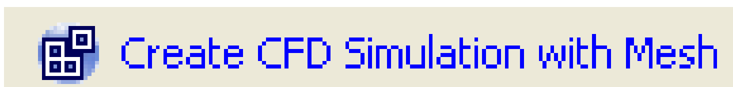
- ***Toolbar – Generate Volume Mesh***
- Save file as **pipe.gtm**.
- Mesh takes a few seconds to generate.

22. Save Project

- On ***Project Page***, select ***File – Save All***.

23. Open CFX

- On ***Project Page*** with ***pipe.gtm*** highlighted, select ***Create CFD Simulation with Mesh*** icon.



24. Define Model Data

- When CFX finally opens, in ***Tree Outline*** double-click on ***Default Domain***.
- ***Domain: Default Domain*** Tab appears.
- ***General Options – Domain Models – Pressure – Reference Pressure***. Ensure it is set to **1 atm**.
- ***Fluid Models – Heat Transfer – Option***. Ensure it is set to **Isothermal**.
- ***Fluid Models – Heat Transfer – Fluid Temperature***. Ensure it is set to **25 C**.
- ***Fluid Models – Turbulence – Option***. Ensure it is set to **None (Laminar)**.
- Click ***Apply*****.** Click ***OK*****.**

Material Library

- In addition to defining your own materials, CFX has the capability of importing material properties from a large library.
- Further, created materials may be stored here for later import.
- Material types include:

  - Constant Property and Ideal Gases

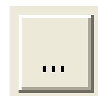
  - Constant Property Liquids

  - Combustion Products

  - Solids

  - Interphase Water

25. Import Material from Library

- ***Outline – Simulation***. Right-click ***Materials***. Select ***Import Library Data***.
- ***Select Library Data to Import*** window appears.
- Under ***Constant Property Liquids***, select **Mercury Hg**.
- Click ***OK***.
- ***Mercury Hg*** now appears under ***Materials*** in ***Tree Outline***.
- ***Tree Outline – Simulation***. Double-click on ***Default Domain***.
- ***General Options – Fluids List***. Left-click on box. 

- ***Fluids List*** window appears.
- Under ***Constant Property Liquids***, select ***Mercury Hg***.
- Click ***OK***.

26. Create Inlet Boundary Condition

- ***Toolbar – Create a Boundary Condition***. Enter **LetIn** for ***Name***. Click ***OK***.
- ***Boundary: LetIn – Basic Settings – Boundary Type***. Select **Inlet**.
- ***Boundary: LetIn – Basic Settings – Location***. Select **LetIn**.
- ***Boundary: LetIn – Boundary Details – Mass and Momentum – Option***. Ensure it is set to **Normal Speed**.
- ***Boundary: LetIn – Boundary Details – Mass and Momentum – Normal Speed***. Enter **1.0 mm/s**.
- Click ***Apply***. Click ***OK***.

27. Create Outlet Boundary Condition

- ***Toolbar – Create a Boundary Condition***. Enter **LetOut** for ***Name***. Click ***OK***.
- ***Boundary: LetOut – Basic Settings – Boundary Type***. Select **Outlet**.
- ***Boundary: LetOut – Basic Settings – Location***. Select **LetOut**.
- ***Boundary: LetOut – Boundary Details – Mass and Momentum – Option***. Select **Average Static Pressure**.
- ***Boundary: LetOut – Boundary Details – Mass and Momentum – Relative Pressure***. Enter **0 Pa.**
- Click ***Apply***. Click ***OK***.

28. Create Wall Boundary Condition

- ***Toolbar – Create a Boundary Condition***. Enter **ThatWall** for ***Name***. Click ***OK***.
- ***Boundary: ThatWall – Basic Settings – Boundary Type***. Select **Wall**.
- ***Boundary: ThatWall – Basic Settings – Location***. Select **Wall**.
- Click ***Apply***. Click ***OK***.

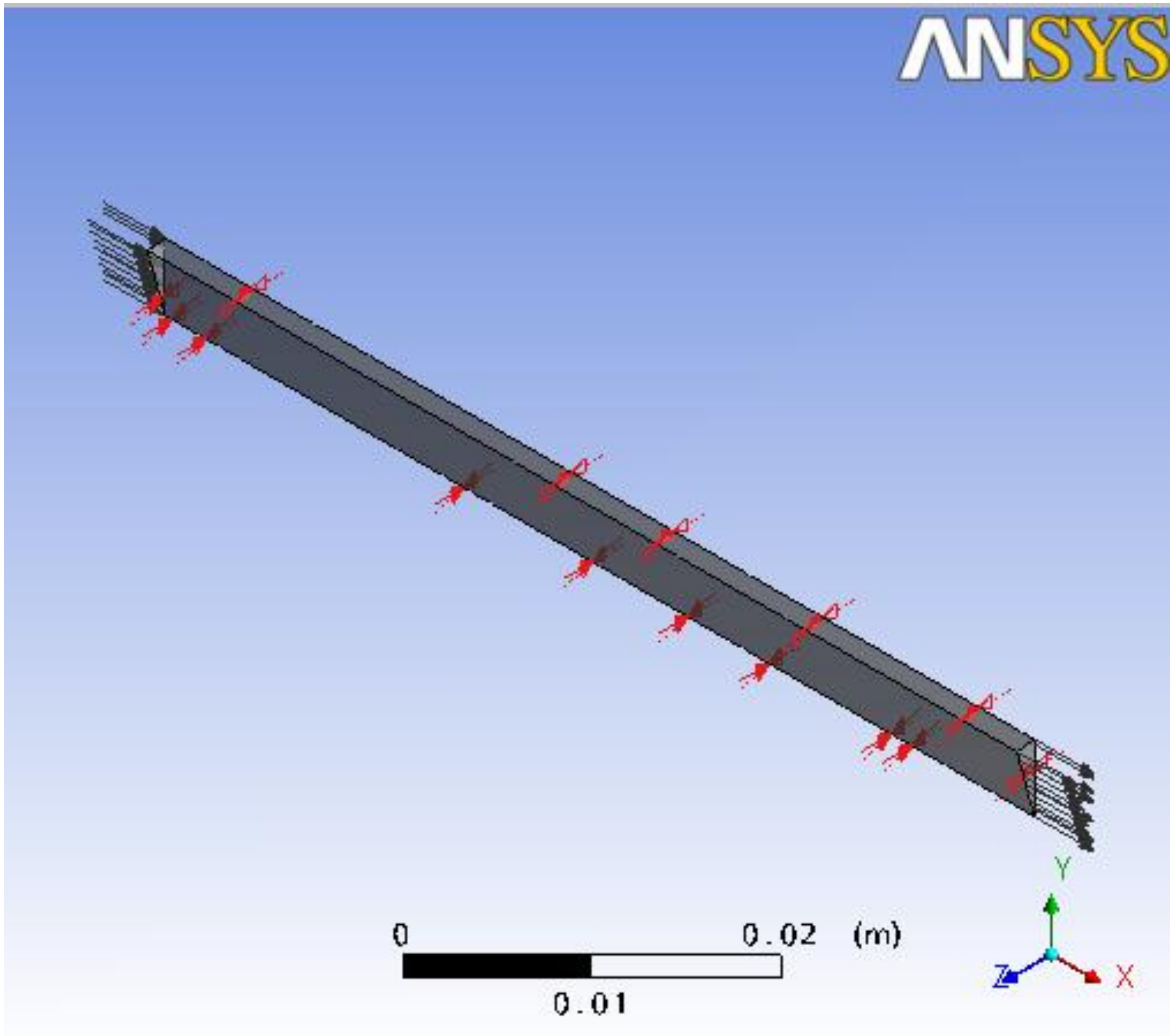
29. Create First Symmetry Boundary Condition

- ***Toolbar – Create a Boundary Condition***. Enter **Sym1** for ***Name***. Click ***OK***.
- ***Boundary: Sym1 – Basic Settings – Boundary Type***. Select **Symmetry**.
- ***Boundary: Sym1 – Basic Settings – Location***. Select **Sym1**.
- Click ***Apply***. Click ***OK***.

30. Create Second Symmetry Boundary Condition

- ***Toolbar – Create a Boundary Condition***. Enter **Sym2** for ***Name***. Click ***OK***.
- ***Boundary: Sym2 – Basic Settings – Boundary Type***. Select **Symmetry**.
- ***Boundary: Sym2 – Basic Settings – Location***. Select **Sym2**.
- Click ***Apply***. Click ***OK***.

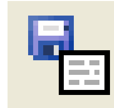
File – Save Simulation. pipe.cfx.

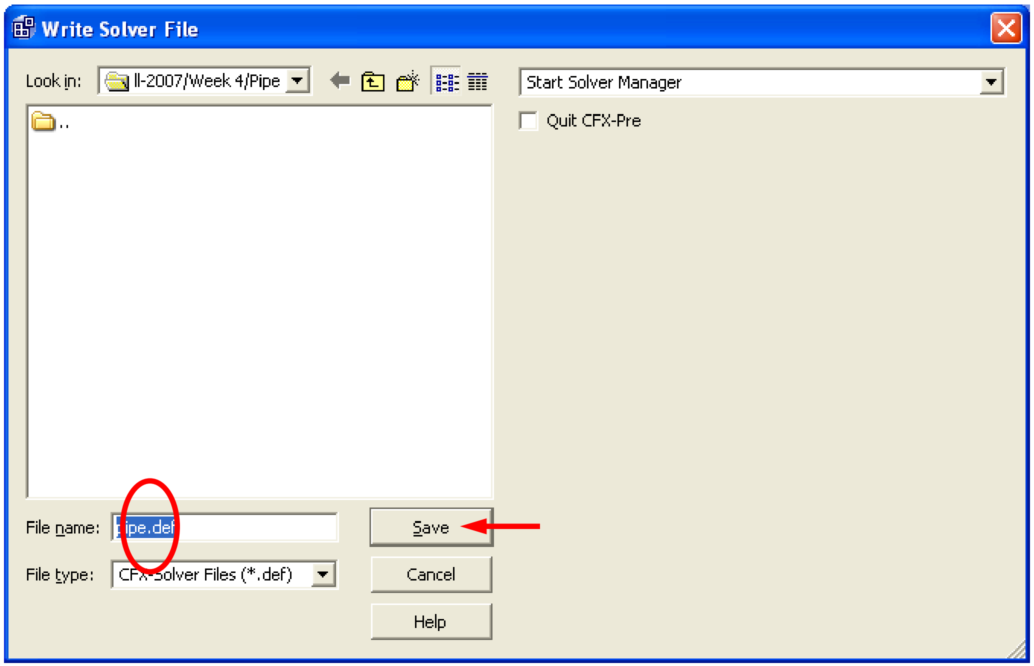

ANSYS
0
0.02
(m)
0.01
Y
Z
X

31. Set Solver Controls

- ***Toolbar – Solver Control***.
- ***Solver Control – Basic Settings – Max. Iterations***. Ensure it is set to **100**.
- ***Solver Control – Basic Settings – Fluid Timescale Control – Timescale Control***. Ensure is set to **Auto Timescale**.
- ***Solver Control – Basic Settings – Convergence Criteria – Residual Target***. Ensure is set to **1E-4**.
- Click ***Apply***. Click ***OK***.

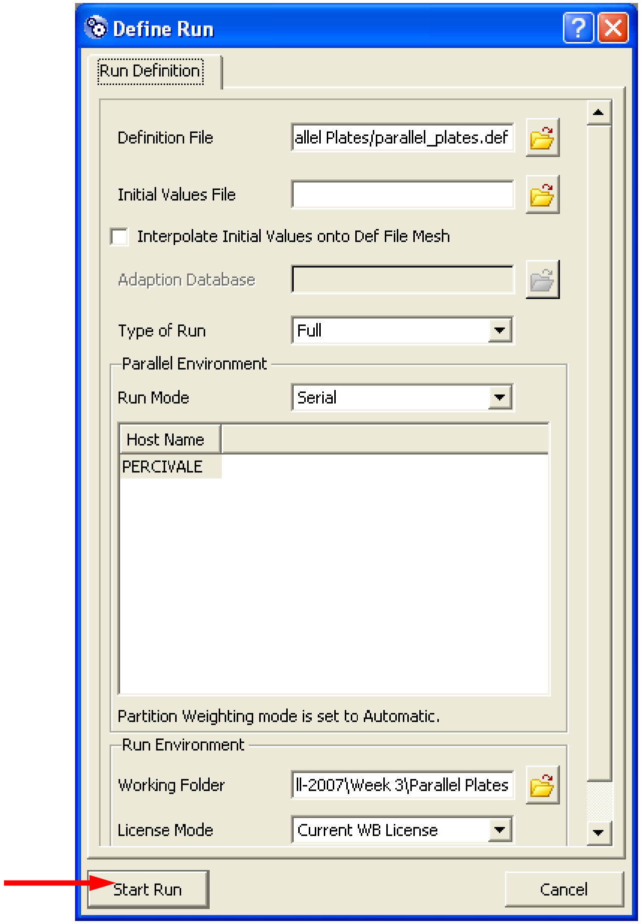
32. Write Solver File

- In ***Toolbar***, select ***Write Solver File***  icon.
- ***Write Solver File*** window appears.
- Save file as **pipe.def**.
- Click ***Save***.



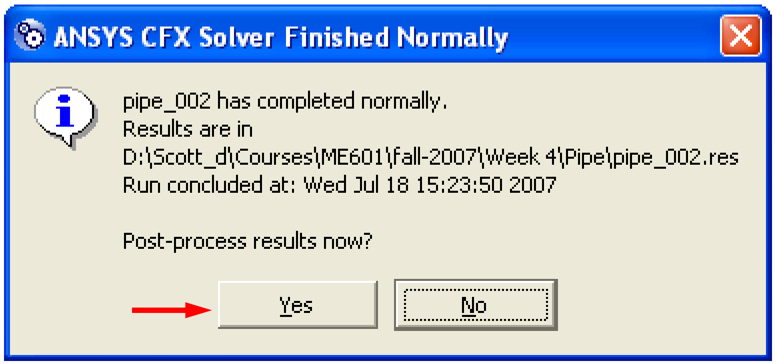
34. Start Run

- ***Define Run*** window appears.
- Select ***Start Run***.



35. Open CFX-Post

- Once solution is completed, ***ANSYS CFX Solver Finished Normally*** window will appear.
- Select ***Yes*** when asked whether you'd like to post-process results now.

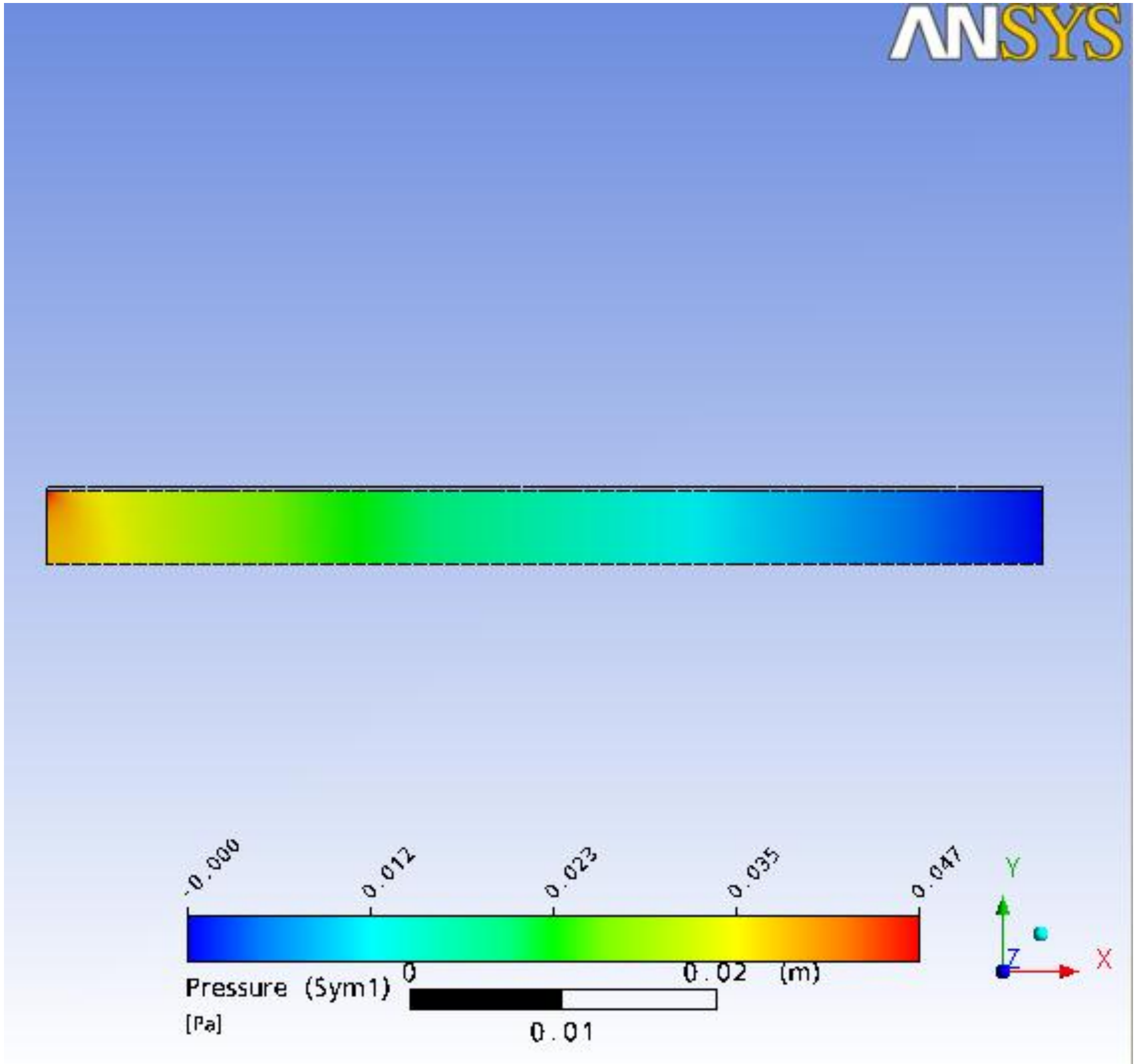


36. Create Contour Plot on Symmetry Boundary

- Contour plots of variables may be applied directly to boundaries. This is a convenient tool for quick post-processing.
- Right-click in ***Viewer***. Select ***Predefined Camera – View Towards –Z***.
- In ***Outline*** tab, double-click on ***Sym1***.
- ***Details View – Colour – Mode***. Select **Variable**.
- ***Details View – Colour – Variable***. Ensure **Pressure** is selected.
- Click Apply.

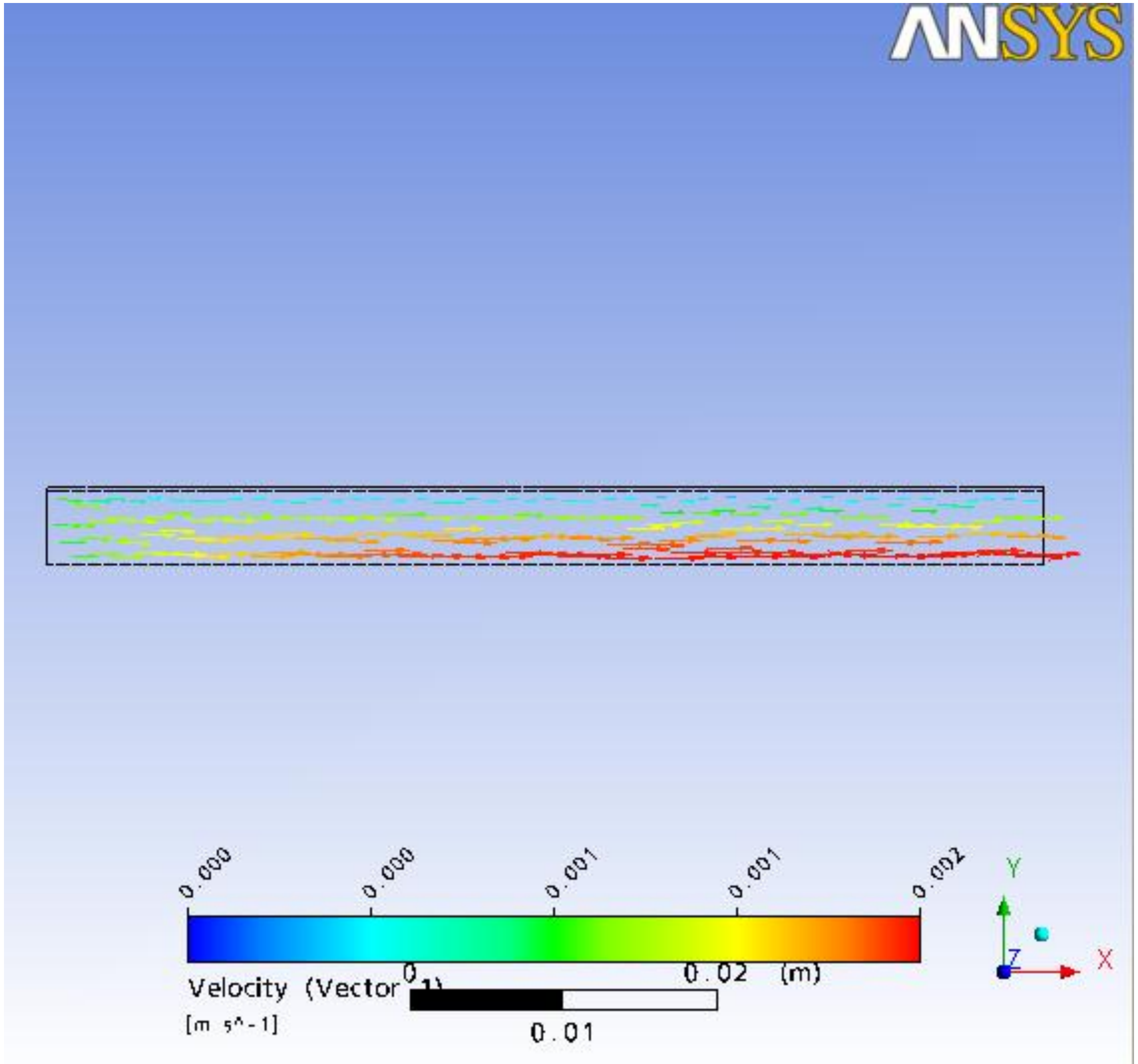
37. Alter Legend

- In ***Outline Workspace*** under ***User Locations and Plots***, double-click on ***Default Legend View 1***.
- ***Details View – Definition***. Select **Horizontal**.
- ***Details View – Definition – Location – X Justification***. Select **Center.**
- ***Details View – Definition – Location – Y Justification***. Select **Bottom**.
- ***Details View – Appearance – Text Parameters – Precision***. Select **Fixed**.
- Click ***Apply***.


ANSYS
-0.000
0.012
0.023
0.035
0.047
Pressure (Sym1)
[Pa]
0
0.02 (m)
0.01
Y
Z
X

38. Create Vector Plot

- ***Toolbar - Vector***
- ***Insert Vector*** window appears. Select ***OK***.
- ***Details View – Geometry – Definition – Locations***. Select **Sym1**.
- ***Details View – Geometry – Sampling***. Select **Equally Spaced**.
- ***Details View – Geometry - # of Points***. Enter **200**.
- Click ***Apply***.
- Deselect check box next to ***Sym1*** in ***Outline Workspace***.


ANSYS
0.000
0.000
0.001
0.001
0.002
Velocity (Vector 1)
[m s^-1]
0
0.02 (m)
0.01
Y
Z
X

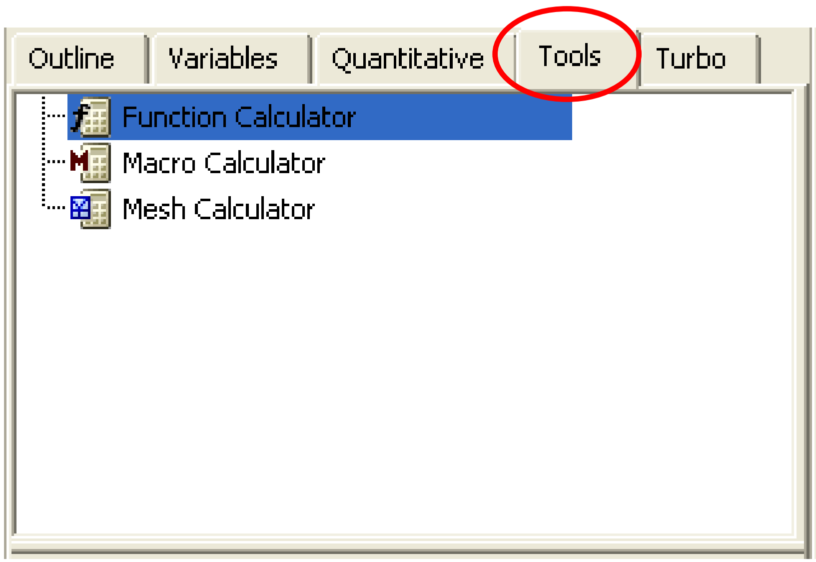
39. Create Point

- In ***Toolbar***, select ***Location*** 

icon. In drop-down menu, select ***Point***.
- ***Insert Point*** window appears. Click ***OK***.
- ***Details View – Geometry – Definition – Method***. Ensure it is set to **XYZ**.
- ***Details View – Geometry – Definition – Point***. Set to **0.05, 0, 0**.
- Click ***Apply***.
- Note that the point appears in the ***Viewer*** as a yellow cross-hairs.

CFX-Post Tools

- The Tools tab in CFX-Post is a powerful post-processing utility.
- Function Calculator allows the calculation of many useful quantities across boundaries and user-specified locations.
- Mesh Calculator shows mesh quality and statistics throughout domain.

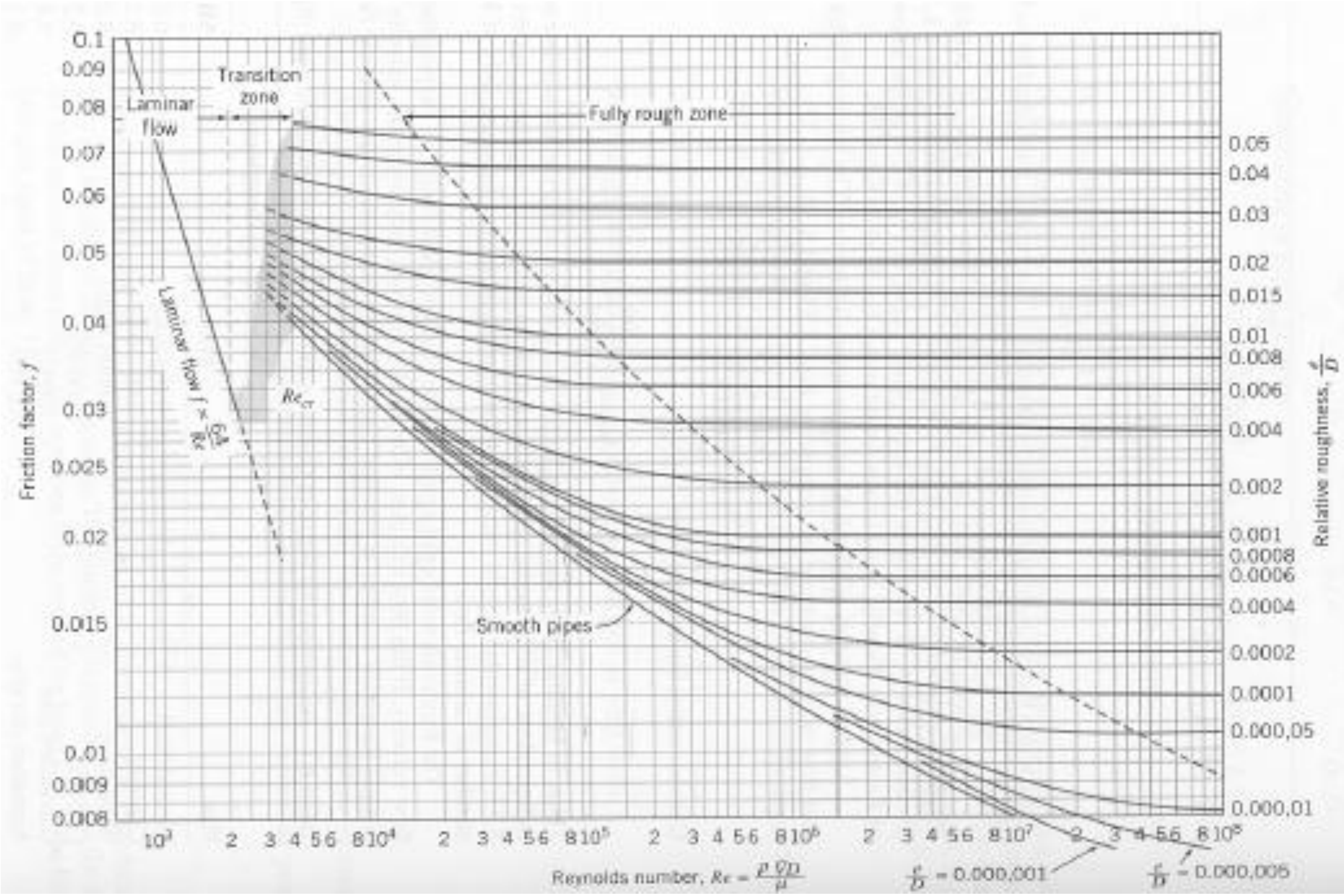


40. Use Function Calculator

- In ***Outline Workspace***, select ***Tools*** tab. Double-click on ***Function Calculator***.
- ***Details View – Function***. Select **Probe**.
- ***Details View – Location***. Ensure **Point 1** is selected.
- ***Details View – Variable***. Select **Velocity u**.
- Click ***Calculate***.
- Note that the calculated value appears in the ***Result*** field.

0.00192929 [m s^-1]

Moody Diagram



Friction Factor

- Relates friction factor (f) to Reynolds number.
- Friction factor is independent of surface roughness in laminar flow.

$$f = \frac{64}{Re}$$

- Friction factor related to pressure differential by:

$$\Delta p = \frac{1}{2} f \frac{L}{D} \rho u_m^2$$

41. Calculate Pressure Drop

- In ***Outline Workspace***, select ***Tools*** tab. Double-click on ***Function Calculator***.
- ***Details View – Function***. Select **areaAve**.
- ***Details View – Location***. Select ***LetIn***.
- ***Details View – Variable***. Select **Pressure**.
- Click ***Calculate***.
- Note that the calculated value appears in the ***Result*** field.

0.041738 [Pa]

42. Save State file

- Postprocessing method may be saved and reopened later in the form of a state file (*.cst).
- Select ***File – Save State As***, pipe.cst.

Practice

- What is the value of Velocity v at x = 0.06 m, y = 0.003 m?
- How would this value change with increased viscosity?
- Compare the Velocity u along the axis of symmetry at the outlet with the analytical value for fully-developed, internal pipe flow:

$$u_{max} = 2u_m$$

- Compare the numerical pressure rise from that calculated from the Moody diagram.