Created in COMSOL Multiphysics 6.0

1D Plane Slider Bearing

Introduction

This benchmark model computes the load-carrying capacity of a one-dimensional hydrodynamic slider bearing. The results are compared with analytic expressions obtained by solving the Reynolds equations directly in this simple case (Ref. 1 provides the derivation of the results used).

Model Definition

Although the model is defined in 2D within COMSOL, the Thin-Film Flow, Edge interface is used, which means that it is effectively 1D. The Thin-Film Flow interfaces are defined in this manner to facilitate easy coupling to structural problems in higher dimensions.

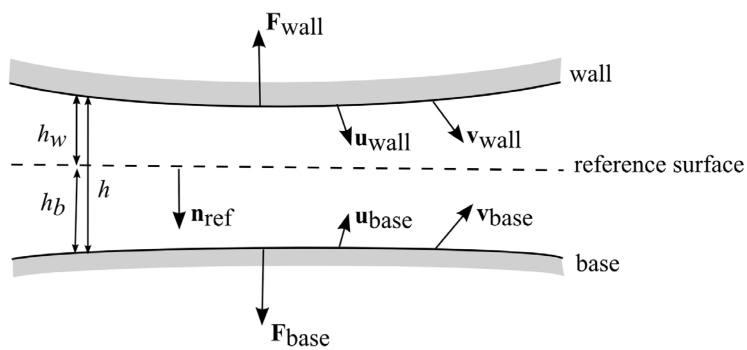

Figure 1: An example illustrating the configuration and definitions used in the Thin-Film Flow, Edge interface. Here **u** *denotes a displacement vector and* **v** *a velocity vector.*

When Thin-Film Flow is assigned to a boundary, the boundary represents a reference surface in the physical device. In practice a small gap exists at the boundary and two impermeable structures, the wall and the base, are located either side of it. The problem formulation, including definitions of the terms used, is shown in Figure 1.

In this example, the model geometry consists of a single line, with length, L (set to 1 mm in the model parameters). The line is located at the origin and aligned with the x-axis. The base is coincident with the reference surface and the height of the wall varies linearly along the line. At the origin the wall height is h_0+s_h (2.2 µm in the initial configuration) and at position (L,0) it is h_0 (0.2 µm in the initial configuration). The wall height can therefore be written as:

$$h_w = h_0 + s_h\left(1 - \frac{x}{L}\right)$$

The model defines a number of dimensionless parameters to facilitate easy comparison with theory, and h_0 and s_h are defined in terms of these parameters. A pressure is generated in the bearing by a tangential velocity of the base along the reference plane ($v_{b,x}$).

For no slip boundary conditions at the wall and the base, the Reynolds equation takes the following form for a general stationary problem:

$$\nabla_t \cdot (h\rho\mathbf{v}_{av}) - \rho(\mathbf{v}_w \cdot \nabla_t h_w + \mathbf{v}_b \cdot \nabla_t h_b) = 0$$

$$\mathbf{v}_{av} = \frac{1}{2}(\mathbf{I} - \mathbf{n}_r\mathbf{n}_r{}^T)(\mathbf{v}_w + \mathbf{v}_b) - \frac{h^2}{12\mu}\nabla_t p_f$$

here ρ is the fluid density, μ is its viscosity and p_f is the pressure developed as a result of the flow (this is the dependent variable in COMSOL). Other terms are defined in Figure 1. For this 1D problem the Reynolds equation is greatly simplified and can be written as:

$$\frac{d}{dx}\left(\frac{\rho h v_{b,x}}{2} - \frac{\rho h^3}{12\mu}\frac{dp_f}{dx}\right) = 0$$

This equation can be integrated directly to give:

$$\frac{1}{\mu}\frac{dp_f}{dx} = \frac{6v_{b,x}}{h^2} + \frac{C}{\rho h^3}$$

where C is a constant of integration. C is sometimes expressed in terms of the density (ρ_m) and height (h_m) at the position in the bearing (x_m) which the pressure gradient is zero. So given that:

$$\frac{dp_f}{dx} = 0 \qquad \text{when } x = x_m,\ \rho = \rho_m \text{ and } h = h_m$$

C is given by:

$$C = -6\rho_m v_{b,x} h_m$$

Using this notation the Reynolds equation becomes:

$$\frac{dp_f}{dx} = 6\mu v_{b,x}\frac{\rho h - \rho_m h_m}{\rho h^3}$$

If ρ and μ are assumed to be independent of the pressure p_f then:

$$\frac{dp_f}{dx} = 6\mu v_{b,x}\frac{h - h_m}{h^3}$$

In Ref. 1 this equation is solved in a dimensionless form, using the dimensionless variables:

$$P = \frac{ps_h^2}{\mu v_{b,x}L} \qquad H = \frac{h}{s_h} \qquad H_m = \frac{h_m}{s_h} \qquad H_0 = \frac{h_0}{s_h} \qquad X = \frac{x}{L}$$

The dimensionless form of the Reynolds equation is therefore:

$$\frac{dP}{dx} = 6\frac{H - H_m}{H^3}$$

Which can be solved to give:

$$P = \frac{6X(1-X)}{(H_0 + 1 - X)^2(1 + 2H_0)}$$

With this pressure distribution it is straightforward to show that the pressure takes its maximum value (P_m) at position X_m, where X_m and P_m are given by:

$$X_m = \frac{1 + H_0}{1 + 2H_0} \qquad P_m = \frac{3}{2H_0(1 + H_0)(1 + 2H_0)}$$

The dimensionless flow rate (Q=$2q/(s_h v_{b,x})$, where q is the flow rate per unit depth, q=v_{av}×h) can be shown to be:

$$Q = \frac{2H_0(1 + H_0)}{1 + 2H_0}$$

Finally the dimensionless total vertical load (L_v) and the horizontal loads acting on the wall ($L_{w,h}$) and the base ($L_{b,h}$) are given by:

$$L_v = -\frac{s_h^2}{\mu v_{b,x} L^2}\int_0^L F_{w,y}dx = 6\ln\left(\frac{1+H_0}{H_0}\right) - \frac{12}{1+2H_0}$$

$$L_{w,h} = \frac{s_h}{\mu v_{b,x} L}\int_0^L F_{w,x}dx = 4\ln\left(\frac{1+H_0}{H_0}\right) - \frac{6}{1+2H_0}$$

$$L_{b,h} = \frac{s_h}{\mu v_{b,x} L}\int_0^L F_{b,x}dx = 4\ln\left(\frac{H_0}{1+H_0}\right) + \frac{6}{1+2H_0}$$

The vertical load results from the pressure, while the horizontal loads result from the shear forces from the fluid and the in plane components of the pressure forces. For details of the derivation of these loads, see Ref. 1.

In this model COMSOL solves the bearing problem on a specific geometry, but the results are expressed in the dimensionless forms given above, for ease of comparison with the expressions and plots shown in Ref. 1.

Results and Discussion

The results of the simulation are compared with the analytic expressions discussed above in Figure 2 to Figure 7. In all cases the agreement between COMSOL and the analytic results is excellent. The ratio $H_0 = h_0/s_h$ is a measure of the slope of the wall surface–for smaller values of H_0 the slope is greater in relation to the exit height of the bearing. For smaller H_0 larger pressures can be produced in the bearing and higher loads can be sustained by the bearing. The flow rate of gas through the bearing increases with increasing H_0 as the flow tends toward a pure Couette flow, which produces no back pressure.

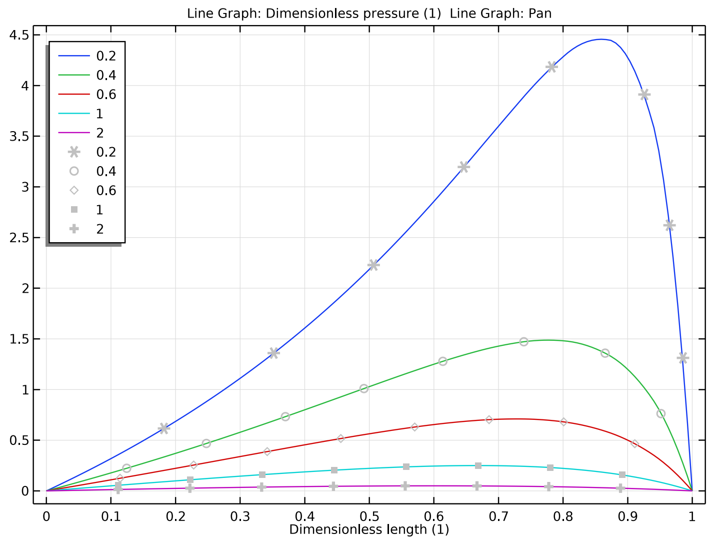

Figure 2: Nondimensional pressure vs distance along the bearing, plotted for different values of the film thickness ratio, $H_0 = h_0/s_h$. The computed results are shown as the continuous curves and the theoretical results as the gray symbols.

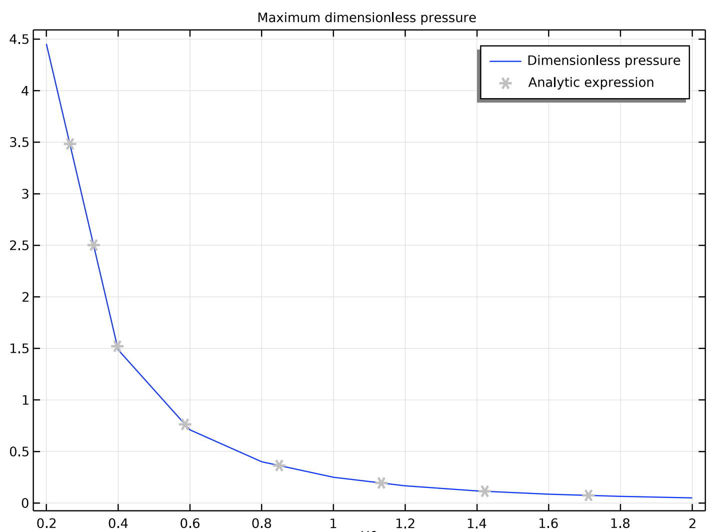

Figure 3: Nondimensional maximum pressure vs film thickness ratio, $H_0 = h_0/s_h$. The computed results are shown as the continuous curve and the theoretical result as the gray symbols.

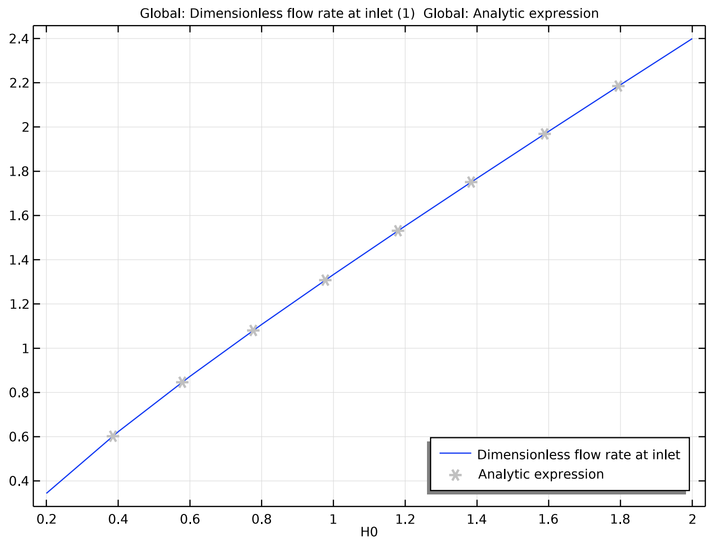

Figure 4: Nondimensional flow rate vs film thickness ratio, H_0=h_0/s_h. The computed results are shown as the continuous curve and the theoretical result as the gray symbols.

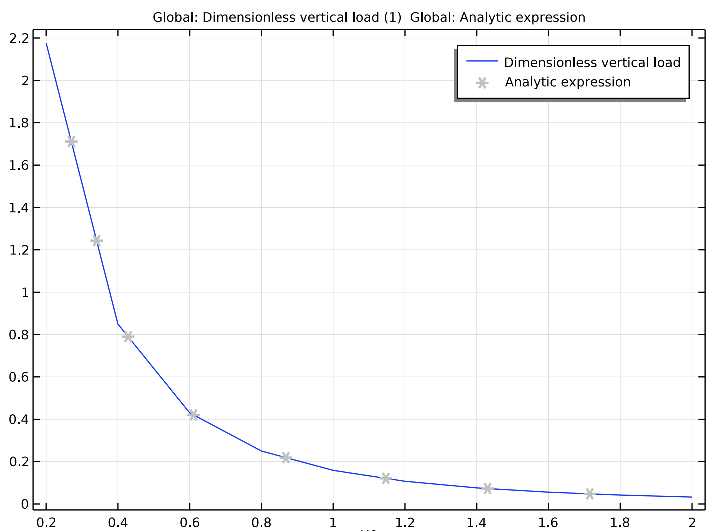

Figure 5: Nondimensional vertical load vs film thickness ratio, H_0=h_0/s_h. The computed results are shown as the continuous curve and the theoretical result as the gray symbols.

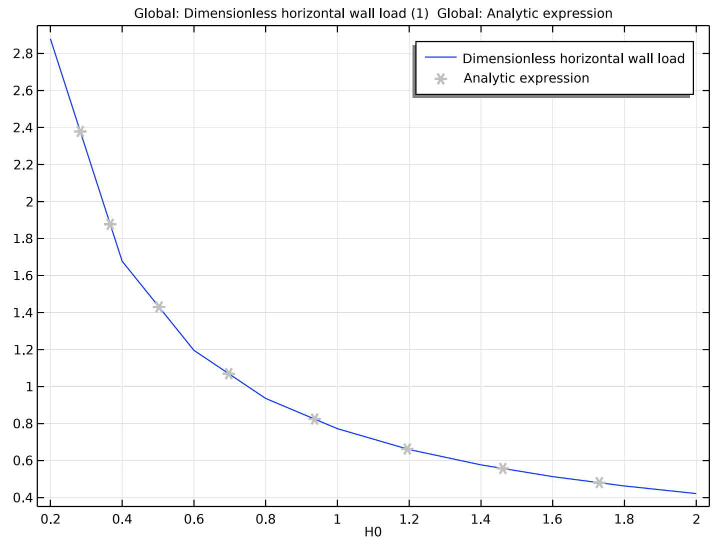

Figure 6: Nondimensional horizontal wall load vs film thickness ratio, $H_0=h_0/s_h$. The computed results are shown as the continuous curve and the theoretical result as the gray symbols.

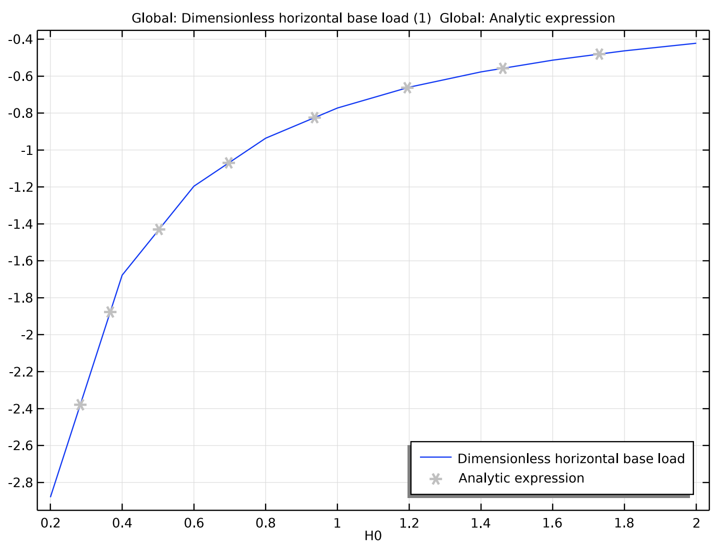

Figure 7: Nondimensional horizontal base load vs film thickness ratio, $H_0=h_0/s_h$. The computed results are shown as the continuous curve and the theoretical result as the gray symbols.

Reference

1. B.J. Hamrock, S.R. Schmid, and B.O. Jacobson, *Fundamentals of Fluid Film Lubrication*, Marcel Dekker, New York, 2004.

This example is based on the discussion entitled *Fixed-Incline Slider Bearing* in section 8.5 of the above reference.

Application Library path: `CFD_Module/Thin-Film_Flow/slider_bearing_1d`

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>Thin-Film Flow>Thin-Film Flow, Edge (tffs)**.

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
L	1[mm]	0.001 m	Bearing length
H0	0.1	0.1	Dimensionless height at end
sh	2[um]	2E-6 m	Additional height at start

Name	Expression	Value	Description
h0	sh*H0	2E-7 m	Height at end
Vb	0.1[mm/s]	1E-4 m/s	Velocity of base
mu0	0.8[Pa*s]	0.8 Pa·s	Fluid viscosity
rho0	900[kg/m^3]	900 kg/m³	Fluid density

GEOMETRY 1

Line Segment 1 (ls1)

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

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 In the **x** text field, type L.

6 Click **Build Selected**.

DEFINITIONS

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 1 only.

Integration 2 (intop2)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Point**.

4 Select Point 1 only.

Variables 1

1 In the **Definitions** toolbar, click a= **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
hw	h0+sh*(1-x/L)	m	Wall height
Xd	x/L		Dimensionless length
Pd	pfilm*sh^2/(mu0*Vb*L)		Dimensionless pressure
Hd	hw/sh		Dimensionless height
Qd	2*intop2(tffs.vavex* tffs.h)/(sh*Vb)		Dimensionless flow rate at inlet
VLd	-intop1(tffs.fwally)*sh^2/ (mu0*Vb*L^2)		Dimensionless vertical load
HLwd	intop1(tffs.fwallx)*sh/ (mu0*Vb*L)		Dimensionless horizontal wall load
HLbd	intop1(tffs.fbasex)*sh/ (mu0*Vb*L)		Dimensionless horizontal base load
Pmaxan	3/(2*H0*(1+H0)*(1+2*H0))		Analytic maximum pressure
Qan	2*H0*(1+H0)/(1+2*H0)		Analytic dimensionless flow rate
VLan	6*log((H0+1)/H0)-12/(1+2* H0)		Analytic dimensionless vertical load
HLwan	4*log((H0+1)/H0)-6/(1+2*H0)		Analytic dimensionless horizontal wall load
HLban	4*log(H0/(H0+1))+6/(1+2*H0)		Analytic dimensionless horizontal base load

Pan

1 In the **Definitions** toolbar, click **Analytic**.

2 In the **Settings** window for **Analytic**, type Pan in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type 6*X0*(1-X0)/((h0+1-X0)^2*(1+2*h0)).

4 In the **Arguments** text field, type h0, X0.

5 Right-click **Analytic 1 (an1)** and choose **Rename**.

6 In the **Rename Analytic** dialog box, type Pan in the **New label** text field.

7 Click **OK**.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Dynamic viscosity	mu	mu0	Pa·s	Basic
Density	rho	rho0	kg/m³	Basic

THIN-FILM FLOW, EDGE (TFFS)

Fluid-Film Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Thin-Film Flow, Edge (tffs)** click **Fluid-Film Properties 1**.

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

3 In the h_{w1} text field, type hw.

4 Locate the **Base Properties** section. From the $\mathbf{v}_b$ list, choose **User defined**. Specify the vector as

Vb	x
0	y

STUDY 1

Step 1: Stationary

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

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

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 From the list in the **Parameter name** column, choose **H0 (Dimensionless height at end)**.

6 Click **Range**.

7 In the **Range** dialog box, type 0.2 in the **Start** text field.

8 In the **Step** text field, type 0.2.

9 In the **Stop** text field, type 2.

10 Click **Replace**.

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

RESULTS

1D Plot Group 2

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

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Parameter selection (H0)** list, choose **From list**.

4 In the **Parameter values (H0)** list, choose **0.2**, **0.4**, **0.6**, **1**, and **2**.

5 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Line Graph 1

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

2 Select Boundary 1 only.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `Pd`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `Xd`.

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

Line Graph 2

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

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `Pan(H0,Xd)`.

4 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Gray**.

5 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

6 Find the **Line style** subsection. From the **Line** list, choose **None**.

Pressure Distribution

1 In the **Model Builder** window, right-click **1D Plot Group 2** and choose **Rename**.

2 In the **Rename 1D Plot Group** dialog box, type `Pressure Distribution` in the **New label** text field.

3 Click **OK**.

Maximum 1

1 In the **Results** toolbar, click **More Datasets** and choose **Evaluation>Maximum**.

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

3 From the **Geometry level** list, choose **Line**.

1D Plot Group 3

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Maximum 1**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Maximum dimensionless pressure`.

Global 1

1 Right-click **1D Plot Group 3** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
Pd	1	Dimensionless pressure

Global 2

1 In the **Model Builder** window, right-click **1D Plot Group 3** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
Pmaxan		Analytic expression

4 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Gray**.

5 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

6 Find the **Line style** subsection. From the **Line** list, choose **None**.

Maximum Pressure

1 Right-click **1D Plot Group 3** and choose **Rename**.

2 In the **Rename 1D Plot Group** dialog box, type `Maximum Pressure` in the **New label** text field.

3 Click **OK**.

1D Plot Group 4

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 From the **Position** list, choose **Lower right**.

Global 1

1 Right-click **1D Plot Group 4** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`Qd`	`1`	`Dimensionless flow rate at inlet`

Global 2

1 In the **Model Builder** window, right-click **1D Plot Group 4** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`Qan`		`Analytic expression`

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

5 From the **Color** list, choose **Gray**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

Flow Rate

1 Right-click **1D Plot Group 4** and choose **Rename**.

2 In the **Rename 1D Plot Group** dialog box, type `Flow Rate` in the **New label** text field.

3 Click **OK**.

1D Plot Group 5

In the **Results** toolbar, click **1D Plot Group**.

Global 1

1 Right-click **1D Plot Group 5** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`VLd`	`1`	`Dimensionless vertical load`

Global 2

1 In the **Model Builder** window, right-click **1D Plot Group 5** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`VLan`		`Analytic expression`

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

5 From the **Color** list, choose **Gray**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

Vertical Load

1 Right-click **1D Plot Group 5** and choose **Rename**.

2 In the **Rename 1D Plot Group** dialog box, type `Vertical Load` in the **New label** text field.

3 Click **OK**.

1D Plot Group 6

In the **Results** toolbar, click **1D Plot Group**.

Global 1

1 Right-click **1D Plot Group 6** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`HLwd`	`1`	`Dimensionless horizontal wall load`

Global 2

1 In the **Model Builder** window, right-click **1D Plot Group 6** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`HLwan`		`Analytic expression`

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

5. From the **Color** list, choose **Gray**.
6. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

Horizontal Load, Wall

1. Right-click **1D Plot Group 6** and choose **Rename**.
2. In the **Rename 1D Plot Group** dialog box, type `Horizontal Load, Wall` in the **New label** text field.
3. Click **OK**.

1D Plot Group 7

In the **Results** toolbar, click **1D Plot Group**.

Global 1

1. Right-click **1D Plot Group 7** and choose **Global**.
2. In the **Settings** window for **Global**, locate the **y-Axis Data** section.
3. In the table, enter the following settings:

Expression	**Unit**	**Description**
`HLbd`	`1`	`Dimensionless horizontal base load`

Global 2

1. In the **Model Builder** window, right-click **1D Plot Group 7** and choose **Global**.
2. In the **Settings** window for **Global**, locate the **y-Axis Data** section.
3. In the table, enter the following settings:

Expression	**Unit**	**Description**
`HLban`		`Analytic expression`

4. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.
5. From the **Color** list, choose **Gray**.
6. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

Horizontal Load, Base

1. In the **Model Builder** window, click **1D Plot Group 7**.
2. In the **Settings** window for **1D Plot Group**, locate the **Legend** section.
3. From the **Position** list, choose **Lower right**.
4. Right-click **1D Plot Group 7** and choose **Rename**.

5 In the **Rename 1D Plot Group** dialog box, type `Horizontal Load, Base` in the **New label** text field.

6 Click **OK**.