Created in COMSOL Multiphysics 5.5

Slurry Transport

Introduction

The Pipe Flow interface allows you to simulate non-Newtonian fluids flowing in pipes. You can easily switch between Newtonian and non-Newtonian fluid models using a Fluid Model property setting.

This example models a coal slurry being transported in a pipe system where the pipe diameter changes in different sections. The slurry behaves as a non-Newtonian fluid described by the power law model. Results provide the flow rate in different sections of the piping and the total pressure drop across the system.

Model Definition

A coal slurry enters a 150 mm diameter pipe at a flow rate of 3.4 m^3/min. The main pipe branches of into two loops, where the pipe diameter is 100 mm in the upper loop and 75 mm in the lower loop. The branches then rejoin in a single pipe that has a diameter of 175 mm.

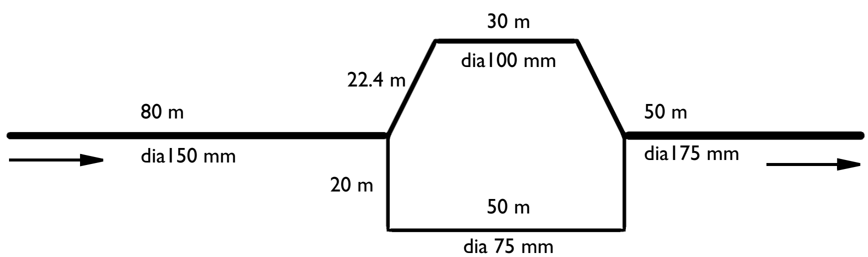

Figure 1: A non-Newtonian coal slurry is transported in a system with two pipe loops.

The slurry properties are summarized below:

TABLE 1: SLURRY PROPERTIES.

PROPERTY	VALUE
Density	1020 kg/m^3
Fluid consistency index (m)	1.4
Flow behavior index (n)	0.4

The mass conservation and momentum equations below describe the stationary flow of a fluid in a horizontal pipe:

$$\nabla \cdot (A\rho u) = 0 \tag{1}$$

$$\rho\mathbf{u} \cdot \nabla\mathbf{u} = -\nabla p - f_{\mathrm{D}}\frac{\rho}{2d_h}\mathbf{u}|\mathbf{u}| \tag{2}$$

The pressure losses due to pipe friction is a function of the Darcy friction factor, f_{D}, that will be different depending upon whether the fluid is Newtonian or non-Newtonian.

The present example treats a Power-law fluid, where the apparent viscosity is related to the shear rate as

$$\mu = m(\dot{\gamma})^{n-1} \tag{3}$$

Above, where m and n are two empirical curve fitting parameters known as the fluid consistency coefficient and the flow behavior index, respectively.

In the laminar regime the friction factor for power law fluids is calculated by the Stokes equation using the modified Reynolds number proposed by Metzner and Reed (Ref. 1):

$$f_D = \frac{64}{\mathrm{Re}_{\mathrm{MR}}} \tag{4}$$

$$\mathrm{Re}_{\mathrm{MR}} = \frac{\rho u^{(2-n)} d_h^n}{8^{(n-1)} m\left(\frac{3n+1}{4n}\right)^n} \tag{5}$$

For the turbulent regime the friction model proposed by Irvine (Ref. 2) is available:

$$f_D = 4\left(\frac{D(n)}{\mathrm{Re}_{\mathrm{MR}}}\right)^{\left(\frac{1}{3n+1}\right)} \tag{6}$$

where

$$D(n) = \frac{2^{(n+4)}}{7^{7n}}\left(\frac{4n}{3n+1}\right)^{3n^2} \tag{7}$$

The Pipe Flow interface calculates a critical Reynolds number (Ref. 3) internally, which serves as a criterion for the transition between laminar and turbulent flow

$$\mathrm{Re}_{\mathrm{crit}} = \frac{6464n}{(3n+1)^2}(2+n)^{\left(\frac{2+n}{1+n}\right)} \tag{8}$$

If the Irvine friction model is selected and ReMR < Recrit, then the friction factor is evaluated using Equation 4. Nevertheless, it is always good practice to evaluate the Reynolds number in a pipe flow simulation to confirm that the selected friction model applies.

The friction model is specified in the Pipe Properties feature. It is possible set up multiple Pipe Properties features, and if needed assign different friction models to different parts of a pipe system.

Results and Discussion

Figure 2 shows the calculated pressure in the pipe system. The total pressure drop is around $2.7 \cdot 10^5$ Pa.

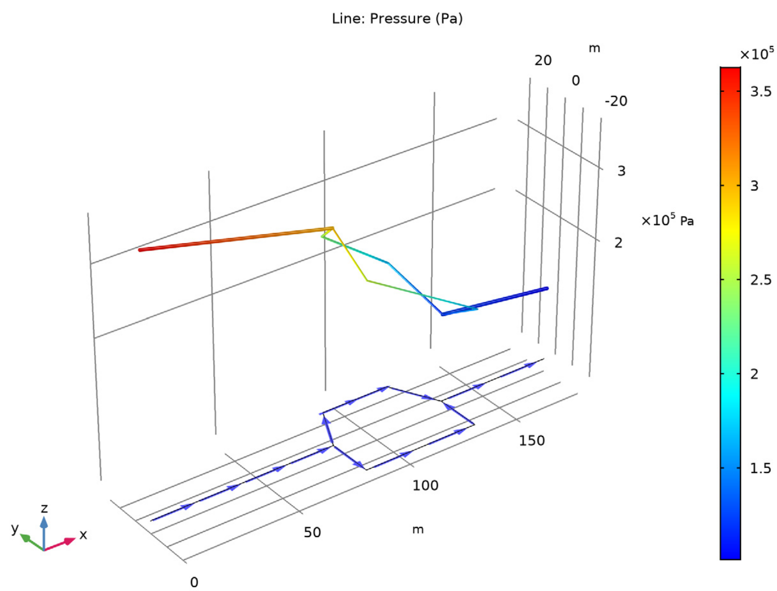

Figure 2: Pipe pressure as function of position in the pipe system.

The flow rate is evaluated to 2.48 m^3/min in the upper pipe loop and to 0.92 m^3/min in the lower loop. Figure 3 shows a plot of $\mathrm{Re}_{\mathrm{MR}}$ in the different pipe sections, that can be used to find if the flow is laminar or turbulent. Since $Re_{MR} > Re_{crit}$, calculated to 2434 from Equation 8, it can be concluded that the flow is turbulent everywhere.

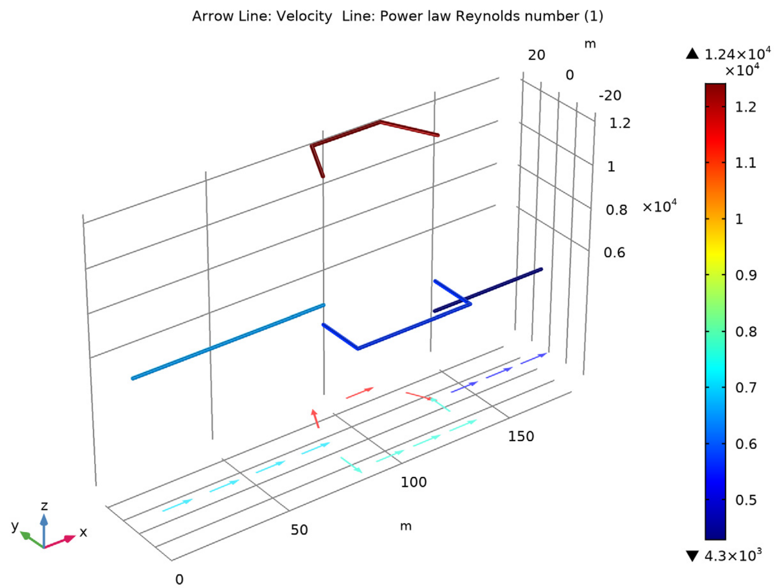

Figure 3: The Reynolds number indicates turbulent conditions in the entire pipe network.

References

1. A.B. Metzner and J.C. Reed, *AICHE Journal*, vol. 1, p. 434, 1955.

2. T.F. Irvine, *Chem. Eng. Commun.*, vol. 65, p. 39, 1988.

3. N.W. Ryan and M.M. Johnson, *AICHE Journal*, vol. 5, p. 433, 1959.

Application Library path: `Pipe_Flow_Module/Tutorials/slurry_transport`

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 **2D**.
2 In the **Select Physics** tree, select **Fluid Flow>Single-Phase Flow>Pipe Flow (pfl)**.
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
Qv0	3.4[m^3/min]	0.056667 m³/s	Volumetric flow rate
d1	150[mm]	0.15 m	Pipe diameter, pipe 1
d2	100[mm]	0.1 m	Pipe diameter, pipe 2
d3	75[mm]	0.075 m	Pipe diameter, pipe 3
d4	175[mm]	0.175 m	Pipe diameter, pipe 4

GEOMETRY 1

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **Polygon**.
2 In the **Settings** window for **Polygon**, locate the **Object Type** section.
3 From the **Type** list, choose **Open curve**.
4 Locate the **Coordinates** section. From the **Data source** list, choose **Vectors**.
5 In the **x** text field, type `0 80 80 90 90 120 120 130 130 180` .
6 In the **y** text field, type `0 0 0 20 20 20 20 0 0 0`.

Polygon 2 (pol2)

1 In the **Geometry** toolbar, click **Polygon**.
2 In the **Settings** window for **Polygon**, locate the **Object Type** section.
3 From the **Type** list, choose **Open curve**.

4 Locate the **Coordinates** section. From the **Data source** list, choose **Vectors**.

5 In the **x** text field, type `80 80 80 130 130 130`.

6 In the **y** text field, type `0 -20 -20 -20 -20 0`.

7 Click **Build All Objects**.

PIPE FLOW (PFL)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pipe Flow (pfl)**.

2 In the **Settings** window for **Pipe Flow**, locate the **Fluid Model** section.

3 From the **Fluid model** list, choose **Power law**.

Fluid Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Pipe Flow (pfl)** click **Fluid Properties 1**.

2 In the **Settings** window for **Fluid Properties**, locate the **Fluid Properties** section.

3 In the m text field, type `1.4`.

4 In the n text field, type `0.4`.

Pipe Properties 1

1 In the **Model Builder** window, click **Pipe Properties 1**.

2 In the **Settings** window for **Pipe Properties**, locate the **Pipe Shape** section.

3 From the list, choose **Circular**.

4 In the d_i text field, type `d1`.

Inlet 1

1 In the **Physics** toolbar, click **Points** and choose **Inlet**.

2 Select Point 1 only.

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

4 From the **Specification** list, choose **Volumetric flow rate**.

5 In the $q_{v,\,0}$ text field, type `Qv0`.

Pipe Properties 2

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

2 Select Boundaries 4–6 only.

3 In the **Settings** window for **Pipe Properties**, locate the **Pipe Shape** section.

4 From the list, choose **Circular**.

5 In the d_i text field, type `d2`.

Pipe Properties 3

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

2 Select Boundaries 2, 3, and 7 only.

3 In the **Settings** window for **Pipe Properties**, locate the **Pipe Shape** section.

4 From the list, choose **Circular**.

5 In the d_i text field, type `d3`.

Pipe Properties 4

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

2 Select Boundary 8 only.

3 In the **Settings** window for **Pipe Properties**, locate the **Pipe Shape** section.

4 From the list, choose **Circular**.

5 In the d_i text field, type `d4`.

STUDY 1

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

RESULTS

Height Expression 1

1 In the **Model Builder** window, expand the **Pressure (pfl)** node.

2 Right-click **Line 1** and choose **Height Expression**.

3 In the **Settings** window for **Height Expression**, locate the **Expression** section.

4 From the **Height data** list, choose **Expression**.

5 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Pipe Flow>pfl.ptot - Total pressure - Pa**.

Arrow Line 1

1 In the **Model Builder** window, right-click **Pressure (pfl)** and choose **Arrow Line**.

2 In the **Settings** window for **Arrow Line**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

4 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Normalized**.

5 From the **Arrow base** list, choose **Center**.

6 Select the **Scale factor** check box.

7 In the associated text field, type `3.5`.

8 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `14`.

9 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

10 In the **Pressure (pfl)** toolbar, click **Plot**.

The plot should like look that shown in Figure 2.

Velocity (pfl)

To reproduce Figure 3 do as follows.

Line 1

1 In the **Model Builder** window, right-click **Velocity (pfl)** and choose **Line**.

2 In the **Settings** window for **Line**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Pipe Flow>pfl.ReMR - Power law Reynolds number**.

3 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.

Height Expression 1

Right-click **Line 1** and choose **Height Expression**.

Color Expression 1

1 In the **Model Builder** window, expand the **Arrow Line 1** node, then click **Color Expression 1**.

2 In the **Settings** window for **Color Expression**, locate the **Coloring and Style** section.

3 Clear the **Color legend** check box.

Velocity (pfl)

1 In the **Model Builder** window, click **Velocity (pfl)**.

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

3 Clear the **Plot dataset edges** check box.

4 In the **Velocity (pfl)** toolbar, click **Plot**.

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

6 Select the **Show units** check box.

Line 1

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

2 In the **Settings** window for **Line**, click to expand the **Quality** section.

3 From the **Smoothing** list, choose **Inside geometry domains**.

4 In the **Velocity (pfl)** toolbar, click **Plot**.

Point Evaluation 1

1 In the **Results** toolbar, click **Point Evaluation**.

   You can evaluate the volumetric flow rate for the upper and lower loops:

2 Select Points 2 and 4 only.

3 In the **Settings** window for **Point Evaluation**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	**Unit**	**Description**
`pfl.Qv`	`m^3/min`	`Volumetric flow rate magnitude`

5 Click **Evaluate**.