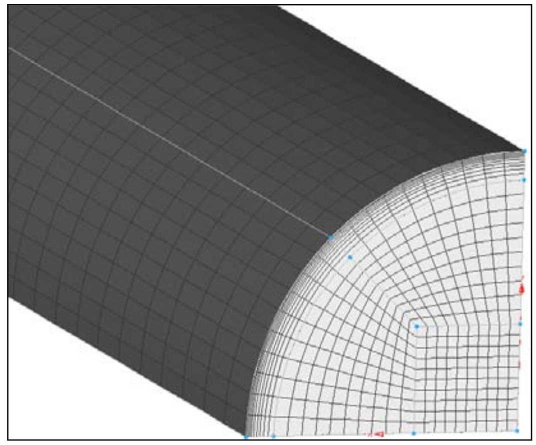
Tutorial – Minor Losses using Pointwise and FLUENT

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

Introduction



This tutorial provides instructions for meshing two internal flows. Pointwise software will be used to create the meshes, and FLUENT will be used as the CFD solver. 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 "extrusion" to generate a fine grid near a wall and a structured mesh everywhere else.

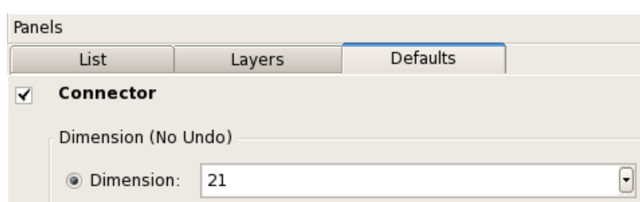
Part 1: Straight Pipe

Choose solver

1. This tutorial assumes that Pointwise is running. If it is not running, open it before proceeding to the next step.
2. 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.
3. <u>CAE</u>-<u>Select Solver…</u>
4. <u>ANSYS FLUENT</u>.
5. <u>OK</u>.
- The current solver is displayed in the lower left of the Pointwise window.
6. <u>File</u>-<u>Save</u>.
7. After selecting an appropriate folder in which to save the project, enter "Straight_Pipe_MinorLoss" for the *File Name*. <u>Save</u>.

Create geometry in Pointwise

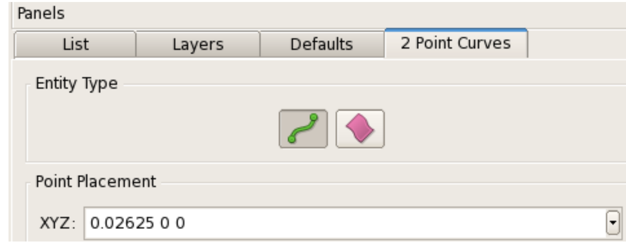
1. Because of symmetry, it is not necessary to model the entire pipe. Only one-quarter of the pipe will be analyzed; this change in geometry reduces the analysis time by approximately 75%.
2. <u>View</u>-<u>Toolbars</u> and verify that all toolbars are activated (a check by each title).
3. *<u>Defaults</u>* tab. Make sure the box to the left of *Connector* is checked.
4. Enter "21" in as the default dimension. *<u>List</u>* tab.
   - The default dimension is the number of nodes assigned to each connector created after setting the default.



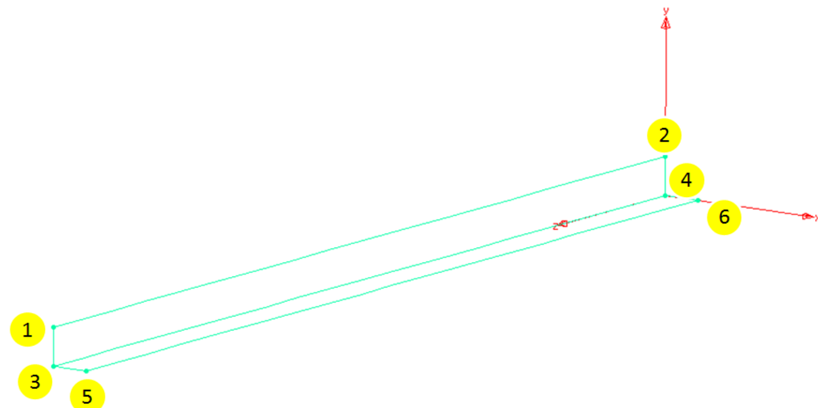
5.  <u>2 Point Curves</u> from the Create toolbar.
   - Hover the cursor over any toolbar button to display its name.
   - Alternatively, <u>Create</u>-<u>2 Point Curves...</u>
6. Under *Point Placement*, type "0 0.02625 0" as the XYZ coordinates for the first endpoint. <Enter>.
   - Use the following format entering *Point Placement* data: "x-coordinate" <Space> "y-coordinate" <Space> "z-coordinate".
7. Type "0 0 0" as the XYZ coordinates for the second endpoint. <Enter>.

- Delete Last Point if a wrong coordinate is entered.

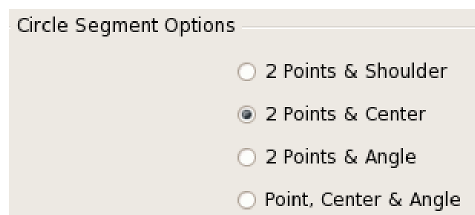
8. <Enter> again to place another point at "0 0 0".
9. Type "0.02625 0 0". <Enter>.
10. Type "0 0.02625 0.68247". <Enter>.
11. Type "0 0 0.68247". <Enter>.
12. <Enter> again to place another point at "0 0 0.68247".
13. Type "0.02625 0 0.68247". <Enter>.
14. OK.



15. Rotate the axis to an isometric view by holding down <Ctrl> and the RMB while moving the cursor to the left.
16. View-Zoom-Zoom to Fit.
    - Alternatively, press <F2> on the keyboard.
17. 2 Point Curves.
18. LMB Point1, then LMB Point2 to create a long connector.
19. LMB Point3, then LMB Point4 to create a second long connector.
20. LMB Point5, then LMB Point6 to create a third long connector
21. OK.

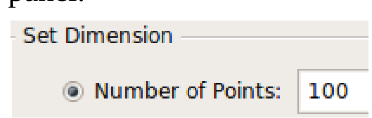


22. Curve from the Create toolbar.
    - Alternatively, Create-Draw Curves-Curve.
23. Circle. LMB "2 Points & Center" under *Circle Segment Options*.
24. LMB Point1, then LMB Point5. LMB Point3 to place the center of the circle. Apply.
25. LMB Point2, then LMB Point6. LMB Point4 to place the center of the circle. Apply.
26. OK.
27. LMB the "+" next to Connectors in the *List* side panel. The expanded list shows the type and number of nodes assigned to every connector.
    - A "connector" is the Pointwise terminology for a construction line.



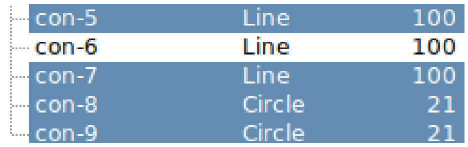
Modify axial dimension

1. Simultaneously select the four longest connectors ("con-5", "con-6", and "con-7").
    - Hold <Ctrl> while LMB to make multiple selections.
    - Selections can be made in the graphics window or from the *List* side panel.
2. Grid-Dimension…
3. Type "100" for *Number of Points*. Dimension-OK. These points will space the mesh cells in the axial direction.
    - Sometimes it is helpful to make the spacing nodes visible. To do this, select all the connectors by <Ctrl> + <A>. Choose "Points On" from the drop down menu beside "Points Off" in the attributes toolbar.

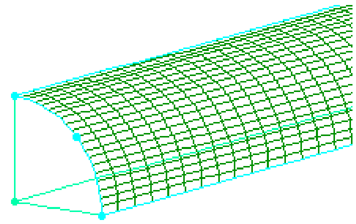


Create surface mesh on pipe wall

1. Simultaneously select the closed loop of four connectors that form the perimeter of the curved pipe surface. These connectors form the framework for a mesh that will cover the curved wall of the pipe.
   - Hold <Ctrl> while LMB to make multiple selections.
   - Try selecting the connectors from the list under *Connectors* in the side panel. Select "con-5", "con-7", "con-8", and "con-9".

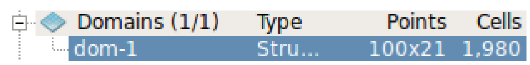


2. Structured in the Mesh toolbar.
   - Alternatively, Grid-Set Type-Structured.
3. Assemble Domains in the Create toolbar. A structured mesh with 100 axial divisions and 20 circumferential divisions is created on the pipe wall.

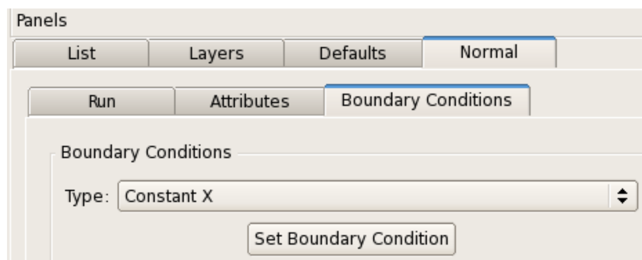


Extrude surface mesh

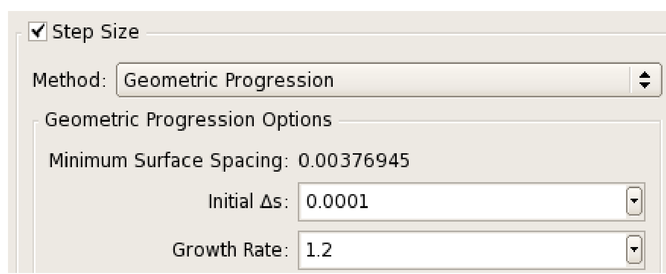
1. LMB the "+" next to Domains in the *List* panel.
   - A "domain" is a surface grid in Pointwise.
2. Select "dom-1".



3. Create-Extrude-Normal…
4. *Boundary Conditions* tab in the side panel.
5. LMB the long edge with the largest y-coordinate.
6. In the side panel, change *Type* from "Splay" to "Constant X". Set Boundary Conditions.

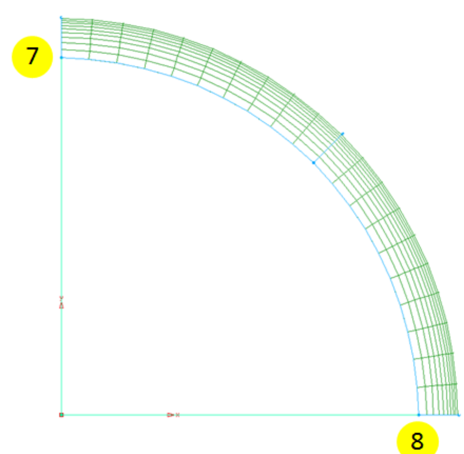


7. LMB the long edge with the largest x-coordinate.
8. Change *Type* from "Splay" to "Constant Y". Set Boundary Conditions.
9. Simultaneously select the circular edges on each end of the pipe (total of 2).
10. Change *Type* from "Splay" to "Constant Z". Set Boundary Conditions.
11. *Attributes* tab in the side panel.
12. LMB the box next to *Step Size*.
13. Enter "0.0001" for *Initial* Δ*s*.
14. Change the *Growth Rate* to "1.2".
15. LMB the box next to *Orientation*.
16. Flip only if the direction arrows in the graphics window are pointing away from the center of the pipe.
17. *Run* tab in the side panel.
18. Enter "10" for *Steps*. Run-OK.

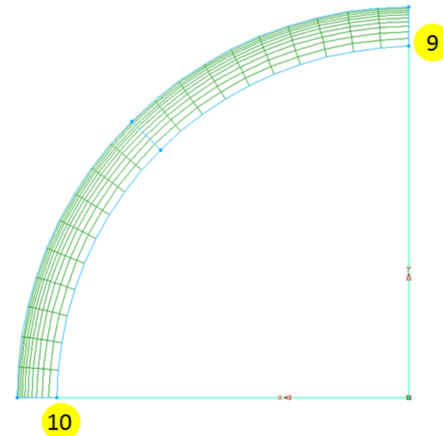


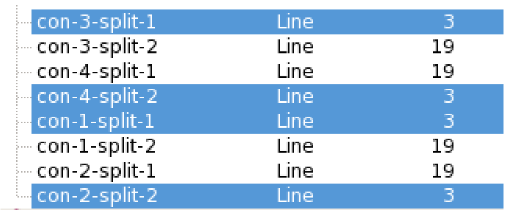
Split connectors

1. Recall View -Z from the View toolbar.
2. View-Zoom-Zoom to Fit (or <F2>).
3. You are now looking down the Z axis of the pipe. Notice the 10 layers of cells along the wall of the pipe. The surface mesh has been extruded into a block of cells. There should now be a (1) after *Blocks* in the *List* panel.
   - A "block" refers to a volume grid in Pointwise.
4. LMB the vertical connector (con-3).
5. Edit-Split…
6. LMB Point7.

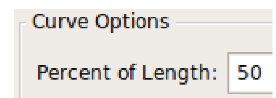


7. OK.
8. LMB the horizontal connector (con-4).
9. Edit-Split…
10. LMB Point8.
11. OK.
12. Recall View +Z from the View toolbar.
13. View-Zoom-Zoom to fit (or <F2>).
14. LMB the vertical connector (con-1).
15. Edit-Split…
16. LMB Point9.
17. OK.
18. LMB the horizontal connector (con-2).
19. Edit-Split…
20. LMB Point10.
21. OK.
22. Simultaneously select "con-3-split-1", "con-4-split-2", "con-1-split-1", and "con-2-split-2" from *Connectors* in the *List* panel.
23. <Delete>.
24. Select "con-14" from *Connectors* in the *List* panel.
25. Edit-Split...
26. Enter "50" for the *Percent of Length*. <Enter>.
27. OK. The connector has been split into two separate connectors with 11 nodes on each.

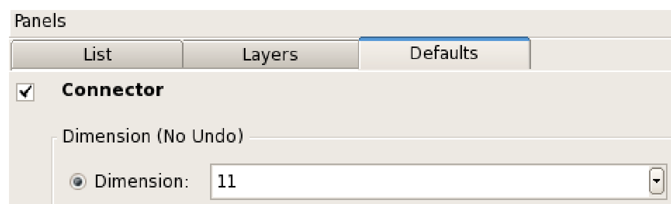




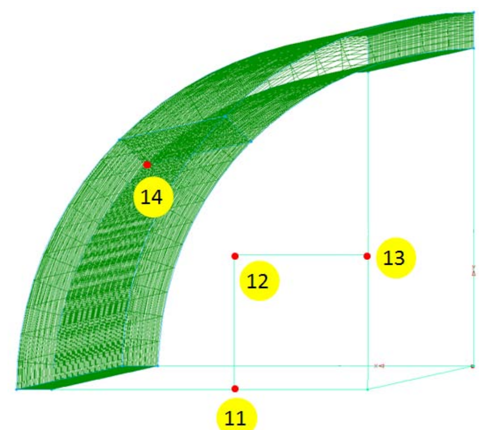
con-3-split-1	Line	3
con-3-split-2	Line	19
con-4-split-1	Line	19
con-4-split-2	Line	3
con-1-split-1	Line	3
con-1-split-2	Line	19
con-2-split-1	Line	19
con-2-split-2	Line	3

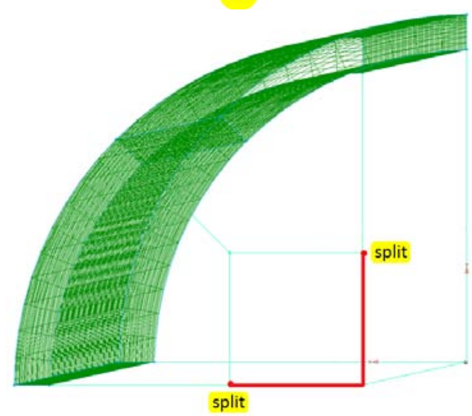


Create divisions on pipe end cap



1. *Defaults* tab. Make sure the box to the left of *Connector* is checked.
2. Enter "11" as the new default dimension. *List* tab.
3. 2 Point Curves.
4. Type "0.01 0 0". <Enter>.
5. Type "0.01 0.01 0". <Enter>.
6. <Enter> again to place another point at "0.01 0.01 0".
7. Type "0 0.01 0". <Enter>.
8. OK.
9. Rotate and zoom to get the perspective shown to the right.
✚ 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.
10. 2 Point Curves.
11. Connect Point12 and Point14 by successively LMB on each point.
12. Follow the basic steps used in the section Split connectors to split the four connectors at the Point11 and Point13 as shown to the right.
    ✚ Select line to be split. Edit-Split… Select point where split will occur. OK.
13. Simultaneously select the two newly split connectors that only have 9 nodes.





- ✚ Select ("con-1-split-2-split-2" and "con-2-split-1-split-1) from under *Connectors* in the *List* side panel.

con-1-split-2-split-2	Line	9
con-2-split-1-split-1	Line	9

14. Grid-Dimension…
15. Enter "11" as the *Number of Points*. Dimension-OK.
    - Connectors opposite from each other must have the same number of nodes to create a structured grid.

Create structured surface meshes on pipe end cap

1. Simultaneously select the 9 connectors shown to the right from the List side panel.
   - To make selecting an entire list of connectors easier, hold <Shift> while LMB on the first and last entry of a list.
2. 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.

con-14-split-1	Line	11
con-14-split-2	Line	11
con-1	Line	11
con-2	Line	11
con-3	Line	11
con-1-split-2-split-1	Line	11
con-1-split-2-split-2	Line	11
con-2-split-1-split-1	Line	11
con-2-split-1-split-2	Line	11

Extrude domains along pipe length

1. Simultaneously select "dom-7", "dom-8", and "dom-9" from the *List* side panel.
2. Create-Extrude-Path…
3. LMB the box next to *Assemble Special*.
4. Delete All Faces.
5. Auto-Assemble.
6. Assemble.
7. Done.
8. *List* tab.
9. Select "con-11". "Con-11" is an axial connector that will guide the extrusion path.
10. *Path* tab.
11. Run-OK.

dom-7	Structured	...	100
dom-8	Structured	...	100
dom-9	Structured	...	100

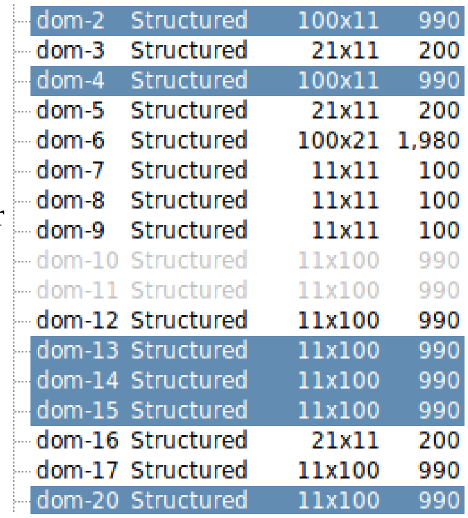
Define boundary conditions

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 "Velocity Inlet" from the drop down menu.
5. *List* tab from the side panel.
6. Simultaneously select "dom-5", "dom-16", and "dom-21".
7. *Set BC* tab from the side panel.
8. LMB in the empty box next to the second row. A 3 should appear under the "#" column. This indicates that three domains have been defined as an inlet.
9. New.
10. **LMB** "bc-3" and change the name to "outlet".
11. In the same row, "**Unspecified**" and choose "Pressure Outlet" from the drop down menu.
12. *List* tab.
13. Select "dom-3", "dom-7", "dom-8", and "dom-9".
14. *Set BC* tab.
15. LMB in the empty box next to the third row. A 4 should appear under the "#" column.

dom-5	Structured	21x11	200
dom-6	Structured	100x21	1,980
dom-7	Structured	11x11	100
dom-8	Structured	11x11	100
dom-9	Structured	11x11	100
dom-10	Structured	11x100	990
dom-11	Structured	11x100	990
dom-12	Structured	11x100	990
dom-13	Structured	11x100	990
dom-14	Structured	11x100	990
dom-15	Structured	11x100	990
dom-16	Structured	21x11	200
dom-17	Structured	11x100	990
dom-20	Structured	11x100	990
dom-21	Structured	11x11	100

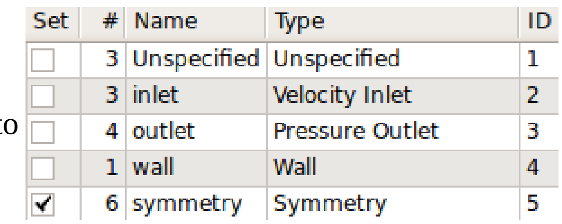
dom-3	Structured	21x11	200
dom-4	Structured	100x11	990
dom-5	Structured	21x11	200
dom-6	Structured	100x21	1,980
dom-7	Structured	11x11	100
dom-8	Structured	11x11	100
dom-9	Structured	11x11	100

16. New.
17. **LMB** “bc-4” and change the name to “wall”.
18. In the same row, “**Unspecified**” and choose “Wall” from the drop down menu.
19. *List* tab.
20. Select “dom-1”.
21. *Set BC* tab.
22. LMB in the empty box next to the fourth row. A 1 should appear under the “#” column.
23. New.
24. **LMB** “bc-5” and change the name to “symmetry”.
25. In the same row, “**Unspecified**” and choose “Symmetry” from the drop down menu.
26. *List* tab.
27. Simultaneously select “dom-2”, “dom-4”, “dom-13”, “dom-14”, “dom-15”, and “dom-20”.



dom-2	Structured	100x11	990
dom-3	Structured	21x11	200
dom-4	Structured	100x11	990
dom-5	Structured	21x11	200
dom-6	Structured	100x21	1,980
dom-7	Structured	11x11	100
dom-8	Structured	11x11	100
dom-9	Structured	11x11	100
dom-10	Structured	11x100	990
dom-11	Structured	11x100	990
dom-12	Structured	11x100	990
dom-13	Structured	11x100	990
dom-14	Structured	11x100	990
dom-15	Structured	11x100	990
dom-16	Structured	21x11	200
dom-17	Structured	11x100	990
dom-20	Structured	11x100	990

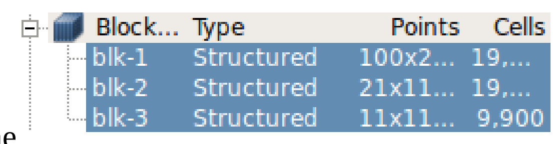
28. *Set BC* tab.
29. LMB in the empty box next to the fifth row. A 6 should appear under the “#” column.
30. The boundary conditions should now be set as indicated to the right.
31. Close the BC tab.



Set	#	Name	Type	ID
	3	Unspecified	Unspecified	1
	3	inlet	Velocity Inlet	2
	4	outlet	Pressure Outlet	3
	1	wall	Wall	4
✓	6	symmetry	Symmetry	5

Export CAE as FLUENT case

1. LMB the “+” next to *Blocks* in the side panel.
2. Simultaneously select all the blocks (total of 3).
3. File-Export-CAE…
4. After selecting an appropriate folder in which to export the project, enter “Straight_Pipe_MinorLoss” for the *File Name*. Save.
5. File-Save the Pointwise file.
6. File-Exit Pointwise.

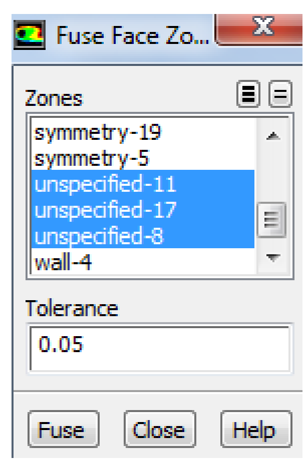


Block...	Type	Points	Cells
blk-1	Structured	100x2...	19,...
blk-2	Structured	21x11...	19,...
blk-3	Structured	11x11...	9,900

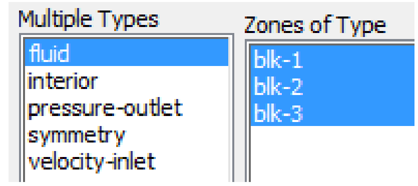
Launch FLUENT and open case

1. Locate and launch FLUENT. Verify that “Double Precision” is selected in *FLUENT Launcher*. 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*.
3. File-Read-Case…
4. Locate “*Straight_Pipe_MinorLoss.cas*”, and OK.
5. FLUENT reads and builds the mesh that was created in Pointwise.

Merge fluid zones and surfaces

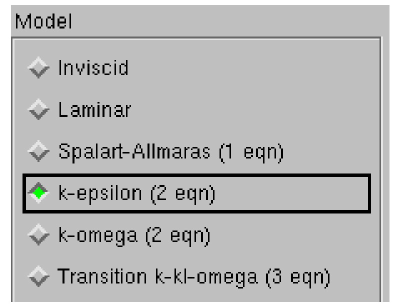


1. The mesh imported from Pointwise consists of three fluid zones (*blocks* in Pointwise terminology) because the mesh was created in sections. Multiple zones result in numerous surfaces with identical boundary conditions. Post-processing is simplified by combining similar entities.
2. Mesh-Fuse…
3. Select all three of the “unspecified” zones by LMB on each of them.
4. Fuse-Close.
   - Fusing combines common nodes and surfaces in the mesh. In this case, internal surfaces between zones are combined and designated as an “interior” boundary condition.



5. Mesh-Merge…
6. LMB "fluid", then Merge in the *Merge Zones* window that opens.
   - ✚ The fluid zones must be combined before merging any other type.
7. LMB "interior", then Merge.
8. LMB "pressure-outlet", then Merge.
9. LMB "symmetry", then Merge.
10. LMB "velocity-inlet", then Merge.
11. Close the *Merge Zones* window.

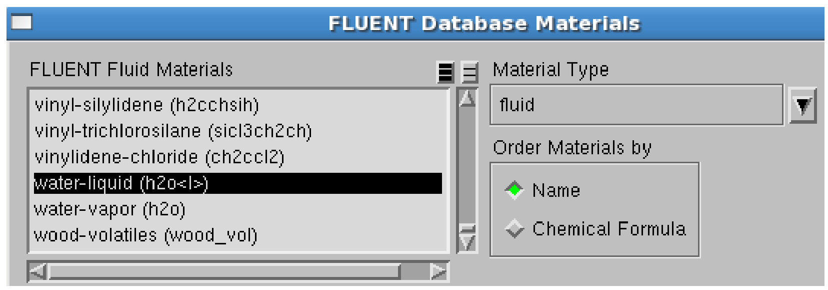
Set solver model



1. The default model is laminar flow, but we want to model flow through the pipe as turbulent flow.
2. In the main *FLUENT* menu, Define-Models-Viscous-Laminar-Edit.
3. 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

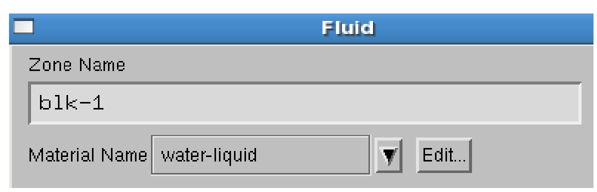
1. The default fluid is air, but we want to define the fluid as water.
2. In the main *FLUENT* menu, Define-Materials-Create/Edit-FLUENT Database.
3. Select water-liquid from the list of *FLUENT Fluid Materials*.
4. Copy.
5. Close *Fluent Database Materials*.
6. Close the *Create/Edit Materials* window.



   - *Caution*: This has added liquid water into the list of available fluids, but it has not actually changed the fluid from air to water. This will be done next.

Define the cell zones and boundary conditions

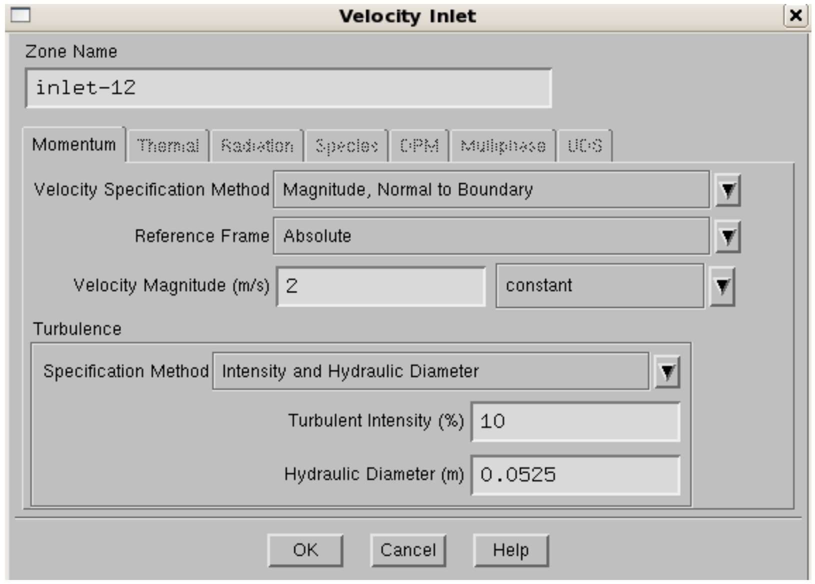
1. In the *FLUENT* menu, Define-Cell Zone Conditions.
2. There should be one zone ("blk-1").
3. Select "blk-1".
4. Make sure *Type* is set as fluid.
5. Edit.
6. Select water-liquid instead of "air" as the material.
7. OK.



Set up boundary conditions

1. Now the boundary conditions need to be specified. In Pointwise the boundary conditions were named (inlet, outlet, etc.) but actual values for inlet velocity were never defined. This must be done in FLUENT.
2. In the main *FLUENT* menu, Define-Boundary Conditions.
3. The default boundary conditions for the walls (pipe wall), the symmetry planes, and the outlet (pressure-outlet) are okay, so nothing needs to be done to them.

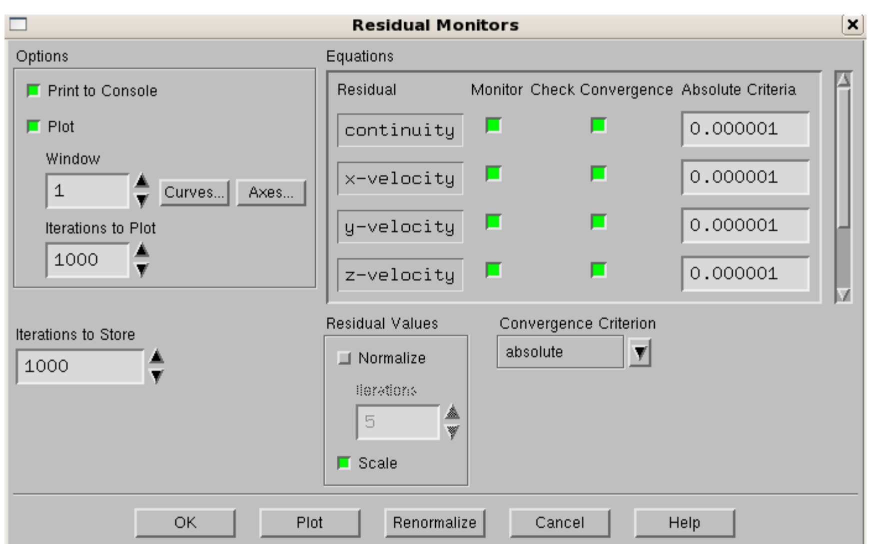
4. Select inlet-12.
   - The numbers beside each label may differ from the tutorial. The numbering is unimportant; make changes based on the type of boundary conditions rather than the numbering scheme.
5. Edit.
6. Change *Zone Name* to “inlet”.
7. In the *Velocity Inlet* window, change *Velocity Magnitude* to “2” m/s.
8. Change the turbulence specification method to Intensity and Hydraulic Diameter.



9. Set the turbulence intensity as “10” % and the hydraulic diameter as the pipe diameter, which we had specified previously as “0.0525 m”.
10. OK.

Set up some more parameters and initialize

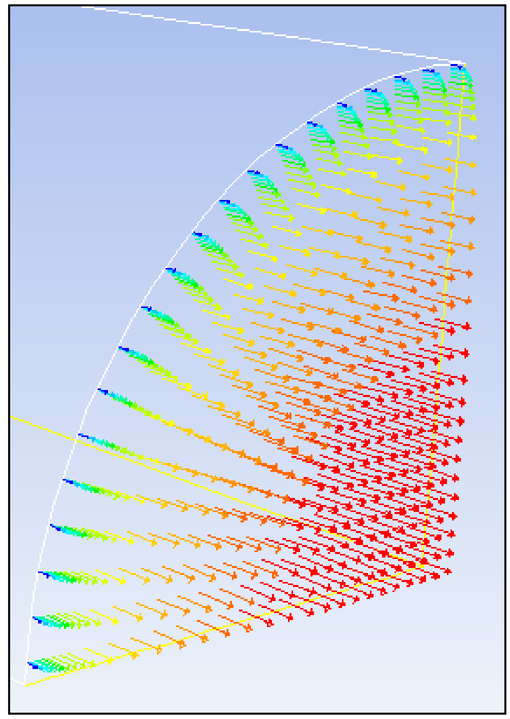
4. Now the convergence criteria need to be set. In the main *FLUENT* menu, Solve-Monitors-Residuals…-Edit.
5. In *Residual Monitors*, make sure both Plot and Print to Console options are specified in the *Options* portion of the window.
   - “Print” refers to text printed in the main *FLUENT* window, 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.



6. There are six differential equations to be solved in a 3-D incompressible k-ε turbulence model flow problem. Therefore, 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).
7. OK.
8. In the main *FLUENT* menu, Solve-Initialization. The default initial values are good enough for this problem. Initialize.

Iterate towards a solution

1. In the main *FLUENT* menu, Solve-Run Calculation.
2. Change *Number of Iterations* to 700, and change *Reporting Interval* to 10.
3. 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.
   - 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.
4. OK when the calculations are completed or converged.
5. 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].
6. Select the outlet surface under *Surfaces* in the *Vectors* window that opens.
7. Display.
8. Fit to Window button to better see the velocity vectors.
9. In *Vectors*, Draw Mesh.
10. In the *Mesh Display* window that opens, select Outline as the *Edge Type*.
11. Display. This will draw an outline of the geometry.
12. Close *Mesh Display*.
13. In *Vectors*, change *Scale* to ".5".
14. Display to see the vectors at the outlet, this time with the geometry outline shown.



15. Zoom and rotate on the velocity field to see it in more detail. It should look similar to the velocity field pictured here. Close the *Vectors* window.
    - To zoom in, drag a box with the MMB from the lower left to upper right.
    - To zoom out, drag a box with the MMB from the upper right to the lower left.
    - To rotate, hold LMB while moving the cursor.
16. Close the *Vectors* window.

Calculate the pressure drop

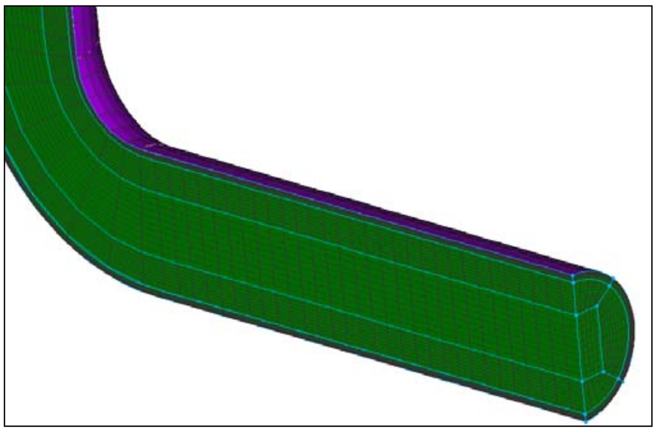
1. On the left side of the main *FLUENT* window under *Results*, Reports-Surface Integrals-Set Up.
2. Set *Report Type* as Area-Weighted Average.
3. *Field Variable* should be Pressure and Static Pressure by default, so leave those settings alone.
4. Select all the inlet and outlet surfaces under *Surfaces*.
5. Compute. The results are written to the main *FLUENT* window.
6. 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 the outlet boundary condition was specified (by default) as zero gage pressure.
7. Close the *Surface Integrals* window.

Save your calculations and exit FLUENT

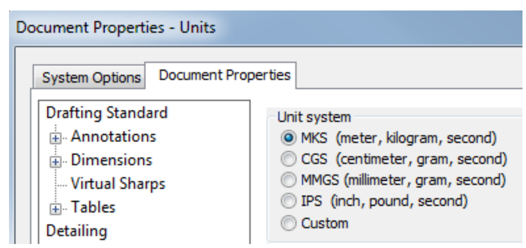
1. In the main *FLUENT* menu, File-Write-Case&Data. OK.
   - Fluent writes two files. The *case* file "…cas.gz" (the grid plus all boundary conditions and other specified parameters), and the *data* file "…dat.gz" (the velocity, turbulence, and pressure fields calculated by the code).
2. OK.
3. File-Exit.

Part 2: Pipe with Elbow

A geometry and mesh like the one shown below will be created in Part 2 of this tutorial. The curved pipe's wall surface will be created in SolidWorks and exported to Pointwise for modification and grid generation.

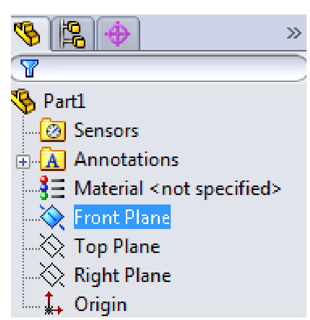


Open new SolidWorks part and set options

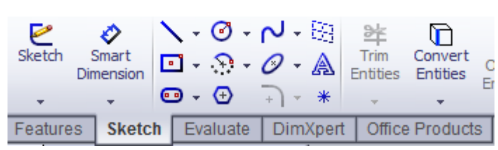


1. This tutorial assumes that SolidWorks is running.
2. File-New.
3. Part. OK.
4. Tools-Options.
5. *Document Properties* tab.
6. Units-MKS (meter, kilogram, second)-OK.
7. View-Toolbars. LMB the buttons in front of "CommandManager" ( ) and "Surfaces" ( ) if they are not already toggled on.

Create the sweep path

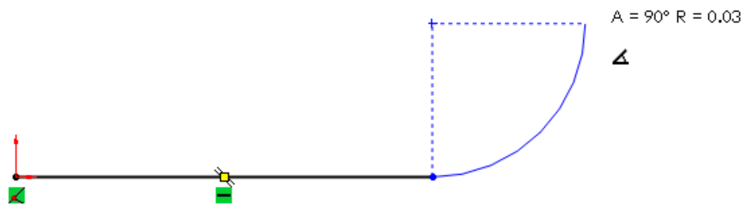


1. First, a horizontal line will be created. Then, a 90° curve will be inserted at one end of the horizontal line. Finally, a vertical line will be connected to the free end of the curve.
2. Front Plane from the *Design Tree*.
3. Sketch from the *Sketch* tab of the *CommandManager*.

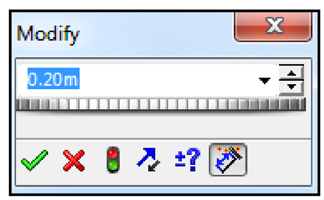


4. Line.
5. LMB on the origin when a appears.
   - In SolidWorks a symbol means that the construction geometry is being constrained to a point.

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 it 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^o (A = 90^o) 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.

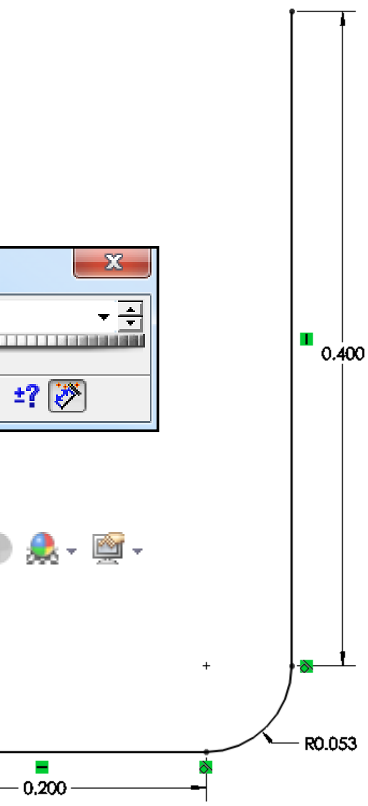


12. <Esc> to unselect the arc tool.
13. Line.
14. LMB the free endpoint of the arc when a constraint appears. This places the first endpoint of a line.
15. Move the cursor up and LMB again when a appears. This places the second endpoint and creates a vertical line.
   - The symbol means that the line 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 .



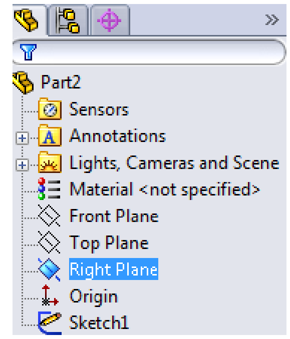
22. Zoom to Fit in the *Heads-Up View Toolbar*.
   - Alternatively, View-Modify-Zoom to Fit.

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>.

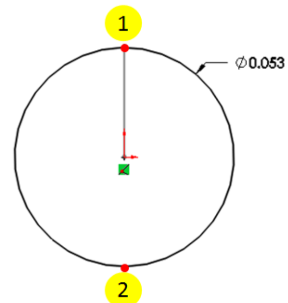


29. Zoom to Fit.
30. The sketch should now look like the one shown here. Lines in a sketch change color from blue to black after they are fully constrained. The sketch is fully defined since all its lines are black.
31. LMB in the upper right corner of the graphics display to exit the sketch.
   - Exit 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 the 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 again 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 current trim option.
15. LMB on the perimeter of the right side of the circle to delete it.
16. LMB on the vertical line to delete it.
    - In this case, the sketch needs to be an open semi-circle to complete a surface sweep.
17. <Esc> to unselect the trim tool.

18. LMB to exit the sketch.
19. View Orientation-Dimetric from the *Heads-Up View Toolbar*.
20. Sketch1 will become the path of the sweep, and Sketch2 will become the profile.

Use a swept surface to create half of the pipe wall

1. Swept Surface from the surfaces toolbar.
   - View-Toolbars-Surfaces to display the surfaces toolbar.
   - Alternatively, Insert-Surface-Sweep…
2. 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*.
3. LMB in the field next to (path icon), then LMB on Sketch1 to assign it as the path.
4. LMB in the upper right of the graphics display to apply the sweep.
5. 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_MinorLoss" 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. Save.

- An IGES file stores data in a universally recognized format so any CAD software can access the data.

5. OK to save all bodies.
6. File-Exit.

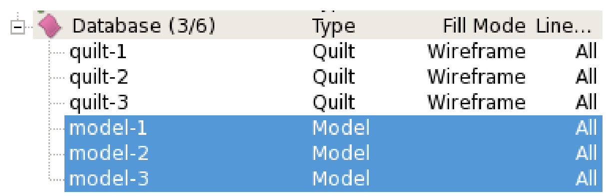
Open Pointwise and import case

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

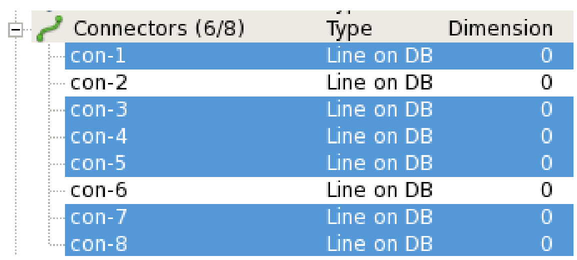
Choose solver

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. ANSYS FLUENT.
4. OK.
5. File-Save.
6. After selecting an appropriate folder in which to save the project, enter "Elbow_Pipe_MinorLoss" for the *File Name*. Save.

Modify geometry and create surface mesh



1. LMB the "+" next to "Database" in the *List* panel.
2. Simultaneously select "model-1", "model-2", and "model-3".
3. Assemble Models.
   - Alternatively, Create-Assemble-Models. Assemble. OK.
4. Simultaneously select "quilt-1", "quilt-2", and "quilt-3".
5. Assemble Quilts.
   - Alternatively, Create-Assemble-Models. Assemble. OK.
6. This reduced the number of databases from six to two ("model-1" and "quilt-1").
7. LMB "quilt-1".

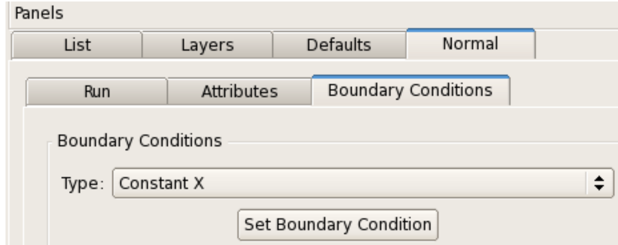


8. Connectors on Database Entities.
   - Alternatively, Create-On Database Entities. OK.
9. LMB the "+" next to "Connectors" in the *List* panel.
10. Simultaneously select all the connectors **except** "con-2" and "con-6".
11. Edit-Join.
12. With the two newly joined connectors still selected, enter "100" in the dimensions box. <Enter>.

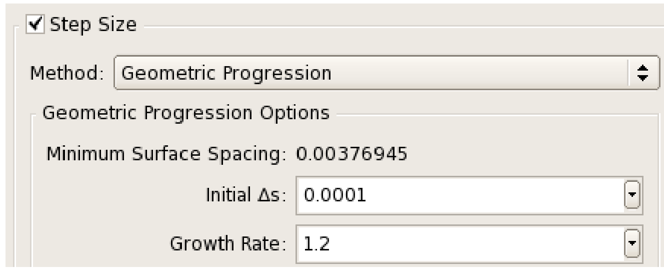
13. Simultaneously select "con-2" and "con-6".
14. Enter "21" in the dimensions box. <Enter>.
15. Rotate the model to get a perspective view of the geometry.
16. LMB "con-2".
17. Edit-Split…
18. Enter "25" for the *Percent of Length*. <Enter>.
19. Enter "75" for the *Percent of Length*. <Enter>.
20. OK. The circular connecter has been separated into three shorter connectors.
21. Simultaneously select all the connectors. There should be a total of six connectors.

22. Assemble Domains.

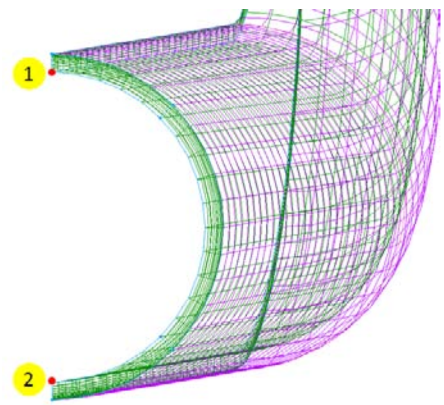
Extrude surface mesh



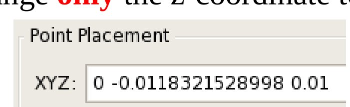
1. LMB the "+" next to "Domains" in the *List* panel.
2. Select "dom-1".
3. Create-Extrude-Normal…
4. *Boundary Conditions* tab in the side panel.
5. Simultaneously select the three split connectors.
6. In the side panel, change *Type* from "Splay" to "Constant X". Set Boundary Conditions.
7. Simultaneously select the two longest connectors.
8. Change *Type* from "Splay" to "Constant Z". Set Boundary Conditions.
9. Select the remaining circular connector.
10. Change *Type* from "Splay" to "Constant Y". Set Boundary Conditions.
11. *Attributes* tab in the side panel.
12. LMB the box next to *Step Size*.
13. Enter "0.0001" for *Initial* Δ*s*.
14. Change the *Growth Rate* to "1.2".
15. LMB the box next to *Orientation*.
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.

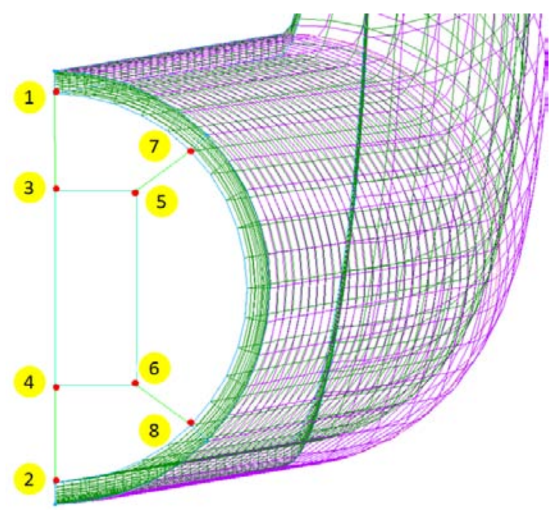


Add aditional connectors

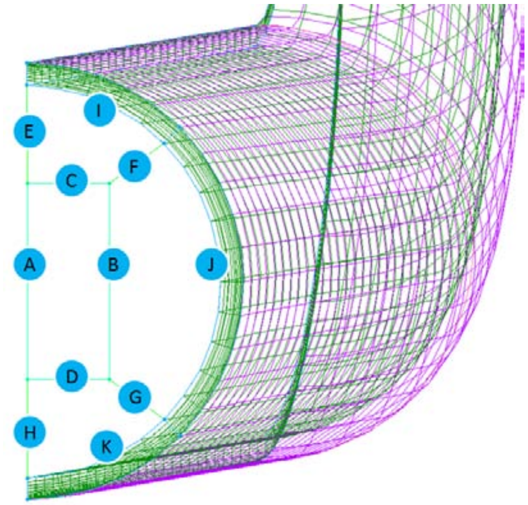


1. Recall View +X from the View toolbar.
2. Zoom in on the end of the pipe perpendicular to the viewing plane.
   ✚ Scroll the MMB to zoom.
3. Notice the 10 layers of cells along the wall of the pipe. The surface mesh has been extruded into a block of cells. There should now be a (1) after *Blocks* in the side panel.
4. 2 Point Curves.
5. Sequentially LMB on Point1 and Point2 to insert a connector between them.
6. OK.
7. Select the newly formed connector ("con-16").
8. Edit-Split…
9. Enter "25" for the *Percent of Length*. <Enter>.
10. Enter "75" for the *Percent of Length*. <Enter>.
11. OK.
12. 2 Point Curves.
13. LMB on Point3.
14. Under *Point Placement*, change **only** the z-coordinate to "0.01" as shown. <Enter>.
15. LMB on Point4.
16. Under *Point Placement*, change **only** the z-coordinate to "0.01". <Enter>.
17. <Enter> again to place another point.





18. LMB on Point5.
19. Connect Point5 and Point7 by sequentially LMB on them.
20. Connect Point6 and Point8 by sequentially LMB on them.
21. OK.



Dimension connectors and create domains

1. Simultaneously select LineA and LineB.
2. Enter “11” in the dimensions box. <Enter>.
3. Simultaneously select LineC and LineD.
4. Enter “6” in the dimensions box. <Enter>.
5. Simultaneously select LineE, LineF, LineG, and LineH.
6. Enter “10” in the dimensions box. <Enter>.
7. Simultaneously select LineB through LineK.
8. Assemble Domains.
9. Simultaneously select LineA through LineD.
10. Assemble Domains.

Extrude domains along pipe length

dom-9	Structured	6x10	45
dom-10	Structured	6x10	45
dom-11	Structured	11x10	90
dom-12	Structured	11x6	50

12. Simultaneously select “dom-9” through “dom-12” from the *List* panel.
13. Create-Extrude-Path…
14. LMB the box next to *Assemble Special*.
15. Delete All Faces.
16. Auto-Assemble.
17. Assemble.
18. Done.
19. List tab.
20. Select “con-7”.
21. Path tab.
22. LMB the box next to *Reverse Path*.
23. Run-OK.

dom-2	Structured	21x11	200
dom-3	Structured	100x11	990
dom-4	Structured	6x11	50
dom-5	Structured	11x11	100
dom-6	Structured	6x11	50
dom-7	Structured	100x11	990
dom-8	Structured	21x100	1,980
dom-9	Structured	6x10	45
dom-10	Structured	6x10	45
dom-11	Structured	11x10	90
dom-12	Structured	11x6	50
dom-13	Structured	6x100	495
dom-14	Structured	11x100	990
dom-15	Structured	6x100	495
dom-16	Structured	10x100	891
dom-17	Structured	6x100	495
dom-18	Structured	11x100	990
dom-19	Structured	6x100	495
dom-20	Structured	10x100	891
dom-21	Structured	21x10	180
dom-22	Structured	11x100	990
dom-26	Structured	11x6	50

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 “Velocity Inlet” from the drop down menu.
5. *List* tab from the side panel.
6. Simultaneously select“dom-2”, “dom-21”, and “dom-26”.
7. *Set BC* tab from the side panel.
8. LMB in the empty box next to the second row. A 3 should appear under the “#” column. This indicates that three domains have been defined as an inlet.
9. New.
10. **LMB** “bc-3” and change the name to “outlet”.
11. In the same row, “**Unspecified**” and choose “Pressure Outlet” from the drop down menu.
12. *List* tab.

13. Select "dom-4", "dom-5", "dom-6", "dom-9", "dom-10", "dom-11", and "dom-12".
14. *Set BC* tab.
15. LMB in the empty box next to the third row. A 7 should appear under the "#" column.
16. New.
17. **LMB** "bc-4" and change the name to "wall".
18. In the same row, "**Unspecified**" and choose "Wall" from the drop down menu.
19. *List* tab.
20. Select "dom-1".
21. *Set BC* tab.
22. LMB in the empty box next to the fourth row. A 1 should appear under the "#" column.
23. New.
24. **LMB** "bc-5" and change the name to "symmetry".
25. In the same row, "**Unspecified**" and choose "Symmetry" from the drop down menu.
26. *List* tab.
27. Simultaneously select "dom-3", "dom-7", "dom-16", "dom-20", and "dom-22".
28. *Set BC* tab.
29. LMB in the empty box next to the fifth row. A 5 should appear under the "#" column.
30. The boundary conditions should now be set as indicated to the right.
31. Close.

dom-4	Structured	6x11	50
dom-5	Structured	11x11	100
dom-6	Structured	6x11	50
dom-7	Structured	100x11	990
dom-8	Structured	21x100	1,980
dom-9	Structured	6x10	45
dom-10	Structured	6x10	45
dom-11	Structured	11x10	90
dom-12	Structured	11x6	50

dom-3	Structured	100x11	990
dom-4	Structured	6x11	50
dom-5	Structured	11x11	100
dom-6	Structured	6x11	50
dom-7	Structured	100x11	990
dom-8	Structured	21x100	1,980
dom-9	Structured	6x10	45
dom-10	Structured	6x10	45
dom-11	Structured	11x10	90
dom-12	Structured	11x6	50
dom-13	Structured	6x100	495
dom-14	Structured	11x100	990
dom-15	Structured	6x100	495
dom-16	Structured	10x100	891
dom-17	Structured	6x100	495
dom-18	Structured	11x100	990
dom-19	Structured	6x100	495
dom-20	Structured	10x100	891
dom-21	Structured	21x10	180
dom-22	Structured	11x100	990

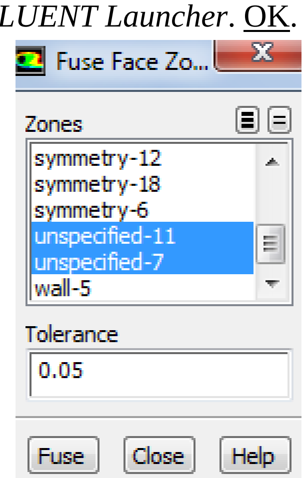
Set	#	Name	Type	ID
☐	4	Unspecified	Unspecified	1
☐	3	inlet	Velocity Inlet	2
☐	7	outlet	Pressure Outlet	3
☐	1	wall	Wall	4
☑	5	symmetry	Symmetry	5

Export CAE to FLUENT case

1. LMB the "+" next to *Blocks* in the side panel.
2. Simultaneously select all the blocks.
3. File-Export-CAE…
4. After selecting an appropriate folder in which to export the project, enter "Elbow_Pipe_MinorLoss" for the *File Name*. Save.
5. File-Save.
6. File-Exit Pointwise.

Launch FLUENT and open case

1. Locate and launch FLUENT. Verify that "Double Precision" is selected in *FLUENT Launcher*. 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*.
3. File-Read-Case…
4. Locate "*Elbow_Pipe_MinorLoss.cas*", and OK.
5. FLUENT reads and builds the mesh that was created in Pointwise.

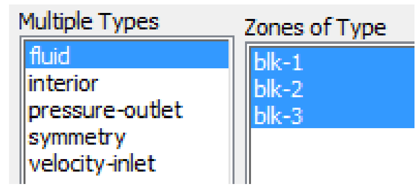


Merge fluid zones and surfaces

1. The mesh imported from Pointwise consists of three fluid zones (*blocks* in Pointwise terminology) because the mesh was created in sections. Multiple zones result in numerous surfaces with identical boundary conditions. Post-

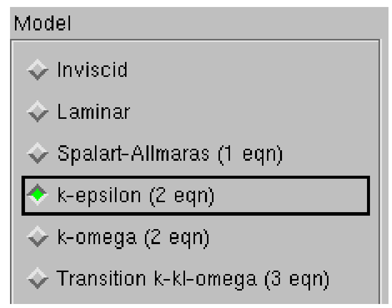
processing is simplified by combining similar entities.

2. Mesh-Fuse…
3. Select all three of the “unspecified” zones by LMB on each of them.
4. Fuse-Close.
   - Fusing combines common nodes and assembles a single surface. In this case, internal surfaces between zones were combined and designated as an “interior” boundary condition.
5. Mesh-Merge…



6. LMB “fluid”, then Merge in the *Merge Zones* window that opens.
   - ✚ The fluid zones must be combined before merging any other type.
7. LMB “interior”, then Merge.
8. LMB “pressure-outlet”, then Merge.
9. LMB “symmetry”, then Merge.
10. LMB “velocity-inlet”, then Merge.
11. Close the *Merge Zones* window.

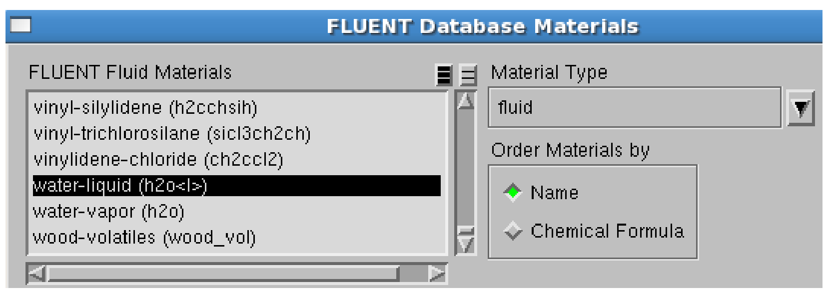
Set solver model



1. The default model is laminar flow, but we want to model flow through the pipe as turbulent flow.
2. In the main *FLUENT* menu, Define-Models-Viscous-Laminar-Edit.
3. 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

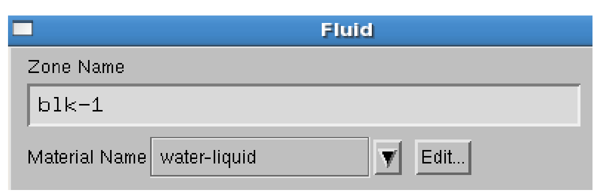
1. The default fluid is air, but we want to define the fluid as water.
2. In the main *FLUENT* menu, Define-Materials-Create/Edit-FLUENT Database.



3. Select water-liquid from the list of *FLUENT Fluid Materials*.
4. Copy.
5. Close *Fluent Database Materials*.
6. 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

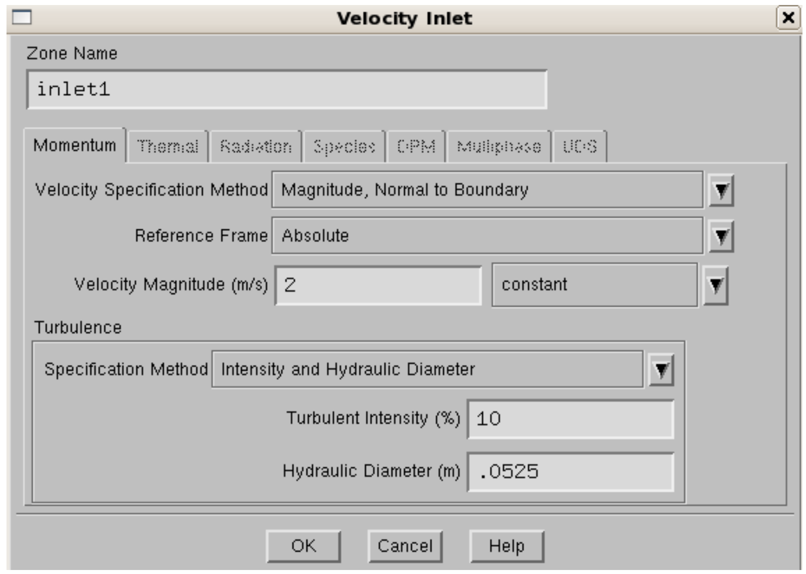
1. In the *FLUENT* menu, Define-Cell Zone Conditions.
2. There should be one zone (“blk-1”).
3. Select “blk-1”.



4. Make sure *Type* is set as fluid.
5. Edit.
6. Select water-liquid instead of “air” as the material.
7. OK.

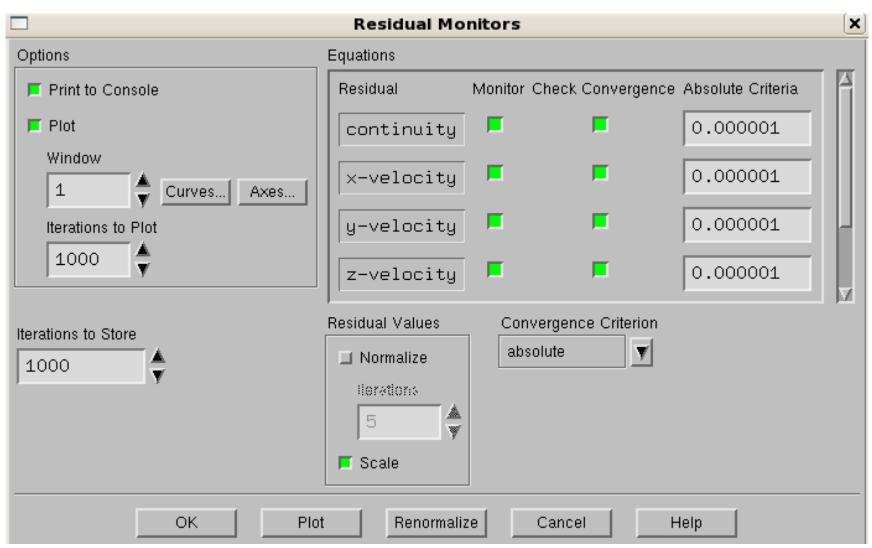
Set up boundary conditions

1. Now the boundary conditions need to be specified. In Pointwise, the boundary conditions were named (inlet, outlet, etc.) but actual values for inlet velocity were never defined. This must be done in FLUENT.
2. In the main *FLUENT* menu, Define-Boundary Conditions.
3. The default boundary conditions for the walls (pipe wall), the symmetry planes, and the outlet (pressure-outlet) are okay, so nothing needs to be done to them.
4. Select inlet-13.
5. Edit.
6. Change *Zone Name* to "inlet1".
7. In the *Velocity Inlet* window, change *Velocity Magnitude* to "2" m/s.
8. Change the turbulence specification method to Intensity and Hydraulic Diameter.
9. Set the turbulence intensity as "10" % and the hydraulic diameter as the pipe diameter, which we had specified previously as "0.0525 m".
10. OK.



Set up some more parameters and initialize

1. Now the convergence criteria need to be set. In the main *FLUENT* menu, Solve-Monitors-Residuals…-Edit.
2. 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.



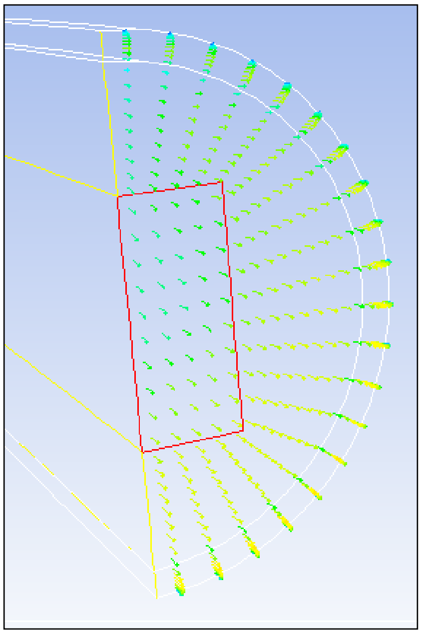
3. There are six differential equations to be solved in a 3-D incompressible k-ε turbulence model flow problem, and so there are six residuals to be monitored for convergence: continuity, x-, y-, and z-velocity, k (turbulent kinetic energy), and ε (turbulent dissipation rate). The default convergence 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).

4. OK.
5. In the main *FLUENT* menu, Solve-Initialization. The default initial values are good enough for this problem. Initialize.

Iterate towards a solution

1. In the main *FLUENT* menu, Solve-Run Calculation.
2. Change *Number of Iterations* to 500, and change *Reporting Interval* to 10.
3. 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.
   - 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.



4. OK when the calculations are completed or converged.
5. 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].
6. In the *Vectors* window that opens, under *Surfaces*, select only "outlet-14", "oulet-17", and "outlet-8".
7. Display.
8. Fit to Window button to better see the velocity vectors.
9. In *Vectors*, Draw Mesh.
10. In the *Mesh Display* window that opens, select Outline as the *Edge Type*.
11. Display. This will draw an outline of the geometry.
12. Close *Mesh Display*.
13. In *Vectors*, change *Scale* to ".5".
14. Display to see the vectors at the outlet, this time with the geometry outline shown.
15. Zoom and rotate on the velocity field to see it in more detail. It should look similar to the velocity field to the right. Close the *Vectors* window.
   - To zoom in, drag a box with the MMB from the lower left to upper right.
   - To zoom out, drag a box with the MMB from the upper right to the lower left.
   - To rotate, hold LMB while moving the cursor.
16. Close the *Vectors* window.

Calculate the pressure drop

1. On the left side of the main *FLUENT* window under *Results*, Reports-Surface Integrals-Set Up.
2. Set *Report Type* as Area-Weighted Average.
3. *Field Variable* should be Pressure and Static Pressure by default, so leave those settings alone.
4. Select all the inlet and outlet surfaces under *Surfaces*. There are three inlet and three outlet surfaces.
5. Compute. The results are written to the main *FLUENT* window.
6. 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 the outlet boundary condition was specified (by default) as zero gage pressure.
7. Close the *Surface Integrals* window.

Save your calculations and exit FLUENT

1. In the main *FLUENT* menu, File-Write-Case&Data. OK.
   - Fluent writes two files. 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).
2. OK to overwrite existing file.
3. File-Exit.