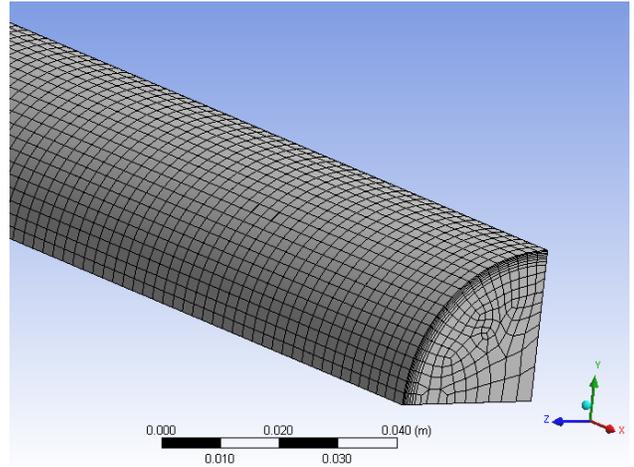


ANSYS Workbench Tutorial – Minor Losses

Authors: Scott Richards, Keith Martin, and John M. Cimbala, Penn State University
Latest revision: 11 June 2010

Introduction

This tutorial provides instructions for meshing two internal flows and for analyzing both of them in FLUENT. The first flow is in a straight pipe, and the second flow is in a pipe of the same length and diameter, but with a 90° elbow. The pressure drop is calculated for both cases. The difference in pressure drop between the two cases is due solely to the elbow since the pipes are the same length; this is the so-called “minor loss”. In Part 1 of this tutorial, a geometry and mesh like the one shown to the right will be created. This tutorial shows how to use “inflation” to generate a fine grid near a wall, how to generate an unstructured quad mesh everywhere else, and how to sweep the mesh along the pipe.



Part 1: Straight Pipe

Set Workbench options for a new FLUENT project

1. This tutorial assumes that ANSYS Workbench is running but no projects are open. If it is not running, open it before proceeding to the next step.
2. Under *View*, make sure that “Toolbox”, “Toolbox Customization”, and “Project Schematic” each have a check mark next to them.

✚ Check marks can be inserted by placing the cursor over the menu item and **LMB** (click the left mouse button).

💡 Placing a check mark next to a menu item opens its window. If you close the window, the corresponding check mark will be removed from the *View* menu.

3. In *Toolbox Customization*, verify that “Fluid Flow (FLUENT)” has a check mark next to it.

✚ If Column A is not shown in *Toolbox Customization*, **1** located to the left of “Name”.

✚ If the box to the left of “Fluid Flow (FLUENT)” is unchecked, **LMB** in the box to make it checked.

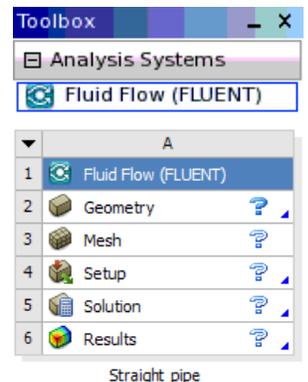
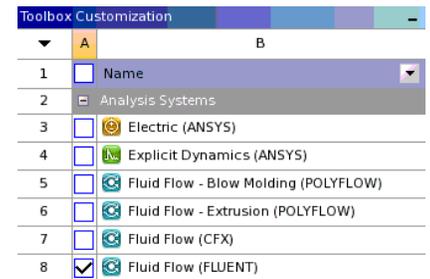
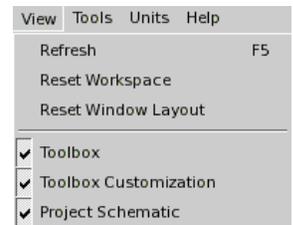
✚ Depending on your system settings, it may be necessary to make Column B wider in order to see the entirety of each entry.

💡 To de-clutter your Workbench workspace, close or minimize the *Toolbox Customization*; it is not needed after this step.

4. In the *Toolbox*, not *Toolbox Customization*, **Fluid Flow (FLUENT)** and hold the **LMB** to drag it into the box that will appear in *Project Schematic*. In *Project Schematic*, rename “Fluid Flow (FLUENT)” to “Straight_pipe” as the title for the first *Analysis System*, then <Enter>.

✚ If there are no Analysis Systems visible in the *Toolbox*, try **±** (clicking “+”) to the left of *Analysis Systems* under the *Toolbox*.

✚ On some platforms you may have to double click on the title. On other platforms you may have to **RMB** on “Fluid Flow (FLUENT)” and **Rename**.



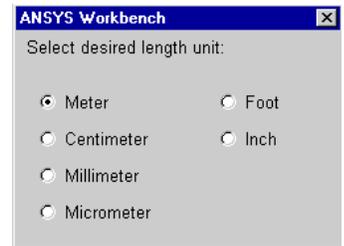
Create the straight pipe geometry by extrusion

1. **Geometry** [bold underline means double click] in the *Fluid Flow (FLUENT)* template to open *ANSYS DesignModeler*.

2. At startup DM prompts you to select the desired length unit, which is meter by default. If not, **LMB** on the button next to “Meter”. **OK**.

💡 It is wise to always use meters as the unit to avoid conversion errors.

- ✚ If this prompt does not appear, it means that your version of DM was set to always use a certain unit of length. If this is the case, try **Tools-Options-Units**, and set “Display Units Pop-up Window” to **Yes**. You may need to close and re-start DM to get the units window to pop up.



3. In *Tree Outline*, **YZPlane**. Then in the *Display Toolbar*, **Look at Face/Plane/Sketch** .

💡 Hover the cursor over a toolbar button to display the name of the button.

4. **Sketching** which is located near the bottom of *Tree Outline*. This will bring up the *Draw* tab of *Sketching Toolboxes*.

5. Zoom in until the scale at the bottom of the *Graphics* window goes from 0 to about 0.05 m as shown to the right.



💡 Zoom and pan via tools in the menu bar , or **LMB** anywhere in the *Graphics* window and scroll the MMB.

💡 Zooming in now is necessary for easier construction of the pipe.

6. In *Draw*, **Circle** and hover over the origin. When a “P” is displayed, **LMB**. Move the cursor an arbitrary distance in any direction, then **LMB** to create a circle.

💡 The size of the circle isn’t critical because you will supply a dimension for its radius later.

💡 The cursor coordinates are displayed at the bottom right corner of *Graphics*.

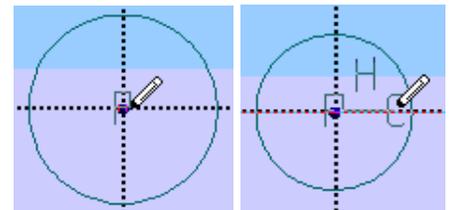
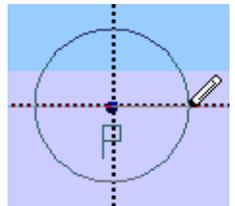
- ✚ If you make a mistake, <Esc> to exit the selected drawing tool or **Undo** .

7. In *Draw*, **Line** and hover over the origin. When a “P” is displayed, **LMB**. Move the cursor along the positive horizontal axis; when the cursor is $\frac{3}{4}$ of the way to the edge of the circle and both a “C” and an “H” are displayed, **LMB** to place the other end of the line.

💡 “P” means that one end of the line will be constrained to the point. “H” means a horizontal line. “C” means the axis.

8. Repeat the process from the previous step to create a line along the vertical axis that goes from the origin to about $\frac{3}{4}$ of the way to the top of the circle.

✚ The constraint that is displayed for this line will be a “V” instead of an “H”.



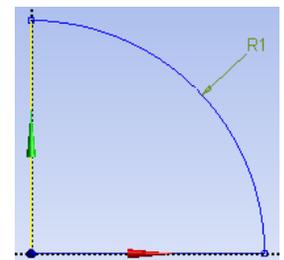
Trim, dimension, and constrain the profile

1. In *Sketching Toolboxes*, **Modify-Trim**. **LMB** on the edges of the 3 quarter circles that are NOT in the first quadrant of the YZ plane; this will delete them.

💡 Because of symmetry, it is not necessary to model the entire pipe. In this tutorial, a quarter of the pipe will be analyzed; this will reduce the analysis time by 75%.

💡 In the bottom left corner of the *DesignModeler* window there is a prompt that gives you a brief description of how to use the tool that is selected.

2. In *Sketching Toolboxes*, **Dimensions**. **LMB** on the edge of the remaining quarter circle, and **LMB** again somewhere farther out to place the radial dimension at the cursor’s location. Its default name is “R1”.



- In *Details View*, dimension *R1* corresponds to the radial dimension for the extrusion profile. Set the value to “0.02625” m, which is the radius of a typical two-inch pipe.

✚ If the quarter circle’s radius was not the first dimension used in the sketch, DM will assign the radial dimension label with a number other than “1”.

- In *Sketching Toolboxes*, Constraints-Coincident.
- LMB on the edge of the quarter circle to select it, then LMB on the right endpoint of the line (on the *point*, not the *line* itself) on the Y-axis. This should make the two selections coincident.

✚ If it does not work, you may have to LMB on the bottom right endpoint of the quarter circle one more time.

✚ *Note*: Since you are sketching in the YZ plane, the Y-axis is the horizontal axis.

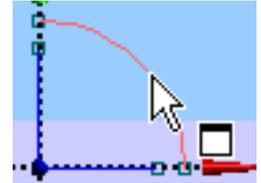
✚ If you are having trouble selecting the endpoint of a line, zoom in by drawing a box around the area of interest while holding down the RMB.

💡 The lines along the Y and Z axes were drawn to be $\frac{3}{4}$ of their final length to facilitate constraining their endpoints.

- Repeat the process from the previous step to constrain to top endpoint of the line on the Z-axis to the quarter circle.

💡 This is the last constraint/dimension needed to fully define the sketch.

Details View	
Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 1	
R1	.02625



Extrude the profile

- From the *Features Toolbar*, Extrude . Then in *Details View*, make sure that “Sketch1” is selected as the *Base Object*.

✚ If DM didn’t select a sketch for the *Base Object*, LMB on “Not Selected”. Then LMB on the sketch in the *Tree Outline* that will be used for the extrusion profile, and Apply.

✚ If the extrusion profile sketch wasn’t the first sketch to be created, DM will assign the profile sketch a number other than “1”.

- In *Details View*, set the value of *FD1*, *Depth* to “0.68247” m. <Enter>.

💡 The length of the extrusion is chosen to be the same as the length of the sweep path for the geometry that will be created in the second part of this tutorial (0.2 m upstream + 0.08247 m elbow + 0.4 m downstream).

- Either  Generate or <F5> to generate the extrusion. Click on the floating green sphere in the lower right corner of *Graphics* to rotate the geometry to an isometric view. You should see a long quarter-pipe similar to the one shown here.

💡 You may have to zoom in to see the geometry. The  Zoom to Fit button is useful here.

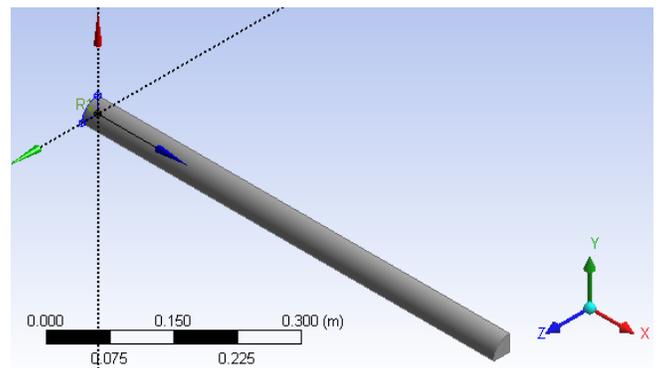
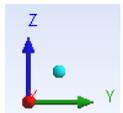
💡 The  Rotate button is useful to orient the geometry as you wish.

💡 The <F5> key updates the model after making any number changes.

- In the upper left corner of *DM*, File-Save Project. After selecting an appropriate folder in which to save the project, enter “MinorLoss” for the *File Name*, Save. Back in the *DM* window, close *DM*: File-Close DesignModeler.

💡 If all goes well, “Geometry” in *Project Schematic* should now have a check mark next to it.

Details View	
Details of Extrude1	
Extrude	Extrude1
Base Object	Sketch1
Operation	Add Material
Direction Vector	None (Normal)
Direction	Normal
Extent Type	Fixed
FD1, Depth (>0)	.68247



Set meshing options and label the faces

1. Open *ANSYS Meshing* by **M**esh (double LMB click) in *Project Schematic*.
2. In the *Meshing Options* window that opens, select “Tetrahedrons (Patch Independent)” as the *Mesh Method*. **O**K.

 - 💡 The *Physics Preference* was automatically set to *CFD* when “Fluid Flow (FLUENT)” was chosen for the *Analysis System*, thus it is not necessary to specify a preference in *Meshing Options*.
3. **L**MB on the face with the largest X coordinate, then **R**MB in the same spot. **C**reate **N**amed **S**election. Type “Outlet” as the face’s name. **O**K (or <Enter>).

 - ✚ If the face can’t be selected, be sure that the “Face” button  is selected in the *Selection Toolbar*.
4. **L**MB on the curved face that will represent the pipe wall, then **R**MB in the same spot. **C**reate **N**amed **S**election, and enter “Wall” as the name for this face. **O**K.
5. In *Graphics*, rotate the object until you can see the face with the smallest X coordinate.
 - 💡 To rotate, click and hold the MMB then drag the cursor anywhere in the *Graphics* window.
 - 💡 It is typical to rotate the view in order to select a particular face. It is possible to select a hidden face in DM, but the procedure is not covered in this tutorial.
6. **L**MB on the quarter circle face with the smallest X coordinate, **R**MB in the same spot. **C**reate **N**amed **S**election, and enter “Inlet” as the name for that face. **O**K.
7. **L**MB on the face with the smallest Y coordinate; it will serve as one of the symmetry planes. **R**MB in the same spot. **C**reate **N**amed **S**election, and enter “Symmetry1” as the name for that face. **O**K.
 - ✚ Depending on the view, you may need to rotate to select the face, and you may also need to re-select the “Face” button .
8. Repeat the process of the previous step to name the face with the smallest Z coordinate. Name it “Symmetry2”.
 - 💡 In *Outline*, **L**MB on the “+” to the left of *Named Selections* to list them. Make sure you have named all five as shown to the right.
9. **F**ile-**S**ave **P**roject.

Insert inflation and generate the mesh

1. In *Outline*, **L**MB on the “+” next to “Mesh”, then **P**atch **I**ndependent.
 - ✚ If there is a “-” instead of a “+” don’t worry, it just means that “Mesh” has already been expanded.
2. In *Details of “Patch Independent”*, use the drop down menu to change *Method* from “Tetrahedrons” to “Sweep”.
3. In an empty area of *Graphics*, **R**MB-**V**iew-**L**eft. This will orient the geometry to look down the X axis and show the face that was named “Inlet”.
4. In *Details of “Sweep Method”*, **A**utomatic to change *Src/Trg Selection* to “Manual Source”. **L**MB on “No Selection” which is highlighted in yellow. In the graphics window, **L**MB on the quarter circle face named “Inlet” and **A**pply. The face turns brownish-red.

Definition	
Suppressed	No
Method	Tetrahedrons
Algorithm	Automatic
Element Midside Nodes	Tetrahedrons
	Hex Dominant
Advanced	Sweep
Defined By	MultiZone

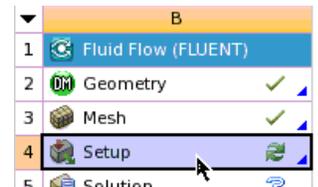
 - ✚ It may helpful to zoom in before trying to select the source face.
5. In *Details of “Sweep Method”*, **L**MB in “Sweep Num Divs”. Enter “100” as the desired number of axial divisions.
6. In *Details of “Sweep Method”*, change *Free Face Mesh Type* from “Quad/Tri” to “All Quad”.
7. **L**MB on the inlet face to select it, then **R**MB in the same spot, **I**nsert-**I**nflation.
 - 💡 “Inflation” means that a finer mesh will be created along walls to help FLUENT resolve the boundary layer.

8. From the *Selection Toolbar*, Edge () to change the selection filter from “Face” to “Edge”. LMB on the edge of the inlet face that is a quarter circle; it should turn green to indicate it has been selected. Apply in *Details of “Inflation”*.
9. For Maximum Thickness, LMB on “Please Define” and enter “.0025” m.
10. In *Outline*, LMB on “Mesh” and expand *Sizing* in *Details of “Mesh”*. Set *Use Advanced Size Function* to “On Proximity and Curvature” for a less distorted mesh.
11. LMB on “Default” for *Curvature Normal Angle*. Enter “4.5” degrees as the angle, which creates 20 evenly spaced circumferential divisions along the 90° circular arc that defines the pipe wall.
12. The rest of the settings are suitable for this tutorial, so in *Outline*, R-Mesh  Generate Mesh.
 -  Mesh generation may take some time. Be patient.
 -  Zoom in and rotate as necessary to see the mesh; it should look something like the one shown at the beginning of this tutorial.
13. In the upper left corner of *ANSYS Meshing*, File-Save Project. Close *ANSYS Meshing*.
14. In *Project Schematic* of ANSYS Workbench, R-Mesh from the template entitled “Straight_pipe”, then  Update. After some calculations, a check mark should appear beside “Mesh”.

Definition	
Suppressed	No
Boundary	1 Edge
Inflation Option	Total Thickness
Number of Layers	5
Growth Rate	1.2
Maximum Thickness	0.0025

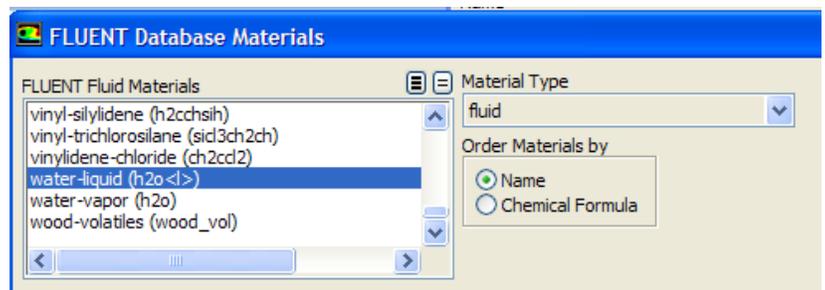
Launch FLUENT and define the flow as turbulent instead of laminar

1. Setup from the *Project Schematic* to open *FLUENT Launcher*. Verify that “Double Precision” is selected. OK.
 -  If “Double Precision” is not selected, select it first (LMB in its box) before OK.
2. The next screen will be the main *FLUENT Window* with your mesh visible in the *Graphics Window*.
3. The default viscous model is laminar flow, but we want to model flow through the pipe as turbulent flow. In the main *FLUENT* menu, Define-Models-Viscous-Laminar-Edit. Select k-epsilon (2 eqn) from the *Viscous Model* window that opens. The default constants and options are fine for this problem, so OK.



Define the fluid as liquid water

1. The default fluid is air, but we want to define the fluid as water. In the main *FLUENT* menu, Define-Materials>Create/Edit-FLUENT Database. Select water-liquid from the list of *FLUENT Fluid Materials*. Copy.
2. Write down the density and viscosity of liquid water. These properties are needed later to calculate Reynolds numbers, etc. Close *Fluent Database Materials*.
3. Close the *Create/Edit Materials* window. *Caution*: This has added liquid water into the list of available fluids, but has not actually changed the fluid from air to water. This will be done next.



Define the cell zones and boundary conditions

1. In the *FLUENT* menu, Define-Cell Zone Conditions. There should be only one zone, “solid” by default. Make sure *Type* is set as fluid. Edit. Select water-liquid instead of “air” as the material. OK.
2. Now the boundary conditions need to be specified. Previously, the boundary conditions were named (inlet, outlet, etc.) but actual values for inlet velocity were never defined. This must be done in *FLUENT*. In the main *FLUENT* menu, Define-Boundary Conditions.
3. Select interior-solid and Edit. Change the name to “interior-fluid”. OK.

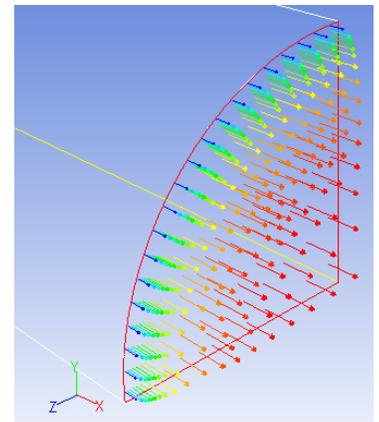
- The default boundary conditions for the wall (pipe wall), the symmetry planes (symmetry1 and symmetry2), and the outlet (pressure-outlet) are okay, so nothing needs to be done to them.
- Select inlet, which is the left side of the computational domain. Edit. In the *Velocity Inlet* window, change *Velocity Magnitude* to “2” m/s. Change the turbulence specification method to Intensity and Hydraulic Diameter. Set the turbulence intensity as “10” % and the hydraulic diameter as the pipe diameter, which we had specified previously as “0.0525 m”. OK.

Set up some more parameters and initialize

- Now the convergence criteria need to be set. In the main *FLUENT* menu, Solve-Monitors-Residuals...-Edit. In *Residual Monitors*, make sure both Plot and Print to Console options are specified in the *Options* portion of the window. “Print” refers to text printed in the main *FLUENT* window, and “Plot” causes the code to plot the residuals on the screen while the code is iterating.
 - 💡 As the code iterates, residuals are calculated for each flow equation. Residuals represent a kind of average error in the solution – the smaller the residual, the more converged the solution.
- There are six differential equations to be solved in a 3-D incompressible $k-\varepsilon$ turbulence model flow problem, and so there are six residuals to be monitored for convergence: continuity, x -, y -, and z -velocity, k (turbulent kinetic energy), and ε (turbulent dissipation rate). The default convergence criteria are 0.001 for all six of these. Experience has shown that this value is generally not low enough for proper convergence. Change the *Convergence Criterion* for all six residuals from 0.001 to 0.000001 (click between two zeroes of the number, and insert three more zeroes).
- To apply the changes, OK, which will also close the *Residual Monitors* window.
- In the main *FLUENT* menu, Solve-Initialization. The default initial values are good enough for this problem. Initialize.
- At this point, and every so often, it is wise to save your work. In the main *FLUENT* menu, File-Save Project. Fluent writes two files in addition to the Workbench file: the *case* file “...cas.gz” (the grid plus all boundary conditions and other specified parameters), and the *data* file “...dat.gz” (the velocity, turbulence, and pressure fields calculated by the code).

Iterate towards a solution

- In the main *FLUENT* menu, Solve-Run Calculation. In the *Run Calculation* sub-window, change *Number of Iterations* to 500, and change *Reporting Interval* to 10. Calculate. The main screen will list the residuals after every 10 iterations, while the graphical display window will plot the residuals as a function of iteration number. When the calculations are complete or converged, OK.
 - 💡 Sometimes you have to click Calculate twice to get it to work.
 - 💡 The residuals may rise at first, but should slowly start to fall. It is normal for the residuals to fluctuate up and down. Do not be concerned if there are reverse flow warnings; these will disappear in time.
- At the end of these iterations, check to see how the solution is progressing. In the main *FLUENT* menu, Display-Graphics and Animations-Vectors-Set Up [the *upper Set Up*, not the lower one].
- In the *Vectors* window that opens, under *Surfaces*, select only outlet. Display. The graphical display window will show the velocity vectors.
 - 💡 The Fit to Window  button is often useful here.
- In *Vectors*, Draw Mesh. In the *Mesh Display* window that opens, select *Edge Type* to be Outline. Display. This will draw an outline of the geometry. Close Mesh Display.
- In *Vectors*, Display again to see the vectors at the outlet, this time with the geometry outline shown.
- Zoom in with the middle mouse [drag a box with MMB from lower



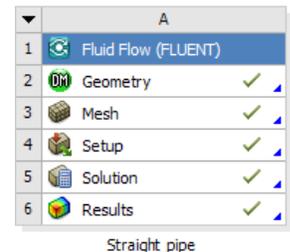
left to upper right to zoom in on the box you draw; drag the MMB from upper right to lower left to zoom out], pan (move) [MMB on the location you want to be at the center], and rotate [drag LMB] as necessary to view the velocity field in more detail. It should look like the vector field shown above. Close the *Vectors* window.

Calculate the pressure drop

1. In the main *FLUENT* window, on the left side under *Results*, Reports-Surface Integrals-Set Up. Set *Report Type* as Area-Weighted Average. *Field Variable* should be Pressure and Static Pressure by default, so leave those settings alone. Select both inlet and outlet under *Surfaces*, and Compute. The results are written to the main *FLUENT* window.
2. Record the inlet and outlet pressures – these are gage pressures averaged across the inlet and outlet faces, respectively.
 - 💡 *Note:* The outlet gage pressure should be zero or nearly zero, since we had specified (by default) the outlet boundary condition as zero gage pressure.
3. Close the *Surface Integrals* window.

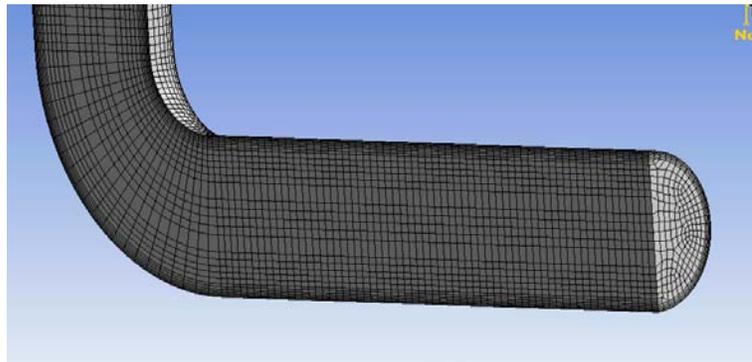
Save your calculations and exit FLUENT

1. In the main *FLUENT* menu, File-Save Project.
 - ✚ If a prompt appears at this point, select the option called Store the modified settings... OK.
2. Exit *FLUENT* by File-Close FLUENT. This will return you to Workbench.
3. In Workbench, U Update Project. After some calculations, check marks should appear on all components of the *Fluid Flow (FLUENT)* template in *Project Schematic*. File-Save.



Part 2: Pipe with Elbow

A geometry and mesh like the one shown below will be created in Part 2 of this tutorial.



Set Workbench options for a second FLUENT project – the pipe with an elbow

1. This tutorial assumes that ANSYS Workbench is running, and the straight pipe project from Part 1 of this tutorial has been completed. **If so, skip to Step 5.** If Workbench is not running, do Steps 2-4.
2. Start ANSYS Workbench. Under *View* make sure that “Toolbox”, “Toolbox Customization” and “Project Schematic” all have check marks next to them.
 - ✚ Check marks can be inserted by placing the cursor over the menu item and LMB.
3. In *Toolbox Customization* under *Analysis Systems*, verify that “Fluid Flow (FLUENT)” has a check mark next to it.
 - ✚ If column “A” is not shown in *Toolbox Customization*, 1 located to the left of “Name” in *Toolbox Customization*.
 - ✚ If the box to the left of “Fluid Flow (FLUENT)” is unchecked, LMB in the box next to “Fluid Flow (FLUENT)”.

- ✚ Depending on your system settings, it may be necessary to make Column B wider in order to see the entirety of each entry.
 - 💡 To de-clutter your Workbench workspace, close or minimize *Toolbox Customization*, it is not needed after this step.
4. File-Open, then navigate to “MinorLoss”, the Workbench project created in Part 1 of this tutorial. Open.
 5. In the *Toolbox*, not *Toolbox Customization*, Fluid Flow (FLUENT) and hold the LMB to drag it into the large box that will appear in *Project Schematic*. In the *Project Schematic*, LMB on “Fluid Flow (FLUENT)” located below the new template, and enter “Elbow_pipe” as the name for the second *Analysis System*, <Enter>.

 - ✚ In some versions of Workbench, “Fluid Flow” is displayed for the name of the analysis template instead of “Fluid Flow (FLUENT)”.
 - ✚ Multiple boxes may appear when you drag the *Analysis System* into *Project Schematic*, let go of the LMB when the cursor is in the biggest box.

Create the sweep path line

1. Geometry in the *Fluid Flow (Fluent)* template entitled “Elbow_pipe” to open *ANSYS DesignModeler*.
2. At startup DM will prompt the user to select the desired length unit, LMB on the circle next to “Meter” if it is not selected already, then OK.

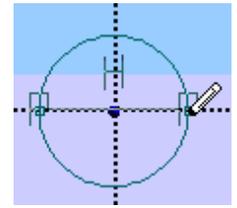
 - ✚ If this prompt does not appear, it means that your version of DM was set to always use a certain unit of length. If this is the case, try Tools-Options-Units, and set *Display Units Pop-up Window* to “Yes”. You may need to restart DM to get the window to pop up.
3. In *Tree Outline*, XYPlane. Then in *Display Toolbar*, Look at Face/Plane/Sketch .
4. Sketching which is located near the bottom of *Tree Outline*. This will bring up the *Draw* tab of *Sketching Toolboxes*.
5. Line and hover over the origin. When a “P” is displayed, LMB. Move the cursor along the positive X-axis. When the cursor is about half the way to the edge of *Graphics* and both a “C” and an “H” are displayed, LMB to place the other end of the line.

Note: The length of the line isn’t critical because you will supply a dimension for its length later.
6. Hover over the right endpoint of the line that was just created. When a “P” is displayed, LMB. Move the cursor straight up to make a line that is about the same length as the first line. Before LMB to place the second endpoint, make sure that a “V” is displayed near the center of the line.
 - 💡 The “V” indicates that a vertical constraint will be added to the line.
7. In *Sketching Toolboxes* Modify-Fillet, then LMB on the vertex where the lines intersect.

Create the sweep profile

1. Modeling from the bottom of *Sketching Toolboxes*. Then in *Tree Outline*, YZPlane and Look at Face/Plane/Sketch  from *Display Toolbar*.
2. Zoom in until the scale at the bottom of the *Graphics* window goes from 0 to about 0.05 m as shown to the right.
3. Sketching to bring up the *Draw* tab of *Sketching Toolboxes*.
4. Circle and hover over the origin. When a “P” is displayed, LMB. Move the cursor approximately .03 m any direction then LMB.
 - 💡 The coordinates of your cursor are displayed at the bottom right corner of *Graphics*.
5. In *Sketching Toolboxes*, Modify-Split at Select. RMB on an open area in *Graphics* and Split Edge into n Equal Segments.

6. **LMB** on the edge of the circle to create two new points on the (horizontal) Y-axis. These points split the circle into two segments, top and bottom.
7. In *Sketching Toolboxes*, **Draw-Line**. Hover over one of the newly created points. When a “P” appears, **LMB**. Move the cursor and hover over the other point. When a “P” appears, **LMB**.
8. In *Sketching Toolboxes*, **Modify-Trim**. **LMB** on both the lower right and lower left edges of the circle to remove them.



- 💡 Using a semicircle for the profile of the sweep and a symmetry plane will yield the same results as using a circular profile since the flow is symmetric about the symmetry plane. However, using a semicircular profile reduces the analysis time by approximately 50%.

Constrain and dimension the lines

1. In *Sketching Toolboxes*, **Dimensions**. **LMB** on the edge of the semicircle. With the cursor in a reasonable location, **LMB** again to place the radial dimension at the cursor’s location.
 - 💡 In the bottom left corner of the DesignModeler window there is a prompt that gives you a brief description of how to use the tool that is selected.
2. In the *Dimensions* section of *Details View*, the dimension listed corresponds to the radius of the circle. Set this value to “.02625” m.
3. At the bottom of *Sketching Toolboxes*, **Modeling**. From the *Tree Outline*, **+** (click “+”) to the left of “XYPlane”, then **Sketch1**.
4. **Look at Face/Plane/Sketch**. Then reopen the *Sketching Toolbox* by **Sketching** at the bottom of *Tree Outline*.
5. In *Sketching Toolboxes*, **Dimensions**. **LMB** on the horizontal line in *Geometry*. With the cursor in a reasonable location, **LMB** again to place the dimension at the cursor’s location.
6. Repeat the process from the previous step to dimension the vertical line and the quarter circle.
7. In the *Dimensions* section of *Details View*, the first dimension listed corresponds to the length of the horizontal line. Set this value to “0.20” m. The second dimension listed corresponds to the radius of the quarter circle; set it to “.0525” m. The last dimension listed corresponds to the length of the vertical line; set it to “0.40” m.
 - 💡 DM lists dimensions in alphabetical order. Although “R3” was the third dimension to be created, it is the second dimension listed in *Details View*.
8. Zoom in and pan to see the line that will define the geometry of the pipe: a straight entrance region, the elbow, and a straight exit region.

Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 1	
<input checked="" type="checkbox"/> R1	0.02625 m

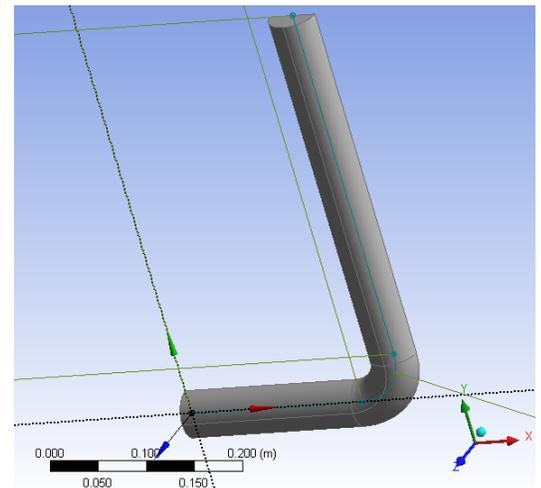


Details View	
Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 3	
<input type="checkbox"/> H1	0.2 m
<input type="checkbox"/> R3	0.0525 m
<input checked="" type="checkbox"/> V2	0.4 m

Use a sweep to create half of the pipe

1. From the *Features Toolbar*, **Sweep** (). **LMB** on “Sketch2”, which is under “YZPlane”, and **Apply** from *Details View* to set “Sketch2” as the sweep profile.
 - ✚ If “Sketch2” isn’t visible, **+** to the left of “YZPlane” in *Tree Outline*.
 - ✚ If the sweep profile sketch wasn’t the second sketch to be created, DM will assign the profile sketch a number other than “2”.
2. **LMB** on “**Not Selected**” in *Details View*. **LMB** on “Sketch1”, which is under “XYPlane”, and **Apply** to set “Sketch1” as the sweep path.
 - ✚ If “Sketch1” isn’t visible, **+** (click “+”) to the left of “XYPlane” in *Tree Outline*.
 - ✚ If the sweep path sketch wasn’t the first sketch to be created, DM will assign the path sketch a number other than “1”.
3. **Generate**, which is above *Tree Outline*, to perform the sweep.

- ✚ If DM displays a red “!” next to “Sweep1” in *Tree Outline*, **RMB** on “Sweep1” and **Show Errors or Warnings** to find out the reason why the sweep couldn’t be performed.
- 4. Zoom in, pan, and rotate as necessary to see if the sweep worked. The pipe/elbow should look like the one to the right.
- 5. In the upper left corner of the *DM*, **File-Save Project**, then close *DM*.



Set meshing options and label the faces

1. **Mesh** in the *Fluid Flow (Fluent)* template entitled “Elbow_pipe” to open *ANSYS Meshing*.
2. From the *Meshing Options* window that opens, **LMB** on the circle next to “Tetrahedrons (Patch Independent)” if it isn’t selected already to set the *Mesh Method*, then **OK**.

💡 The *Physics Preference* was automatically set to *CFD* when “Fluid Flow (FLUENT)” was chosen for the *Analysis System*, thus it is not necessary to specify a preference in *Meshing Options*.

3. **Units-Metric (m, kg, N, s, V, A)**.
4. **LMB** on the face with the largest Y coordinate, then **RMB** in the same spot. **Create Named Selection** and enter “Outlet” as the name for that face, then **OK**.



✚ If the face can’t be selected, be sure that the “Face” button is selected in the *Selection Toolbar*.

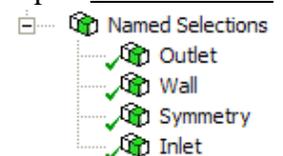
5. Hold <Ctrl> and **LMB** on all the curved faces that will represent the pipe wall. **RMB** anywhere in *Graphics*. **Create Named Selection** and enter “Wall” as the name for those faces, then **OK**.
6. In *Graphics*, click and hold the MMB while moving the cursor to rotate the view until the symmetry plane is visible.

7. **LMB** on any of the three faces that will be used as a symmetry boundary condition. From the *Selection Toolbar*, **Extend Selection-Extend to Limits** to select all three faces. **RMB** anywhere in *Graphics*. **Create Named Selection** and enter “Symmetry” as the name for those faces, then **OK**.



💡 The Extend to Limits option is a quick way to select multiple faces.

8. **LMB** on the semicircular face with the smallest X coordinate. **RMB** in the same spot. **Create Named Selection** and enter “Inlet” as the name for that face, then **OK**.
9. In *Outline*, **LMB** on the “+” to the left of *Named Selections* to list them. Make sure you have named all four as shown to the right.

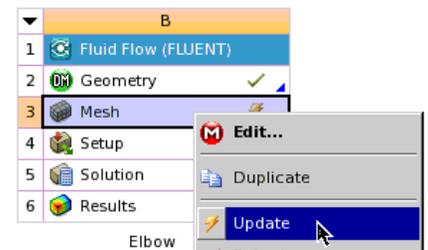


Insert inflation and generate the mesh

1. In *Outline* **LMB** on the “+” next to “Mesh”, then **LMB** on “Patch Independent”.
 - ✚ If there is a “-“ instead of a “+“ don’t worry, it just means that “Mesh” has already been expanded.
2. In *Details of “Patch Independent”*, use the drop down menu to change *Method* from “Tetrahedrons” to “Sweep”.
3. In an empty area of *Graphics*, **RMB-View-Left**. This will orient the geometry to show the face that was named “Inlet” (looking directly down the pipe from the inlet).
4. **LMB** on “Automatic” to change *Src/Trg Selection* to “Manual Source”. **LMB** on “No Selection”. **LMB** on the semicircle face named “Inlet”. It should turn green. **Apply**.
 - 💡 It may helpful to zoom in or rotate a little before trying to select the source face.

5. Change *Free Face Mesh Type* from “Quad/Tri” to “All Quad” using the drop down menu.
6. **LMB** on the semicircle face named “Inlet” to select it, then **RMB** in the same spot, **Insert-Inflation**.
7. From the *Selection Toolbar*, **Edge** () to change the selection filter from “Face” to “Edge”. **LMB** on the edge of the inlet face that is a semicircle; it should turn green to indicate it has been selected. **Apply** in *Details of “Inflation”*.
8. For **Maximum Thickness**, **Please Define** and enter “.0025” m.
💡 The “Inflation” that will be added along the pipe wall will help FLUENT resolve the boundary layer.
9. In *Outline*, **Mesh**. Expand *Sizing* in *Details of “Mesh”* by clicking on the “+”. Set *Use Advanced Size Function* to “On Proximity and Curvature” for a less distorted mesh.
10. **LMB** on “Default” for *Curvature Normal Angle*. Enter “4.5” degrees as the desired angle, which creates 40 circumferential divisions. [This is the same resolution as the straight pipe case.]
11. In *Outline*, under *Mesh*, **Sweep Method**. In *Details of “Sweep Method”*, **LMB** in “Sweep Num Divs”. Enter “100” as the desired number of axial divisions. [This is the same number of axial divisions that were used for the straight pipe.]
12. The rest of the settings are suitable for this tutorial, so **R-Mesh-Generate Mesh**. (It may take a while, so be patient.)
13. In the upper left corner of *ANSYS Meshing*, **File-Save Project** and close *ANSYS Meshing*.
14. In the *Project Schematic* of ANSYS Workbench, **R-Mesh** from the template entitled “Elbow_pipe”, then **Update**. After some calculations, a check mark should appear beside “Mesh”.

Definition	
Suppressed	No
Boundary	1 Edge
Inflation Option	Total Thickness
Number of Layers	5
Growth Rate	1.2
Maximum Thickness	2.5e-003 m



Run FLUENT for the pipe with elbow

1. To run the CFD analysis, follow the FLUENT instructions for the straight pipe above. Everything is identical except that there is only one symmetry plane instead of two separate symmetry planes.
2. Calculate the pressure drop from the inlet to the outlet as was previously done for the straight pipe and compare. The difference between the two pressures is due to the effects of the elbow alone, since the two pipes are the same diameter and length, and since the fluid, flow rate, etc. are identical.