Created in COMSOL Multiphysics 5.6

Flow Through a Pipe Elbow

Introduction

In an engineering context, pipes and fittings are not normally the subject for turbulence modeling. There is a vast amount of tabulated data and correlations that can be used in simplified approaches such as those used in the Pipe Flow Module. But now more and more detailed studies are required, for example, to generate additional correlation data or to understand phenomena caused by the flow. One such example is found in Ref. 1, which models the flow in a 90° pipe elbow in order to understand corrosion and erosion.

This example simulates the same pipe flow as that studied in Ref. 1 using the k-ω turbulence model. The result is compared to experimental correlations.

Model Definition

The model geometry is shown in Figure 1. It includes a 90° pipe elbow of constant diameter, D, equal to 35.5 mm, and coil radius, R_c, equal to 50 mm. The straight inlet and outlet pipe sections are respectively 70 mm and 350 mm long. Only half the pipe is modeled because the xy-plane is a symmetry plane.

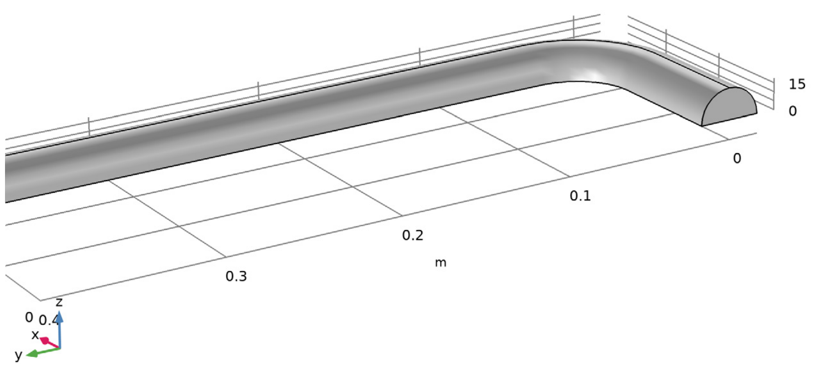


Figure 1: Pipe elbow geometry.

The flow conditions are typical to those that can be found in the piping system of a nuclear power plant. The working fluid is water at temperature T=90 °C and the absolute outlet pressure is 20 bar. At this state, the density, ρ, is $965.35\ \mathrm{kg/m^3}$ and the dynamic viscosity, μ, is 3.145×10^{-4} Pa·s. The water is approximated as incompressible.

The flow at the inlet is a fully developed turbulent flow with an average inlet velocity of 5 m/s.

Modeling Considerations

Ref. 1 shows separation after the bend which can be due to the fact that the k-ε model has a tendency to perform poorly for flows involving strong pressure gradients, separation and strong streamline curvature (Ref. 2). Therefore the k-ω model is selected model over the k-ε model.

The mesh is constructed to approximately match that used Ref. 1, with the exception that the mesh in the bend itself is unstructured.

Results and Discussion

The resulting streamline pattern is shown in Figure 2. There is a separation zone after the bend which is consistent with the results in Ref. 1. Further downstream, two counterrotating vortices form, caused by the centripetal force (Ref. 3). Observe that only one of the vortices is visible in Figure 2 since the other is located on the other side of the symmetry plane.

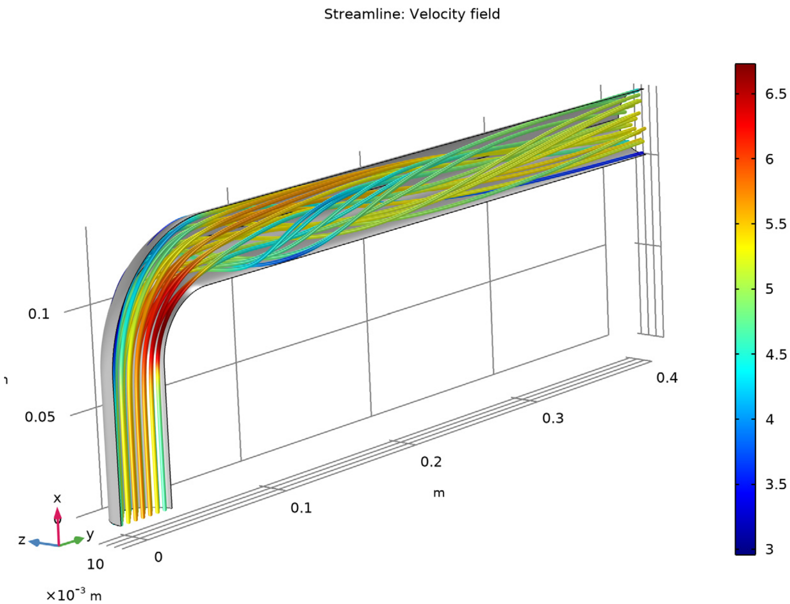


Figure 2: Streamlines colored by the velocity magnitude.

A quantitative assessment of the results is given by comparing the so-called diametrical pressure coefficient, c_k, to a engineering correlation as is done in Ref. 1. The definition of c_k is

$$c_k = \frac{p_o - p_i}{\frac{1}{2}\rho U_{\mathrm{avg}}^2} \tag{1}$$

where p_o and p_i are the pressures where the outer and inner radii of the bend intersect the symmetry plane respectively. The diametrical pressure coefficient is commonly measured at half the bend angle, in this case at 45°. Figure 3 shows a surface plot of the pressure in a plane at 45°. p_o and p_i are the maximum and minimum values of the pressure in Figure 3 respectively which gives $p_o - p_i \approx 1.99 \times 10^4$ Pa. Since U_{avg} is 5 m/s, Equation 1 evaluates to 1.6.

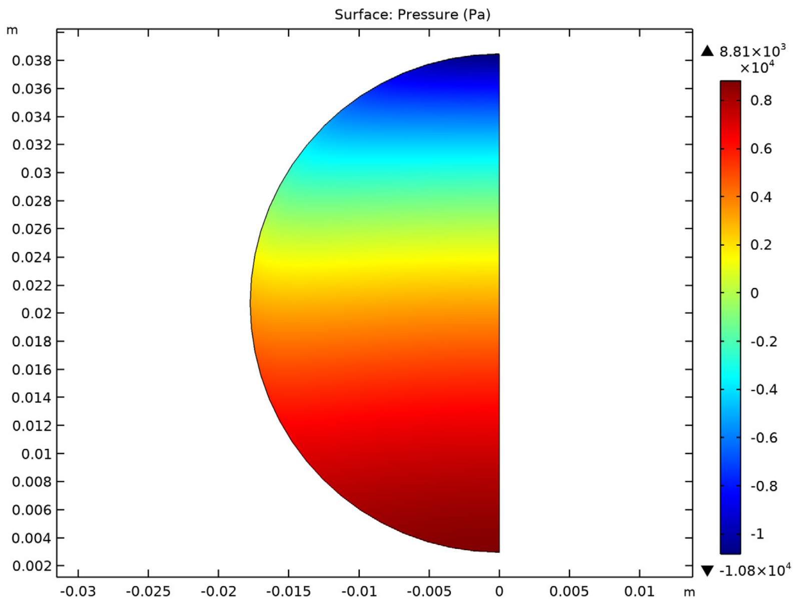


Figure 3: Pressure in a plane at $45°$.

Ref. 1 compares Equation 1 to the following correlation based on curvatures where $R_c/D > 2$:

$$c_k = \frac{2D}{R_c} \tag{2}$$

For the pipe model studied here this evaluates to 1.42. The agreement with the value 1.6 computed above is reasonably good considering that R_c/D in this case is 1.41.

A more common engineering measure is the friction factor, f_f, which is related to the head loss, h_L, via

$$h_L = 2f_f\frac{L}{D}\frac{U_{\text{avg}}^2}{g} \tag{3}$$

where L is the length of the pipe segment and g is the gravity constant. In the case of a bend, L is the distance along the centerline. The head loss is related to the pressure drop over the pipe segment, Δp, via

$$h_L = \frac{\Delta p}{g\rho} \tag{4}$$

The friction factor can hence be calculated as:

$$f_f = \frac{\Delta p}{2\rho U_{\mathrm{avg}}^2}\frac{D}{L} \tag{5}$$

Equation 5 cannot be evaluated all the way from the start to the end of the bend since there is an exit effect (see Ref. 4 for more details). It is instead evaluated from the start of the bend, 0°, to half way through the bend, that is 45°. This gives $L = \pi R_c/4$ and Δp evaluates to approximately 400 Pa, which results in a friction factor:

$$f_f = \frac{400}{\rho U_{\mathrm{avg}}^2}\frac{2D}{\pi R_c} = \frac{400}{\rho 25}\frac{1.42}{\pi} = 7.49 \cdot 10^{-3}$$

Ref. 4 gives the following correlation for the friction factor through a curved pipe

$$f_f = \frac{0.079}{\mathrm{Re}^{0.25}} + \frac{0.0073}{\sqrt{D_c/D}} \tag{6}$$

where $D_c = 2{\cdot}R_c$ is the coil diameter. In this case, Equation 6 gives $f_f = 7.26 \times 10^{-3}$. The difference of 3% is well within the rage of the accuracy of Equation 6 which is 15%.

References

1. G.F. Homicz, "Computational Fluid Dynamic Simulations of Pipe Elbow Flow," *SAND REPORT*, SAND2004-3467, Sandia National Laboratories, 2004.

2. F. Menter, "Zonal Two Equation *k*-ω Turbulence Models for Aerodynamic Flows," AIAA Paper #93-2906, *24th Fluid Dynamics Conference*, July 1993.

3. M.J. Pattison, "Secondary Flows," *Thermopedia*,DOI: 10.1615/AtoZ.s.secondary_flows, http://www.thermopedia.com/content/1113/?tid=104&sn=1420

4. R.H. Perry and D.W. Green, *Perry's Chemical Engineers' Handbook*, 7th ed., McGraw-Hill, 1997.

Application Library path: `CFD_Module/Verification_Examples/pipe_elbow`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Fluid Flow>Single-Phase Flow>Turbulent Flow>Turbulent Flow, k-ω (spf)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
D	35.5[mm]	0.0355 m	Pipe diameter
Lin	70[mm]	0.07 m	Inlet length
Lout	350[mm]	0.35 m	Outlet length
Rc	50[mm]	0.05 m	Coil radius
rhof	965.35[kg/m^3]	965.35 kg/m³	Density
muf	3.145e-4[Pa*s]	3.145E-4 Pa·s	Dynamic viscosity
Uavg	5[m/s]	5 m/s	Average velocity

GEOMETRY 1

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **yz-plane**.

4 In the **x-coordinate** text field, type `Lin`.

5 Click **Show Work Plane**.

Work Plane 1 (wp1)>Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `D/2`.

Work Plane 1 (wp1)>Square 1 (sq1)

1 In the **Work Plane** toolbar, click **Square**.

2 In the **Settings** window for **Square**, locate the **Size** section.

3 In the **Side length** text field, type `D`.

4 Locate the **Position** section. In the **xw** text field, type `-D/2`.

5 In the **yw** text field, type `-D`.

Work Plane 1 (wp1)>Difference 1 (dif1)

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Click the **Zoom Extents** button in the **Graphics** toolbar.

3 Select the object **c1** only.

4 In the **Settings** window for **Difference**, locate the **Difference** section.

5 Find the **Objects to subtract** subsection. Select the **Activate Selection** toggle button.

6 Select the object **sq1** only.

Work Plane 1 (wp1)

1 In the **Model Builder** window, click **Work Plane 1 (wp1)**.

2 In the **Settings** window for **Work Plane**, click **Build Selected**.

Revolve 1 (rev1)

1 In the **Geometry** toolbar, click **Revolve**.

2 In the **Settings** window for **Revolve**, locate the **Revolution Angles** section.

3 Click the **Angles** button.

4 In the **End angle** text field, type `90`.

5 Select the object **wp1** only.

6 Locate the **General** section. From the **Input object handling** list, choose **Keep**.

7 Locate the **Revolution Axis** section. Find the **Point on the revolution axis** subsection. In the **xw** text field, type `Rc`.

8 In the **yw** text field, type `Lin`.

9 Click **Build Selected**.

10 Click the **Zoom Extents** button in the **Graphics** toolbar.

Extrude 1 (ext1)

1 In the **Geometry** toolbar, click **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (m)
`Lin`

4 Select the **Reverse direction** check box.

Work Plane 1 (wp1)

Right-click **Work Plane 1 (wp1)** and choose **Copy**.

Work Plane 2 (wp2)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Paste Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **xz-plane**.

4 In the **y-coordinate** text field, type `Rc`.

5 In the **Model Builder** window, expand the **Work Plane 2 (wp2)** node.

Work Plane 2 (wp2)>Circle 1 (c1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Geometry 1>Work Plane 2 (wp2)>Plane Geometry** node, then click **Circle 1 (c1)**.

2 In the **Settings** window for **Circle**, locate the **Position** section.

3 In the **xw** text field, type `Lin+Rc`.

Work Plane 2 (wp2)>Square 1 (sq1)

1 In the **Model Builder** window, click **Square 1 (sq1)**.

2 In the **Settings** window for **Square**, locate the **Position** section.

3 In the **xw** text field, type `-D/2+Lin+Rc`.

4 In the **Model Builder** window, click **Geometry 1**.

Extrude 2 (ext2)

1 In the **Geometry** toolbar, click **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 Select the **Reverse direction** check box.

4 In the table, enter the following settings:

Distances (m)
Lout

5 In the **Geometry** toolbar, click **Build All**.

Mesh Control Faces 1 (mcf1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Mesh Control Faces**.

2 In the **Settings** window for **Mesh Control Faces**, locate the **Input** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `5 11` in the **Selection** text field.

5 Click **OK**.

6 In the **Geometry** toolbar, click **Build All**.

7 In the **Model Builder** window, collapse the **Geometry 1** node.

MATERIALS

Water

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	rhof	kg/m^3	Basic
Dynamic viscosity	mu	muf	Pa·s	Basic

4 Right-click **Material 1 (mat1)** and choose **Rename**.

5 In the **Rename Material** dialog box, type `Water` in the **New label** text field.

6 Click **OK**.

TURBULENT FLOW, K-ω (SPF)

Inlet 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Turbulent Flow, k-ω (spf)** and choose **Inlet**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.

4 From the list, choose **Fully developed flow**.

5 Locate the **Fully Developed Flow** section. In the U_{av} text field, type `Uavg`.

Outlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 In the **Settings** window for **Outlet**, locate the **Boundary Condition** section.

3 From the list, choose **Fully developed flow**.

4 Select Boundary 5 only.

Symmetry 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 Select Boundary 3 only.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Extra fine**.

Size

1 Right-click **Component 1 (comp1)>Mesh 1** and choose **Edit Physics-Induced Sequence**.

2 In the **Settings** window for **Size**, click to expand the **Element Size Parameters** section.

3 In the **Maximum element size** text field, type `D/20`.

Size 1

In the **Model Builder** window, right-click **Size 1** and choose **Disable**.

Free Tetrahedral 1

In the **Model Builder** window, right-click **Free Tetrahedral 1** and choose **Disable**.

Swept 1

In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 1 only.

5 Locate the **Distribution** section. In the **Number of elements** text field, type 40.

Distribution 2

1 In the **Model Builder** window, right-click **Swept 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 2 only.

5 Locate the **Distribution** section. In the **Number of elements** text field, type 50.

Distribution 3

1 Right-click **Swept 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 3 only.

5 Locate the **Distribution** section. From the **Distribution type** list, choose **Predefined**.

6 In the **Number of elements** text field, type 70.

7 In the **Element ratio** text field, type 10.

8 From the **Growth formula** list, choose **Geometric sequence**.

Boundary Layers 1

1 In the **Model Builder** window, click **Boundary Layers 1**.

2 In the **Settings** window for **Boundary Layers**, click to expand the **Transition** section.

3 Clear the **Smooth transition to interior mesh** check box.

4 Click **Build All**.

STUDY 1

In the **Home** toolbar, click **Compute**.

RESULTS

Deactivate the automatic update of plots when working with large 3D models.

1 In the **Model Builder** window, click **Results**.

2 In the **Settings** window for **Results**, locate the **Update of Results** section.

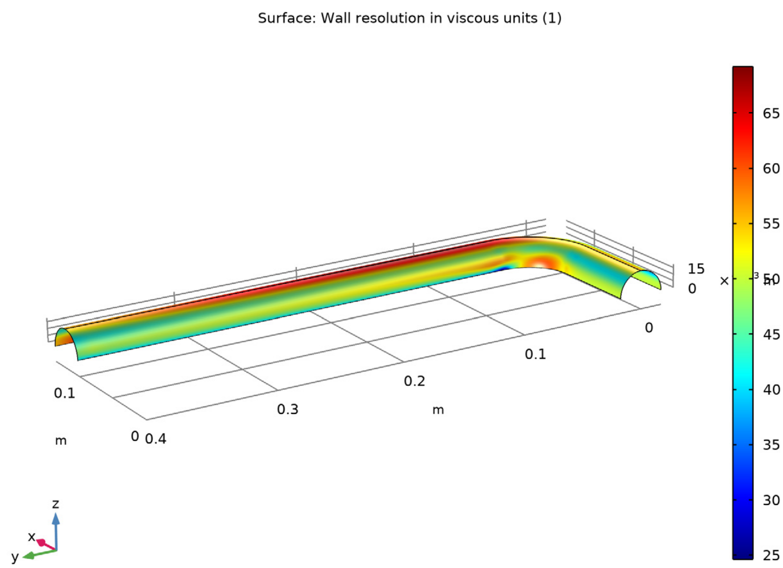
3 Select the **Only plot when requested** check box.

Check that the **Wall Resolution** is sufficient.

Wall Resolution (spf)

1 In the **Model Builder** window, under **Results** click **Wall Resolution (spf)**.

2 In the **Wall Resolution (spf)** toolbar, click **Plot**.



The wall lift-off is well below 100 viscous units everywhere and the flow can therefore be regarded as well-resolved at the walls.

Proceed to reproduce Figure 2.

Pressure

1 In the **Model Builder** window, expand the **Results>Pressure (spf)** node.

2 Right-click **Pressure** and choose **Disable**.

Streamline 1

1 In the **Model Builder** window, right-click **Pressure (spf)** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

4 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **On selected boundaries**.

5 Locate the **Selection** section. Click **Paste Selection**.

6 In the **Paste Selection** dialog box, type `1` in the **Selection** text field.

7 Click **OK**.

8 In the **Settings** window for **Streamline**, locate the **Coloring and Style** section.

9 Find the **Line style** subsection. From the **Type** list, choose **Tube**.

10 Select the **Radius scale factor** check box.

11 In the associated text field, type `0.001`.

Color Expression 1

1 Right-click **Streamline 1** and choose **Color Expression**.

2 In the **Pressure (spf)** toolbar, click **Plot**.

Velocity Streamlines

1 In the **Model Builder** window, right-click **Pressure (spf)** and choose **Rename**.

2 In the **Rename 3D Plot Group** dialog box, type `Velocity Streamlines` in the **New label** text field.

3 Click **OK**.

Execute the following steps to reproduce Figure 3.

Cut Plane 1

1 In the **Results** toolbar, click **Cut Plane**.

2 In the **Settings** window for **Cut Plane**, locate the **Plane Data** section.

3 From the **Plane type** list, choose **General**.

4 From the **Plane entry method** list, choose **Point and normal**.

5 Find the **Point** subsection. In the **x** text field, type `Lin`.

6 Find the **Normal** subsection. In the **x** text field, type `1`.

7 In the **z** text field, type `0`.

Cut Plane 2

1 Right-click **Cut Plane 1** and choose **Duplicate**.

2 In the **Settings** window for **Cut Plane**, locate the **Plane Data** section.

3 Find the **Point** subsection. In the **x** text field, type `Lin+Rc`.

4 Find the **Normal** subsection. In the **y** text field, type `1`.

2D Plot Group 4

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Cut Plane 2**.

4 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Surface 1

1 Right-click **2D Plot Group 4** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `p`.

4 In the **2D Plot Group 4** toolbar, click **Plot**.

Cut Plane Pressure Distribution

1 In the **Model Builder** window, right-click **2D Plot Group 4** and choose **Rename**.

2 In the **Rename 2D Plot Group** dialog box, type `Cut Plane Pressure Distribution` in the **New label** text field.

3 Click **OK**.

The following steps provides the pressure loss Δp in Equation 4.

Surface Average 1

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Average>Surface Average**.

2 In the **Settings** window for **Surface Average**, locate the **Data** section.

3 From the **Dataset** list, choose **Cut Plane 1**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
p	Pa	Pressure

5 Click **Evaluate**.

6 Locate the **Data** section. From the **Dataset** list, choose **Cut Plane 2**.

7 Click **Evaluate**.

The pressure drop can now computed by subtracting the mean pressure, evaluated at the cut plane in the middle of bend, from the pressure just before the bend (boundary 5).