Created in COMSOL Multiphysics 6.0

Geothermal Heating from a Pond Loop

Introduction

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. 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.

Figure 1: A sled carrying pipe coils shown before the system is submerged.

Model Definition

GEOMETRY

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 2). The coil groups are 2.4 m in height an sit at the bottom of a pond that is 6 m deep. The total length of the piping is 1446 m.

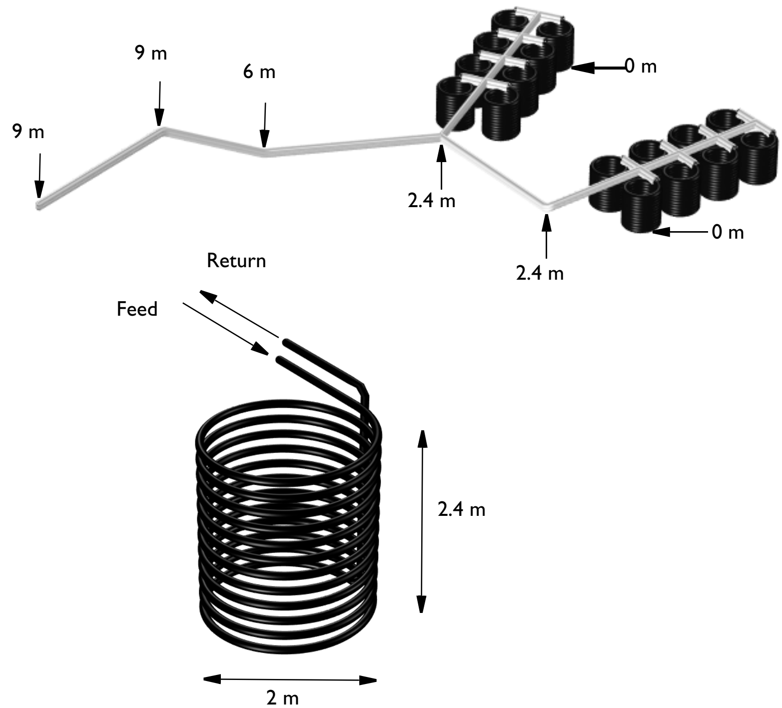


Figure 2: Polyethylene pipe system. Elevation above the pond bottom is indicated. Feed and return piping (gray) is 50 mm in diameter while coils (black) are 20 mm in diameter. The pipes are insulated above the pond surface.

The heat exchange between pond water and pipe fluid depends on the temperature difference between the two. A slow current in the pond makes the heat transfer more effective than water at rest. The pond is warmer closer to the surface, as shown by the temperature data in Table 1 below.

TABLE 1: 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 continuity and momentum equations below describe the stationary flow inside the pipe system:

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

$$0 = -\nabla p - f_{\mathrm{D}}\frac{\rho}{2d_h}\mathbf{u}|\mathbf{u}| + \mathbf{F} \qquad (1)$$

Above, A (SI unit: m^2) is the cross section area of the pipe, ρ (SI unit: $\mathrm{kg/m}^3$) is the density, $\mathbf{u}$ (SI unit: m/s) is the fluid velocity, and p (SI unit: $\mathrm{N/m}^2$) is the pressure and $\mathbf{F}$ (SI unit: $\mathrm{N/m}^3$) 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 by setting $\mathbf{F}$=0 and interpret the pressure variable as the reduced pressure $p_{\mathrm{r}} = p - \rho g(z_0 - z)$, where z_0 is the datum level of the free liquid surface. This reduces the model complexity and yields the same results.

Expressions for the Darcy Friction Factor

The right-hand side of Equation 1 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, $\mathrm{e}/d_{\mathrm{h}}$. The

Nonisothermal Pipe Flow interface provides a library of built-in expressions for the Darcy friction factor, f_D.

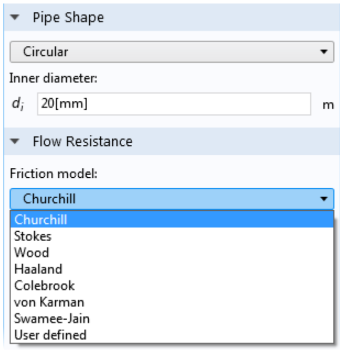


Figure 3: Select from different predefined Friction models in the Pipe Properties node.

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:

$$f_D = 8\left[\left(\frac{8}{\mathrm{Re}}\right)^{12} + (A+B)^{-1.5}\right]^{1/12} \tag{2}$$

where

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

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

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.

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

$$\mathrm{Re} = \frac{\rho u d}{\mu}$$

The physical properties of water as function of temperature are directly available from the software's built-in material library. Inspection of Equation 2 reveals that for low Reynold's number (at laminar flow), the friction factor is 64/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:

$$\rho A C_p \mathbf{u} \cdot \nabla T = \nabla \cdot A k \nabla T + f_D \frac{\rho}{2 d_h} |\mathbf{u}|^3 + Q_{\text{wall}} \tag{3}$$

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

$$Q_{\text{wall}} = hZ(T_{\text{ext}} - T) \tag{4}$$

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

The overall heat transfer coefficient includes contribution from internal film resistance, wall resistance, and external film resistance. For a circular pipe, under assumption that the heat transfer through the wall is quasi static and that the temperature is equal around the circumference of the pipe, an effective hZ in Equation 4 is given by

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

where r_n is the outer radius of wall n, h_{int} and h_{ext} are the film heat transfer coefficients on the inside and outside of the tube, and k_n is the thermal conductivity of wall n.

The film resistance inside the pipe is given by:

$$h_{\text{int}} = \text{Nu}_{\text{int}} \frac{k_{\text{water}}}{d}$$

The internal Nusselt number is taken as 3.66 for the laminar flow regime (Ref. 2), and for the turbulent flow regime the Gnielinski correlation for internal pipe flow is used (Ref. 3):

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

The external film resistance around the pipe is:

$$h_{\mathrm{ext}} = \mathrm{Nu}_{\mathrm{ext}}\frac{k_{\mathrm{water}}}{d}$$

A slow current is present in the pond. For external forced convection around a pipe, COMSOL uses the Churchill and Bernstein (Ref. 4) relation for Nu, valid for all Re and for $\mathrm{Pr} > 0.2$,:

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

where $\mathrm{Pr} = C_p\mu/k$.

Properties of the pipe wall is given in the table below.

TABLE 2: PIPE PROPERTIES.

Name	**Value**	**Description**
d_{wall}	2 mm	Pipe wall thickness
k_{wall}	0.46 W/(m·K)	Pipe wall thermal conductivity

Results and Discussion

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

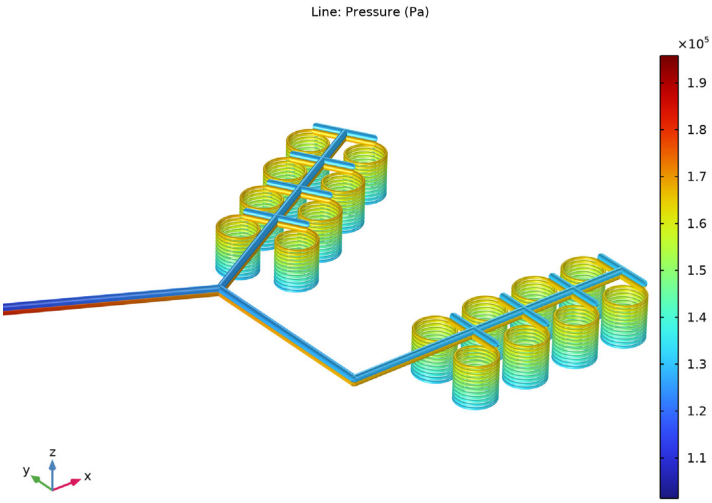


Figure 4: Pressure drop over the pipe system.

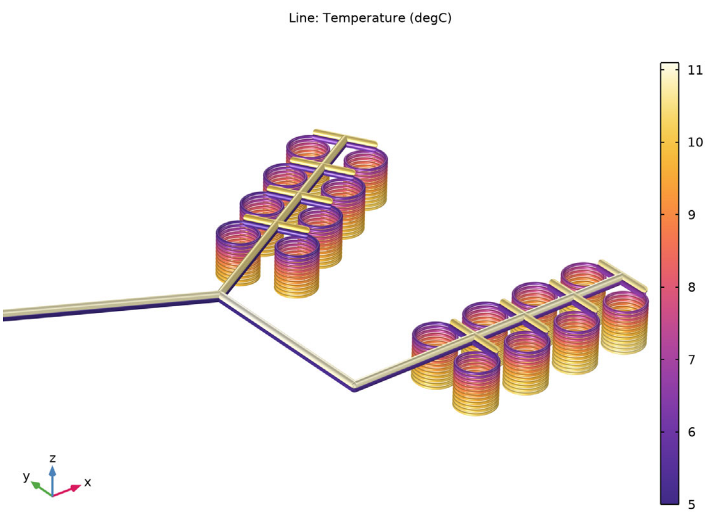


Figure 5: Temperature of the pipe fluid.

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

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 6, confirming that flow is turbulent (Re > 3000) throughout the system.

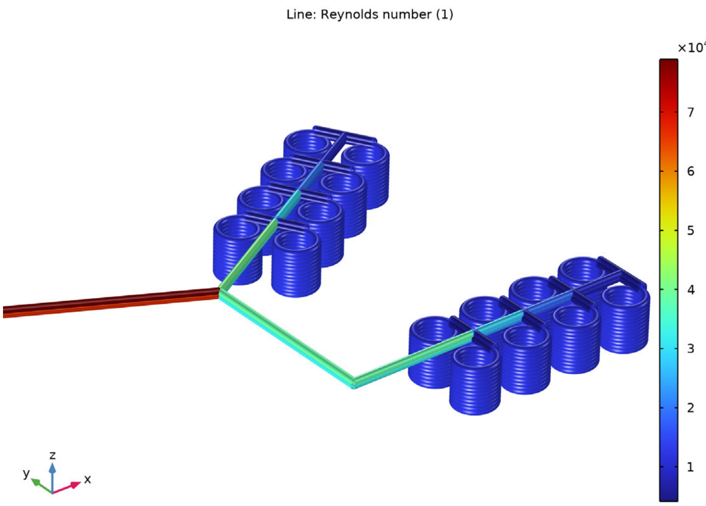


Figure 6: The Reynolds number in the pipe loop confirms that the flow conditions are turbulent.

Note: This model is also available in an extended version in the *Introduction to Pipe Flow Module* booklet.

References

1. S.W. Churchill, “Friction factor equation spans 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, "New Equation for Heat and Mass Transfer in Turbulent Pipe and Channel Flow," *Int. Chem. Eng.*, vol. 16, pp. 359–368, 1976.

4. S.W. Churchill and M. Bernstein, "A Correlating Equation for Forced Convection from Gases and Liquids to a Circular Cylinder in Crossflow," *J Heat Transfer*, vol. 99, p. 300, 1977.

Application Library path: `Pipe_Flow_Module/Heat_Transfer/geothermal_heating`

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 **3D**.

2 In the **Select Physics** tree, select **Fluid Flow>Nonisothermal Flow>Nonisothermal Pipe Flow (nipfl)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 Click **Done**.

GEOMETRY 1

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

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `geothermal_heating_geom_sequence.mph`.

3 In the **Geometry** toolbar, click **Build All**.

The complete instructions for creating this geometry can be found in the appendix at the end of this document.

DEFINITIONS

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

Interpolation 1 (int1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the table, enter the following settings:

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

4 Locate the **Units** section. In the **Argument** table, enter the following settings:

Argument	Unit
t	m

5 In the **Function** table, enter the following settings:

Function	Unit
int1	K

Variables 1

1 In the **Home** toolbar, click a= **Variables** and choose **Local Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `geothermal_heating_variables.txt`.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in>Water, liquid**.

4 Click **Add to Component** in the window toolbar.

5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

NONISOTHERMAL PIPE FLOW (NIPFL)

Pipe Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)> Nonisothermal Pipe Flow (nipfl)** 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 `20[mm]`.

Temperature 1

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

2 In the **Settings** window for **Temperature**, locate the **Temperature** section.

3 In the T_{in} text field, type `5[degC]`.

Pipe Properties 2

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

2 Select 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 only.

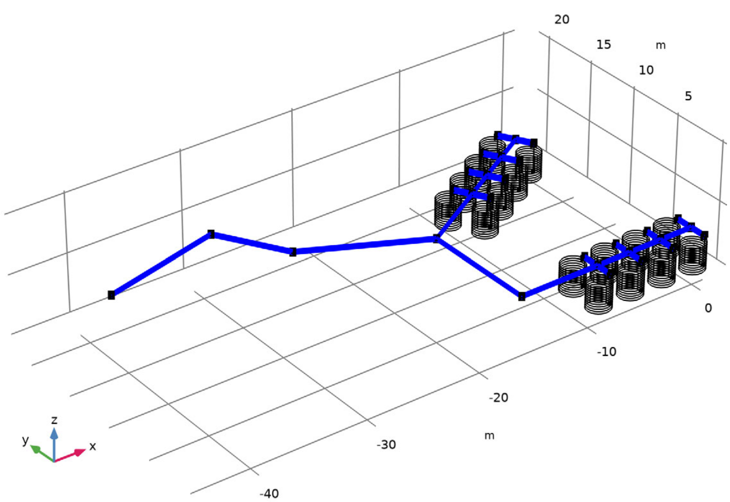
   To make the selection easily, first click **Go to XZ View** and then click **Select Box**. In the **Graphics** window, draw a box around the pipes to select them. Click the **Go to Default View** button. Alternatively, copy the entity numbers from the text, click in the selection box, and then press Ctrl+V.

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 `50[mm]`.

6 Click the **Zoom to Selection** button in the **Graphics** toolbar.

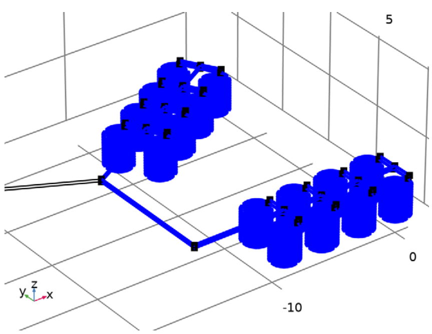


Wall Heat Transfer 1

1 In the **Physics** toolbar, click **Edges** and choose **Wall Heat Transfer**.

2 Select Edges 7–104 only.

Use the same rubber band technique as in the previous selection step.



3 In the **Model Builder** window, click **Wall Heat Transfer 1**.

4 In the **Settings** window for **Wall Heat Transfer**, locate the **Heat Transfer Model** section.

5 In the T_{ext} text field, type `T_pond`.

Internal Film Resistance 1

In the **Physics** toolbar, click **Attributes** and choose **Internal Film Resistance**.

Wall Heat Transfer 1

In the **Model Builder** window, click **Wall Heat Transfer 1**.

Wall Layer 1

1 In the **Physics** toolbar, click **Attributes** and choose **Wall Layer**.

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

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

4 In the text field, type `k_wall`.

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

6 In the text field, type `d_wall`.

Wall Heat Transfer 1

In the **Model Builder** window, click **Wall Heat Transfer 1**.

External Film Resistance 1

1 In the **Physics** toolbar, click **Attributes** and choose **External Film Resistance**.

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.

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

3 From the **Surrounding fluid** list, choose **Water, liquid (mat1)**.

4 In the u_{ext} text field, type `0.2[m/s]`.

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 `4[l/s]`.

Heat Outflow 1

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

2 Select Point 2 only.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

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

4 Click **Build All**.

5 Click the **Go to Default View** button in the **Graphics** toolbar.

STUDY 1

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

RESULTS

Pressure (nipfl)

Default plot groups show the pressure (Figure 4), velocity, and temperature (Figure 5) in the pipe system. To get a better view, do as follows:

1 Click the **Zoom Box** button in the **Graphics** toolbar. Draw a box in the **Graphics** window to zoom in on the coils.

DEFINITIONS

View 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **View 1**.

2 In the **Settings** window for **View**, locate the **View** section.

3 Clear the **Show grid** check box.

RESULTS

Temperature (nipfl)

The following instructions reproduce the plot on Figure 5.

Line 1

1 In the **Model Builder** window, expand the **Temperature (nipfl)** node, then click **Line 1**.

2 In the **Settings** window for **Line**, locate the **Expression** section.

3 From the **Unit** list, choose **degC**.

4 In the **Temperature (nipfl)** toolbar, click **Plot**.

Reproduce the Reynolds' number plot in Figure 6 with the following steps.

Reynolds' number

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Reynolds' number` in the **Label** text field.

Line 1

1 Right-click **Reynolds' number** 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 **Component 1 (comp1)>Nonisothermal Pipe Flow>nipfl.Re - Reynolds number**.

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

4 In the **Reynolds' number** toolbar, click **Plot**.

CREATING THE GEOMETRY

The previously inserted geometry can be created from scratch like this:

ADD COMPONENT

In the **Home** toolbar, click **Add Component** and choose **3D**.

GEOMETRY 1

Parametric Curve 1 (pc1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Parametric Curve**.

2 In the **Settings** window for **Parametric Curve**, locate the **Parameter** section.

3 In the **Maximum** text field, type `24`.

4 Locate the **Expressions** section. In the **x** text field, type `cos(pi*s)`.

5 In the **y** text field, type `sin(pi*s)`.

6 In the **z** text field, type `0.1*s`.

7 Click **Build Selected**.

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

x (m)	y (m)	z (m)
1	0	0
1.1	0	0
1.1	0	2.6
1.1	1.5	2.6

4 Click **Build Selected**.

Polygon 2 (pol2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

x (m)	y (m)	z (m)
1	0	2.4
1	1.5	2.4

4 Click **Build Selected**.

Mirror 1 (mir1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

3 In the **Settings** window for **Mirror**, locate the **Point on Plane of Reflection** section.

4 In the **y** text field, type 1.5.

5 Locate the **Normal Vector to Plane of Reflection** section. In the **y** text field, type 1.

6 In the **z** text field, type 0.

7 Locate the **Input** section. Select the **Keep input objects** check box.

8 Click **Build Selected**.

Array 1 (arr1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **x size** text field, type 4.

5 Locate the **Displacement** section. In the **x** text field, type -3.

6 Click **Build Selected**.

Polygon 3 (pol3)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

x (m)	y (m)	z (m)
1	1.5	2.4
-15	1.5	2.4

4 Click Build Selected.

Polygon 4 (pol4)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

x (m)	y (m)	z (m)
1.1	1.5	2.6
-15	1.5	2.6

4 Click **Build Selected**.

Array 2 (arr2)

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **y size** text field, type 2.

5 Locate the **Displacement** section. In the **y** text field, type 10.

6 Click **Build Selected**.

Polygon 5 (pol5)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

x (m)	y (m)	z (m)
-15	1.5	2.4
-15	11.5	2.4
-28	11.5	6
-35	11.5	10
-45	9	10

4 Click **Build Selected**.

Polygon 6 (pol6)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

x (m)	y (m)	z (m)
-15	1.5	2.6
-15	11.5	2.6
-28	11.5	6.2
-35	11.5	10.2
-45	9	10.2

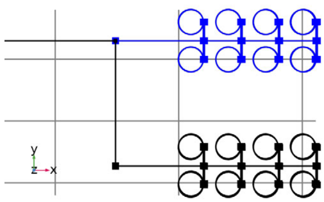
4 Click **Build Selected**.

5 Click the **Zoom Extents** button in the **Graphics** toolbar.

Rotate 1 (rot1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Click the **Go to XY View** button in the **Graphics** toolbar.



3 In the **Settings** window for **Rotate**, locate the **Rotation** section.

4 In the **Angle** text field, type 30.

5 Locate the **Point on Axis of Rotation** section. In the **x** text field, type -15.

6 In the **y** text field, type 11.5.

7 Click **Build Selected**.

8 Click the **Go to Default View** button in the **Graphics** toolbar.