COMSOL

Introduction to the Pipe Flow Module

Version: COMSOL 5.5

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries, contact Technical Support, or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located at the COMSOL Access page at www.comsol.com/support/case. Other useful links include:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/community
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Contents

Introduction

The Pipe Flow Module is an optional add-on package for COMSOL Multiphysics designed to model and simulate fluid flow, heat, and mass transfer in pipes and channels. Compressible hydraulic transients and acoustic waves can also be modeled using the Water Hammer interface and Pipe Acoustics interface, respectively. To analyze the stresses and deformation in the pipes, the Pipe Mechanics interface is available. The Pipe Flow Module can address problems involving flow velocity, pressure, temperature, stresses, deformation and sound waves in pipes and channels.

Modeling pipes as curves in 2D or 3D gives a great advantage in computational efficiency over meshing and computing 3D pipes with finite diameter.
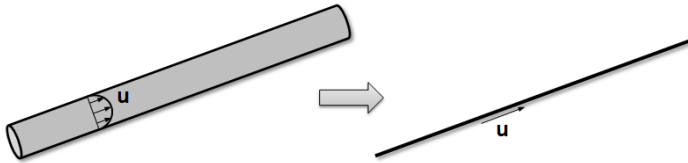

Figure 1: The Pipe Flow interface reduces the 3D flow problem to a 2D or 3D curve.

Pipe systems for which the ratio length/diameter is large enough that the flow inside of each pipe segment can be considered fully developed are suitable for the Pipe Flow Module.

The physics interfaces in the module define the conservation of momentum, energy, and mass of a fluid inside a pipe or channel. The flow rate, pressure, temperature, and concentration fields are modeled as cross-section averaged quantities, so that they only vary along the length of the pipes. The pressure losses along the length of a pipe or in a pipe component are described using friction factors. A broad range of built-in expressions for Darcy and Fanning friction factors cover the entire flow regime from laminar to turbulent flow, Newtonian and non-Newtonian fluids, different cross-sectional geometries, and a wide range of relative surface roughness values. In addition to the continuous frictional pressure drop along pipe stretches, pressure drops due to irreversible losses in components such as bends, contractions, expansions, T-junctions, Y-junctions, and valves are computed through an extensive library of industry standard loss coefficients. Pumps are also available as flow inducing devices.

The features in this module are intended for modeling and simulating incompressible and weakly compressible fluid flow in pipes and channel systems, as well as compressible hydraulic transients and acoustic waves. Typical simulations yield the velocity, pressure variation, and temperature in systems of pipes and channels. Hydraulic transients are also possible to model. These can be the result

of a valve that is closed rapidly in a pipe network, which is referred to as a water hammer.

The Pipe Mechanics interface is intended for modeling slender pipes with arbitrary cross sections. Among the computed results are displacements, rotations, stresses, strains, and section forces

The Applications

The module can be used to design and optimize complex cooling systems in turbines, analyze ventilation systems in buildings, pipe systems in the chemical process industry, and pipelines in the oil and gas industry, just to mention a few applications.
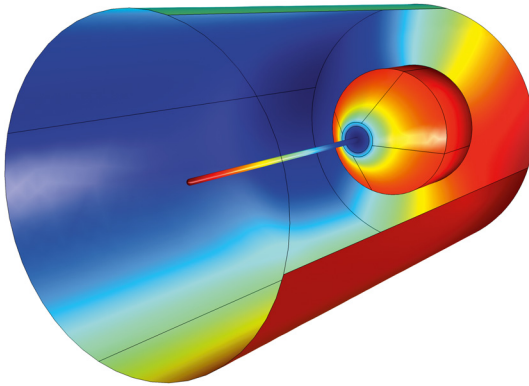

Figure 2: A Probe Tube Microphone modeled using the "Pipe Acoustics, Transient" interface in the Pipe Flow Module.

Any devices in which you find flow, waves, mass or heat transfer phenomena in narrow pipes or ducts are candidates for simulation with the Pipe Flow Module.

Classical pressure-drop and mass flow calculations through piping with bends, valves, tanks and so forth are well suited for the Pipe Flow Module.
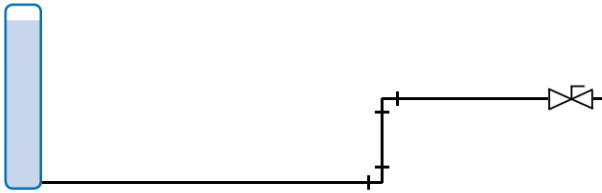

Figure 3: Tutorial example describing the discharging of a water tank through a simple pipe system with bends and valves.

For heat transfer studies, the Pipe Flow Module includes several automatic couplings to the surrounding heat sinks or sources; both shortcut methods with semi-empirical correlations for forced and convection, and also direct coupling to a 3D solid, in which the pipe is embedded.
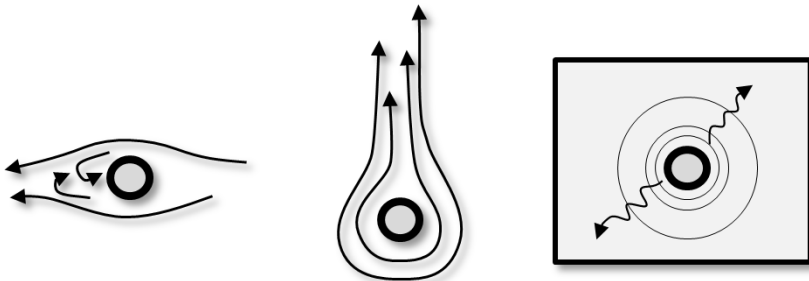

Figure 4: Methods for heat transfer to the surroundings, from left to right: forced convection, natural convection, solid conduction.

An example of an application which demonstrates the capabilities of pipe-solid coupling is the Mold Cooling model in the Pipe Flow application library. When

manufacturing devices in polymeric materials, the structural integrity of the end product is very sensitive to the cooling history in the mold.
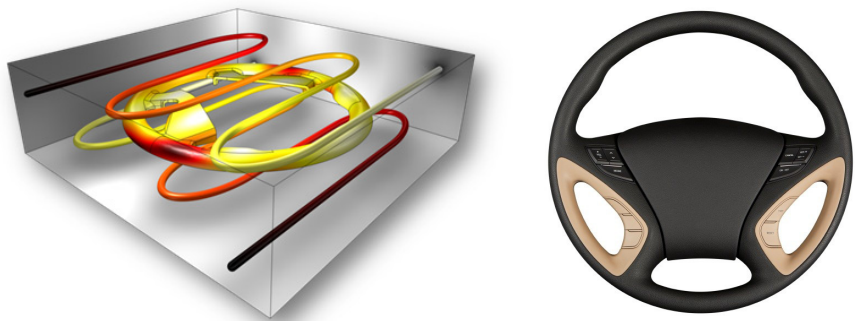

Figure 5: The heat transfer to the cooling channels embedded in a cooling mold is simulated to understand the controlled cooling of a polyurethane steering wheel. The Nonisothermal Pipe Flow interface is used in the model.

With the capabilities of modeling the transfer of chemical compounds diluted in fluids flowing through thin pipes, the pipe flow module allows for complex chemical reaction modeling. This can include mass transfer, chemical kinetics, heat transfer, and pressure drop calculations in the same model.
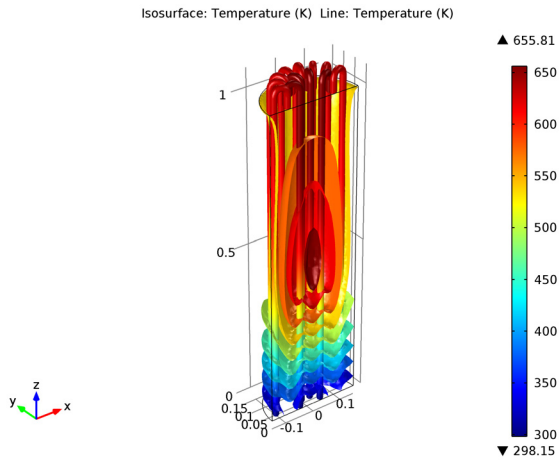

Figure 6: Temperature distribution in an autothermal chemical reactor. The model includes mass transport, chemical kinetics, heat transfer, and pressure drop and flow calculations.

Thanks to COMSOL's strong capabilities in handling so-called nonlinear materials, such as non-Newtonian fluids and materials with highly temperature dependent physical properties, oil and gas applications can be readily modeled.

One example is crude oil pipelines, where the viscous heating effects combined with the temperature dependent viscosity have great impact on the possibilities to convey oil through pumping. The Pipe Connection multiphysics coupling feature can connect a pipe segment to a 3D flow domain seamlessly and easily.

Figure 7: The Pipeline Insulation model in the application library simulates the effect of viscous heating on the cooling and transport properties of oil in a pipeline.

The Pipe Flow Module Interfaces

The module includes the following physics interfaces. See also the graphical view from the Model Wizard, shown in Figure 8.

PIPE FLOW

The Pipe Flow interface ( ) computes the pressure and velocity field in isothermal pipe systems.

HEAT TRANSFER IN PIPES

The Heat Transfer in Pipes interface ( ) computes the energy balance in pipe systems but receives the flow field as a value or as a known solved field. Wall heat transfer to the surroundings is included.

TRANSPORT OF DILUTED SPECIES IN PIPES

The Transport of Diluted Species in Pipes interface ( ) solves a mass balance equation for pipes in order to compute the concentration distribution of a solute in a dilute solution, considering diffusion, dispersion, convection, and chemical reactions.

NONISOTHERMAL PIPE FLOW

The Nonisothermal Pipe Flow interface ( ) solves the flow, pressure, and temperature simultaneously and fully coupled.

REACTING PIPE FLOW

The Reacting Pipe Flow interface ( ) solves the velocity fields, pressure, temperature, and reacting species transport simultaneously and fully coupled.

WATER HAMMER

The Water Hammer interface ( ) solves rapid hydraulic transients in pipe systems, taking the elastic properties of both the fluid and pipe wall into account.

PIPE ACOUSTICS, FREQUENCY DOMAIN

The Pipe Acoustics, Frequency Domain interface ( ) models sound waves in flexible pipe systems, with the assumption of harmonic vibrations.

Pipe Acoustics, Transient

The Pipe Acoustics, Transient interface (▥) models sound waves in flexible pipe systems, for arbitrary transient variations in pressure.

Pipe Mechanics

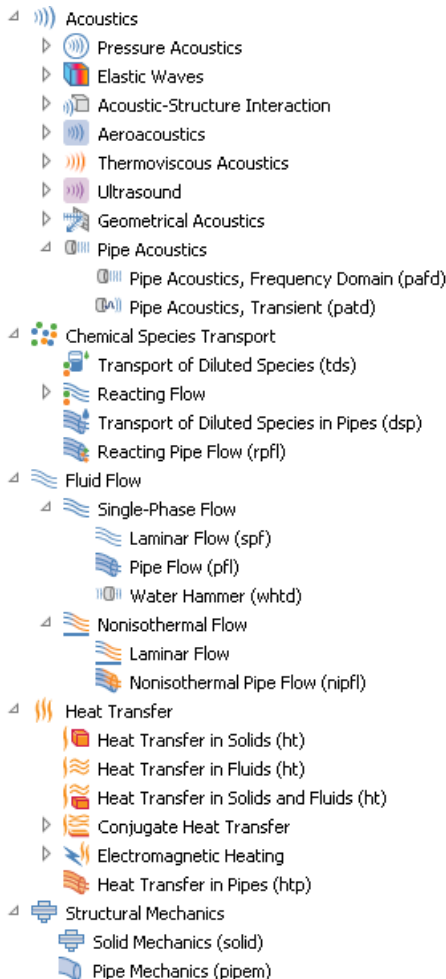The Pipe Mechanics interface (▬) is used for analysis of stresses and deformation in pipes.

Figure 8: The physics interfaces for the Pipe Flow Module as shown in the Model Wizard. The interfaces are located in different groups.

Physics Interface Guide by Space Dimension and Study Type

The table lists the physics interfaces available with this module in addition to those included with the COMSOL basic license.

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
))) **Acoustics**				
⊚ **Pipe Acoustics**				
Pipe Acoustics, Frequency Domain[1]		pafd	3D, 2D	eigenfrequency; frequency domain
Pipe Acoustics, Transient[1]		patd	3D, 2D	time dependent
⁘ **Chemical Species Transport**				
Reacting Pipe Flow		rpfl	3D, 2D	stationary; time dependent
Transport of Diluted Species in Pipes		dsp	3D, 2D	stationary; time dependent
≋ **Fluid Flow**				
≋ **Single-Phase Flow**				
Pipe Flow		pfl	3D, 2D	stationary; time dependent
Water Hammer		whtd	3D, 2D	time dependent
Nonisothermal Pipe Flow		nipfl	3D, 2D	stationary; time dependent
⦚ **Heat Transfer**				
Heat Transfer in Pipes		htp	3D, 2D	stationary; time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Pipe Mechanics		pipem	3D, 2D	stationary; eigenfrequency; time dependent, modal; time dependent, modal reduced-order model; frequency domain; frequency domain, modal; frequency domain, modal reduced-order model; time dependent; response spectrum; random vibration (PSD)
[1] Requires both the Pipe Flow Module and the Acoustics Module.				

Tutorial Example: Geothermal Heating from a Pond Loop

Ponds and lakes can serve as thermal reservoirs in geothermal heating applications. In this example, fluid circulates underwater through polyethylene piping in a closed system. Inside the pipes, cold water is flowing and picking up low grade heat from the pond. The return water will have a slightly higher temperature than the feed. A heat pump (not included in the model) will upgrade the heat for use in dwellings. Chilled water is then returned back to the pond loop. The pipes are coiled in a slinky shape and mounted onto sleds. The Nonisothermal Pipe Flow interface sets up and solves the equations for the temperature and fluid flow in the pipe system, where the geometry is represented by lines in 3D. A forced convection correlation is used (see Figure 4 above) for the external cooling effect.

Figure 9: A sled carrying pipe coils shown before the system is submerged.

Problem Specification

High density polyethylene pipe (20 mm diameter) is rolled into sixteen coils. Groups of eight coils are mounted on two sleds. Each coil has a radius of 1 m and a length of approximately 75 m. The coil groups are connected to feed and return piping with a diameter of 50 mm (see Figure 10). The coil groups are 2.4 m in

height and sit at the bottom of a pond that is 6 m deep. The total length of the piping is 1446 m.
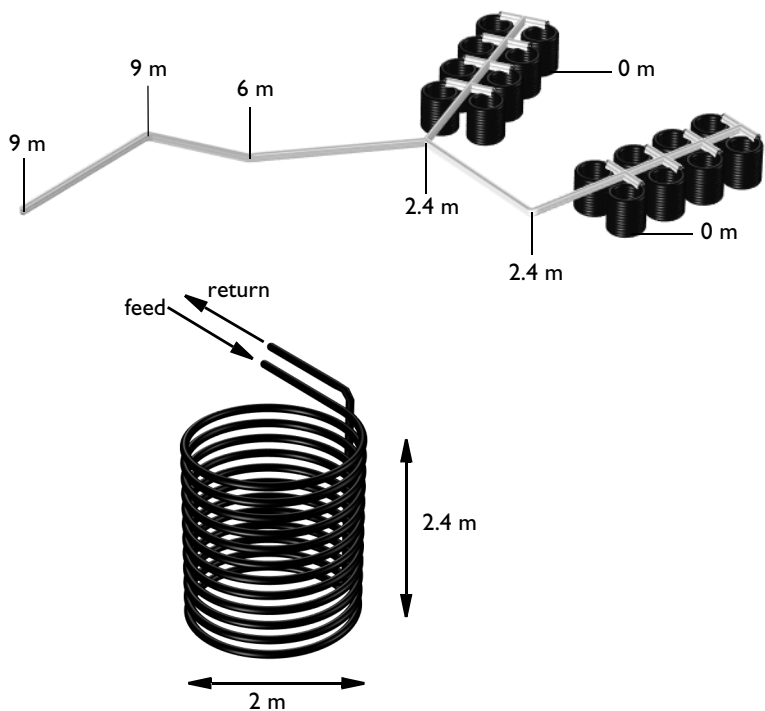

Figure 10: Polyethylene pipe system. Elevation above the pond bottom is indicated. Feed and return piping (gray) is 50 mm in diameter while coils (black) are 20 mm in diameter. The pipes are insulated above the pond surface.

The heat exchange between pond water and pipe fluid will depend, among other things, on the temperature difference between the two. A slow current in the pond will make the heat transfer more effective than water at rest. The pond is warmer closer to the surface, as shown by the temperature data in the table below.

POND TEMPERATURE

ELEVATION (M)	TEMPERATURE (K)
0	284
2	288
4	291
6	293

It is easy to set up a function in the software with linear interpolation between points so that the varying pond temperature can be taken into account in the simulation.

FLOW EQUATIONS

The stationary momentum and continuity equations below describe the stationary flow inside the pipe system, and are solved by the Nonisothermal Pipe Flow interface:

$$0 = -\nabla p - f_D \frac{\rho}{2d_h} \mathbf{u}|\mathbf{u}| + \mathbf{F} \tag{1}$$

$$\nabla \cdot (A\rho\mathbf{u}) = 0 \tag{2}$$

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 in the tangential direction of the pipe curve segment, and p (SI unit: N/m^2) is the pressure. $\mathbf{F}$ (SI unit: N/m^3) is a volume force, like gravity.

Gravity can be included explicitly in the model, but since the variation in density is negligible, and the model is not pressure driven, the only effect of including gravity is a change in the total pressure level. It is therefore common modeling practice to exclude gravity from $\mathbf{F}$ and interpret the pressure variable as the reduced pressure $p_r = p - \rho g(z_0 - z)$, where z_0 is the datum level of the free liquid surface. This reduces the model complexity and yields the same results. For models where you want to include gravity effects you can add a Volume Force node.

EXPRESSIONS FOR THE DARCY FRICTION FACTOR

The last two terms of Equation 1 describes the pressure drop due to internal viscous shear and gravity. One of the terms contain the Darcy friction factor, f_D, which is a function of the Reynolds number and the surface roughness divided by the

hydraulic pipe diameter, e/d_h. The Nonisothermal Pipe Flow interface provides a library of built-in expressions for the Darcy friction factor, f_D.
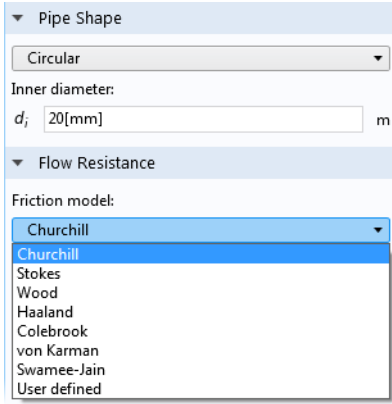

Figure 11: Select a shape and different predefined friction models in the Pipe Properties node.

This example uses the Churchill relation (Ref. 1) that is valid for laminar flow, turbulent flow, and the transitional region in between. The Churchill relation is:

$$f_D = 8\left[\left(\frac{8}{\mathrm{Re}}\right)^{12} + (A + B)^{-1.5}\right]^{1/12} \tag{3}$$

where

$$A = \left[-2.457\ln\left(\left(\frac{7}{\mathrm{Re}}\right)^{0.9} + 0.27(e/d)\right)\right]^{16} \tag{4}$$

$$B = \left(\frac{37530}{\mathrm{Re}}\right)^{16} \tag{5}$$

As seen from the equations above, the friction factor is a function of the surface roughness divided by diameter of the pipe. Surface roughness data can be selected from a predefined list in the Pipe Properties feature.
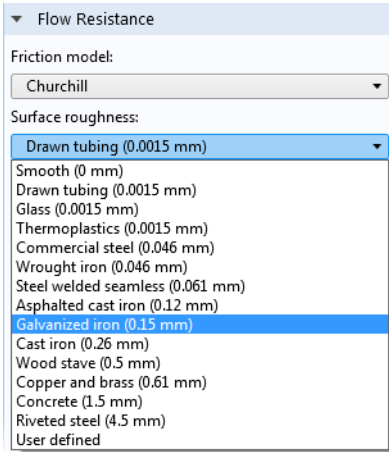

Figure 12: Specify the surface roughness in the Pipe Properties node.

The Churchill equation is also a function of the fluid properties, through the Reynolds number:

$$\text{Re} = \frac{\rho u d}{\mu} \tag{6}$$

The physical properties of water as a function of temperature are directly available from the software's built-in material library. Inspection of Equation 3 reveals that for low Reynold's number (at laminar flow), the friction factor is $64/\text{Re}$, and for very high Reynolds number, the friction factor is independent of Re.

HEAT TRANSFER EQUATIONS

The energy equation for the pipeline flow is:

$$\rho A C_p \mathbf{u} \cdot \nabla T = \nabla \cdot A k \nabla T + f_\text{D} \frac{\rho A}{2 d_\text{h}} |\mathbf{u}|^3 + Q_\text{wall} \tag{7}$$

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 7 corresponds to friction heat dissipated due to viscous shear. Q_wall (SI unit: W/m) is a source/sink term due to heat exchange with the surroundings through the pipe wall:

$$Q_{\text{wall}} = hZ(T_{\text{ext}} - T) \tag{8}$$

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

If there are several wall layers, the overall heat transfer coefficient includes contributions from internal film resistance, layered wall resistance, and external film resistance. The heat transfer coefficient is automatically calculated taking all of these resistances into account.

In this model we only have one wall and the properties of the pipe wall are given in the table below.

NAME	VALUE	DESCRIPTION
d_{wall}	2 mm	Pipe wall thickness
k_{wall}	0.46 W/(m·K)	Pipe wall thermal conductivity

Results

Figure 13 shows the pressure (Pa) in the 1446 m pipe system assuming that water enters the system at a rate of 4 l/s.
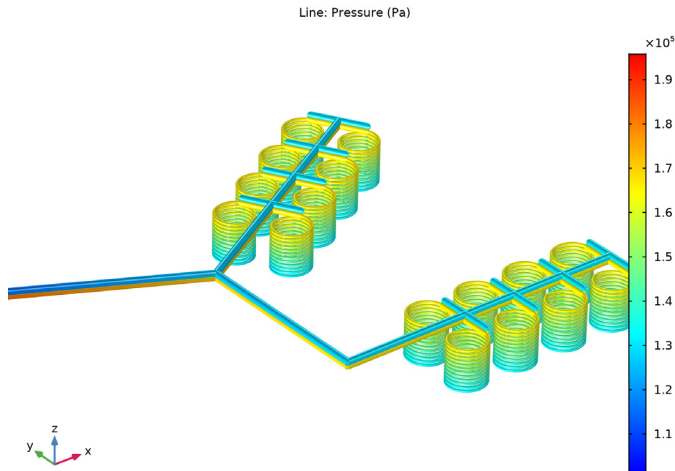

Figure 13: Pressure drop over the pipe system due to flow losses.

The plot below shows the temperature distribution for the pipe fluid. It enters the pipe system at 5 °C and exits with a temperature of approximately 11 °C.
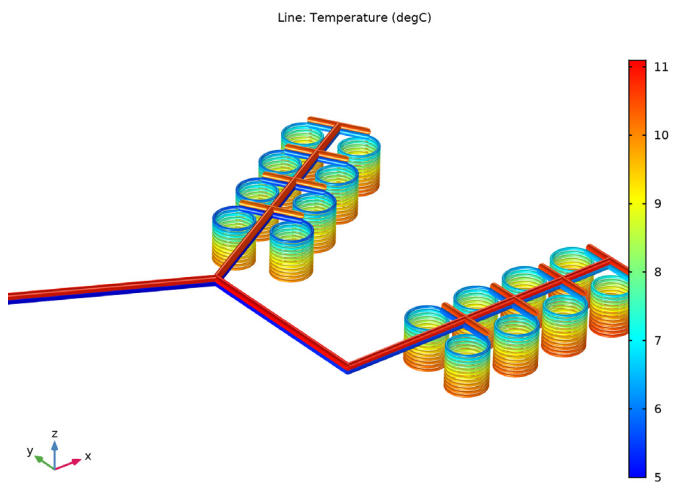
Line: Temperature (degC)

Figure 14: Temperature of the pipe fluid.

Turbulent flow conditions in the loop are important for good heat exchange between the pipes and the surroundings. A plot of the Reynolds number is shown in Figure 15, confirming that flow is turbulent (Re > 3000) throughout the system.
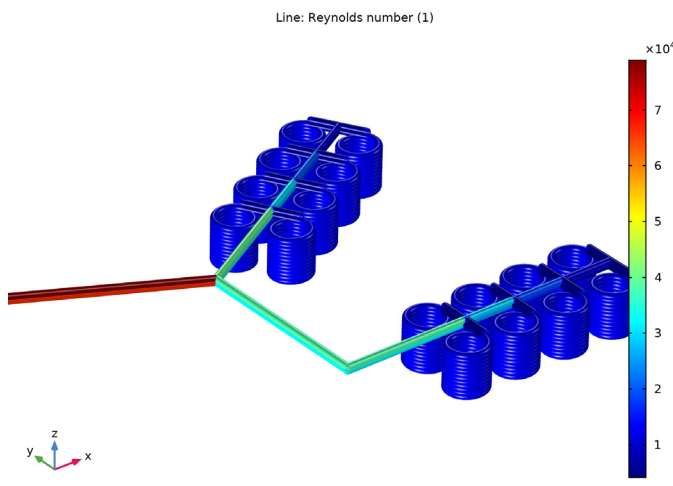
Line: Reynolds number (1)

Figure 15: The Reynolds number in the pipe loop confirms that the flow conditions are turbulent.

References

1. S.W. Churchill, "Friction factor equations span all fluid-flow regimes," *Chem. Eng.*, vol. 84, no. 24, p. 91, 1997.

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 9.34, respectively.

3. V. Gnielinski, *Int. Chem. Eng.* vol. 16, p. 359, 1976.

4. S. W. Churchill, M. Bernstein, *J Heat Transfer,* vol. 99, p. 300, 1977.

Model Wizard

The first step to build a model is to open COMSOL and then select the physics interface and specify the type of analysis you want to do—in this case, a stationary, Nonisothermal Flow analysis.

Note: These instructions are for the user interface on Windows but apply, with minor differences, also to Linux and Mac.

1 To start the software, double-click the COMSOL icon on the desktop. When the software opens, you can choose to use the Model Wizard to create a new COMSOL model or Blank Model to create one manually. For this tutorial, click the Model Wizard button.

If COMSOL is already open, you can start the Model Wizard by selecting New from the File menu and then click Model Wizard .

The Model Wizard guides you through the first steps of setting up a model. The next window lets you select the dimension of the modeling space.
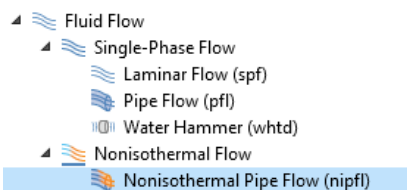
2 In the Select Space Dimension window click 3D .

3 In the Select Physics tree under Fluid Flow>Nonisothermal Flow, click Nonisothermal Pipe Flow (nipfl) .

4 Click Add and then click Study .

5 In the Select Study window under General Studies, click Stationary .

6 Click Done .

- Fluid Flow
  - Single-Phase Flow
    - Laminar Flow (spf)
    - Pipe Flow (pfl)
    - Water Hammer (whtd)
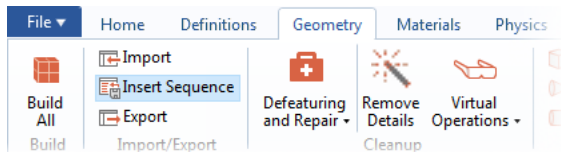 - Nonisothermal Flow
    - Nonisothermal Pipe Flow (nipfl)

Geometry

Note: The location of the files used in this exercise varies based on your installation. For example, if the installation is on your hard drive, the file path is similar to `C:\Program Files\COMSOL55\Multiphysics\applications\`.

Start by creating the pipe system geometry. You can simplify this by inserting a prepared geometry sequence from a file.

1 On the Geometry toolbar click Insert Sequence 📑.

2 Browse to the application library folder and under
`Pipe_Flow_Module\Heat_Transfer` double-click the file
`geothermal_heating_geom_sequence.mph`.

3 On the Geometry toolbar click Build All 🔳 .

You can at a later time inspect the geometry sequence in the Model Builder tree in detail if you want to learn how to build such a geometry from scratch.

Definitions

Now add some data in the form of interpolation tables and variables.

Interpolation 1

The water in the pond is somewhat stratified and has a temperature gradient along the depth direction. This has been measured earlier and will be added as an interpolation table.

1 On the Definitions toolbar click Interpolation 📉.

Note: On Linux and Mac, the Home toolbar refers to the specific set of controls near the top of the Desktop.

2 In the Settings window for Interpolation under Definition, enter the following in the table:

T	F(T)
0	284
2	288
4	291
6	293

3 Locate the Units section.

- In the Arguments text field, type m.
- In the Function text field, type K.

Settings

Interpolation

Plot Create Plot

Label: Interpolation 1

▼ Definition

Data source:	Local table
Function name:	int1

t	f(t)
0	284
2	288
4	291
6	293

▷ Interpolation and Extrapolation

▼ Units

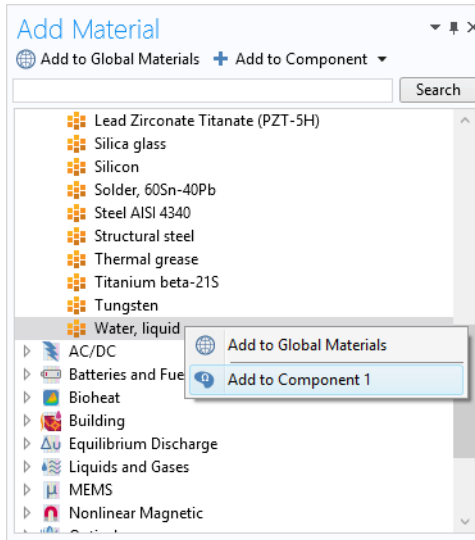| Arguments:	m
Function:	K

Variables 1

1 On the Home toolbar click Variables a= and choose Local Variables a= .

2 In the Settings window for Variables locate the Variables section. Click Load from File .

3 Browse to the application library folder and double-click the file geothermal_heating_variables.txt, under Pipe_Flow_Module\Heat_Transfer.

▼ Variables

Name	Expression	Unit	Description
T_pond	int1(z)	K	Water temperature
d_wall	2[mm]	m	Thickness of pipe wall
k_wall	0.46[W/m/K]	W/(m·K)	Thermal conductivity of high density polyethylene

This should be the result of the file import for variable definitions. Note that the expression int1(z) refers to the interpolation table that was imported in the previous step, and z is the name of the height coordinate.

Materials

1 On the Materials toolbar click Add Material ![icon].

2 In the tree under Built-In, click Water, liquid ![icon] .
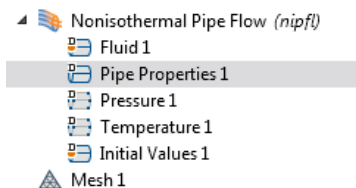
3 Click ➕ Add to Component 1.

Nonisothermal Pipe Flow

Now is the time to specify the physics feature settings, which include the flow and temperature conditions, as well as the pressure drop correlations to use in the pipe segments. Since piping systems typically consist of a large number of different segments, it can be efficient to define selections to get a manageable model.

Pipe Properties 1

1 In the Model Builder under Nonisothermal Pipe Flow, click Pipe Properties 1.

Note: In many of the Settings windows in COMSOL, there is an *Equation* section that you can expand. This shows you the equations employed by the current setting. In this case, the correlations for the default friction factor model *Churchill* is displayed. Also, if you are interested in the underlying theory of any Settings window, press F1 or the Help button ? . This takes you to the documentation section describing the current Settings window.

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

- From the Shape list, choose Circular.
- In the d_i text field, type 20[mm].

Temperature 1

1 In the Model Builder under Nonisothermal Pipe Flow, click Temperature 1.

2 In the Settings window for Temperature locate the Temperature section. In the T_{in} text field, type 5[degC].
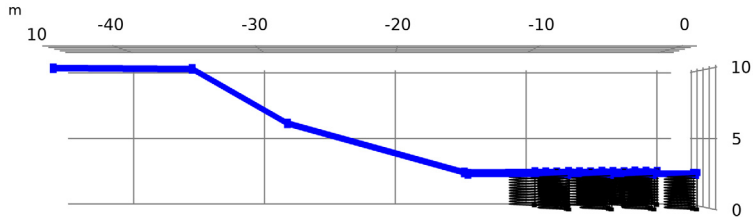
Pipe Properties 2

Now you can assign additional different properties to a new group of pipe segments.

1 Right-click Nonisothermal Pipe Flow 🝰 and choose Pipe Properties.

2 Select the edges 1–12, 15, 17–21, 27–29, 33–35, 39, 41–45, 51–53, 57–59, 63, 65–69, 75–77, 81–83, 87, 89–91, 97, 98, 102, and 103.

This selection represents the feed and return pipes from the coils. It can be somewhat tedious to select them only by looking at the numbered list above. However, since we know visually which part this is, you can select it graphically.
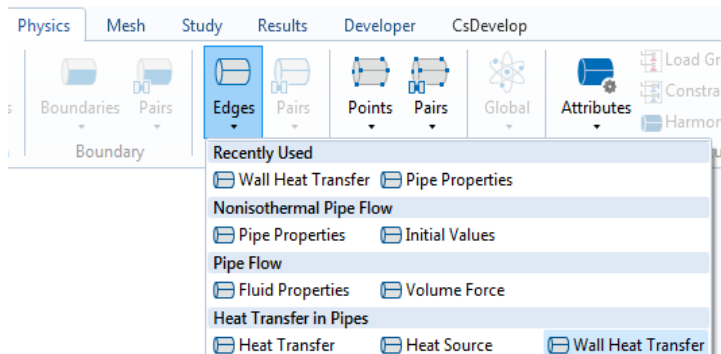
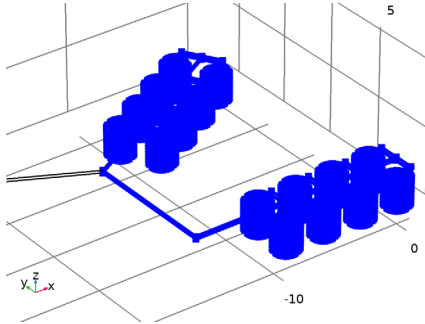- In the Graphics window, click Go to ZX View ⌊ᶻˣ and then Select Box 🖾.

- In the Graphics window, draw a box around the pipes highlighted in blue in the picture above.
- Click Go to Default 3D View ↙ button.

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

- From the list, choose Circular.
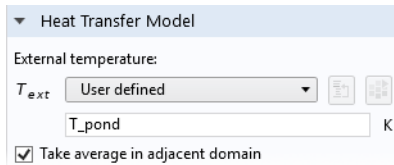- In the d_i text field, type 50[mm].

Wall Heat Transfer 1

1 On the Physics toolbar click Edges ⊟ and choose Wall Heat Transfer.

2 Select Edges 7–104 only, using the Select Box tool as before. The parts corresponding to these edges are shown in blue color below.

3 In the Settings window for Wall Heat Transfer locate the Heat Transfer Model section. In the T_{ext} text field, type T_pond.

4 Right-click Wall Heat Transfer 1 ⬚ and choose Internal Film Resistance ⬚ , This adds an automatic calculation of the heat transfer coefficient, based on the Nusselt number.

Wall Layer 1

Now the polyethene pipe is defined by adding a wall layer to the Wall Heat Transfer feature.

1 In the Model Builder under Nonisothermal Pipe Flow, right-click Wall Heat Transfer 1 ⬚ and choose Wall Layer ⬚ .

2 In the Settings window for Wall Layer locate the Specification section.

- From the k list, choose User defined.
- In the k text field, type k_wall.
- From the Δw list, choose User defined.
- In the text field, type d_wall.

More than one Wall Layer can be added if you want a multilayered pipe wall.

External Film Resistance

1 In the Model Builder right-click Wall Heat Transfer 1 ⊟ and choose External Film Resistance ⊟.

> ◢ ⊟ Wall Heat Transfer 1
>   ⊟ Internal Film Resistance 1
>   ⊟ Wall Layer 1
>   ⊟ External Film Resistance 1

2 In the Settings window for External Film Resistance locate the Specification section.

- In the External film heat transfer model list, External forced convection should now be active by default.
- From the Surrounding fluid list, choose Water, liquid.
- In the u_{ext} text field, type 0.2[m/s].

The external slow flow of 0.2 m/s is the mild current in the pond. This is enough to consider it forced convection outside the tubes. The temperature dependent material properties of water are all taken from the Materials node as added earlier.

> ▼ Specification
>
> External film heat transfer model:
> | External forced convection ▼ |
>
> Surrounding fluid:
> | Water, liquid (mat1) ▼ |
>
> Heat capacity at constant pressure:
> c_p | From material ▼ |
>
> Thermal conductivity:
> k | From material ▼ |
>
> Density:
> ρ | From material ▼ |
>
> Dynamic viscosity:
> μ | From material ▼ |
>
> External velocity:
> u_{ext} | User defined ▼ |
> | 0.2[m/s] | m/s
>
> External pressure:
> p_{ext} | User defined ▼ |
> | 1[atm] | Pa

Inlet 1

1 On 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.

- From the Specification list, choose Volumetric flow rate.
- In the $q_{v,0}$ text field, type 4[l/s] (as in liters/second).

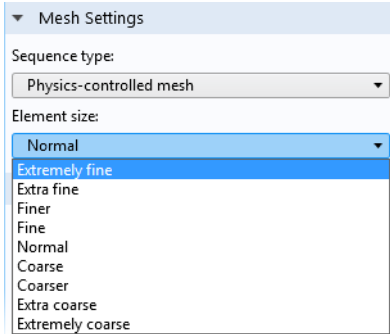The outlet condition is already set in the default node Pressure 1.

Heat Outflow 1

1 On the Physics toolbar click Points ⊟ and choose Heat Outflow ⊟.

2 Select Point 2 only.

Mesh

1 In the Model Builder under Component 1, click Mesh 1 ⬚.

2 In the Settings window for Mesh locate the Mesh Settings section. From the Element size list, choose Extremely fine.

3 Click the Build All button ⬚ .

4 Click the Go to Default 3D View button ↳ on the Graphics toolbar.
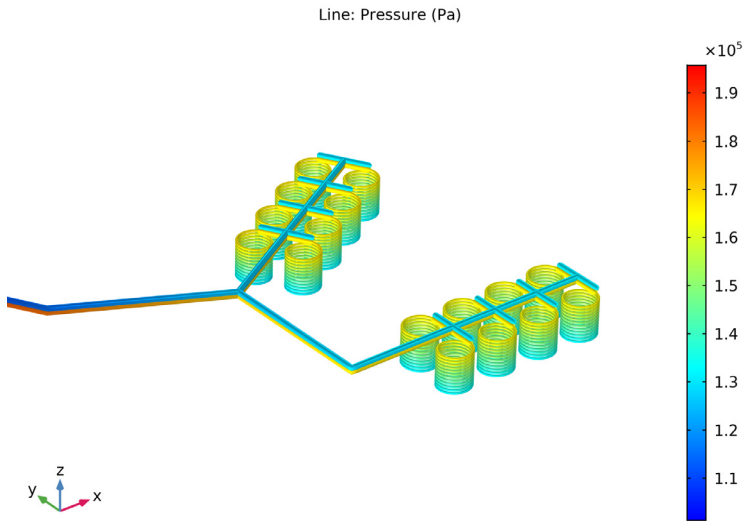
Solve

On the Home toolbar click Compute button ═ .

View the Results

Pressure

The default plot groups show the pressure (Figure 13), velocity, and temperature (Figure 14) in the pipe system. To get a better view, do as follows:

1 Click the Zoom Box ⊕ button on the Graphics toolbar.

2 Draw a box in the Graphics window to zoom in on the two coil packages.

Line: Pressure (Pa)

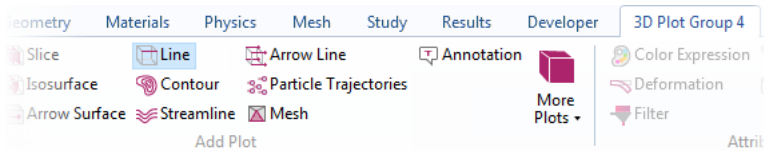*Temperature*

The following instructions reproduce plot Figure 14:

1 Expand the Temperature node, then click Line 1.

2 In the Settings window for Line, locate the Expression section. From the Unit list, choose degC.

3 Click the Plot button ⬛ .

Reproduce the Reynolds number plot in Figure 15 with the following steps.

3D Plot Group 4

1 On the Results toolbar click 3D Plot Group 🟥 .

2 When the 3D Plot Group 4 is selected in the Toolbar, an additional toolbar containing Plot Tools for the 3D Plot Group displays. On the 3D Plot Group 4 toolbar, click Line 🔲 .

3 In the Settings window for Line, click Replace Expression ◤ ▾ in the upper-right corner of the Expression section.

4 From the menu under Nonisothermal Pipe Flow, double-click Reynolds number (nipfl.Re).

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

6 Click the Plot button .

Note: In this model the bend features in the pipe corners are deliberately left out as it is assumed these do not affect the results much. As an exercise, you can add bend point features and see if this assumption is correct.