

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 $k-\omega$ 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 both 200 mm long. Only half the pipe is modeled because the xy -plane is a symmetry plane.

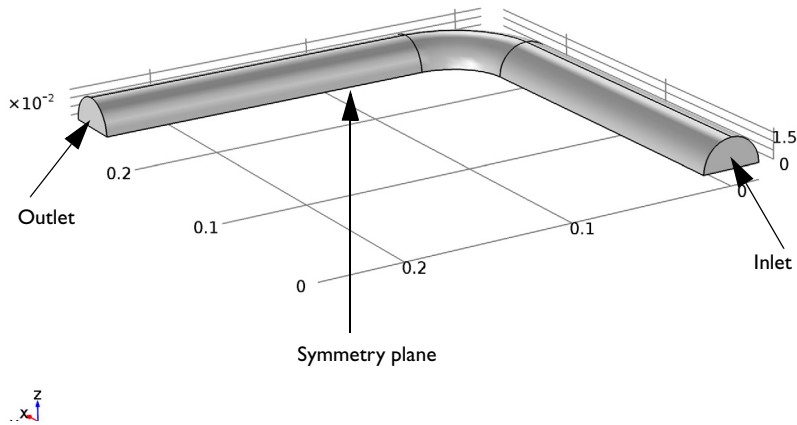


Figure 1: Pipe elbow geometry.

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 $T=90\text{ }^{\circ}\text{C}$ and the absolute outlet pressure is 20 bar. At this stat, the density, ρ , is 965.35 kg/m^3 and the dynamic viscosity, μ , is $3.145 \times 10^{-4}\text{ Pa}\cdot\text{s}$. 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

The high Reynolds number, $\text{Re}_D=5.45 \times 10^5$ based on the pipe diameter, calls for a turbulence model with wall functions. Here the $k-\omega$ model is selected over the $k-\epsilon$ model. The reason for this is that Ref. 1 shows separation after the bend and the $k-\epsilon$ model has a tendency to perform poorly for flows involving strong pressure gradients, separation and strong streamline curvature (Ref. 2).

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

The inlet conditions are obtained from a 2D axisymmetry model where the inlet profiles in turn are plug flow profiles. The entrance length for this Reynolds number is in [Ref. 1](#) approximated to 35 pipe diameters, but to ensure that the flow really becomes fully developed, the 2D pipe is 100 diameters long. The outlet results from the 2D pipe are mapped onto the inlet boundary of the 3D geometry by using coupling variables.

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 counter-rotating 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.

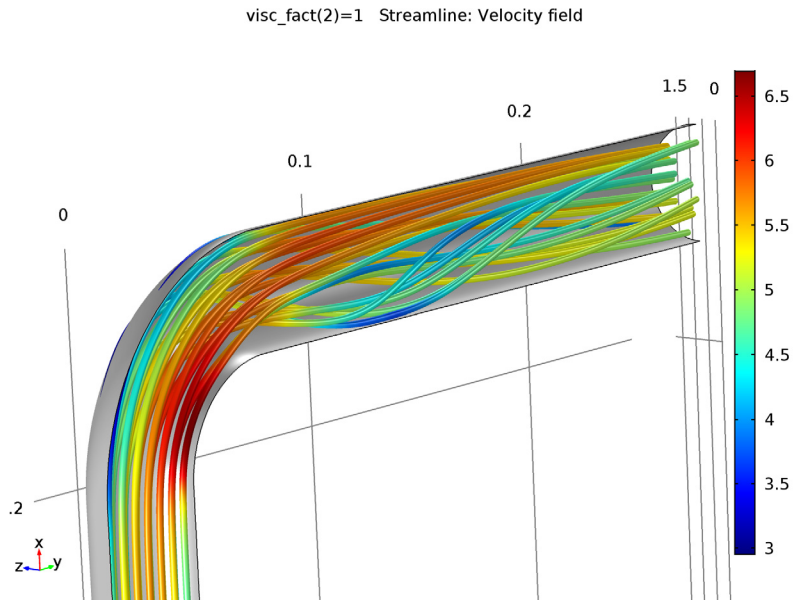


Figure 2: Streamlines colored by the velocity magnitude.

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

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

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 $p_o - p_i \approx 1.87 \times 10^4$ Pa. Since U_{avg} is 5 m/s, Equation 1 evaluates to 1.55.

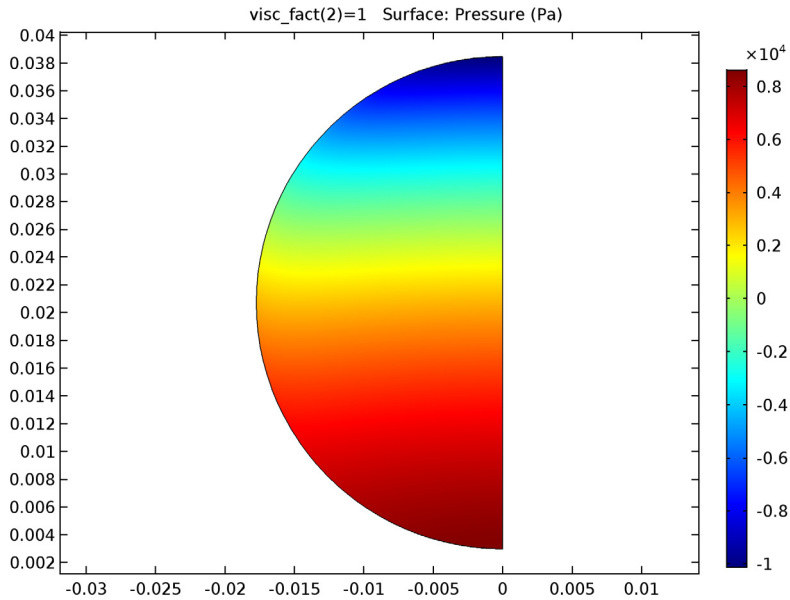


Figure 3: Pressure in a plane at 45° .

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

$$c_k = \frac{2D}{R_c} \quad (2)$$

For the pipe model studied here this evaluates to 1.42. The agreement with the value 1.55 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

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

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 center line. The head loss is related to the pressure drop over the pipe segment, Δp , via

$$h_L = \frac{\Delta p}{g\rho} \quad (4)$$

The friction factor can hence be calculated as:

$$f_f = \frac{\Delta p}{2\rho U_{\text{avg}}^2} \frac{D}{L} \quad (5)$$

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). They are 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 398 Pa, which results in a friction factor equal to 7.4×10^{-3} .

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

$$f_f = \frac{0.079}{\text{Re}^{0.25}} + \frac{0.0073}{\sqrt{D_c/D}} \quad (6)$$

where $D_c = 2 \cdot R_c$ is the coil diameter. In this case, Equation 6 gives $f_f = 7.6 \times 10^{-3}$. The difference of 2.7% 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/Single-Phase_Benchmarks/pipe_elbow

Modeling Instructions

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

NEW

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

MODEL WIZARD

- 1 In the **Model Wizard** window, click **2D Axisymmetric**.
- 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 **Preset Studies>Stationary**.
- 6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

- 1 On the **Home** toolbar, click **Parameters**.
- 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
L	200[mm]	0.2 m	Inlet/outlet length
Rc	50[mm]	0.05 m	Coil radius
rho	965.35[kg/m ³]	965.4 kg/m ³	Density
mu	3.145e-4[Pa*s]	3.145E-4 Pa*s	Dynamic viscosity
Uavg	5[m/s]	5 m/s	Average velocity

GEOMETRY I

Rectangle 1 (r1)

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
- 2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.
- 3 In the **Width** text field, type $D/2$.
- 4 In the **Height** text field, type $100*D$.

Turbulent Flow, k- ω (spf)

Fluid Properties 1

- 1 In the **Model Builder** window, expand the **Component 1 (comp1)>Turbulent Flow, k- ω (spf)** node, then click **Fluid Properties 1**.
- 2 In the **Settings** window for Fluid Properties, locate the **Fluid Properties** section.
- 3 From the ρ list, choose **User defined**. In the associated text field, type ρ .
- 4 From the μ list, choose **User defined**. In the associated text field, type μ .

Inlet 1

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
- 2 Select Boundary 2 only.
- 3 In the **Settings** window for Inlet, locate the **Velocity** section.
- 4 In the U_0 text field, type U_{avg} .

Outlet 1

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.
- 2 Select Boundary 3 only.
- 3 In the **Settings** window for Outlet, locate the **Pressure Conditions** section.
- 4 Select the **Normal flow** check box.

MESH I

Mapped 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.

Distribution 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Mapped 1** and choose **Distribution**.

- 2 Select Boundaries 2 and 3 only.
- 3 In the **Settings** window for Distribution, locate the **Distribution** section.
- 4 From the **Distribution properties** list, choose **Predefined distribution type**.
- 5 In the **Number of elements** text field, type 40.
- 6 In the **Element ratio** text field, type 40.
- 7 From the **Distribution method** list, choose **Geometric sequence**.

Distribution 2

- 1 Right-click **Mapped 1** and choose **Distribution**.
- 2 Select Boundaries 1 and 4 only.
- 3 In the **Settings** window for Distribution, locate the **Distribution** section.
- 4 From the **Distribution properties** list, choose **Predefined distribution type**.
- 5 In the **Number of elements** text field, type 300.
- 6 In the **Element ratio** text field, type 5.
- 7 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

STUDY 1

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

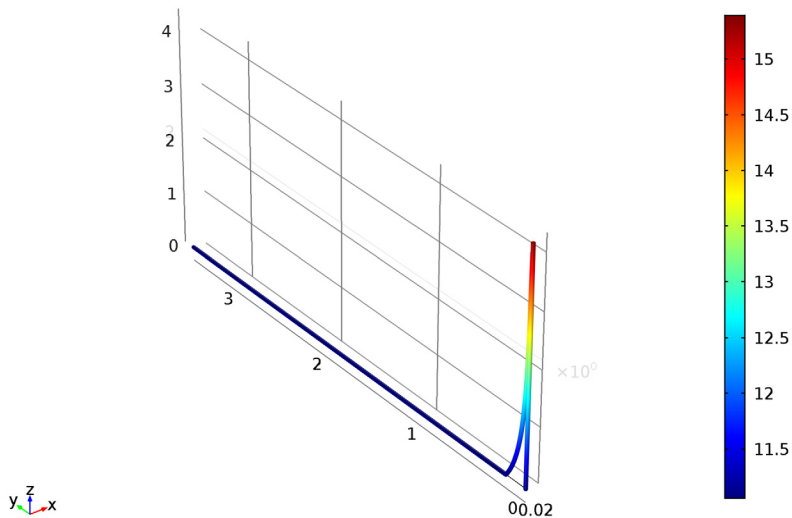
RESULTS

Wall Resolution (spf)

Check that the wall resolution is sufficient.

- 1 On the **Wall Resolution (spf)** toolbar, click **Plot**.

- Click the **Zoom Extents** button on the **Graphics** toolbar.
Line: Wall lift-off in viscous units (1)



Verify that the wall lift-off in viscous units is not much larger than 11.06 some distance behind the inlet.

Assert that the flow really is fully developed by plotting the turbulent viscosity along the symmetry axis.

ID Plot Group 5

On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.

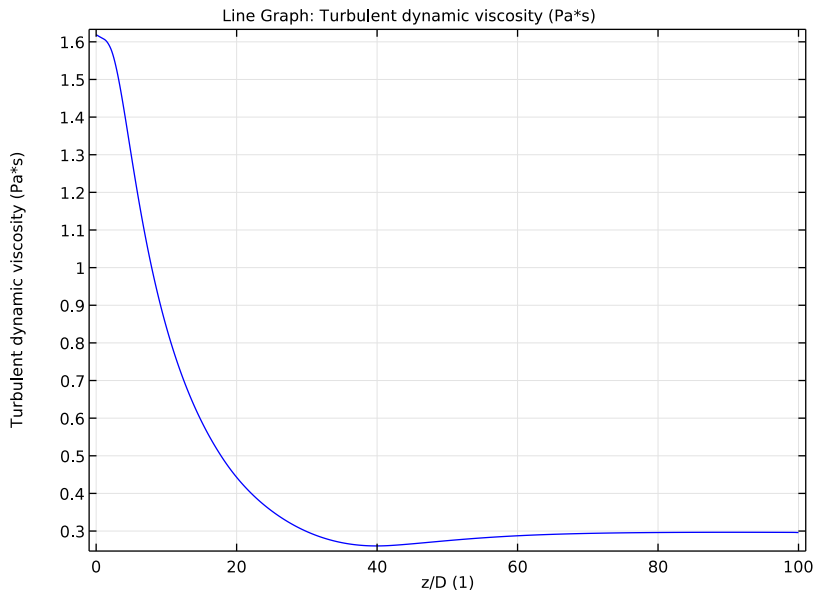
Line Graph 1

On the **ID Plot Group 5** toolbar, click **Line Graph**.

ID Plot Group 5

- Select Boundary 1 only.
- In the **Settings** window for Line Graph, locate the **y-Axis Data** section.
- In the **Expression** text field, type $\text{spf} \cdot \mu T$.
- Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- In the **Expression** text field, type z/D .

- 6 On the **ID Plot Group 5** toolbar, click **Plot**.



The turbulent viscosity at the centerline is seen to have reached an approximately constant value well before the outlet. The flow can therefore be considered fully developed at the outlet.

DEFINITIONS

General Extrusion 1 (genext1)

- 1 On the **Definitions** toolbar, click **Component Couplings** and choose **General Extrusion**.
- 2 In the **Settings** window for General Extrusion, locate the **Source Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Locate the **Destination Map** section. In the **r-expression** text field, type $\sqrt{y^2+z^2}$.
- 5 In the **z-expression** text field, type $100*D$.
- 6 Select Boundary 3 only.

ROOT

- 1 In the **Model Builder** window, click the root node.
- 2 On the **Home** toolbar, click **Add Component** and choose **3D**.

ADD PHYSICS

- 1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.
- 2 Go to the **Add Physics** window.
- 3 In the **Add physics** tree, select **Fluid Flow>Single-Phase Flow>Turbulent Flow>Turbulent Flow, k- ω (spf)**.
- 4 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Studies	Solve
Study 1	

- 5 Click **Add to Component** in the window toolbar.
- 6 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ADD STUDY

- 1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.
- 2 Go to the **Add Study** window.
- 3 Find the **Studies** subsection. In the **Select study** tree, select **Preset Studies>Stationary**.
- 4 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Physics	Solve
Turbulent Flow, k- ω (spf)	

- 5 Click **Add Study** in the window toolbar.
- 6 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

GEOMETRY 2*Work Plane 1 (wp1)*

- 1 On 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 L.

Circle 1 (c1)

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
- 2 In the **Settings** window for Circle, locate the **Size and Shape** section.

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

Square 1 (sq1)

1 On the **Geometry** toolbar, click **Primitives** and choose **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 $-D$.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Click the **Zoom Extents** button on 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 **Active** toggle button.

6 Select the object **sq1** only.

Revolve 1 (rev1)

1 In the **Model Builder** window, under **Component 2 (comp2)>Geometry 2** right-click **Work Plane 1 (wp1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Revolve**.

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

4 Click the **Angles** button.

5 In the **End angle** text field, type 90 .

6 Select the object **wp1** only.

7 Locate the **General** section. Clear the **Unite with input objects** check box.

8 Locate the **Revolution Axis** section. Find the **Point on the revolution axis** subsection.

In the **xw** text field, type Rc .

9 In the **yw** text field, type L .

10 Right-click **Revolve 1 (rev1)** and choose **Build Selected**.

11 Click the **Zoom Extents** button on the **Graphics** toolbar.

Extrude 1 (ext1)

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

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

3 In the table, enter the following settings:

Distances (m)
L

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

Work Plane 1 (wp1)

In the **Model Builder** window, under **Component 2 (comp2)>Geometry 2** right-click **Work Plane 1 (wp1)** and choose **Copy**.

Work Plane 2 (wp2)

1 Right-click **Geometry 2** 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 R_c .

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

Circle 1 (c1)

1 In the **Model Builder** window, expand the **Component 2 (comp2)>Geometry 2>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 $L+R_c$.

Square 1 (sq1)

1 In the **Model Builder** window, under **Component 2 (comp2)>Geometry 2>Work Plane 2 (wp2)>Plane Geometry** click **Square 1 (sq1)**.

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

3 In the **xw** text field, type $-D/2+L+R_c$.

Extrude 2 (ext2)

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

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

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

4 In the table, enter the following settings:

Distances (m)
L

- 5 On the **Geometry** toolbar, click **Build All**.
- 6 In the **Model Builder** window, collapse the **Geometry 2** node.

GLOBAL DEFINITIONS

Parameters

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.
- 2 In the **Settings** window for Parameters, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Value	Description
visc_fact	1	1	Ramping factor on viscosity

Turbulent Flow, k- ω 2 (spf2)

On the **Physics** toolbar, click **Turbulent Flow, k- ω (spf)** and choose **Turbulent Flow, k- ω 2 (spf2)**.

Fluid Properties 1

- 1 In the **Model Builder** window, expand the **Component 2 (comp2)>Turbulent Flow, k- ω 2 (spf2)** node, then click **Fluid Properties 1**.
- 2 In the **Settings** window for Fluid Properties, locate the **Fluid Properties** section.
- 3 From the ρ list, choose **User defined**. In the associated text field, type ρ .
- 4 From the μ list, choose **User defined**. In the associated text field, type $\mu \cdot \text{visc_fact}$.

Inlet 1

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
- 2 Select Boundary 1 only.
- 3 In the **Settings** window for Inlet, locate the **Turbulence Conditions** section.
- 4 Click the **Specify turbulence variables** button.
- 5 In the k_0 text field, type $\text{comp1.genext1}(k)$.
- 6 In the ω_0 text field, type $\text{comp1.genext1}(\omega)$.
- 7 Locate the **Velocity** section. Click the **Velocity field** button.

8 Specify the \mathbf{u}_0 vector as

comp1.genext1 (w)	x
0	y
0	z

Outlet 1

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.
- 2 Select Boundary 12 only.

Symmetry 1

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
- 2 Select Boundaries 3, 7, and 10 only.

MESH 2

- 1 In the **Model Builder** window, under **Component 2 (comp2)** click **Mesh 2**.
- 2 In the **Settings** window for Mesh, locate the **Mesh Settings** section.
- 3 From the **Element size** list, choose **Extra fine**.

Size

- 1 Right-click **Component 2 (comp2)>Mesh 2** and choose **Edit Physics-Induced Sequence**.
- 2 In the **Model Builder** window, under **Component 2 (comp2)>Mesh 2** click **Size**.
- 3 In the **Settings** window for Size, click to expand the **Element size parameters** section.
- 4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type 0.0012.

Free Tetrahedral 1

- 1 In the **Model Builder** window, under **Component 2 (comp2)>Mesh 2** right-click **Size 1** and choose **Disable**.
- 2 In the **Model Builder** window, under **Component 2 (comp2)>Mesh 2** click **Free Tetrahedral 1**.
- 3 In the **Settings** window for Free Tetrahedral, locate the **Domain Selection** section.
- 4 From the **Geometric entity level** list, choose **Domain**.
- 5 Select Domain 2 only.
- 6 Click the **Build Selected** button.

Swept 1

- 1 In the **Model Builder** window, right-click **Mesh 2** and choose **Swept**.

- 2 In the **Settings** window for Swept, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domains 1 and 3 only.

Distribution 1

- 1 Right-click **Component 2 (comp2)>Mesh 2>Swept 1** and choose **Distribution**.
- 2 In the **Settings** window for Distribution, locate the **Distribution** section.
- 3 From the **Distribution properties** list, choose **Predefined distribution type**.
- 4 In the **Element ratio** text field, type 5.
- 5 In the **Number of elements** text field, type 80.
- 6 Select Domains 2 and 3 only.

Distribution 2

- 1 Right-click **Swept 1** and choose **Distribution**.
- 2 Select Domain 1 only.
- 3 In the **Settings** window for Distribution, locate the **Distribution** section.
- 4 From the **Distribution properties** list, choose **Predefined distribution type**.
- 5 In the **Number of elements** text field, type 40.
- 6 In the **Element ratio** text field, type 10.

Boundary Layer Properties 1

- 1 In the **Model Builder** window, expand the **Component 2 (comp2)>Mesh 2>Boundary Layers 1** node, then click **Boundary Layer Properties 1**.
- 2 In the **Settings** window for Boundary Layer Properties, locate the **Boundary Layer Properties** section.
- 3 In the **Number of boundary layers** text field, type 8.
- 4 In the **Thickness adjustment factor** text field, type 1.
- 5 In the **Model Builder** window, right-click **Mesh 2** and choose **Build All**.
- 6 In the **Model Builder** window, collapse the **Mesh 2** node.

COMPONENT 2 (COMP2)

Mesh 3

- On the **Mesh** toolbar, click **Add Mesh**.

MESH 3

Reference 1

- 1 In the **Model Builder** window, under **Component 2 (comp2)>Meshes** right-click **Mesh 3** and choose **More Operations>Reference**.
- 2 In the **Settings** window for Reference, locate the **Reference** section.
- 3 From the **Mesh** list, choose **Mesh 2**.

Scale 1

- 1 Right-click **Component 2 (comp2)>Meshes>Mesh 3>Reference 1** and choose **Scale**.
- 2 In the **Settings** window for Scale, locate the **Scale** section.
- 3 In the **Element size scale** text field, type 2.

Reference 1

Right-click **Reference 1** and choose **Expand**.

MESH 3

Size

- 1 In the **Model Builder** window, under **Component 2 (comp2)>Meshes>Mesh 3** click **Size**.
- 2 In the **Settings** window for Size, locate the **Element Size Parameters** section.
- 3 In the **Maximum element growth rate** text field, type 1.1.

Boundary Layer Properties 1

- 1 In the **Model Builder** window, expand the **Component 2 (comp2)>Meshes>Mesh 3>Boundary Layers 1** node, then click **Boundary Layer Properties 1**.
- 2 In the **Settings** window for Boundary Layer Properties, locate the **Boundary Layer Properties** section.
- 3 In the **Number of boundary layers** text field, type 6.
- 4 In the **Boundary layer stretching factor** text field, type 1.25.
- 5 In the **Model Builder** window, right-click **Mesh 3** and choose **Build All**.
- 6 In the **Model Builder** window, collapse the **Mesh 3** node.

COMPONENT 2 (COMP2)

Mesh 4

On the **Mesh** toolbar, click **Add Mesh**.

MESH 4

Reference 1

- 1 In the **Model Builder** window, under **Component 2 (comp2)>Meshes** right-click **Mesh 4** and choose **More Operations>Reference**.
- 2 In the **Settings** window for Reference, locate the **Reference** section.
- 3 From the **Mesh** list, choose **Mesh 3**.

Scale 1

- 1 Right-click **Component 2 (comp2)>Meshes>Mesh 4>Reference 1** and choose **Scale**.
- 2 In the **Settings** window for Scale, locate the **Scale** section.
- 3 In the **Element size scale** text field, type 2.

Reference 1

Right-click **Reference 1** and choose **Expand**.

MESH 4

Size

- 1 In the **Model Builder** window, under **Component 2 (comp2)>Meshes>Mesh 4** click **Size**.
- 2 In the **Settings** window for Size, locate the **Element Size Parameters** section.
- 3 In the **Maximum element growth rate** text field, type 1.12.

Boundary Layer Properties 1

- 1 In the **Model Builder** window, expand the **Component 2 (comp2)>Meshes>Mesh 4>Boundary Layers 1** node, then click **Boundary Layer Properties 1**.
- 2 In the **Settings** window for Boundary Layer Properties, locate the **Boundary Layer Properties** section.
- 3 In the **Number of boundary layers** text field, type 4.
- 4 In the **Boundary layer stretching factor** text field, type 1.3.
- 5 In the **Model Builder** window, right-click **Mesh 4** and choose **Build All**.
- 6 In the **Model Builder** window, collapse the **Mesh 4** node.
- 7 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Study Options** in the menu.

STUDY 2

Step 1: Stationary

Set up an auxiliary continuation sweep for the `visc_fact` parameter.

- 1 In the **Model Builder** window, expand the **Study 2** node, then click **Step 1: Stationary**.
- 2 In the **Settings** window for Stationary, click to expand the **Study extensions** section.
- 3 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
- 4 Click **Add**.
- 5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
visc_fact	10 1	

- 6 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.
- 7 From the **Method** list, choose **Solution**.
- 8 From the **Study** list, choose **Study 1, Stationary**.
- 9 Right-click **Study 2>Step 1: Stationary** and choose **Multigrid Level**.
- 10 In the **Settings** window for Multigrid Level, locate the **Mesh Selection** section.
- 11 In the table, enter the following settings:

Geometry	Mesh
Geometry 2	mesh3

Solution 2 (sol2)

- 1 In the **Model Builder** window, right-click **Step 1: Stationary** and choose **Multigrid Level**.
- 2 On the **Study** toolbar, click **Show Default Solver**.
- 3 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.
- 4 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1>Iterative 1** node, then click **Multigrid 1**.
- 5 In the **Settings** window for Multigrid, locate the **General** section.
- 6 From the **Hierarchy generation method** list, choose **Manual**.
- 7 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1>Iterative 2** node, then click **Multigrid 1**.
- 8 In the **Settings** window for Multigrid, locate the **General** section.
- 9 From the **Hierarchy generation method** list, choose **Manual**.
- 10 In the **Model Builder** window, collapse the **Solution 2 (sol2)** node.

STUDY 2*Solution 2 (sol2)*

In the **Model Builder** window, under **Study 2>Solver Configurations** right-click **Solution 2 (sol2)** and choose **Compute**.

RESULTS*Velocity (spf2)*

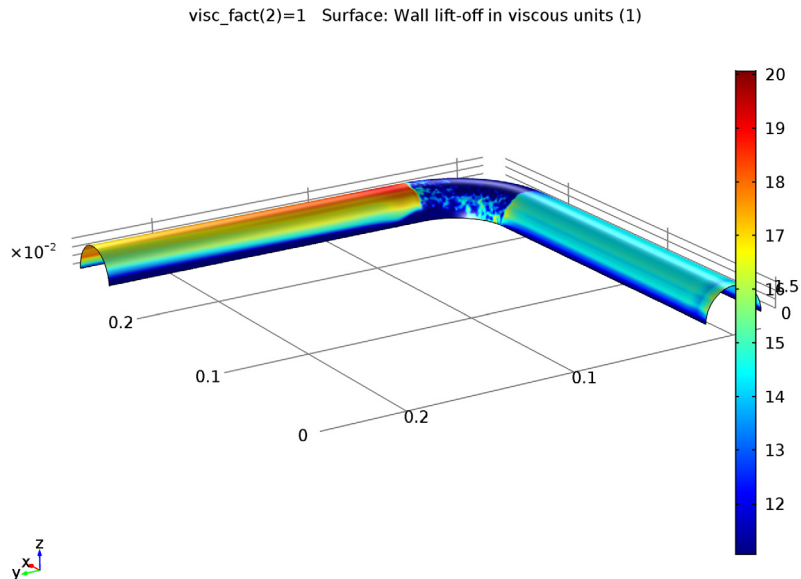
Deactivate the automatic update of plots when working with large 3D models.

- 1 In the **Model Builder** window, click **Results**.
- 2 In the **Settings** window for Results, locate the **Result Settings** section.
- 3 Clear the **Automatic update of plots** check box.

Check that the wall resolution is sufficient.

Wall Resolution (spf2)

- 1 In the **Model Builder** window, under **Results** click **Wall Resolution (spf2)**.
- 1 On the **Wall Resolution (spf2)** 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 (spf2)

- 1** In the **Model Builder** window, expand the **Results>Pressure (spf2)** node.
- 2** Right-click **Pressure** and choose **Disable**.
- 3** Right-click **Pressure (spf2)** and choose **Streamline**.
- 4** In the **Settings** window for Streamline, locate the **Data** section.
- 5** From the **Data set** list, choose **Study 2/Solution 2 (3) (sol2)**.
- 6** Locate the **Streamline Positioning** section. From the **Positioning** list, choose **On selected boundaries**.
- 7** Locate the **Selection** section. Select the **Active** toggle button.
- 8** Select Boundary 1 only.
- 9** Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.
- 10** Select the **Radius scale factor** check box.
- 11** In the associated text field, type 0.001.
- 12** Right-click **Results>Pressure (spf2)>Streamline 1** and choose **Color Expression**.
- 13** On the **Pressure (spf2)** toolbar, click **Plot**.

Execute the following steps to reproduce [Figure 3](#).

Cut Plane 1

On the **Results** toolbar, click **Cut Plane**.

Data Sets

- 1** In the **Settings** window for Cut Plane, locate the **Plane Data** section.
- 2** From the **Plane type** list, choose **General**.
- 3** From the **Plane entry method** list, choose **Point and normal**.
- 4** Find the **Point** subsection. In the **x** text field, type L+Rc.
- 5** Find the **Normal** subsection. In the **y** text field, type 1.
- 6** In the **x** text field, type 1.
- 7** In the **z** text field, type 0.

2D Plot Group 9

- 1** On the **Results** toolbar, click **2D Plot Group**.
- 2** In the **Settings** window for 2D Plot Group, locate the **Data** section.
- 3** From the **Data set** list, choose **Cut Plane 1**.

- 4 Right-click **2D Plot Group 9** and choose **Surface**.
- 5 In the **Settings** window for Surface, locate the **Expression** section.
- 6 In the **Expression** text field, type p_2 .
- 7 On the **2D Plot Group 9** toolbar, click **Plot**.

The following steps evaluate the pressure drop Δp in [Equation 4](#).

Derived Values

In the **Model Builder** window, under **Results** click **Derived Values**.

Surface Average 1

On the **Results** toolbar, click **More Derived Values** and choose **Average>Surface Average**.

Derived Values

- 1 In the **Settings** window for Surface Average, locate the **Data** section.
- 2 From the **Data set** list, choose **Study 2/Solution 2 (3) (sol2)**.
- 3 From the **Parameter selection (visc_fact)** list, choose **Last**.
- 4 Select Boundary 5 only.
- 5 Locate the **Expression** section. In the **Expression** text field, type p_2 .
- 6 Click the **Evaluate** button.
- 7 In the **Settings** window for Surface Average, locate the **Data** section.
- 8 From the **Data set** list, choose **Cut Plane 1**.
- 9 Click the **Evaluate** button.

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