Insulation of a Pipeline Section

Introduction

As oil flows through a pipeline heat is dissipated due to internal viscous shear in the fluid. With good insulation of the pipeline, this generated heat can be used to avoid the need of preheating of the oil, despite the fact that it needs to be transported in a cold environment over long distances.

This model uses the Non-isothermal Pipe Flow interface to set up and solve the flow and energy equations describing oil transport in a pipeline section. With the addition of an Optimization interface, the thickness of the pipeline insulation can be found such that the temperature is constant along the pipe stretch.

Note: The third study of this model requires the Optimization Module.

©2013 COMSOL

Model Definition

Oil flowing at a rate of 2500 m^3/h enters a 150 km pipeline section with a temperature of 25 °C. The average temperature outside of the pipeline is -10 °C.

Figure 1: A section of pipeline transporting crude oil

FLOW EQUATIONS

The continuity and momentum equations below describe the flow of oil inside a horizontal pipe:

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

$$0 = -\nabla p - f_D \frac{\rho}{2d_h}\mathbf{u}|\mathbf{u}| \tag{2}$$

Above, A (SI unit: m^2) is the cross section area of the pipe, ρ (SI unit: kg/m^3) is the density, $\mathbf{u}$ (SI unit: m/s) is the fluid velocity, and p (SI unit: N/m^2) is the pressure.

The second term on the right-hand side of Equation 2 describes the pressure drop due to internal viscous shear. The term contains the Darcy friction factor, f_D, which is a function of the Reynolds number and the surface roughness divided by the hydraulic pipe diameter. In this example, f_D is calculated from the Haaland equation (Ref. 1). It can recover both small and large relative roughness limits for a wide range of Reynolds numbers ($4 \cdot 10^3 < Re < 1 \cdot 10^8$):

 ©2013 COMSOL

$$\sqrt{\frac{1}{f_D}} = -1.8\log_{10}\left(\left(\frac{e/d}{3.7}\right)^{1.11} + \left(\frac{6.9}{\mathrm{Re}}\right)\right) \quad (2\text{-}2)$$

The pipe flow interfaces have the Haaland equation and several other friction models predefined and automatically calculate the friction factor based on the local properties of the pipe, the fluid physical properties, and the fluid velocity.

HEAT TRANSFER EQUATIONS

The energy equation the pipeline flow is:

$$\rho A C_p \mathbf{u} \cdot \nabla T = \nabla \cdot Ak\nabla T + f_D \frac{\rho A}{2d_h}|\mathbf{u}|^3 + Q_{\mathrm{wall}} \quad (3)$$

where C_p (SI unit: J/(kg·K)) is the heat capacity at constant pressure, T is the temperature (SI unit: K), and k (SI unit: W/(m·K)) is the thermal conductivity. The second term on the right-hand side of Equation 3 corresponds to heat released due to the work of internal friction forces. Q_{wall} (SI unit: W/m) is a source/sink term due to heat exchange with the surroundings through the pipe wall:

$$Q_{\mathrm{wall}} = hZ(T_{\mathrm{ext}} - T) \quad (4)$$

Where Z (m) is the perimeter of the pipe, h (W/(m^2·K)) an overall heat transfer coefficient and T_{ext} (K) the external temperature outside the pipe.

The overall heat transfer coefficient includes contribution from internal film resistance, wall resistance, and external film resistance.

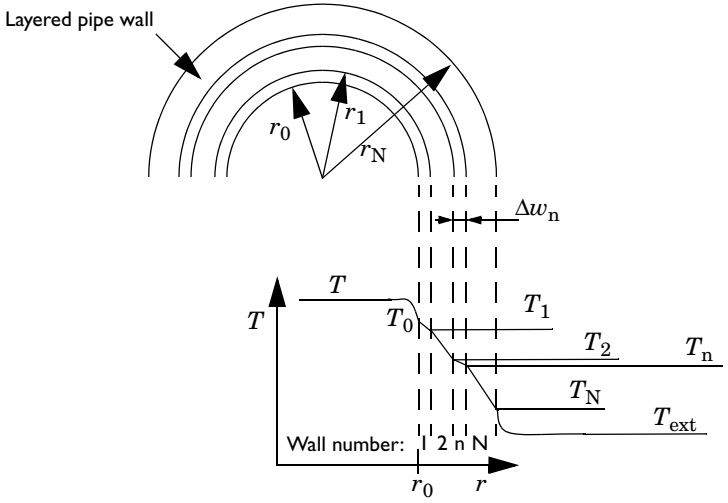

Figure 3: Wall heat transfer resistance

- r_n (m) is the outer radius of wall n
- $w = r - r_0$ (m) a wall coordinate, starting at the inner radius r_0
- $\Delta w_n = r_n - r_{n-1}$ (m) the wall thickness of wall n
- Z_n (m) is the outer perimeter of wall n
- h_{int} and h_{ext} are the film heat transfer coefficients on the inside and outside of the tube, respectively (W/(m^2, K)).
- k_n is the thermal conductivity (W/(m, K)) of wall n

For a circular tube, an effective hZ in Equation 4 can be used such that

$$(hZ)_{eff} = \frac{2\pi}{\frac{1}{r_0 h_{int}} + \frac{1}{r_N h_{ext}} + \sum_{n=1}^{N}\left(\frac{\ln\left(\frac{r_n}{r_{n-1}}\right)}{k_n}\right)} \qquad (4\text{-}4)$$

The assumption is equal temperature around the circumference of the pipe, and that the heat transfer through the wall is quasi-static.

 ©2013 COMSOL

For this particular case, with one pipe wall and one layer of insulation, this simplifies to:

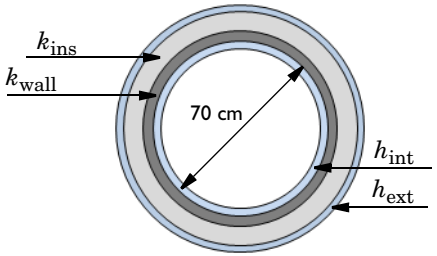

Figure 5: Pipeline cross section. A two layered wall (dark and light gray) and the film resistances on the inside and outside (light blue).

Properties of the pipe and insulation material is given in the table below.

TABLE 4-1: PIPE PROPERTIES

NAME	VALUE	DESCRIPTION
d_{wall}	2 cm	Pipe wall thickness
k_{wall}	45 W/(m·K)	Pipe wall thermal conductivity
d_{ins}	to be determined	Insulation thickness
k_{ins}	0.025 W/(m·K)	Insulation thermal conductivity

The film resistance inside the pipe is given by:

$$h_{int} = \mathrm{Nu}_{int}\frac{k_{oil}}{d} \tag{5}$$

with the following Nusselt correlation (Ref. 2):

$$\mathrm{Nu}_{int} = \frac{(f_D/8)(\mathrm{Re}-1000)\mathrm{Pr}}{1+\sqrt{12.7}(\mathrm{Pr}^{2/3}-1)} \tag{6}$$

The film resistance due to the external flow of air around the pipeline is:

$$h_{ext} = \mathrm{Nu}_{ext}\frac{k_{air}}{d} \tag{7}$$

where Nu_{ext} is calculated with a forced convection relation, assuming an average air speed of 5 m/s.

$$\mathrm{Nu}_{\mathrm{ext}} = 0.3 + \frac{0.62\sqrt{\mathrm{Re}}\mathrm{Pr}^{1/3}}{[1+(0.4/\mathrm{Pr})^{2/3}]^{1/4}}[1+(\mathrm{Re}/282000)^{5/8}]^{4/5} \quad (8)$$

Results and Discussion

A first study calculates the temperature along the pipeline assuming perfect insulation, as well as case where the pipeline in uninsulated. Figure 6 shows that the heat generated due to friction increases the temperature of the oil by approximately 3 °C over 150 km. With no pipeline insulation the temperature at the outlet is close to that of the surroundings.

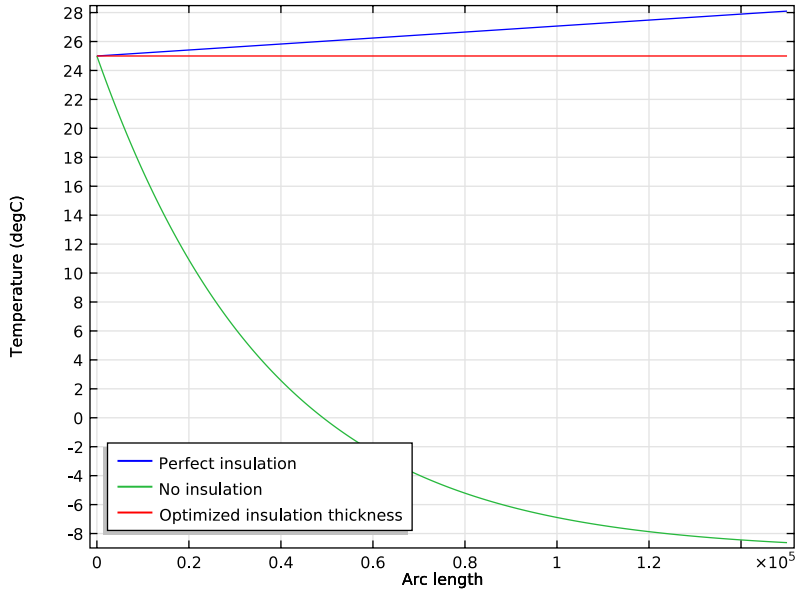

Figure 6: Oil temperature as a function of position in the pipeline assuming no heat transfer across the pipe wall, heat transfer with no insulating material around the pipe, and with an optimized insulation thickness.

Optimization calculations performed under the constraint that the temperature at the outlet should be the same as inlet temperature, predicts that the thickness of the insulating material should be approximately 8.9 cm.

It is always advisable to check the Reynolds number and verify that the friction model is valid under the present flow conditions. In this case Re = $1.1 \cdot 10^5$ confirming that the Haaland equation is valid for calculating f_D.

 ©2013 COMSOL

Notes About the COMSOL Implementation

The film heat transfer coefficients and wall resistance are automatically computed by COMSOL's Wall Heat Transfer feature.

This modeling example involves three study.

1. The oil's temperature is calculated accounting for friction heating and heat lost to the surroundings when the pipeline is uninsulated.
2. The other extreme case is computed: perfect insulation (no heat leakage to the surroundings).
3. The optimal thickness of an insulating layer is computed through optimization. The insulation thickness is sought so that the outlet temperature is equal to the inlet temperature.

References

1. S.E. Haaland, "Simple and Explicit Formulas for the Friction Factor in Turbulent Flow," *Journal of Fluids Engineering (ASME)*, vol. 103, no. 5, pp. 89–90, 1983.

2. F.P. Incropera and D.P. DeWitt, *Fundamentals of Heat and Mass Transfer*, 4th ed., John Wiley & Sons, 1996. Eq 8.62 and Eq 7.55, respectively.

Model Library path: `Pipe_Flow_Module/Heat_Transfer/pipeline_insulation`

Modeling Instructions

MODEL WIZARD

1. Go to the **Model Wizard** window.
2. Click the **3D** button.
3. Click **Next**.
4. In the **Add physics** tree, select **Fluid Flow>Non-Isothermal Flow>Non-Isothermal Pipe Flow (nipfl)**.
5. Click **Next**.
6. Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.

7 Click **Finish**.

GEOMETRY 1

Polygon 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **More Primitives>Polygon**.

2 In the **Polygon** settings window, locate the **Coordinates** section.

3 In the **x** edit field, type `0 0 150e3`.

4 In the **y** edit field, type `0 0 0`.

5 In the **z** edit field, type `0 0 0`.

6 Click the **Build Selected** button.

Next, import parameter values from text file.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 Click **Load from File**.

4 Browse to the model's Model Library folder and double-click the file `pipeline_insulation_parameters.txt`.

MATERIALS

Now create the crude oil material needed for the simulation. The air properties needed for the external forced convection cooling is taken from the built-in materials database.

Material 1

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Material**.

2 Right-click **Material 1** and choose **Rename**.

3 Go to the **Rename Material** dialog box and type `Crude oil` in the **New name** edit field.

4 Click **OK**.

5 In the **Material** settings window, locate the **Material Contents** section.

 ©2013 COMSOL

6 In the table, enter the following settings:

PROPERTY	NAME	VALUE
Density	rho	`rho_oil`
Dynamic viscosity	mu	`mu_oil`
Heat capacity at constant pressure	Cp	`Cp_oil`
Ratio of specific heats	gamma	`gamma_oil`
Thermal conductivity	k	`k_oil`

Material Browser

1 In the **Model Builder** window, right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** window, locate the **Materials** section.

3 In the tree, select **Built-In>Air**.

4 Right-click and choose **Add Material to Model** from the menu.

NON-ISOTHERMAL PIPE FLOW

Use the default settings for the **Non-Isothermal Pipe Flow** and the **Fluid 1** nodes. You can click them if you wish to inspect the defaults.

Pipe Properties 1

1 In the **Model Builder** window, expand the **Model 1>Non-Isothermal Pipe Flow** node, then click **Pipe Properties 1**.

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

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

4 In the d_i edit field, type `70[cm]`.

5 Locate the **Flow Resistance** section. From the **Friction model** list, choose **Haaland**.

6 From the **Surface roughness** list, choose **Commercial steel (0.046 mm)**.

Temperature 1

1 In the **Model Builder** window, under **Non-Isothermal Pipe Flow** click **Temperature 1**.

2 In the **Temperature** settings window, locate the **Temperature** section.

3 In the T_{in} edit field, type `T_in`.

Heat Outflow 1

1 Right-click **Non-Isothermal Pipe Flow** and choose the point condition **Heat Transfer in Pipes>Heat Outflow**.

2 Select Point 2 only.

Inlet 1

1 Right-click **Non-Isothermal Pipe Flow** and choose the point condition **Pipe Flow>Inlet**.

2 Select Point 1 only.

3 In the **Inlet** settings window, locate the **Inlet Specification** section.

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

5 In the $q_{v,0}$ edit field, type `oil_rate`.

Wall Heat Transfer 1

Next, set up the wall heat transfer components.

1 Right-click **Non-Isothermal Pipe Flow** and choose the edge condition **Heat Transfer in Pipes>Wall Heat Transfer**.

2 Select Edge 1 only.

3 In the **Wall Heat Transfer** settings window, locate the **Heat Transfer Model** section.

4 In the T_{ext} edit field, type `T_ext`.

5 Right-click **Model 1>Non-Isothermal Pipe Flow>Wall Heat Transfer 1** and choose **Internal Film Resistance**.

Add two layers of wall material.

Wall Layer 1

1 In the **Model Builder** window, under **Non-Isothermal Pipe Flow** right-click **Wall Heat Transfer 1** and choose **Wall Layer**.

2 Right-click **Wall Layer 1** and choose **Rename**.

3 Go to the **Rename Wall Layer** dialog box and type `Steel pipe wall` in the **New name** edit field.

4 Click **OK**.

5 In the **Wall Layer** settings window, locate the **Specification** section.

6 From the k list, choose **User defined**.

7 In the associated edit field, type `k_wall`.

8 From the Δw list, choose **User defined**.

9 In the associated edit field, type `d_wall`.

Wall Layer 2

1 In the **Model Builder** window, under **Model 1>Non-Isothermal Pipe Flow** right-click **Wall Heat Transfer 1** and choose **Wall Layer**.

2 Right-click **Wall Layer 2** and choose **Rename**.

 ©2013 COMSOL

3 Go to the **Rename Wall Layer** dialog box and type `Insulation layer` in the **New name** edit field.

4 Click **OK**.

5 In the **Wall Layer** settings window, locate the **Specification** section.

6 From the *k* list, choose **User defined**.

7 In the associated edit field, type `k_ins`.

8 From the Δw list, choose **User defined**.

9 In the associated edit field, type `d_ins`.

Now add the external film resistance to model the forced convection.

External Film Resistance 1

1 In the **Model Builder** window, under **Model 1>Non-Isothermal Pipe Flow** right-click **Wall Heat Transfer 1** and choose **External Film Resistance**.

2 In the **External Film Resistance** settings window, locate the **Specification** section.

3 From the **External material** list, choose **Air**.

4 In the u_{ext} edit field, type `v_air`.

MESH 1

1 In the **Model Builder** window, under **Model 1** click **Mesh 1**.

2 In the **Mesh** settings window, locate the **Mesh Settings** section.

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

4 Click the **Build All** button.

STUDY 1

The first study calculates the temperature for the uninsulated pipeline.

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.

2 In the **Stationary** settings window, locate the **Physics and Variables Selection** section.

3 Select the **Modify physics tree and variables for study step** check box.

4 In the **Physics and variables selection** tree, select **Model 1>Non-Isothermal Pipe Flow>Wall Heat Transfer 1>Insulation layer**.

5 Click **Disable**.

6 In the **Model Builder** window, click **Study 1**.

7 In the **Study** settings window, locate the **Study Settings** section.

8 Clear the **Generate default plots** check box.

9 Right-click **Study 1** and choose **Rename**.

10 Go to the **Rename Study** dialog box and type `No insulation` in the **New name** edit field.

11 Click **OK**.

12 Right-click **Study 1** and choose **Compute**.

RESULTS

Plot the temperature distribution along the pipeline.

1D Plot Group 1

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

2 Right-click **1D Plot Group 1** and choose **Line Graph**.

3 In the **Line Graph** settings window, locate the **Selection** section.

4 From the **Selection** list, choose **All edges**.

5 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Non-Isothermal Pipe Flow (Heat Transfer in Pipes)>Temperature (T)**.

6 Locate the **y-Axis Data** section. From the **Unit** list, choose **degC**.

7 Click to expand the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
No insulation

10 Click the **Plot** button.

Add the second study for the perfect insulation case.

ROOT

In the **Model Builder** window, right-click the root node and choose **Add Study**.

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.

3 Click **Finish**.

 ©2013 COMSOL

STUDY 2

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

2 In the **Stationary** settings window, locate the **Physics and Variables Selection** section.

3 Select the **Modify physics tree and variables for study step** check box.

4 In the **Physics and variables selection** tree, select **Model 1>Non-Isothermal Pipe Flow>Wall Heat Transfer 1**.

5 Click **Disable**.

6 In the **Model Builder** window, click **Study 2**.

7 In the **Study** settings window, locate the **Study Settings** section.

8 Clear the **Generate default plots** check box.

9 Right-click **Study 2** and choose **Rename**.

10 Go to the **Rename Study** dialog box and type `Perfect insulation` in the **New name** edit field.

11 Click **OK**.

12 Right-click **Study 2** and choose **Compute**.

RESULTS

1D Plot Group 1

1 In the **Model Builder** window, expand the **Results>1D Plot Group 1** node.

2 Right-click **Line Graph 1** and choose **Duplicate**.

3 In the **Line Graph** settings window, locate the **Data** section.

4 From the **Data set** list, choose **Solution 2**.

5 Click to expand the **Legends** section. In the table, enter the following settings:

Legends
Perfect insulation

6 Click the **Plot** button.

Finish the plot by adjusting the title and legend positioning.

7 In the **Model Builder** window, click **1D Plot Group 1**.

8 In the **1D Plot Group** settings window, click to expand the **Title** section.

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

10 Click to expand the **Legend** section. From the **Position** list, choose **Lower left**.

11 Click the **Plot** button.

Next, add an Optimization interface and set up the study to optimize the insulation thickness.

MODEL 1

In the **Model Builder** window, right-click **Model 1** and choose **Add Physics**.

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 In the **Add physics** tree, select **Mathematics>Optimization and Sensitivity>Optimization (opt)**.

3 Click **Add Selected**.

4 Click **Next**.

5 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Stationary**.

6 Click **Finish**.

OPTIMIZATION

Point Sum Objective 1

1 In the **Model Builder** window, under **Model 1** right-click **Optimization** and choose **Points>Point Sum Objective**.

2 Select Point 2 only.

3 In the **Point Sum Objective** settings window, locate the **Objective** section.

4 In the **Objective expression** edit field, type `(T_in-T)^2`.

Global Control Variables 1

1 In the **Model Builder** window, right-click **Optimization** and choose **Global Control Variables**.

2 In the **Global Control Variables** settings window, locate the **Control Variables** section.

3 In the **Control variables** table, enter the following settings:

Variable	Initial value	Lower bound	Upper bound
d_ins	0	0	

 ©2013 COMSOL

STUDY 3

Step 1: Stationary

1 In the **Model Builder** window, under **Study 3** click **Step 1: Stationary**.

2 In the **Stationary** settings window, click to expand the **Study Extensions** section.

3 Select the **Optimization** check box.

4 In the **Model Builder** window, click **Study 3**.

5 In the **Study** settings window, locate the **Study Settings** section.

6 Clear the **Generate default plots** check box.

7 Right-click **Study 3** and choose **Rename**.

8 Go to the **Rename Study** dialog box and type `Insulation thickness optimization` in the **New name** edit field.

9 Click **OK**.

10 Right-click **Study 3** and choose **Compute**.

RESULTS

1D Plot Group 1

1 In the **Model Builder** window, expand the **Results>1D Plot Group 1** node.

2 Right-click **Line Graph 2** and choose **Duplicate**.

3 In the **Line Graph** settings window, locate the **Data** section.

4 From the **Data set** list, choose **Solution 3**.

5 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Optimized insulation thickness`

6 Click the **Plot** button.

Finally, evaluate the optimal value of `d_ins`.

Derived Values

1 In the **Model Builder** window, under **Results** right-click **Derived Values** and choose **Global Evaluation**.

2 In the **Global Evaluation** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. Locate the **Data** section. From the **Data set** list, choose **Solution 3**.

3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Optimization>Global control variable d_ins (d_ins)**.

4 Click the **Evaluate** button.

 ©2013 COMSOL