Created in COMSOL Multiphysics 6.2

1D Step Bearing

Introduction

This benchmark model computes the load-carrying capacity of a one-dimensional hydrodynamic step 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 Multiphysics, the Thin-Film Flow interface is applied to a geometry consisting of a 1D edge. The Thin-Film Flow interfaces are defined in this manner to facilitate easy coupling to structural problems in higher dimensions.

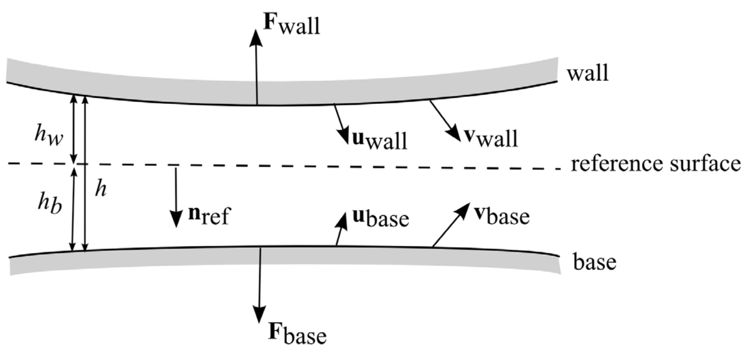

Figure 1: An example illustrating definitions used in the Thin-Film Flow interface. Here $\mathbf{u}$ *denotes a displacement vector and* $\mathbf{v}$ *a velocity vector.*

When Thin-Film Flow is assigned to 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 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. At the origin the wall height is h_0+s_h (2.2 µm in the initial configuration). A step in the wall height is located a fraction n_s along the line at coordinate (L_s,0) where L_s=Ln_s.(initially n_s=0.6 and L_s=0.6 mm). After the step the wall height is h_0 (0.2 µm in the initial configuration). The model defines a number of dimensionless parameters to facilitate easy comparison with theory and h_0 and L_s 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 Multiphysics). Other terms are defined in Figure 1. For this 1D problem the Reynolds equation is greatly simplified and can be written as:

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

For a constant value of h and assuming that ρ and μ are independent of p_f this equation simplifies further to:

$$\frac{d^2 p_f}{dx^2} = 0$$

Thus p_f takes the form of a straight line in the two regions in the bearing. The pressure is maximal at the step, where a discontinuity in the gradient exists. Using the boundary condition that $p_f = 0$ at x=0 and $x = L$ and ensuring that p_f is continuous at the step (with value p_m), gives the following equation:

$$p_m = \left(\frac{dp_f}{dx}\right)_i L_s = -L(1 - n_s)\left(\frac{dp_f}{dx}\right)_o \tag{1}$$

where the subscript i refers to the inlet and the subscript o refers to the outlet. The flow rate q=$v_{av,x}h$ must also be continuous at the step so that:

$$-\frac{(h_0 + s_h)^3}{12\mu}\left(\frac{dp_f}{dx}\right)_i + \frac{(h_0 + s_h)v_{b,x}}{2} = -\frac{h_0^{\ 3}}{12\mu}\left(\frac{dp_f}{dx}\right)_o + \frac{h_0 v_{b,x}}{2} \tag{2}$$

Equation 1 and Equation 2 can be solved simultaneously to give the values of the pressure gradients at the inlet and outlet. The resulting equations are:

$$\left(\frac{dp_f}{dx}\right)_i = \frac{6\mu v_{b,x}(1-n_s)s_h}{(1-n_s)(h_0+s_h)^3+n_s h_0^3}$$

$$\left(\frac{dp_f}{dx}\right)_o = \frac{6\mu v_{b,x} n_s s_h}{(1-n_s)(h_0+s_h)^3+n_s h_0^3}$$

p_{m} is therefore given by:

$$p_m = \frac{6\mu n_s L v_{b,x}(1-n_s)s_h}{(1-n_s)(h_0+s_h)^3+n_s h_0^3}$$

Using the dimensionless variables adopted in Ref. 1:

$$P = \frac{p s_h^2}{\mu v_{b,x} L} \qquad P_m = \frac{p_m s_h^2}{\mu v_{b,x} L} \qquad H_0 = \frac{h_0}{s_h} \qquad X = \frac{x}{L}$$

the dimensionless maximum pressure (P_m) can be expressed as:

$$P_m = \frac{6n_s(1-n_s)}{(1-n_s)(H_0+1)^3+n_s H_0^3}$$

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

$$Q = 1+H_0-\frac{P_m(1+H_0)^3}{6n_s}$$

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

$$L_v = -\frac{s_h^2}{\mu v_{b,x} L^2}\int_0^L F_{w,y}dx = \frac{P_m}{2}$$

$$L_{w,h} = \frac{s_h}{\mu v_{b,x} L}\int_0^L F_{w,x}dx = -\frac{1+H_0-n_s}{H_0(1+H_0)}-\frac{P_m}{2}$$

$$L_{b,h} = \frac{s_h}{\mu v_{b,x} L}\int_0^L F_{b,x}dx = \frac{1+H_0-n_s}{H_0(1+H_0)}-\frac{P_m}{2}$$

The vertical load results from the pressure, while the horizontal loads result from the shear forces from the fluid. An additional horizontal load on the wall results from the pressure acting on the vertical surface of the step, which is not considered by the model. For details of the derivation of these loads, see Ref. 1.

In this example, the COMSOL Multiphysics model 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 step height relative to the outlet height — for smaller values of H_0 the step height is greater in relation to the exit height of the bearing. These results show a trend of increasing load bearing capacity with reduced H_0 and increased n_s, with a corresponding increase in the maximum pressure in the bearing. As discussed in Ref. 1, there is, in fact, an optimum value of H_0 and n_s, but this optimum occurs at larger values of n_s. 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.

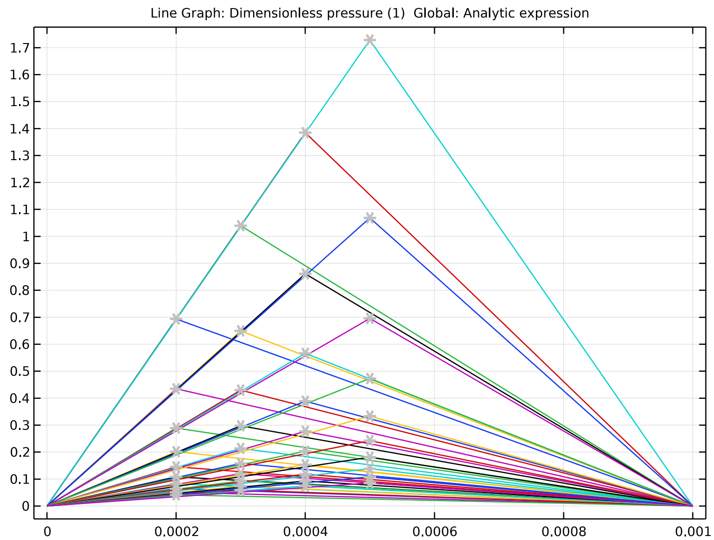

Figure 2: Nondimensional pressure vs distance along the bearing, plotted for different values of the film thickness ratio, $H_0 = h_0/s_h$ The computed results are shown as the continuous curves and the theoretical results as the gray symbols.

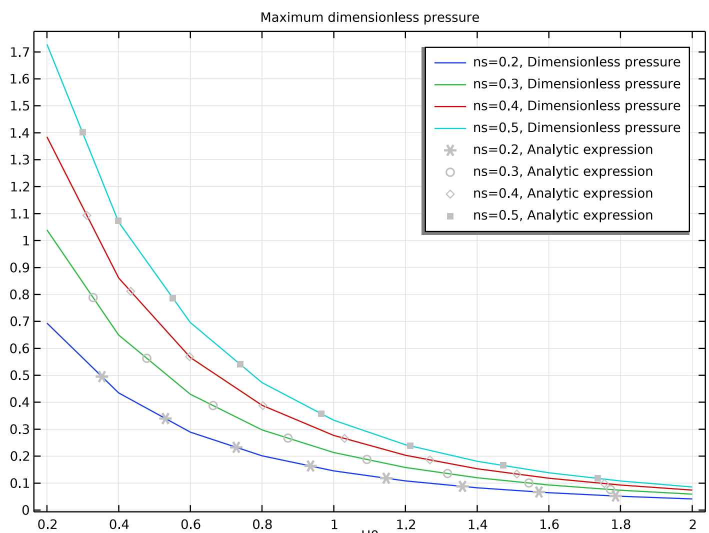

Figure 3: Nondimensional maximum pressure vs film thickness ratio, $H_0 = h_0/s_h$. The computed results are shown as the continuous curve and the theoretical result as gray symbols. Different values of the step location $n_s = L_s/L$ are shown.

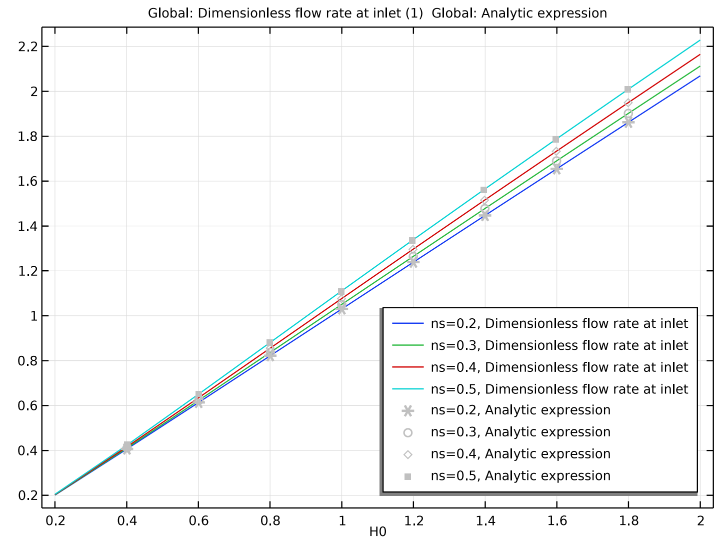

Figure 4: Nondimensional flow rate vs film thickness ratio, $H_0 = h_0/s_h$. The computed results are shown as the continuous curve and the theoretical result as gray symbols.Different values of the step location $n_s = L_s/L$ are shown.

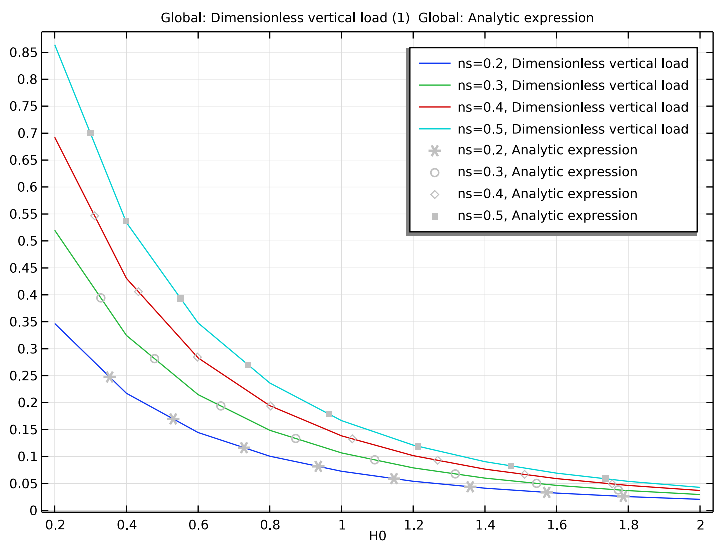

Figure 5: Nondimensional vertical load vs film thickness ratio, $H_0 = h_0/s_h$. The computed results are shown as the continuous curve and the theoretical result as gray symbols. Different values of the step location $n_s = L_s/L$ are shown.

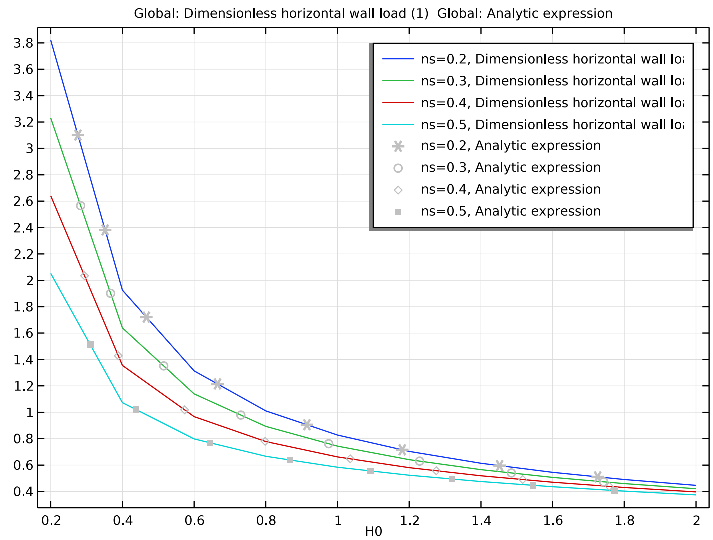

Figure 6: Nondimensional horizontal wall load vs film thickness ratio, $H_0 = h_0/s_h$. The computed results are shown as the continuous curve and the theoretical result as gray symbols. Different values of the step location $n_s = L_s/L$ are shown.

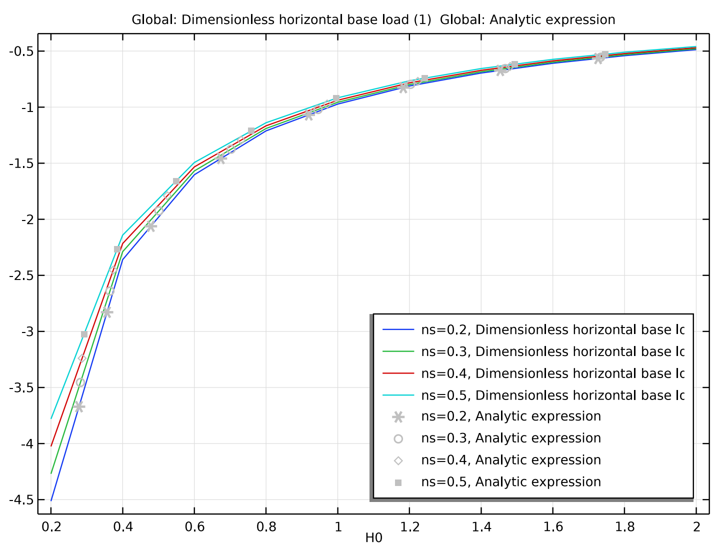

Figure 7: Nondimensional horizontal base load vs film thickness ratio, $H_0 = h_0/s_h$. The computed results are shown as the continuous curve and the theoretical result as gray symbols. Different values of the step location $n_s = L_s/L$ are shown.

Reference

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

This model is based on the discussion entitled *Parallel-Step Slider Bearing* in section 8.6 of the above reference.

Application Library path: `CFD_Module/Thin-Film_Flow/step_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 (tff)**.
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
ns	0.6	0.6	Dimensionless step location
Ls	L*ns	6E-4 m	Step x-coordinate
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

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **Polygon**.

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

3 From the **Type** list, choose **Open curve**.

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

5 In the **x** text field, type `0 L Ls`.

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

7 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 Click in the **Graphics** window and then press Ctrl+A to select both boundaries.

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
Xd	x/L		Dimensionless length
Pd	pfilm*sh^2/(mu0*Vb*L)		Dimensionless pressure
Qd	2*intop2(tff.vavex*tff.h)/(sh*Vb)		Dimensionless flow rate
VLd	-intop1(tff.fwally)*sh^2/(mu0*Vb*L^2)		Dimensionless vertical load
HLwd	intop1(tff.fwallx)*sh/(mu0*Vb*L)		Dimensionless horizontal wall load
HLbd	intop1(tff.fbasex)*sh/(mu0*Vb*L)		Dimensionless horizontal base load
Pmaxan	6*ns*(1-ns)/((1-ns)*(H0+1)^3+ns*H0^3)		Analytic dimensionless maximum pressure
Qan	-Pmaxan*(H0+1)^3/(6*ns)+H0+1		Analytic dimensionless flow rate
VLan	Pmaxan/2		Analytic dimensionless vertical load
HLwan	-Pmaxan/2+(H0+1-ns)/(H0*(1+H0))		Analytic dimensionless horizontal load, wall
HLban	-Pmaxan/2-(H0+1-ns)/(H0*(1+H0))		Analytic dimensionless horizontal load, base

Note that the tangential load at the step is not included in the expression for the analytic horizontal wall load. This is because this load acts at a point in the geometry where the height is discontinuous. COMSOL does not automatically include the additional force acting at this point, but you can add it manually if desired. In this case the force is given by the maximum pressure multiplied by the step height.

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 (TFF)

Fluid-Film Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Thin-Film Flow (tff)** 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 `h0`.

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

Vb	x
0	y

5 Right-click **Component 1 (comp1)>Thin-Film Flow (tff)>Fluid-Film Properties 1** and choose **Duplicate**.

Fluid-Film Properties 2

1 In the **Model Builder** window, click **Fluid-Film Properties 2**.

2 Select Boundary 1 only.

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

4 In the h_{w1} text field, type `h0+sh`.

STUDY 1

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 From the **Sweep type** list, choose **All combinations**.

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 **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

12 Click **Add**.

13 From the list in the **Parameter name** column, choose **ns (Dimensionless step location)**.

14 Click **Range**.

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

16 In the **Step** text field, type `0.1`.

17 In the **Stop** text field, type `0.5`.

18 Click **Replace**.

19 In the **Study** 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 **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

Line Graph 1

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

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

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

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

Global 1

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

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

8 In the **Expression** text field, type `Ls`.

9 Click to expand the **Legends** section. Clear the **Show legends** check box.

Pressure Distribution

1 In the **Model Builder** window, under **Results** click **1D Plot Group 2**.

2 In the **Settings** window for **1D Plot Group**, type `Pressure Distribution` in the **Label** text field.

Maximum 1

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

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 Locate the **Settings** section. 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`

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **H0**.

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 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **H0**.

5 Locate the **Coloring and Style** section. From the **Color** list, choose **Gray**.

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

7 From the **Positioning** list, choose **Interpolated**.

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

Maximum Pressure

1 In the **Model Builder** window, under **Results** click **1D Plot Group 3**.

2 In the **Settings** window for **1D Plot Group**, type `Maximum Pressure` in the **Label** text field.

1D Plot Group 4

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 **Study 1/Parametric Solutions 1 (sol2)**.

4 Locate the **Legend** section. 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`

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **H0**.

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 **x-Axis Data** section. From the **Axis source data** list, choose **H0**.

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

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

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

8 From the **Positioning** list, choose **Interpolated**.

Flow Rate

1 In the **Model Builder** window, under **Results** click **1D Plot Group 4**.

2 In the **Settings** window for **1D Plot Group**, type `Flow Rate` in the **Label** text field.

1D Plot Group 5

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 **Study 1/Parametric Solutions 1 (sol2)**.

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

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **H0**.

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 **x-Axis Data** section. From the **Axis source data** list, choose **H0**.

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

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

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

8 From the **Positioning** list, choose **Interpolated**.

Vertical Load

1 In the **Model Builder** window, under **Results** click **1D Plot Group 5**.

2 In the **Settings** window for **1D Plot Group**, type `Vertical Load` in the **Label** text field.

1D Plot Group 6

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 **Study 1/Parametric Solutions 1 (sol2)**.

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`

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **H0**.

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 **x-Axis Data** section. From the **Axis source data** list, choose **H0**.

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

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

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

8 From the **Positioning** list, choose **Interpolated**.

Horizontal Load, Wall

1 In the **Model Builder** window, under **Results** click **1D Plot Group 6**.

2 In the **Settings** window for **1D Plot Group**, type `Horizontal Load, Wall` in the **Label** text field.

1D Plot Group 7

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 **Study 1/Parametric Solutions 1 (sol2)**.

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

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **H0**.

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 **x-Axis Data** section. From the **Axis source data** list, choose **H0**.

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

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

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

8 From the **Positioning** list, choose **Interpolated**.

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. In the **Label** text field, type `Horizontal Load, Base`.