

# 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 secondary flow pattern. One such example is found in Ref. 1, which models the flow in a 90° pipe elbow in order to achieve a preliminary, before making simulations with particles, understanding of corrosion and erosion processes.

This example simulates the same pipe flow as that studied in Ref. 1 using the k- $\omega$  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 70 mm and 350 mm long, respectively. 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,  $\rho$ , is 965.35 kg/m<sup>3</sup> and the dynamic viscosity,  $\mu$ , is  $3.145 \times 10^{-4}$  Pa·s. The water is approximated as an incompressible fluid.

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

# Modeling Considerations

In Ref. 1 the separation bubble after the bend is predicted and analyzed. However, it is known that the k- $\varepsilon$  model has a tendency to perform poorly for flows involving strong pressure gradients, separation, and strong streamline curvature (Ref. 2). Therefore, the k- $\omega$  model is the preferred turbulence model, compared to the k- $\varepsilon$  model.

The mesh is constructed to approximately match that used Ref. 1.

# 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

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

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 is 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

#### 4 | FLOW THROUGH A PIPE ELBOW



respectively which gives  $p_o - p_i \approx 1.96 \times 10^4$  Pa. Since  $U_{avg}$  is 5 m/s, Equation 1 evaluates to 1.62.

Figure 3: Pressure in a plane at 45°.

As indicated in Ref. 1, Equation 1can be compared to the following correlation based on mild curvatures with  $R_c/D > 2$ :

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

For the pipe model studied here Equation 2 evaluates to 1.42. The agreement with the value 1.62 computed above is reasonably good considering that our bend is rather sharp with  $R_c/D$  equal to 1.41.

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

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

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,  $\Delta p$ , via

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

The friction factor can hence be calculated as:

$$f = \frac{\Delta p}{2\rho U_{\text{avg}}^2 L} \tag{5}$$

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  $\Delta p$ evaluates to approximately 405 Pa, which results in a friction factor:

$$f = \frac{400}{\rho U_{\text{avg}}^2 \pi R_c} = \frac{400}{\rho 25} \frac{1.42}{\pi} = 7.58 \cdot 10^{-3}$$

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

$$f = \frac{0.079}{\text{Re}^{0.25}} + \frac{0.0073}{\sqrt{D_c/D}}$$
(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 4% is well within the range 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- $\omega$  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

- I In the Model Wizard window, click 间 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  $\bigcirc$  Study.
- 5 In the Select Study tree, select General Studies>Stationary.
- 6 Click **M** Done.

## **GLOBAL DEFINITIONS**

#### Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 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 I

## Work Plane I (wp1)

- I 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 I (wp1)>Circle I (c1)

- I 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 I (wp1)>Square I (sq1)

- I 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 I (wp1)>Difference I (dif1)

- I In the Work Plane toolbar, click i Booleans and Partitions and choose Difference.
- **2** Click the  $4 \rightarrow$  **Zoom Extents** button in the **Graphics** toolbar.
- **3** Select the object **cl** only.
- 4 In the Settings window for Difference, locate the Difference section.
- **5** Find the **Objects to subtract** subsection. Click to select the **Selection** toggle button.
- 6 Select the object sql only.

## Work Plane I (wp1)

- I In the Model Builder window, under Component I (compl)>Geometry I click Work Plane I (wpl).
- 2 In the Settings window for Work Plane, click 🖷 Build Selected.

## Revolve I (rev1)

I In the **Geometry** toolbar, click  $\leftarrow$  **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 wpl 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 틤 Build Selected.
- **10** Click the  $\longleftrightarrow$  **Zoom Extents** button in the **Graphics** toolbar.

Extrude I (extI)

- I In the **Geometry** toolbar, click **Sector** 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 I (wp1)

Right-click Work Plane I (wpI) and choose Copy.

Work Plane 2 (wp2)

- I In the Model Builder window, right-click Geometry I 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)

- In the Model Builder window, expand the Component I (compl)>Geometry I>
  Work Plane 2 (wp2)>Plane Geometry node, then click Circle I (cl).
- 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)

- I In the Model Builder window, click Square I (sql).
- 2 In the Settings window for Square, locate the Position section.
- **3** In the **xw** text field, type -D/2+Lin+Rc.

## Extrude 2 (ext2)

- I In the Model Builder window, right-click Geometry I and choose 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 (mcfl)

- I 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 I node.

## MATERIALS

Water

- I In the Model Builder window, under Component I (compl) 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/m³	Basic
Dynamic viscosity	mu	muf	Pa∙s	Basic

4 In the Label text field, type Water.

#### TURBULENT FLOW, $K-\omega$ (SPF)

#### Inlet 1

- I In the Model Builder window, under Component I (compl) right-click Turbulent Flow, k- $\omega$  (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 I

- I 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 I

- I In the Physics toolbar, click 🔚 Boundaries and choose Symmetry.
- **2** Select Boundary **3** only.

## MESH I

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Physics-Controlled Mesh section.
- 3 From the Element size list, choose Extra fine.

#### Size

- I Right-click Component I (comp1)>Mesh I 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.
- 4 In the Minimum element size text field, type 8.03E-5.

## Size I

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

#### Free Tetrahedral I

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

## Corner Refinement I

In the Model Builder window, right-click Corner Refinement I and choose Disable.

#### Boundary Layers 1

- I In the Model Builder window, click Boundary Layers I.
- 2 In the Settings window for Boundary Layers, locate the Geometric Entity Selection section.
- **3** From the Geometric entity level list, choose Boundary.
- **4** Select Boundary 1 only.

#### Boundary Layer Properties 1

- I In the Model Builder window, expand the Boundary Layers I node, then click Boundary Layer Properties I.
- **2** Select Edges 1 and 4 only.
- 3 In the Model Builder window, click Boundary Layer Properties I.
- 4 In the Settings window for Boundary Layer Properties, locate the Layers section.
- 5 In the Number of layers text field, type 6.
- 6 In the Thickness adjustment factor text field, type 2.

#### Swept I

- I In the Mesh toolbar, click A Swept.
- 2 In the Settings window for Swept, click to expand the Control Entities section.
- **3** Clear the **Smooth across removed control entities** check box.

## Distribution I

- I Right-click Swept I and choose Distribution.
- 2 In the Settings window for Distribution, locate the Domain Selection section.
- 3 Click K Clear Selection.
- **4** Select Domain 1 only.
- 5 Locate the Distribution section. In the Number of elements text field, type 40.

## Distribution 2

- I In the Model Builder window, right-click Swept I and choose Distribution.
- 2 In the Settings window for Distribution, locate the Domain Selection section.
- 3 Click K Clear Selection.
- **4** Select Domain 2 only.
- **5** Locate the **Distribution** section. In the **Number of elements** text field, type **50**.

#### Distribution 3

I Right-click Swept I 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 rate list, choose Exponential.

## Free Triangular 1

- I In the Mesh toolbar, click  $\triangle$  Boundary and choose Free Triangular.
- 2 Drag and drop below Free Tetrahedral I.
- **3** Select Boundary 1 only.
- **4** In the Settings window for Free Triangular, click 📗 Build All.

## STUDY I

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

## RESULTS

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

- I 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)

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

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

Surface: Wall resolution in viscous units (1)



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.

#### Surface

- I In the Model Builder window, expand the Results>Pressure (spf) node.
- 2 Right-click Surface and choose Disable.

## Streamline 1

- I 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 I/Solution I (soll).
- 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.

**IO** Select the **Radius scale factor** check box. In the associated text field, type 0.001.

## Color Expression 1

- I Right-click Streamline I and choose Color Expression.
- 2 In the Pressure (spf) toolbar, click **I** Plot.

## **Velocity Streamlines**

- I In the Model Builder window, under Results 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

- I 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 vector**.
- 5 Find the **Point** subsection. In the **x** text field, type Lin.
- 6 Find the Normal vector subsection. In the x text field, type 1.
- **7** In the **z** text field, type **0**.

## Cut Plane 2

- I Right-click Cut Plane I 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 vector subsection. In the y text field, type 1.

#### Average 1

- I In the **Results** toolbar, click **More Datasets** and choose **Evaluation>Average**.
- 2 In the Settings window for Average, locate the Data section.
- 3 From the Dataset list, choose Cut Plane I.

#### Average 2

- I In the **Results** toolbar, click **More Datasets** and choose **Evaluation>Average**.
- 2 In the Settings window for Average, locate the Data section.

3 From the Dataset list, choose Cut Plane 2.

Join I

- I In the **Results** toolbar, click **More Datasets** and choose **Join**.
- 2 In the Settings window for Join, locate the Data I section.
- 3 From the Data list, choose Average I.
- 4 Locate the Data 2 section. From the Data list, choose Average 2.

Evaluation Group 1

In the **Results** toolbar, click **Figure Evaluation Group**.

Pressure drop

I Right-click Evaluation Group I and choose Global Evaluation.

The following steps provides the pressure loss  $\Delta p$  in Equation 4.

- 2 In the Settings window for Global Evaluation, type Pressure drop in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose Join I.
- **4** Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
р	1	Pressure drop

5 In the Evaluation Group I toolbar, click **=** Evaluate.

The pressure drop now is computed by subtracting the mean pressure evaluated at the cut plane just before the bend, from the mean pressure evaluated at the cut plane in the middle of the bend.

## Calculated friction factor

- I In the Model Builder window, right-click Evaluation Group I and choose Global Evaluation.
- **2** In the **Settings** window for **Global Evaluation**, type **Calculated** friction factor in the **Label** text field.
- 3 Locate the Data section. From the Dataset list, choose Join I.
- **4** Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
(D/(0.25*pi*Rc))*(p)/(2*rhof*Uavg^2)	1	Friction factor

Correlation for the friction factor

I Right-click Evaluation Group I and choose Global Evaluation.

- **2** In the **Settings** window for **Global Evaluation**, type Correlation for the friction factor in the **Label** text field.
- 3 Locate the Data section. From the Dataset list, choose Study I/Solution I (soll).
- **4** Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
(0.079/(rhof*Uavg*D/ muf)^0.25)+(0.0073/sqrt(2* Rc/D))	1	Correlation for the friction factor

**5** In the **Evaluation Group I** toolbar, click **= Evaluate**.

2D Plot Group 4

- I In the Home toolbar, click 🚛 Add Plot Group and choose 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

- I 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

- I 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.

18 | FLOW THROUGH A PIPE ELBOW