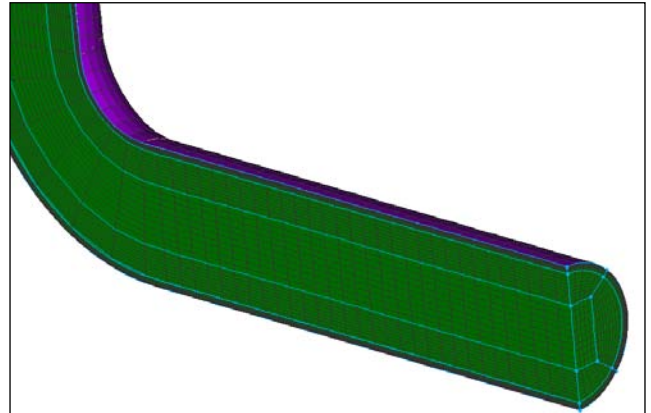


Pointwise to OpenFOAM Tutorial – Minor Losses through a Pipe Elbow

Authors: Keith Martin, and John M. Cimballa, Penn State University
Latest revision: 17 January 2011

Introduction

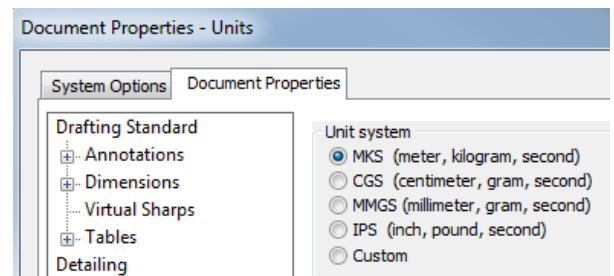
This tutorial provides instructions for meshing an internal flow in a curved pipe. A geometry and mesh similar the one shown to the right will be created. Because of symmetry, it is not necessary to model the entire pipe. Only half of the pipe will be analyzed. Splitting the geometry in this way reduces the analysis time by approximately 50%. The pipe geometry will be created in SolidWorks and exported to Pointwise for modification and grid generation. This tutorial shows how to generate a fine grid near a wall and a structured mesh everywhere else. OpenFOAM will be used as the CFD solver to calculate the pressure drop across the inlet and outlet of the pipe.








Pipe with 90° Elbow

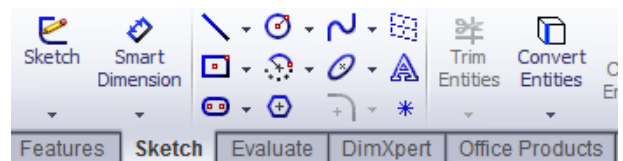
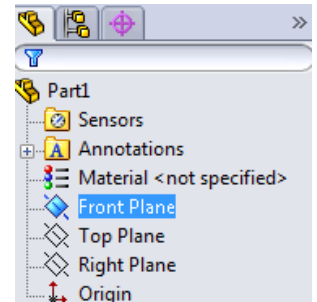
Open new SolidWorks part and set options


1. This tutorial assumes that SolidWorks is running.
2. **File-New.**
3. **Part. OK.**
4. **Tools-Options.**
5. **Document Properties** tab.
6. **Units-MKS (meter, kilogram, second)-OK.**
7. **View-Toolbars. LMB** the button in front of “CommandManager” () if it is not already toggled on.

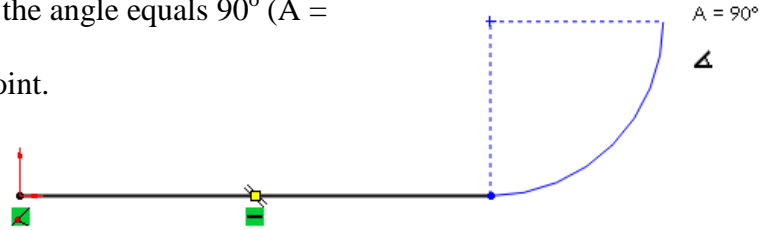


Create a sweep path


1. First, a horizontal line will be created. Then, a 90° curve will be inserted at one end of the horizontal line. Finally, a vertical line will be connected to the free end of the curve.
2. **Front Plane** from the *Design Tree*.
3. **Sketch** from the *Sketch* tab of the *CommandManager*.
4. **Line**.
5. **LMB** on the origin when a  appears.
 - In SolidWorks a  symbol means that the construction geometry is being constrained to a point.
6. Position the second endpoint by moving the cursor to the right and **LMB** when a  constraint appears. This creates a horizontal line. The length of the line is not important because exact dimensions will be assigned later.
 - The  symbol means that the line is constrained as a horizontal line.
7. **<Esc>** to unselect the line tool.
 - Pressing **<Esc>** unselects the current sketching tool but does not exit the sketch.
8. Select **Tangent Arc** from the drop down menu beside  in the *CommandManager*.







9. **LMB** on the right endpoint of the horizontal line when a  constraint appears.
10. Move the cursor up and to the right until the angle equals 90° (A = 90°) as shown.





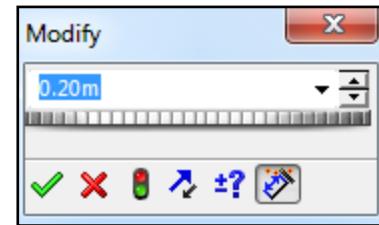
11. **LMB** again to place the second arc endpoint.
The radius of the arc is not important because exact dimensions will be assigned later.


12. <Esc> to unselect the arc tool.
13.  Line.

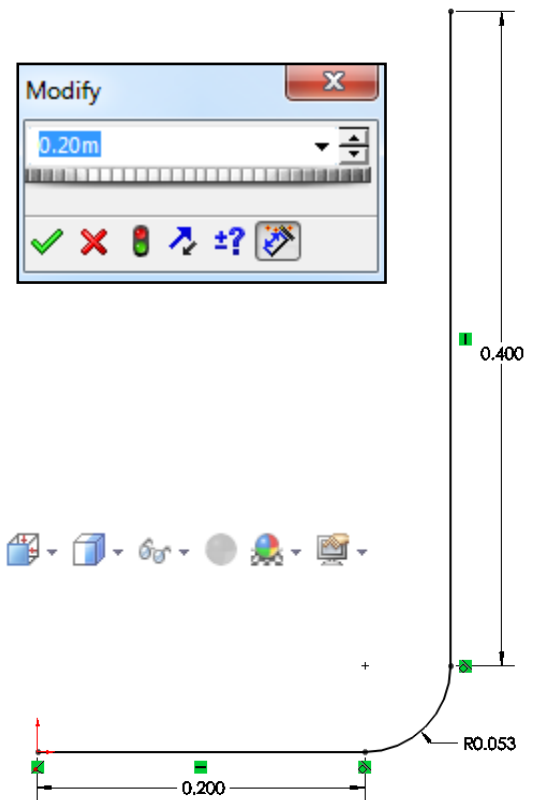
14. **LMB** the free endpoint of the arc when a  constraint appears. This places the first endpoint of a line.
15. Move the cursor up and **LMB** again when a  appears. This places the second endpoint and creates a vertical line.


 The  symbol means that the line is vertical and tangent to an arc.

16. <Esc> to unselect the line tool.
17.  **Smart Dimension** from the *CommandManager*.
18. **LMB** the horizontal line.
19. Move the cursor and **LMB** again to place the dimension.
20. Enter “0.20”m in the *Modify* window that opens.
21. <Enter> or .





22.  **Zoom to Fit** in the *Heads-Up View Toolbar*.
+ Alternatively, **View-Modify-Zoom to Fit**.
23. **LMB** the arc.
24. Move the cursor and **LMB** to place the dimension.
25. Enter “0.0525”m as the dimension, and <Enter>.
26. **LMB** the vertical line.
27. Move the cursor and **LMB** to place the dimension.
28. Enter “0.40”m as the dimension, and <Enter>.







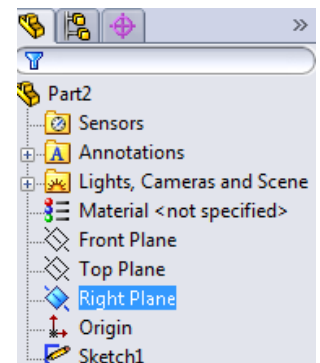
29.  **Zoom to Fit**.
30. The sketch should now look like the one shown here. Lines in a sketch change color from blue to black after they are fully constrained. The sketch is fully defined since all its lines are black.



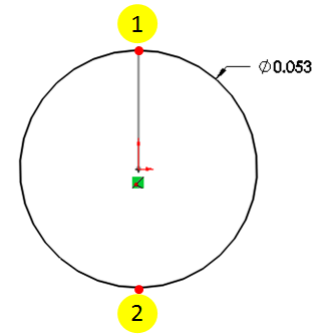
31. **LMB**  in the upper right corner of the graphics display to exit the sketch.
+  **Exit Sketch** from the *CommandManager* is an alternative to using the button in the graphics display window.
32. Notice that “Sketch1” has been added to the *Design Tree*.

Create the sweep profile

1. **R-Right Plane** in the *Design Tree* and select  **Normal To** from the menu that appears. This zooms and rotates the model to make the view orientation normal to the right plane.
2.  **Sketch** from the *CommandManager*.
3.  **Circle**.
4. **LMB** on the origin to place the center of a circle.
+ Verify that a  symbol appears beside the cursor before placing the circle’s center point.

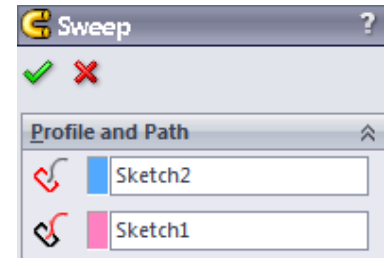


5. Move the cursor in any direction and **LMB** to freeze the circle. The radius of the circle is not important.
6. **Smart Dimension**.
 - + Verify that the *Sketch* tab is open in the *CommandManager*.
7. **LMB** on the perimeter of the circle.
8. Move the cursor and **LMB** to place the dimension.
9. Enter “0.0525”m as the diameter of the circle, and <Enter>.
10. **Line**.
11. **LMB** sequentially on Point1 and Point2 as shown to the right.
12. <Esc> to unselect the line tool.
13. **Trim Entities** in the *Sketch* tab of the *CommandManager*.
14. In the *Trim* window that opens, select “**Trim to closest**” as the trim option.
15. **LMB** on the right perimeter of the circle to delete it.
16. <Esc> to unselect the trim tool.
17. **LMB** to exit the sketch.
18. **View Orientation-Dimetric** from the *Heads-Up View Toolbar*.
19. Sketch1 will become the path of the sweep, and Sketch2 will become the profile.



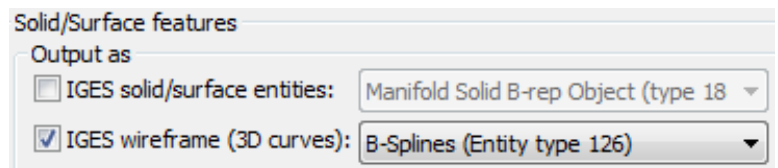
Use a swept tool to create half of the pipe

1. *Features* tab in the *CommandManager*.
2. **Swept Boss/Base**.
 - + Alternatively, **Insert-Boss/Base-Sweep...**
3. **LMB** in the field next to (profile icon), then **LMB** on Sketch2 to assign it as the profile.
 - + Select sketches either by **LMB** on them in the graphics display or by selecting them from the *Design Tree*.
4. **LMB** in the field next to (path icon), then **LMB** on Sketch1 to assign it as the path.
5. **LMB** to apply the sweep.
6. Zoom and rotate the model to better see the geometry.
 - + To zoom, scroll the **MMB**.
 - + To rotate, hold down the **MMB** while moving the cursor.



Save file and close SolidWorks

1. **File-Save**.
2. After selecting an appropriate folder in which to save the file, enter “Elbow_Pipe” for the *File name*. **Save**.
3. **File-Save As**.
4. Set *Save as type* to “IGES (*.igs)” by selecting it from the drop down menu.
 - 💡 An IGES file stores data in a universally recognized format so any CAD software can access the file.
5. **Options...**
6. Uncheck the box next to “IGES solid/surface entities”.
7. Check the box next to “IGES wireframe (3D curves)”.
 - 💡 In this tutorial only a 3D wireframe will be saved for use in Pointwise. In other cases, it is helpful to save the solid/surface entities of a model.
8. **OK**.



9. After selecting a folder in which to save the file, enter “Elbow_Pipe” for the *File Name*.
10. Save.
11. File-Exit SolidWorks.

Open Pointwise and import case

1. Open Pointwise.
2. File-Import-Database...
3. Locate and Open the file titled “Elbow_Pipe.IGS” that you saved in the previous section.
4. OK.

Choose solver

1. It is wise to choose the desired CFD solver before creating any grids. Setting the solver first reduces the possibility of problems later when exporting the mesh.
2. CAE-Select Solver...
3. OpenFOAM.
4. OK.
💡 The current solver is displayed in the lower left corner of the Pointwise window.
5. File-Save.
6. After selecting an appropriate folder in which to save the project, enter “Elbow_Pipe” for the *File name*. Save.

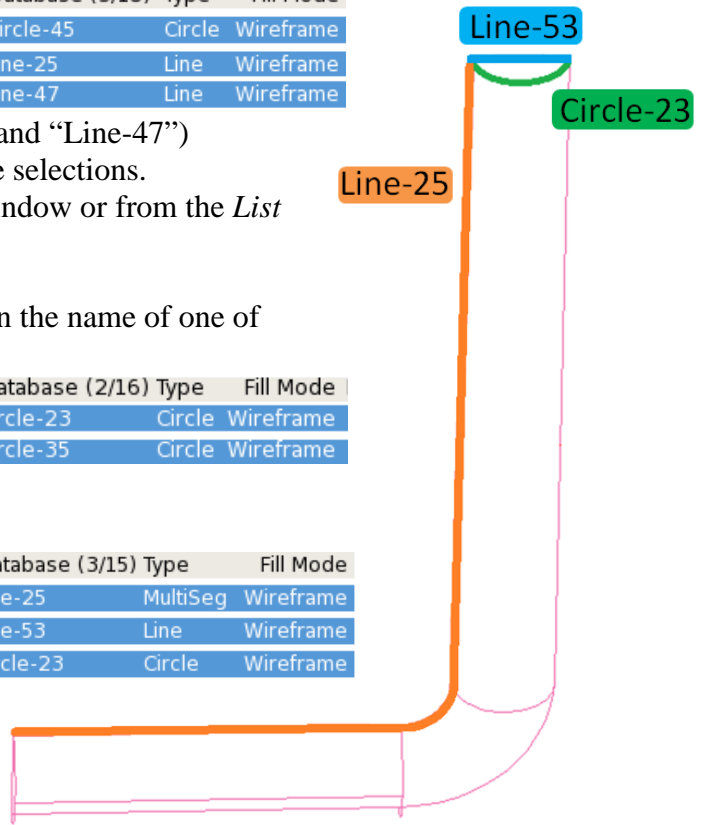
Add connectors to select wireframe geometry

1. LMB the “+” next to “Database” in the *List* panel.

Database (3/18)	Type	Fill Mode
Circle-45	Circle	Wireframe
Line-25	Line	Wireframe
Line-47	Line	Wireframe
2. Simultaneously select the three line segments that form the orange line shown to the right. (“Circle-45”, “Line-25”, and “Line-47”)
 - ➕ Hold <Ctrl> while LMB to make multiple selections.
 - 💡 Selections can be made in the graphics window or from the *List* panel.
3. Edit-Join.
4. The three lines have been combined and given the name of one of the component lines, “Line-25”.

Database (2/16)	Type	Fill Mode
Circle-23	Circle	Wireframe
Circle-35	Circle	Wireframe
5. Simultaneously select the two line segments that form the green semi-circle shown to the right. (“Circle-23” and “Circle-35”)
 - ➕ Hold <Ctrl> and RMB while moving the cursor to rotate the geometry to better select the lines in the graphics window.
6. Edit-Join.
7. The two lines have been combined and given the name of one of the component lines, “Circle-23”.

Database (3/15)	Type	Fill Mode
Line-25	MultiSeg	Wireframe
Line-53	Line	Wireframe
Circle-23	Circle	Wireframe
8. Simultaneously select “Circle-23”, “Line-25”, and “Line-53”. These three lines define the inlet and path of the pipe.
9. Connectors on Database Entities.
 - ➕ Alternatively, Create-On Database Entities. OK.
 - 💡 A “connector” is Pointwise terminology for a construction line on which a mesh is attached.
10. View-Show Database to toggle off the database wireframe.

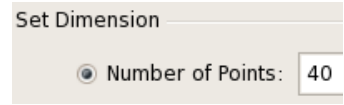


11. Rotate and zoom to get a better perspective of the connectors.

- + Zoom by scrolling the MMB. Rotate by holding <Ctrl> and RMB while moving the cursor. <F2> to center and zoom to fit the geometry in the window.

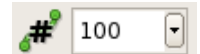
Dimension connectors

1. **LMB** the “+” next to “Connectors” in the *List* panel.
2. Select the semi-circle connector in the graphics window (“con-3”).
3. **Grid-Dimension...**
4. Type “40” for *Number of Points*. **Dimension-OK**. These points will space mesh cells around the perimeter of the pipe.



- 💡 Sometimes it is helpful to make the spacing points visible. To do this, select a connector and choose “Points On” from the drop down menu beside “Points Off” in the Attributes toolbar.

5. Select the connector that follows the path of the pipe (“con-2”).
6. Instead of following the dimensioning procedure used above, enter “100” in the dimensions box on the Grid toolbar. <Enter>.



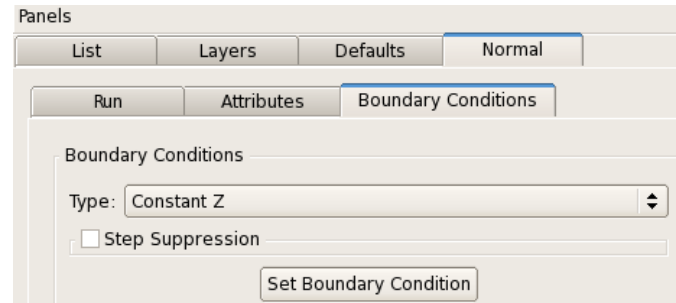
- 💡 The dimensions box is a convenient way to quickly change connector dimensions.

Extrude boundary layer mesh

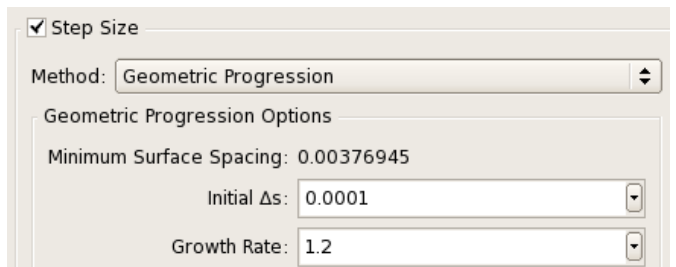
1. **Recall View -Y** from the View toolbar.
2. Zoom in on the end of the pipe parallel to the viewing plane.

- + Scroll the MMB to zoom.

3. **LMB** the “+” next to “Connectors” in the *List* panel if the list is not already expanded.
4. Select the curved connector (“con-3”).
5. **Create-Extrude-Normal...**
6. **Boundary Conditions** tab in the side panel.
7. Simultaneously select the two red endpoints of the semi-circle connector.



8. Change *Type* from “Splay” to “Constant Z”.
9. **Set Boundary Condition**.
10. **Attributes** tab in the side panel.
11. **LMB** the box next to *Step Size*.
12. Enter “0.0001” for *Initial Δs*.
13. Change the *Growth Rate* to “1.2”.
14. **LMB** the box next to *Orientation*.
15. **Set Plane**.





- 💡 The “Set Plane” button constrains the extrusion to a plane parallel to the viewing orientation plane. For this tutorial, it is important that the viewing orientation be set to **Recall View -Y** so the extrusion is perpendicular to the direction of the fluid flow.

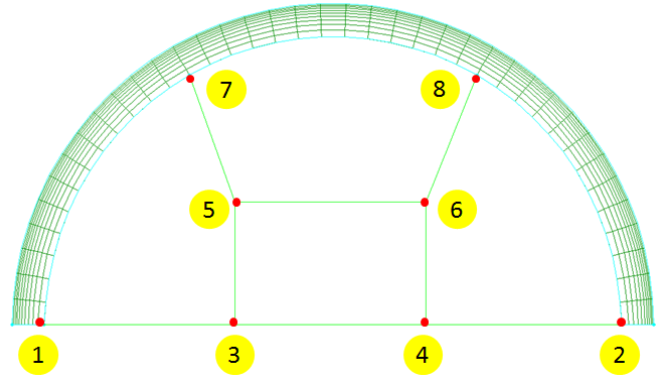
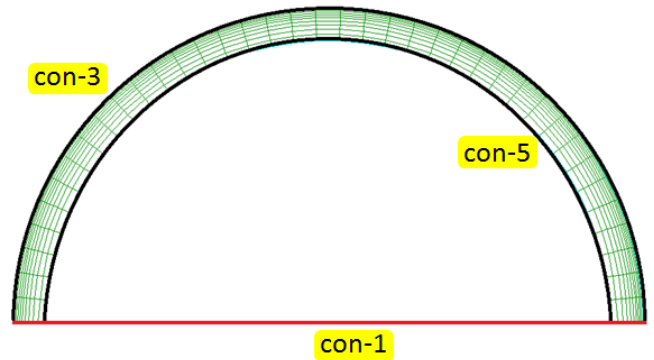
16. **Flip only** if the direction arrows are pointing **away** from the center of the pipe.
17. **Run** tab in the side panel.
18. Enter “10” for *Steps*. **Run-OK**.

Add additional connectors

1. Notice the 10 layers of cells along the wall of the pipe. The semi-circle connector has been extruded into a surface mesh. There should now be a (1) after *Domains* in the side panel.




- 💡 A “domain” is a surface grid in Pointwise.

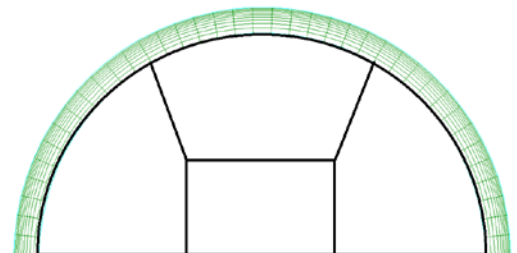
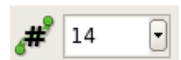
2. Select the inner curved connector (“con-5”).
3. Edit-Split...
4. Enter “33” for the *Percent of Length*. <Enter>.
5. Enter “66” for the *Percent of Length*. <Enter>.
6. LMB the button beside “Split Only at Grid Points”.
 ⚡ This moves the split points to the nearest grid points. The result is three connectors with 14 grid points on each one.
7. OK.
8. Select “con-1”.
9. <Delete>.
10.  2 Point Curves.
11. Sequentially LMB on Point 1 and Point 2 to insert a connector between them.
12. OK.
13. Select the newly formed connector (renamed “con-1”).
14. Edit-Split...
15. LMB the button beside “Split Anywhere”.
16. Enter “33” for the *Percent of Length*. <Enter>.
17. Enter “66” for the *Percent of Length*. <Enter>.
18. OK.
19.  2 Point Curves.
20. LMB on Point 3.
21. Under *Point Placement*, change **only** the z-coordinate to “0.01” as shown. <Enter>.
22. LMB on Point 4.
23. Under *Point Placement*, change **only** the z-coordinate to “0.01”. <Enter>.
24. Connect Point 5 and Point 6 by sequentially LMB on them.
25. Connect Point 5 and Point 7 by sequentially LMB on them.
26. Connect Point 6 and Point 8 by sequentially LMB on them.
27. OK.








Point Placement	
XYZ:	0.260546406851 0.4525 .01

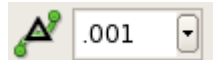
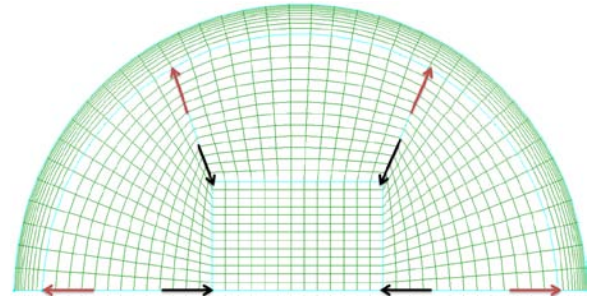
Dimension connectors and create domains

1. Simultaneously select the connectors on the end of the pipe shown here.
 + Select all connectors except “con-2”, “con-3”, “con-4”, and “con-6”.
2. Enter “14” in the dimensions box. <Enter>.
 ⚡ Connectors opposite from each other must have the same number of points to create a structured grid.
1. Verify that  Structured is the current grid type.
 + Make sure the  toggle button is depressed in the Create toolbar, or Grid-Set Type-Structured.
3.  Assemble Domains.






Use spacing to create smooth mesh transition between domains

1. Spacing will be used to match cell sizes across domain boundaries. It is important to eliminate sudden jumps in cell size for accurate solver results.
2.  **All Masks On/Off** to toggle off the selectable mesh features. The masking tool controls which features can or cannot be selected.
3. Now no features can be selected. Convince yourself of this by trying to select a connector or domain in the graphics window.
4.  **Spacing Constraints** in the *Mask* toolbar. This allows spacing constraints to be selected.
5. Simultaneously select the constraints at the outer end of the four radial connectors on the domain boundaries (shown here as red arrows pointing away from the center of the pipe).
 An arrow that shows the direction of the spacing appears when the cursor is hovered over a spacing constraint.
6. Enter “.0005” in the spacing constraints box. <Enter>. This spaces the nodes so cell sizes match across domain boundaries.
7. Simultaneously select the spacing constraints at the inner end of the four radial connectors (shown here as black arrows pointing toward the center of the pipe).
8. Enter “.001” in the spacing constraints box. <Enter>.
9.  **All Masks On/Off** two times. Once to turn off the  **Spacing Constraints** and a second time to make all feature selectable.




Run domain solver to improve mesh quality

1. **LMB** the “+” next to “Domains” in the *List* panel.
2. Simultaneously select all the domains (5 total).
 To make selecting an entire list easier, hold <Shift> while **LMB** on the first and last entry of a list.
3.  **Start Solve**.
4. After the grid seems to converge (a few seconds),  **Stop Solve**.

Domains (5/5)	Type	Points	Cells
dom-1	Structured	40x11	390
dom-2	Structured	14x14	169
dom-3	Structured	14x14	169
dom-4	Structured	14x14	169
dom-5	Structured	14x14	169

Extrude surface domains along pipe length

1. Simultaneously select all the domains (5 total).
2. **Create-Extrude-Path...**
3. **Done** to accept the automatic domain assembly.
4. Select the connector that follows the path of the pipe in the graphics window (“con-25”).
 Alternatively, **List** tab - select “con-25” - **Path** tab.
5. Steps already should be set at “99”.
6. **LMB** the box next to *Reverse Path* **only** if the yellow direction lines do not mimic the direction of the pipe (“con-25”).
7. **Run-OK**.
8. Zoom and rotate to better see the three blocks that have been created.

Define boundary conditions

1. **CAE-Set Boundary Conditions...**
2. **New**.
3. **LMB** “bc-2” and change the name to “inlet”.
4. In the same row, “**Unspecified**” and choose “patch” from the drop down menu (you have to double click to get to the drop down menu).

5. List tab from the side panel.
6. Simultaneously select “dom-1” through “dom-5”.
 ⚡ Alternately, you can select the five domains of the inlet plane manually with the mouse.
7. Set BC tab from the side panel.
8. LMB in the empty box next to the second row. A “5” should appear in the “#” column. This indicates that five domains have been defined as an inlet.
 ⚡ Notice that of the 14 domains, 5 are now specified and 9 remain unspecified.

Domains (5/20)	Type	Points	Cells
dom-1	Structured	40x11	390
dom-2	Structured	14x14	169
dom-3	Structured	14x14	169
dom-4	Structured	14x14	169
dom-5	Structured	14x14	169

9. New.
10. LMB “bc-3” and change the name to “outlet”.
11. In the same row, “Unspecified” and choose “patch” from the drop down menu.

Domains (3/20)	Type	Points	Cells
dom-12	Structured	40x11	390
dom-19	Structured	27x14	338
dom-26	Structured	14x27	338

12. List tab.
13. Select “dom-12”, “dom-19”, and “dom-26”.
14. Set BC tab.
15. LMB in the empty box next to the third row. A “3” should appear in the “#” column.
16. New.
17. LMB “bc-4” and change the name to “symmetry”.
18. In the same row, “Unspecified” and choose “symmetryPlane” from the drop down menu.

Domains (5/20)	Type	Points	Cells
dom-7	Structured	11x100	990
dom-11	Structured	11x100	990
dom-15	Structured	14x100	1,287
dom-24	Structured	14x100	1,287
dom-25	Structured	14x100	1,287



19. List tab.
20. Simultaneously select “dom-7”, “dom-11”, “dom-15”, “dom-24”, and “dom-25”.
21. Set BC tab.
22. LMB in the empty box next to the fourth row. A “5” should appear in the “#” column.
23. New.
24. LMB “bc-5” and change the name to “wall”.
25. In the same row, “Unspecified” and choose “wall” from the drop down menu.

Domains (1/20)	Type	Points	Cells
dom-6	Structured	40x100	3,861

26. List tab.
27. Select “dom-6”.
28. Set BC tab.
29. LMB in the empty box next to the fifth row. A “1” should appear in the “#” column.
30. The boundary conditions should now be set as indicated to the right.
31. Close the BC tab.

Set	#	Name	Type	ID
<input type="checkbox"/>	0	Unspecified	Unspecified	1
<input type="checkbox"/>	5	inlet	patch	2
<input type="checkbox"/>	3	outlet	patch	3
<input type="checkbox"/>	5	symmetry	symmetryPlane	4
<input type="checkbox"/>	1	wall	wall	5

Export CAE as OpenFOAM case

1. LMB the “+” next to Blocks in the side panel if it isn’t already expanded.
2. Simultaneously select all the blocks (3 total).
3. File-Export-CAE...
4. After selecting an appropriate location, create a new folder in which to export the project.
5. Name the folder “polyMesh”  polyMesh.
6.  Choose or Save or OK, depending on your operating system. Pointwise writes OpenFOAM compatible files that define the mesh and preserve the boundary conditions.
7. File-Save the Pointwise file.
8. File-Exit Pointwise.

Blocks (3/3)	Type	Points	Cells
blk-1	Structured	40x11x100	38,610
blk-2	Structured	27x14x100	33,462
blk-3	Structured	14x27x100	33,462

Create OpenFOAM “run” directory and copy tutorials

OpenFOAM is a collection of open source C++ computer programs. Individual programs can be assembled into applications that can solve computational problems and manipulate data. In this tutorial a new mesh and initial conditions are applied to an existing application.

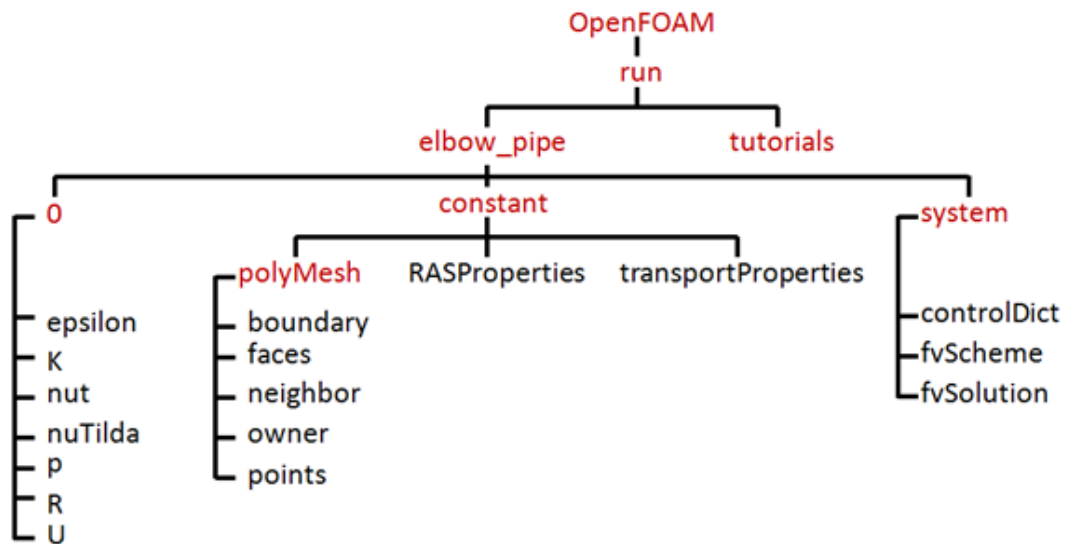
1. This tutorial assumes that a complete installation of OpenFOAM and PyFoam exist on a local computer running on a Linux operating system.
 - 💡 PyFoam is a library of python language scripts that monitors runs in OpenFOAM.
2. Open a new terminal.
 - ➕ The terminal can often be accessed from the graphical interface by Applications-Accessories-Terminal. The exact location of the terminal may differ between computers.
3. In this tutorial instructions are in **black** text and are followed by **blue** text strings that should be typed in the terminal. Press <Enter> to execute the string after it is typed. The entry “[**location string**]” should be replaced with the desired directory location (the same location is used every time [**location string**] appears). An example of a location string is `/home/username` where “username” is replaced with the local user name.
 - ➕ This tutorial can also be completed using a graphical interface. Navigate to desired directories by using the **blue** text as a guide.
 - 💡 A “directory” is the same as a “folder”.
4. Create a directory named “run” within a new directory named “OpenFOAM”.
5. `mkdir -p [location string]/OpenFOAM/run`
6. Copy a folder of OpenFOAM tutorials from the installation directory to the local run directory.
7. `cp -r $FOAM_TUTORIALS [location string]/OpenFOAM/run`
 - 💡 It is helpful to use these tutorials as templates for new cases. The tutorials are organized by their function. To quickly run a new case, modify an existing tutorial from a desired category.

Copy “pitzDaily” tutorial to the “run” directory

1. Navigate to the newly copied “tutorials” folder. Open the “incompressible” directory. Then open the “simpleFoam” directory.
 - 💡 The “incompressible” directory contains examples of standard solvers for incompressible fluids.
 - 💡 The simpleFoam solver is a steady-state solver for incompressible, turbulent flow.
2. `cd [location string]/OpenFOAM/run/tutorials/incompressible/simpleFoam`
3. Move and rename a copy of the “pitzDaily” folder to the “run” directory. The following script copies the folder and renames it “elbow_pipe”.
4. `cp -r cavity [location string]/OpenFOAM/run/elbow_pipe`

Replace existing mesh data files with files exported from Pointwise

1. Locate and open the new directory named “elbow_pipe”. Then open the “constant” folder.
2. `[location string]/OpenFOAM/run/elbow_pipe/constant`
3. Delete the existing directory called “polyMesh”. The directory with files exported from Pointwise will replace this folder.
4. `rm -r polyMesh`
5. Navigate to the location of the “polyMesh” folder that was exported from Pointwise.
6. Move the “polyMesh” directory from Pointwise to the “constant” folder.
7. `mv polyMesh [location string]/OpenFOAM/run/elbow_pipe/constant`
8. The files should now be organized as mapped in the image below. The **red** names are directories, and the **black** names are files.



Change initial pressure conditions in the “p” file

1. Open the “polyMesh” folder.
2. `cd [location string]/OpenFOAM/run/elbow_pipe/constant/polyMesh`
3. Open the “boundary” file in the background with a text editor. Executing a command the background allows a user to enter additional commands in the terminal without completing previous commands.

4. `gedit boundary &`

✚ Here, “gedit” is the text editor used to open the “boundary” file. Other text editors (such as *vi*) can be used if “gedit” isn’t available.

💡 Adding an ampersand (&) after any command causes it to execute in the background.

5. The “boundary” file lists the boundary conditions that were assigned and named in Pointwise. The names of these boundary conditions (inlet, outlet, symmetry, and wall) must be entered in the files contained in the “0” directory. Keep the “boundary” file open so the boundary names are visible.

6. Locate and open the “0” directory.

💡 The “0” directory contains files of initial conditions at the first time (t=0).

💡 After the case is solved, additional directories that correspond to each time step will be added in the case directory.

7. `cd [location string]/OpenFOAM/run/elbow_pipe/0`

8. Open the initial pressure “p” file for editing.

9. `gedit p`

10. Change the boundary names and pressure conditions to match those seen in the “boundary” file. Make changes until the text between the dividing lines matches that shown.

```

// * * * * *
dimensions      [0 2 -2 0 0 0 0];

internalField   uniform 0;

boundaryField
{
    inlet
    {
        type      zeroGradient;
    }

    outlet
    {
        type      fixedValue;
        value     uniform 0;
    }

    symmetry
    {
        type      symmetryPlane;
    }

    wall
    {
        type      zeroGradient;
    }
}
// *****
    
```

- 💡 The dimension is set by indicating the power of each base unit within the [] brackets. The base units are listed in the following order: [mass length time temperature quantity current luminous].
- 💡 The initial value of the internal pressure field is uniform and equal to zero.
- 💡 The inlet and wall boundaries are set to type **zeroGradient**. This means that the values at these boundaries must be constant for progressive elements in the direction normal to the boundary surfaces (the normal gradient equals 0). The actual values will be numerically calculated by the solver.
- 💡 The type **symmetryPlane** indicates that there is a plane of symmetry on the selected faces.
- 💡 Notice that the value for the outlet pressure is a scalar and is set equal to **0** gage pressure (open to the atmosphere).

11. Save when all the changes have been made.

12. Close the “p” file.

Change initial velocity conditions in the “U” file

1. Navigate to initial velocity “U” file in the “0” directory. Open the file with a text editor.
2. `gedit U`
3. In the open text file, change the boundary names and conditions until the text between the dividing lines matches that shown to the right.
 - 💡 Values of the internal velocity field are uniform and each component of the initial vector is set to zero.
 - 💡 Only the outlet velocity is set to type **zeroGradient**. This means that the outlet velocity must be constant for progressive elements normal to the boundary surface (the normal gradient equals 0). The actual values will be numerically calculated by the solver.
 - 💡 The type **symmetryPlane** indicates that there is a plane of symmetry on the selected faces.
 - 💡 Notice that the value for the inlet velocity is a vector and is set to flow at **2** m/s along the negative y-axis, making the flow normal to the inlet face.
 - 💡 The wall velocity vector is (0 0 0) to enforce the no-slip boundary conditions at a stationary wall.
4. Save when all the changes have been made.
5. Close the “U” file.

```
// * * * * *
dimensions      [0 1 -1 0 0 0 0];
internalField   uniform (0 0 0);
boundaryField
{
    inlet
    {
        type          fixedValue;
        value         uniform (0 -2 0)
    }
    outlet
    {
        type          zeroGradient;
    }
    symmetry
    {
        type          symmetryPlane;
    }
    wall
    {
        type          fixedValue;
        value         uniform (0 0 0);
    }
}
// * * * * *
```

Calculate initial turbulent kinetic energy (k) and turbulence dissipation (ε)

1. Calculate a Reynolds number (Re) for this case.
2. Calculate turbulence intensity (I).
3. Calculate turbulence length scale (l).
4. Calculate turbulent kinetic energy (k).
5. Calculate turbulence dissipation (ϵ).

Equations

$$R_e = \frac{L|U|}{\nu} \quad I = .16(R_e)^{-\frac{1}{8}} \quad l = 0.07L \quad k = \frac{3}{2}(U_{avg}I)^2 \quad \varepsilon = \frac{C_u^{.75}k^{1.5}}{l}$$

6. The Reynolds number (R_e) is calculated to be 105,000. This number is much larger than the critical Reynolds number (2,300) and indicates that the flow is turbulent.
7. The calculations have already been completed, so use the k and ε values from the Case-specific Values chart and move to the next section.

Variable Definitions
Case Properties
L = hydraulic diameter
ν = kinematic viscosity
C_u = turbulence model constant
U = velocity normal to inlet
Intermediate Properties
R_e = Reynolds number
I = turbulence intensity
l = turbulence length scale
Initial Conditions
k = turbulent kinetic energy
ε = turbulent dissipation rate

Case-specific Values
Case Properties
L = .0525 m
ν = $1e^{-6}$ m ² /s
C_u = 0.09
U = 2 m/s
Intermediate Calculation Results
R_e = 105000
I = 0.0377
l = 0.00368 m
Calculated Initial Conditions
k = 0.00853 m ² /s ²
ε = 0.0352 m ² /s ³

Set initial turbulent kinetic energy in the “k” file

1. Navigate to “k” file in the “0” directory if not already there.
2. `cd [location string]/OpenFOAM/run/elbow_pipe/0`
3. Open the file with a text editor.
4. `gedit k`
5. In the open text file, change the value of k to “0.00853” m²/s² for the internal field and inlet and wall faces. Make changes until the text between the dividing lines matches that shown to the right.
6. Save when all the changes have been made.
7. Close the “k” file.

```
// * * * * *
dimensions      [0 2 -2 0 0 0];
internalField   uniform 0.00853;
boundaryField
{
    inlet
    {
        type      fixedValue;
        value     uniform 0.00853;
    }
    outlet
    {
        type      zeroGradient;
    }
    symmetry
    {
        type      symmetryPlane;
    }
    wall
    {
        type      kqRWallFunction;
        value     uniform 0.00853;
    }
}
// * * * * *
```

Set initial turbulence dissipation rate in the “epsilon” file

1. Navigate to “epsilon” file in the “0” directory if not already there.
2. `cd [location string]/OpenFOAM/run/elbow_pipe/0`
3. Open the file with a text editor.
4. `gedit epsilon`
5. In the open text file, change the value of epsilon to “0.0352” m^2/s^3 for the internal field and inlet and wall faces. Make changes until the text between the dividing lines matches that shown to the right.
6. Save when all the changes have been made.
7. Close the “epsilon” file.

```
// * * * * *
dimensions      [0 2 -3 0 0 0];
internalField   uniform .0352;

boundaryField
{
    inlet
    {
        type      fixedValue;
        value     uniform .0352;
    }
    outlet
    {
        type      zeroGradient;
    }
    symmetry
    {
        type      symmetryPlane;
    }
    wall
    {
        type      epsilonWallFunction;
        value     uniform .0352;
    }
}

// *****
```

Set initial conditions in the “nut” file

1. Navigate to “nut” file in the “0” directory. Open the file with a text editor.
2. `gedit nut`
3. Make changes until the text between the dividing lines matches that shown to the right.
4. Save when all the changes have been made.
5. Close the “nut” file.

```
// * * * * *
dimensions      [0 2 -1 0 0 0];
internalField   uniform 0;

boundaryField
{
    inlet
    {
        type      calculated;
        value     uniform 0;
    }
    outlet
    {
        type      calculated;
        value     uniform 0;
    }
    symmetry
    {
        type      symmetryPlane;
    }
    wall
    {
        type      nutWallFunction;
        value     uniform 0;
    }
}

// *****
```


Set initial conditions in the “nuTilda” file

1. Navigate to “nuTilda” file in the “0” directory.
Open the file with a text editor.
2. [gedit nuTilda](#)
3. Make changes until the text between the dividing lines matches that shown to the right.
4. Save when all the changes have been made.
5. Close the “nuTilda” file.


```
// * * * * *
dimensions      [0 2 -1 0 0 0];
internalField   uniform 0;
boundaryField
{
    inlet
    {
        type          fixedValue;
        value          uniform 0;
    }

    outlet
    {
        type          zeroGradient;
    }

    symmetry
    {
        type          symmetryPlane;
    }

    wall
    {
        type          zeroGradient;
    }
}
// * * * * *
```

Set initial conditions in the “R” file

1. Navigate to “R” file in the “0” directory.
 R stands for Reynolds stress.
2. Open the file with a text editor.
3. [gedit R](#)
4. Make changes until the text between the dividing lines matches that shown to the right.
5. Save when all the changes have been made.
6. Close the “R” file.

```
// * * * * *
dimensions      [0 2 -2 0 0 0];
internalField   uniform (0 0 0 0 0 0);
boundaryField
{
    inlet
    {
        type          fixedValue;
        value          uniform (0 0 0 0 0 0);
    }

    outlet
    {
        type          zeroGradient;
    }

    symmetry
    {
        type          symmetryPlane;
    }

    wall
    {
        type          kqRWallFunction;
    }
}
// * * * * *
```

Set kinematic viscosity (ν)

1. Navigate to the “constant” directory.
2. `cd [location string]/OpenFOAM/run/elbow_pipe/constant`
3. Open “transportProperties” with a text editor.
4. `gedit transportProperties`
5. Change the value of ν from “1e-05” to “1e-6” as indicated below.
 - 💡 The kinematic viscosity of water at 20° C is approximately $1.0 \times 10^{-6} \text{ m}^2/\text{s}$.
 - 💡 The dimension is set by indicating the power of each base unit within the [] brackets. The base units are listed in the following order: [mass length time temperature quantity current luminous].

```
// * * * * *
nu          nu [ 0 2 -1 0 0 0 0 ] 1e-6;
// ***** //
```

6. Save and then close the “transportProperties” file.

Set the turbulence model

1. Navigate to the “constant” directory if not already there.
2. Open “RASProperties” with a text editor.
 - 💡 RAS is an acronym for the Reynolds-Averaged Stress model.
3. Notice that the RASModel is set to kEpsilon. The k-epsilon turbulence model is a common 2-equation solver.
4. Verify that the turbulence calculation is switched “on”.
5. PrintCoeffs should also be set to “on”. This means values will be displayed in the terminal.
6. The settings here are sufficient for this case, so close the file without making any changes.

Change run-time settings

SimpleFoam is a time dependent solver. A user can define a period over which the solver should iterate a solution. Multiple iterations are performed at each user defined time steps. The solution to each time step is saved in a separate directory within the case directory. This allows users to examine time dependent characteristics such as flow development.

7. Navigate to the “system” directory.
8. `cd [location string]/OpenFOAM/run/elbow_pipe/system`
9. Open “controlDict” with a text editor.
10. `gedit controlDict`
11. Change *endTime* to “0.25”.
12. Change *deltaT* to “0.0005”.
13. Change *writeInterval* to “100”.
 - 💡 The iterations will march at a time step of 0.0005 seconds until 0.25 second is reached (a total of 500 time steps). To reduce memory usage, the write interval is changed so the data of every 100th time step will be written into memory (a total of 5 directories).
14. Save.
15. Close the “controlDict” file.

Set solution parameters in the “fvSolution” file

1. Navigate to the “system” directory.
2. Open “fvSolution” with a text editor.
3. `gedit fvSolution`
4. Change the tolerance of all the variables to **1e-08**.
 - 💡 The tolerance setting is the cut-off residual value at which the solver stops iterating a solution.
 - 💡 Residuals are a measure of average error in a solution – the smaller the residual, the more converged the solution.
5. Set all the relative tolerances (“relTol”) to **0**. This makes the solver tolerance the governing stop condition.
 - 💡 The “relTol” (relative tolerance) value controls how much a residual must be reduced before the next iteration begins.
6. The rest of the settings are sufficient for this case.
 - 💡 The “nNonOrthogonalCorrectors” value controls how many iterative loops are performed on the pressure equation. The “0” indicates that the equation is solved once (no loop). This value can be increased to obtain a more accurate solution at non-orthogonal areas of a mesh.
 - 💡 Relaxation factors affect the stability of a calculation by determining how much a variable can change between iterations. The relaxation factor must be between 0 and 1. A smaller value makes a calculation more stable, but slower.
7. Save.
8. Close the “fvSolution” file.

Run simpleFoam solver

1. Navigate to the “elbow_pipe” directory.
2. `cd [location string]/OpenFOAM/run/elbow_pipe`
3. Run the solver in the background and create a log file.
4. `simpleFoam > log &`
 - 💡 A “log” file is created within the case directory as the solver iterates a solution. This file documents the progress of the solver.
5. Monitor real-time plots of residuals as the simpleFoam iterates a solution.
6. `pyFoamPlotWatcher.py log`
 - ✚ If PyFoam is not installed, skip this step.
7. Wait until the solver is finished iterating a solution before proceeding to the next step . This takes some time, so be patient.
8. After the solver is finished, close the residual plots and <Ctrl>+C to break out of the monitoring.

Create vector component files

1. Velocity vectors are stored in “U” files for each time step. These vectors will be split into their scalar x, y, and z components and stored in U_x , U_y , and U_z files in each time-step directory.
2. `foamCalc components U`
 - 💡 The magnitude of each vector can also be calculated and recorded in each time-step directory by executing the following command: `foamCalc mag U`

Calculate the pressure drop


1. The average inlet and outlet pressures will be calculated. The difference between these pressures is the pressure drop in the pipe.

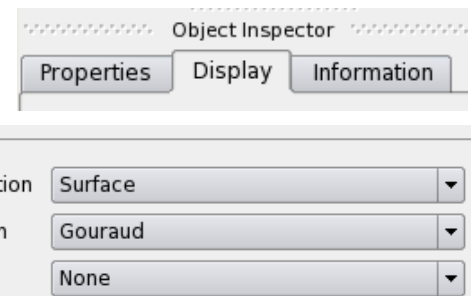
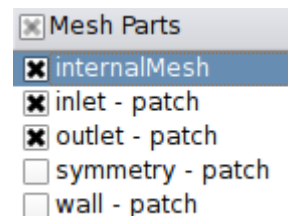
2. Calculate the average pressure at the inlet for each time-step and store the results in a file named “inletAvP”.
3. `patchAverage p inlet > inletAvP`
4. Calculate the average pressure at the outlet for each time-step and store the results in a file named “outletAvP”.
5. `patchAverage p outlet > outletAvP`
6. View the average inlet pressures in the terminal window.
7. `cat inletAvP`
8. OpenFOAM normalizes pressure values by dividing the true pressure by the density of the working fluid. This normalization results in a number with units of m^2/s^2 . To calculate the true gage pressure, multiply the average pressure at the last time-step by the density of the working fluid. Use 998.2 kg/m^3 as the density of the working fluid for this case.
 - 💡 The density of the working fluid was indirectly specified as 998.2 kg/m^3 (density of water at 20°C) when the kinematic viscosity was assigned.
9. View the average outlet pressures.
10. `cat outletAvP`
11. The outlet gage pressure should be zero because the outlet boundary condition was specified as zero gage pressure.

Launch ParaView

1. If not already there, navigate to the “elbow_pipe” directory.
2. `cd [location string]/OpenFOAM/run/elbow_pipe`
3. Launch ParaView.
4. `paraFoam`














View results in ParaView

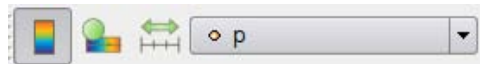
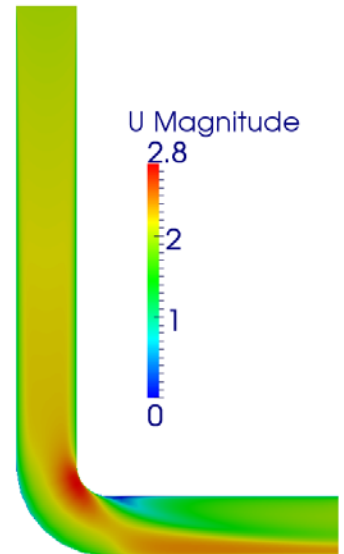
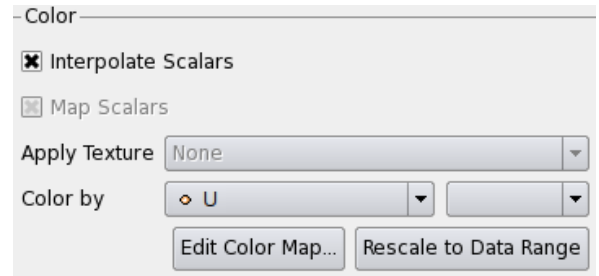
1. In ParaView, make sure that the *Properties* tab is selected in the *Object Inspector* panel.
2. **LMB** “Show Patch Names”.
3. **LMB** the box next to “inlet-patch” and “outlet-patch”.
4.  **Apply**.
5. **LMB** on the *Display* tab in the *Object Inspector* panel.
6. Scroll down to the *Style* section. Change *Representation* from “Outline” to “Surface” by using the dropdown menu.
7. Rotate and zoom in the graphical display to better see the solid model.
 - ➕ Rotate by holding **LMB** while moving the cursor.
 - ➕ Zoom by scrolling the **MMB**.
8. **View-Toolbars**.
9. Activate all the toolbars by **LMB** to place a before any unmarked titles.















View velocity and pressure distribution

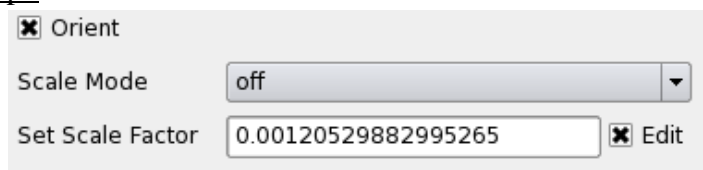
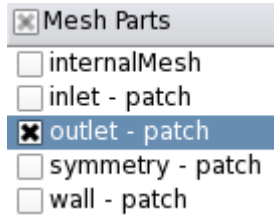
1. Select the *Display* tab.
2. Scroll down to the *Color* section.



3. Change *Color* by “ U” from the dropdown menu.
 -  Nothing has changed in the graphical display because the current time-step is zero.
4.  Last Frame in the *VCR Controls* toolbar to move to the last time step.
 -  Alternatively, enter a specific time-step in the *Current Time Controls* toolbar.
 -   Next Frame to move through individual time-steps.
5.  Rescale to Data Range in the *Active Variable Controls* toolbar.
6.  Toggle Color Legend Visibility.
 -  Alternatively, View- Show Color Legend.
7.  Set view direction to +Z to view the center plane of the pipe (symmetry plane).
8. The resulting velocity distribution should look similar to the one pictured here. A maximum velocity should be located at the bend where separation is visible.
9. To view the pressure distribution, change the active variable to  P and  Toggle Color Legend Visibility in the *Active Variable Controls* toolbar.
10. Maximum pressure should be located at the outside wall of the 90° bend in the pipe.



View velocity field at the outlet

1. Select the *Properties* tab.
2. Unselect “Show Patch Names”.
3. Unselect “internalMesh” and “inlet-patch”.
4. LMB the box next to “U_x”, “U_y”, and “U_z” under “Volume Fields” to select these variables.
5.  Apply.
6. Filters- Alphabetical- Cell Centers.
7.  Apply.
8. With “ CellCenters1” still selected in the *Pipeline Browser*,  Glyph.
 -  Alternatively, Filters- Alphabetical- Glyph.
9. Verify that the *Glyph Type* is set to “Arrow”.
10. Scroll down to the *Orient* section.
11. Change *Scale Mode* from “vector” to “off”.
12. LMB the box next to “Edit”.
13.  Apply.
14.  Toggle Color Legend Visibility and set the variable to  Uz in the *Active Variable Controls* toolbar.



15.  Set view direction to -X.
16.  Rescale to Data Range in the *Active Variable Controls* toolbar.
17. Rotate and zoom in the graphical display to better see the velocity vector field. It should look similar to the vector field to the right.
 - + Rotate by holding LMB while moving the cursor.
 - + Zoom by scrolling the MMB.

Save the ParaView case

1. File-Save State.
2. After selecting the case directory as the location in which to save the ParaView state, enter “Velocity_field” as the *File name*.
3. OK.
4. File-Exit.

