Created in COMSOL Multiphysics 6.2

Squeeze Film Between Rectangular Porous and Nonporous Plates

Introduction

This example describes how to simulate the flow of a thin film of fluid in the gap between two rectangular plates, one of them with a porous facing, when the fluid is squeezed as a consequence of the relative motion between the plates. The model accounts for the ingress and egress of fluid between the porous material and the thin-film region. The simple geometry of the model allows for the comparison of the computed solution with analytic expressions in Ref. 1.

Model Definition

The model is set up with parameterized quantities in a unitless base system. It uses a thin-film flow approximation, described by the Reynolds equation, for the fluid motion in the gap between the plates. The fluid flow in the porous region is described by Darcy's law. The coupling between the Thin-Film Flow interface and the Darcy's Law interface is handled by the Thin-Film and Porous Media multiphysics coupling.

GEOMETRY

The length of the rectangular plates, a, is taken to be of unit value. The width, b is controlled by the aspect ratio parameter, $k = a/b$. Using symmetry planes orthogonal to the x- and y-axes, only one quarter of the plate is modeled, as shown in Figure 1. The height of the porous facing, H, is suitably scaled with respect to the plate length using the parameter, $\overline{H} = H/a$. A rigid body motion is considered for the nonporous plate and it is therefore not included in the model geometry. The relative motion between the porous and nonporous plates can be accounted for by the motion of the "wall" or "base" defined in the Thin-Film Flow interface.

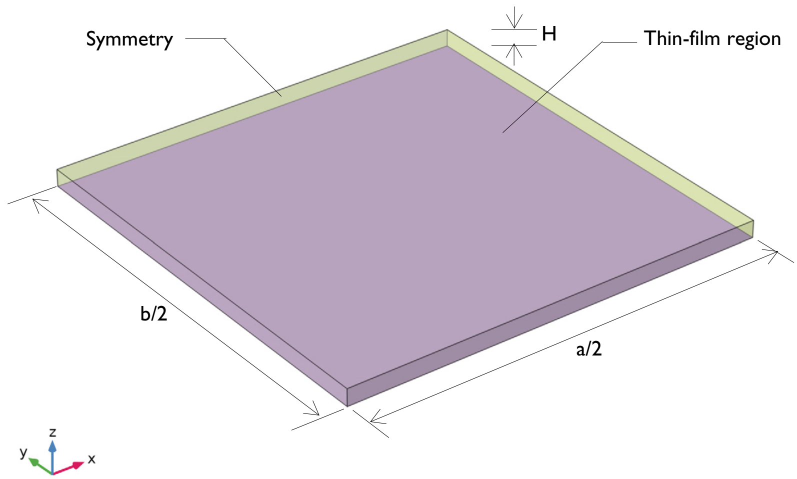

Figure 1: Computational domain with symmetry boundaries (in yellow) and the thin-film region (in purple).

PHYSICS INTERFACE SETTINGS

The Thin-Film Flow interface is set up to solve the Reynolds equation. The reference pressure, p_{ref}, is set to zero. The parameterized fluid density, ρ, and viscosity, η, are both set to unit value. The thin-film thickness is chosen to be, $h_t = 0.001$, and set as the height of the wall above the reference plane. The squeeze action is induced by applying a wall velocity in the positive z direction of unit magnitude controlled by the parameter, v_{sq}. Symmetry condition is applied to edges corresponding to the symmetry boundaries in the geometry. Border condition with zero pressure is applied to other edges as default.

The Darcy's Law interface is set up with zero reference pressure and similar fluid properties to that in the Thin-Film Flow interface. The porosity, ε_p, is set to zero and the permeability, ϕ, is chosen to be isotropic in nature, with a value controlled by the permeability parameter, $\psi = \phi H / h_t^3$. The boundary conditions include a No Flow condition on the top face, a Symmetry condition on the symmetry boundaries, and a Pressure condition with zero prescribed value elsewhere.

The multiphysics coupling, added automatically, acts at the intersection of the two interfaces. It accounts for the fluid motion and the pressure balance in the region.

MESHING

A structured mesh with hexahedral elements is constructed using the Mapped and Swept mesh features. The porous domain thickness is meshed with four elements. The resulting mesh is shown in Figure 2.

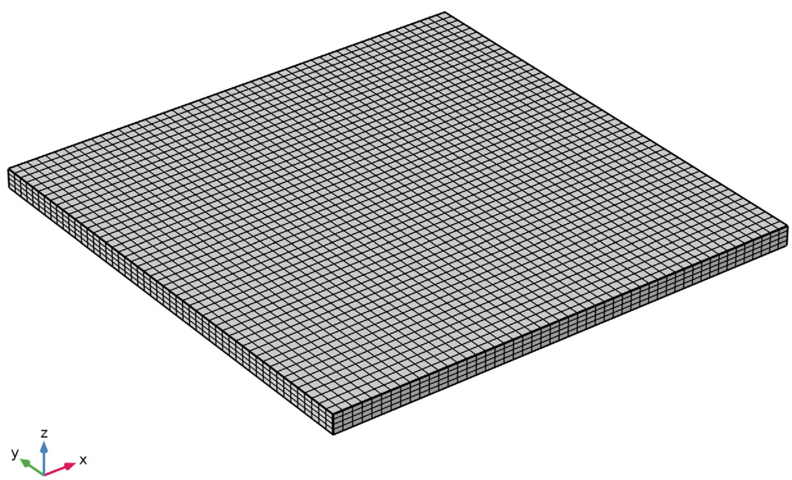

Figure 2: Computational domain with structured, hexahedral mesh.

Results and Discussion

The analysis presented in Ref. 1 provides the expression for the dimensionless pressure at the intersection of the physics interfaces. It is given in the form of infinite series as

$$\frac{h_t^3 p(x,y)}{\eta a^2 v_{\mathrm{sq}}} = \frac{192}{\pi^3} \sum_{m,n\,\mathrm{odd}}^{\infty} \sum^{\infty} \left(\frac{\sin(\alpha_m x)\sin(\beta_n y)}{mn\sqrt{m^2+k^2n^2}(\pi\sqrt{m^2+k^2n^2}+\psi G_{\mathrm{mn}})} \right), \qquad (1)$$

where

$$\alpha_m = (m\pi)/a , \tag{2}$$

$$\beta_n = (n\pi)/b , \tag{3}$$

$$G_{\mathrm{mn}} = \frac{12(e^{2\Upsilon H}-1)}{\overline{H}(e^{2\Upsilon H}+1)} , \tag{4}$$

$$\Upsilon = \frac{\pi}{a}\sqrt{m^2+k^2n^2} . \tag{5}$$

The dimensionless load carrying capacity of the squeeze film is given as

$$\frac{h_t^3 W}{\eta b a^2 v_{\mathrm{sq}}} = \frac{768}{\pi^5}\sum_{m,\, n\,\mathrm{odd}}^{\infty}\sum^{\infty}\left(\frac{1}{m^2n^2\sqrt{m^2+k^2n^2}(\pi\sqrt{m^2+k^2n^2}+\psi G_{\mathrm{mn}})}\right) . \tag{6}$$

In the comparison plots presented in the following paragraphs, the values of m and n are taken to be equal to 29, as suggested in Ref. 1. This ensures accurate pressure and load capacity values up to five significant digits.

Figure 3 shows pressure distribution for different values of ψ on the intersecting boundary of the physics interfaces at the location $y = b/2$, for values $k = 1$ and $\overline{H} = 0.02$. The maximum pressure rapidly decreases with an increase in the permeability parameter. For values of $\psi < 0.001$, the behavior is close to the nonporous limit as reported in Ref. 1.

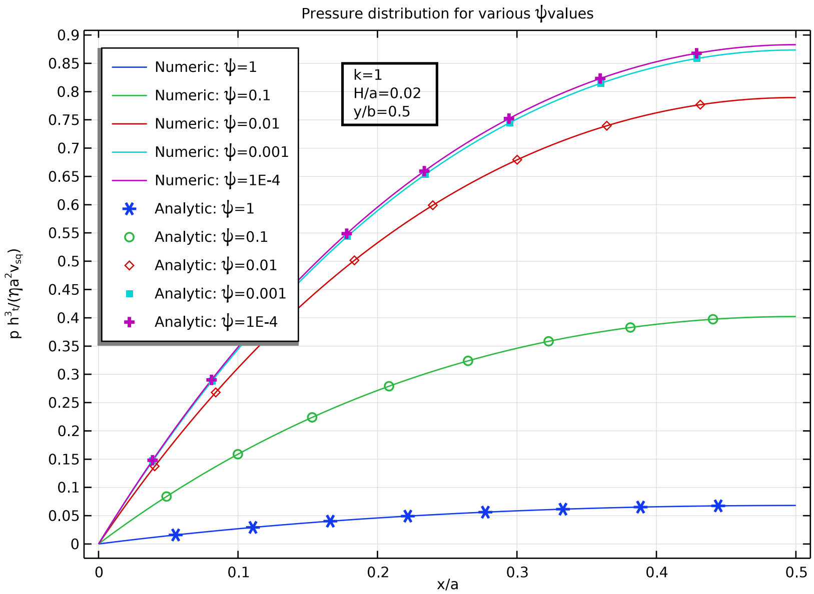

Figure 3: Pressure distribution for various values of ψ.

Figure 4 shows the load capacity for different values of k at the intersecting boundary of the physics interfaces for the value of $\overline{H} = 0.02$. The load-carrying capacity generally decreases with an increase in the aspect ratio, as such plates cannot contain the pressure generated due to increased side leakage. In fact, square plates provide the highest load-bearing capacity for a given quadrilateral area of coverage.

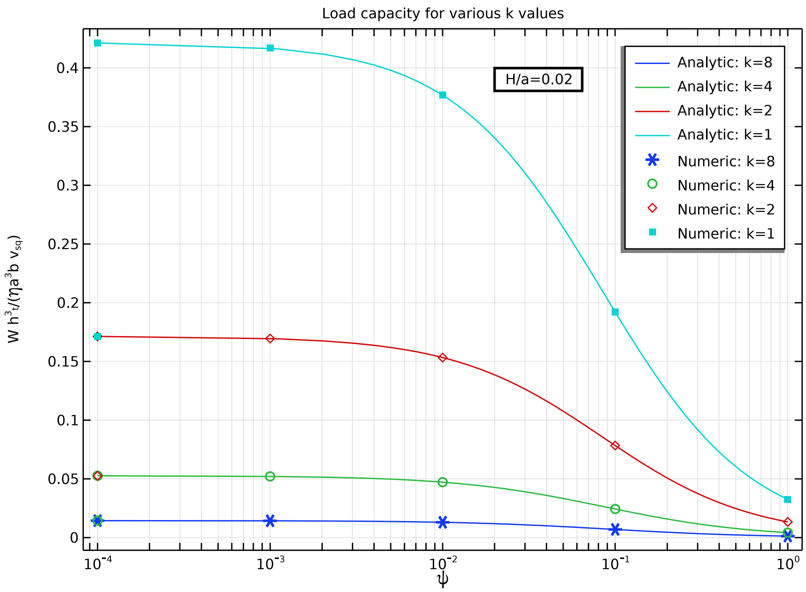

Figure 4: Load capacity for various values of k*.*

Figure 5 shows the loss of load capacity with increasing aspect ratio and permeability parameter.

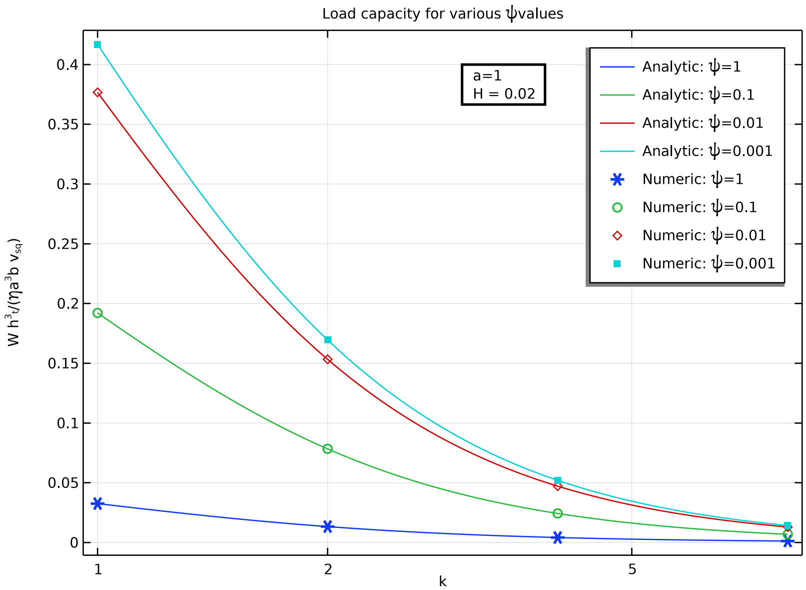

Figure 5: Load capacity for various values of ψ.

Figure 6 shows load capacity for different values of ϕ at the intersection of the physics interfaces for the values of $k = 1$ and $h_t = 0.001$. The load capacity is large and shows little variation with changes in the porous domain thickness for low permeabilities.With an increase in the permeability and porous domain thickness, however, the load capacity rapidly deteriorates.

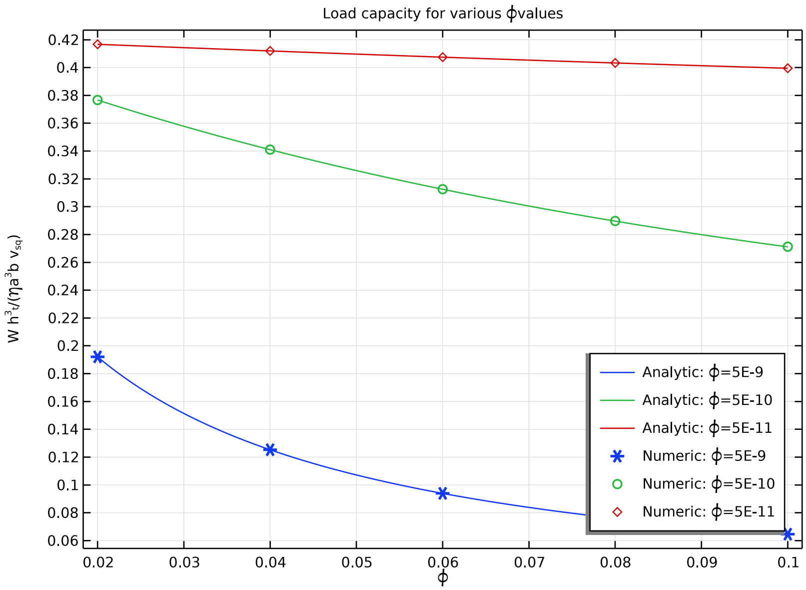

Figure 6: Load capacity for various values of ϕ .

All comparison plots show excellent agreement between computed and analytic solutions.

Figure 7 shows the pressure developed at the intersection of the physics interfaces and the subsequent percolation flow in the porous domain as a consequence of the squeezing action of the plates on the fluid in the thin-film region for the case $k = 1$, $\overline{H} = 0.02$, and $\psi = 0.01$.

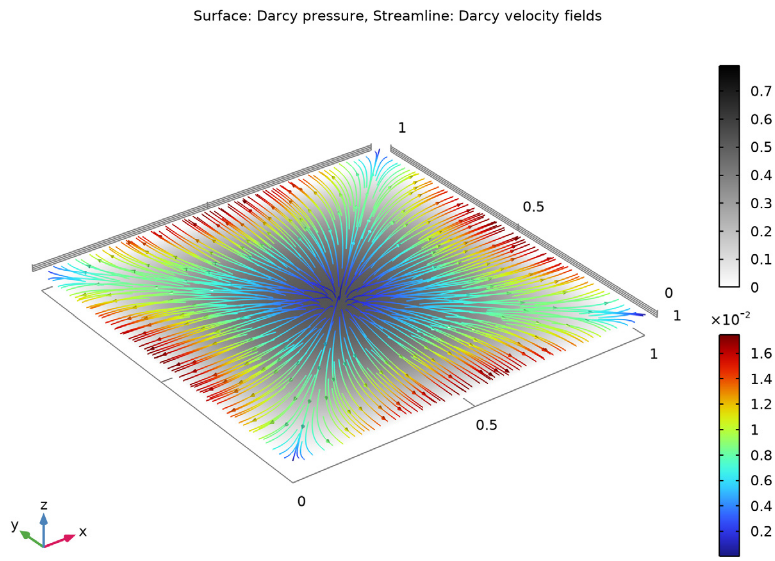

Figure 7: Darcy pressure and velocity streamlines.

Reference

1. H. Wu, “An Analysis of the Squeeze Film Between Porous Rectangular Plates,” *J. Lubr. Technol.*, vol. 94, no. 1, pp. 64–68, 1972.

Application Library path: `CFD_Module/Verification_Examples/squeeze_film_porous`

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>Porous Media and Subsurface Flow>Thin-Film and Porous Media Flow**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Stationary**.
6 Click **Done**.

Choose a unitless base system.

ROOT

1 In the **Model Builder** window, click the root node.
2 In the root node's **Settings** window, locate the **Unit System** section.
3 From the **Unit system** list, choose **None**.

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 Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `squeeze_film_porous_parameters.txt`.

Define analytic expressions of the solution. The expressions are available in Equation 1–Equation 6.

DEFINITIONS

Analytic 1 (an1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Local>Analytic**.
2 In the **Settings** window for **Analytic**, type `a_m` in the **Function name** text field.
3 Locate the **Definition** section. In the **Expression** text field, type `m*pi/a`.
4 In the **Arguments** text field, type `m, a`.

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

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	m	0	29	0	

Analytic 2 (an2)

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

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

3 Locate the **Definition** section. In the **Expression** text field, type `n*pi/b`.

4 In the **Arguments** text field, type `n, b`.

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

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	n	0	29	0	

Analytic 3 (an3)

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

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

3 Locate the **Definition** section. In the **Expression** text field, type `pi*sqrt(m^2+(k*n)^2)/a`.

4 In the **Arguments** text field, type `m, n, k, a`.

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

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	m	0	29	0	
√	n	0	29	0	
√	k	0	8	0	

Analytic 4 (an4)

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

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

3 Locate the **Definition** section. In the **Expression** text field, type `12*(exp(2*gamma_mn*H)-1)/(H*(exp(2*gamma_mn*H)+1)/a)`.

4 In the **Arguments** text field, type `gamma_mn, H, a`.

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

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	gamma_mn	0	1e6	0	

Analytic 5 (an5)

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

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

3 Locate the **Definition** section. In the **Expression** text field, type `(sin(a_m((2*m-1),a)*x)*sin(b_n((2*n-1),b)*y))/((2*m-1)*(2*n-1)*sqrt((2*m-1)^2+(k*(2*n-1))^2)*(pi*sqrt((2*m-1)^2+(k*(2*n-1))^2)+(psi*G_mn(gamma_mn((2*m-1),(2*n-1),k,a),H,a))))`.

4 In the **Arguments** text field, type `m, n, x, y, a, b, psi, k, H`.

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

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	m	0	29	0	
√	n	0	29	0	
√	x	0	a	0	

Analytic 6 (an6)

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

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

3 Locate the **Definition** section. In the **Expression** text field, type `1/(((2*m-1)*(2*n-1))^2*sqrt((2*m-1)^2+(k*(2*n-1))^2)*((pi*sqrt((2*m-1)^2+(k*(2*n-1))^2))+(psi*G_mn(gamma_mn((2*m-1),(2*n-1),k,a),H,a))))`.

4 In the **Arguments** text field, type `m, n, a, psi, k, H`.

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

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	m	0	29	0	
√	n	0	29	0	

Analytic 7 (an7)

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

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

3 Locate the **Definition** section. In the **Expression** text field, type `1/(((2*m-1)*(2*n-1))^2*sqrt((2*m-1)^2+(k*(2*n-1))^2)*((pi*sqrt((2*m-1)^2+(k*(2*n-1))^2))+(phi*H/ht^3*G_mn(gamma_mn((2*m-1),(2*n-1),k,a),H,a))))`.

4 In the **Arguments** text field, type `m, n, a, phi, k, H`.

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

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	m	0	29	0	
√	n	0	29	0	

GEOMETRY 1

Block 1 (blk1)

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

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `a/2`.

4 In the **Depth** text field, type `b/2`.

5 In the **Height** text field, type `H`.

Form Union (fin)

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

Create selections for the thin-film domain and symmetry boundaries.

DEFINITIONS

Thin-Film Boundary

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

2 In the **Settings** window for **Explicit**, type `Thin-Film Boundary` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `3` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Explicit**, locate the **Color** section.

8 From the **Color** list, choose **Color 1**.

Symmetry Boundaries

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

2 In the **Settings** window for **Explicit**, type `Symmetry Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `5-6` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Explicit**, locate the **Color** section.

8 From the **Color** list, choose **Color 4**.

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

Next, specify parameters for the Thin-Film Flow interface: Set the reference pressure to zero and reverse the reference normal to point in the positive z direction.

THIN-FILM FLOW (TFF)

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

2 In the **Settings** window for **Thin-Film Flow**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Thin-Film Boundary**.

4 Locate the **Reference Pressure** section. In the p_{ref} text field, type `0`.

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 **Model Input** section.

3 From the T list, choose **Common model input**.

4 Click to expand the **Reference Surface Properties** section. From the **Reference normal orientation** list, choose **Opposite direction to geometry normal**.

5 Locate the **Wall Properties** section. In the h_{w1} text field, type `ht`.

6 From the $\mathbf{v}_w$ list, choose **User defined**. Specify the vector as

sqvel	z

7 Locate the **Fluid Properties** section. From the μ list, choose **User defined**. In the associated text field, type `eta`.

8 From the ρ list, choose **User defined**. In the associated text field, type `rho`.

Symmetry 1

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

2 In the **Settings** window for **Symmetry**, locate the **Edge Selection** section.

3 Click Paste Selection.

4 In the **Paste Selection** dialog box, type `7, 10` in the **Selection** text field.

5 Click **OK**.

Now specify parameters for the Darcy's Law interface, including a zero reference pressure.

DARCY'S LAW (DL)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Darcy's Law (dl)**.

2 In the **Settings** window for **Darcy's Law**, locate the **Physical Model** section.

3 In the p_{ref} text field, type `0`.

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Darcy's Law (dl)>Porous Medium 1** click **Fluid 1**.

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

3 From the ρ list, choose **User defined**. In the associated text field, type `rho`.

4 From the μ list, choose **User defined**. In the associated text field, type `eta`.

Porous Matrix 1

1 In the **Model Builder** window, click **Porous Matrix 1**.

2 In the **Settings** window for **Porous Matrix**, locate the **Matrix Properties** section.

3 From the ε_{p} list, choose **User defined**. From the κ list, choose **User defined**. In the associated text field, type `phi`.

Symmetry 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Symmetry Boundaries**.

Pressure 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Pressure**.

2 In the **Settings** window for **Pressure**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `1, 2` in the **Selection** text field.

5 Click **OK**.

Create a structured hexahedral mesh.

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 Locate the **Sequence Type** section. From the list, choose **User-controlled mesh**.

Mapped 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Thin-Film Boundary**.

Swept 1

In the **Mesh** toolbar, click **Swept**.

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.

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

3 In the **Number of elements** text field, type 4.

Free Tetrahedral 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Tetrahedral 1** and choose **Disable**.

2 Right-click **Mesh 1** and choose **Build All**.

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

STUDY 1

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

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

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

Add a Parametric Sweep step for computing the Stationary solution for specified parameter combinations.

Parametric Sweep

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

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	**Parameter value list**	**Parameter unit**
k (Ratio of plate length to width)	`8*1^range(1,5) 4*1^range(1,5) 2*1^range(1,5) 1^range(1,5) 1^range(1,15)`	

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	**Parameter value list**	**Parameter unit**
psi (Permeability parameter)	`10^{range(0,-1,-4)} 10^{range(0,-1,-4)} 10^{range(0,-1,-4)} 10^{range(0,-1,-4)} range(0.1,0.1,0.5) range(0.01,0.01,0.05) range(0.001,0.001,0.005)`	

7 Click **Add**.

8 In the table, enter the following settings:

Parameter name	**Parameter value list**	**Parameter unit**
Hbar (Ratio of porous domain thickness to plate length)	`0.02*1^range(1,20) range(0.02,0.02,0.1) range(0.02,0.02,0.1) range(0.02,0.02,0.1)`	

9 In the **Study** toolbar, click **Compute**.

RESULTS

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

2 In the **Settings** window for **Results**, locate the **Update of Results** section.

3 Select the **Only plot when requested** check box.

Prepare datasets for processing the results and analytic functions.

Mirror 3D 1

1 In the **Model Builder** window, expand the **Results** node.

2 Right-click **Results>Datasets** and choose **More 3D Datasets>Mirror 3D**.

3 In the **Settings** window for **Mirror 3D**, locate the **Data** section.

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

5 Locate the **Plane Data** section. From the **Plane** list, choose **XZ-planes**.

6 In the **Y-coordinate** text field, type `b/2`.

Mirror 3D 2

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, locate the **Plane Data** section.

3 In the **X-coordinate** text field, type `a/2`.

4 Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D 1**.

Cut Line 3D 1

1 In the **Results** toolbar, click **Cut Line 3D**.

2 In the **Settings** window for **Cut Line 3D**, locate the **Data** section.

3 From the **Dataset** list, choose **Mirror 3D 2**.

4 Locate the **Line Data** section. In row **Point 1**, set **y** to `b/2`.

5 In row **Point 2**, set **x** to `a`.

6 In row **Point 2**, set **y** to `b/2`.

Grid 1D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Grid>Grid 1D**.

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

3 From the **Source** list, choose **Function**.

4 From the **Function** list, choose **All**.

5 Locate the **Parameter Bounds** section. In the **Name** text field, type `psi`.

6 In the **Minimum** text field, type `1e-4`.

Grid 1D 2

1 In the **Results** toolbar, click **More Datasets** and choose **Grid>Grid 1D**.

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

3 From the **Source** list, choose **Function**.

4 From the **Function** list, choose **All**.

5 Locate the **Parameter Bounds** section. In the **Name** text field, type `k`.

6 In the **Minimum** text field, type `1`.

7 In the **Maximum** text field, type `8`.

Grid 1D 3

1 In the **Results** toolbar, click **More Datasets** and choose **Grid>Grid 1D**.

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

3 From the **Source** list, choose **Function**.

4 From the **Function** list, choose **All**.

5 Locate the **Parameter Bounds** section. In the **Name** text field, type `H`.

6 In the **Minimum** text field, type `0.02`.

7 In the **Maximum** text field, type `0.1`.

Compute the load capacity for different parameter values.

Surface Integration 1

1 In the **Results** toolbar, click **More Derived Values** and choose **Integration>Surface Integration**.

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

3 From the **Dataset** list, choose **Mirror 3D 2**.

4 From the **Parameter selection (k, psi, Hbar)** list, choose **Manual**.

5 In the **Parameter indices (1-35)** text field, type `range(1,1,5)`.

6 Locate the **Selection** section. From the **Selection** list, choose **Thin-Film Boundary**.

7 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`p*ht^3/(eta*a^3*b*sqvel)`		

8 Right-click **Surface Integration 1** and choose **Duplicate**.

Surface Integration 2

1 In the **Model Builder** window, click **Surface Integration 2**.

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

3 In the **Parameter indices (1-35)** text field, type `range(5,1,10)`.

4 Right-click **Surface Integration 2** and choose **Duplicate**.

Surface Integration 3

1 In the **Model Builder** window, click **Surface Integration 3**.

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

3 In the **Parameter indices (1-35)** text field, type `range(10,1,15)`.

4 Right-click **Surface Integration 3** and choose **Duplicate**.

Surface Integration 4

1 In the **Model Builder** window, click **Surface Integration 4**.

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

3 In the **Parameter indices (1-35)** text field, type `range(15,1,20)`.

4 Right-click **Surface Integration 4** and choose **Duplicate**.

Surface Integration 5

1 In the **Model Builder** window, click **Surface Integration 5**.

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

3 In the **Parameter indices (1-35)** text field, type `range(1,5,20)`.

4 Right-click **Surface Integration 5** and choose **Duplicate**.

Surface Integration 6

1 In the **Model Builder** window, click **Surface Integration 6**.

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

3 In the **Parameter indices (1-35)** text field, type `range(2,5,20)`.

4 Right-click **Surface Integration 6** and choose **Duplicate**.

Surface Integration 7

1 In the **Model Builder** window, click **Surface Integration 7**.

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

3 In the **Parameter indices (1-35)** text field, type `range(3,5,20)`.

4 Right-click **Surface Integration 7** and choose **Duplicate**.

Surface Integration 8

1 In the **Model Builder** window, click **Surface Integration 8**.

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

3 In the **Parameter indices (1-35)** text field, type `range(4,5,20)`.

4 Right-click **Surface Integration 8** and choose **Duplicate**.

Surface Integration 9

1 In the **Model Builder** window, click **Surface Integration 9**.

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

3 In the **Parameter indices (1-35)** text field, type `range(21,1,25)`.

4 Right-click **Surface Integration 9** and choose **Duplicate**.

Surface Integration 10

1 In the **Model Builder** window, click **Surface Integration 10**.
2 In the **Settings** window for **Surface Integration**, locate the **Data** section.
3 In the **Parameter indices (1-35)** text field, type range(26,1,30).
4 Right-click **Surface Integration 10** and choose **Duplicate**.

Surface Integration 11

1 In the **Model Builder** window, click **Surface Integration 11**.
2 In the **Settings** window for **Surface Integration**, locate the **Data** section.
3 In the **Parameter indices (1-35)** text field, type range(31,1,35).
4 In the **Results** toolbar, click **Evaluate** and choose **Clear and Evaluate All**.

Pressure Distribution for Various psi Values

1 In the **Results** toolbar, click **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type Pressure Distribution for Various psi Values in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Cut Line 3D 1**.
4 From the **Parameter selection (k, psi, Hbar)** list, choose **From list**.
5 In the **Parameter values (k,psi,Hbar)** list, choose **16: k=1, psi=1, Hbar=0.02**, **17: k=1, psi=0.1, Hbar=0.02**, **18: k=1, psi=0.01, Hbar=0.02**, **19: k=1, psi=0.001, Hbar=0.02**, and **20: k=1, psi=1E-4, Hbar=0.02**.
6 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
7 In the **Title** text area, type Pressure distribution for various \psi values.
8 Locate the **Plot Settings** section.
9 Select the **x-axis label** check box. In the associated text field, type x/a.
10 Select the **y-axis label** check box. In the associated text field, type p h³_t/(\eta a²v_{sq}).
11 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Line Graph 1

1 Right-click **Pressure Distribution for Various psi Values** and choose **Line Graph**.
2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
3 In the **Expression** text field, type p*ht^3/(eta*a^2*sqvel).
4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
5 In the **Expression** text field, type x/a.

6 Click to expand the **Coloring and Style** section. Click to expand the **Legends** section. Select the **Show legends** check box.

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

8 In the table, enter the following settings:

Legends
`Numeric: \psi=1`
`Numeric: \psi=0.1`
`Numeric: \psi=0.01`
`Numeric: \psi=0.001`
`Numeric: \psi=1E-4`

Line Graph 2

1 In the **Model Builder** window, right-click **Pressure Distribution for Various psi Values** and choose **Line Graph**.

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

3 In the **Expression** text field, type `192*sum(sum(p_summand(m, n, x, y, a, b, psi, 1, 0.02), n, 1, 15), m, 1, 15)/(pi^3)`.

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

5 In the **Expression** text field, type `x/a`.

6 Locate the **Coloring and Style** section. From the **Color** list, choose **Cycle (reset)**.

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

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

9 Find the **Line markers** subsection. From the **Positioning** list, choose **Interpolated**.

10 Locate the **Legends** section. Select the **Show legends** check box.

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

12 In the table, enter the following settings:

Legends
`Analytic: \psi=1`
`Analytic: \psi=0.1`
`Analytic: \psi=0.01`
`Analytic: \psi=0.001`
`Analytic: \psi=1E-4`

Annotation 1

1. Right-click **Pressure Distribution for Various psi Values** and choose **Annotation**.
2. In the **Settings** window for **Annotation**, locate the **Annotation** section.
3. In the **Text** text field, type k=eval(k) \\ H/a=eval(Hbar) \\ y/b=0.5.
4. Select the **LaTeX markup** check box.
5. Locate the **Position** section. In the **x** text field, type 0.175.
6. In the **y** text field, type 0.85.
7. Click to expand the **Advanced** section. In the **Expression precision** text field, type 1.
8. Locate the **Coloring and Style** section. Clear the **Show point** check box.
9. Select the **Show frame** check box.

Pressure Distribution for Various psi Values

1. In the **Model Builder** window, click **Pressure Distribution for Various psi Values**.
2. In the **Pressure Distribution for Various psi Values** toolbar, click **Plot**.

Load Capacity for Various k Values

1. In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type Load Capacity for Various k Values in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **None**.
4. Locate the **Title** section. From the **Title type** list, choose **Manual**.
5. In the **Title** text area, type Load capacity for various k values.
6. Locate the **Plot Settings** section.
7. Select the **x-axis label** check box. In the associated text field, type \psi.
8. Select the **y-axis label** check box. In the associated text field, type W h³_t/(\eta a³b v_{sq}).
9. Locate the **Axis** section. Select the **x-axis log scale** check box.

Function 1

1. In the **Load Capacity for Various k Values** toolbar, click **More Plots** and choose **Function**.
2. In the **Settings** window for **Function**, locate the **Data** section.
3. From the **Dataset** list, choose **Grid 1D 1**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `768*sum(sum(W_summand_psi(m, n, a, psi, 8, 0.02), n, 1, 15), m, 1, 15)/(pi^5)`.

5 Locate the **x-Axis Data** section. In the **Expression** text field, type `psi`.

6 In the **Lower bound** text field, type `1e-4`.

7 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Cycle (reset)**.

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

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

10 In the table, enter the following settings:

Legends
`Analytic: k=8`

11 Right-click **Function 1** and choose **Duplicate**.

Function 2

1 In the **Model Builder** window, click **Function 2**.

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

3 In the **Expression** text field, type `768*sum(sum(W_summand_psi(m, n, a, psi, 4, 0.02), n, 1, 15), m, 1, 15)/(pi^5)`.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **Cycle**.

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

Legends
`Analytic: k=4`

6 Right-click **Function 2** and choose **Duplicate**.

Function 3

1 In the **Model Builder** window, click **Function 3**.

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

3 In the **Expression** text field, type `768*sum(sum(W_summand_psi(m, n, a, psi, 2, 0.02), n, 1, 15), m, 1, 15)/(pi^5)`.

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

Legends
`Analytic: k=2`

5 Right-click **Function 3** and choose **Duplicate**.

Function 4

1 In the **Model Builder** window, click **Function 4**.

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

3 In the **Expression** text field, type `768*sum(sum(W_summand_psi(m, n, a, psi, 1, 0.02), n, 1, 15), m, 1, 15)/(pi^5)`.

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

Legends
`Analytic: k=1`

Table Graph 1

1 In the **Model Builder** window, right-click **Load Capacity for Various k Values** and choose **Table Graph**.

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

3 From the **x-axis data** list, choose **psi**.

4 From the **Plot columns** list, choose **Manual**.

5 In the **Columns** list, select **p*ht^3/(eta*a^3*b*sqvel)**.

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

7 From the **Color** list, choose **Cycle (reset)**.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle (reset)**.

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

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

11 In the table, enter the following settings:

Legends
`Numeric: k=8`

12 Right-click **Table Graph 1** and choose **Duplicate**.

Table Graph 2

1 In the **Model Builder** window, click **Table Graph 2**.

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

3 From the **Table** list, choose **Table 2**.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **Cycle**.

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

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

Legends
`Numeric: k=4`

7 Right-click **Table Graph 2** and choose **Duplicate**.

Table Graph 3

1 In the **Model Builder** window, click **Table Graph 3**.

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

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

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

Legends
`Numeric: k=2`

5 Right-click **Table Graph 3** and choose **Duplicate**.

Table Graph 4

1 In the **Model Builder** window, click **Table Graph 4**.

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

3 From the **Table** list, choose **Table 4**.

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

Legends
`Numeric: k=1`

Annotation 1

1 In the **Model Builder** window, right-click **Load Capacity for Various k Values** and choose **Annotation**.

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

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

4 From the **Parameter value (k,psi,Hbar)** list, choose **1: k=8, psi=1, Hbar=0.02**.

5 Locate the **Annotation** section. In the **Text** text field, type `H/a=0.02`.

6 Select the **LaTeX markup** check box.

7 Locate the **Position** section. In the **X** text field, type `0.02`.

8 In the **Y** text field, type `0.4`.

9 Locate the **Advanced** section. In the **Expression precision** text field, type `1`.

10 Locate the **Coloring and Style** section. Clear the **Show point** check box.

11 Select the **Show frame** check box.

Load Capacity for Various k Values

1 In the **Model Builder** window, click **Load Capacity for Various k Values**.

2 In the **Load Capacity for Various k Values** toolbar, click **Plot**.

Load Capacity for Various psi Values

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **None**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Load capacity for various \psi values`.

6 Locate the **Plot Settings** section.

7 Select the **x-axis label** check box. In the associated text field, type `k`.

8 Select the **y-axis label** check box. In the associated text field, type `W h<sup>3</sup><sub>t</sub>/(\eta a<sup>3</sup>b v<sub>sq</sub>)`.

9 Locate the **Axis** section. Select the **x-axis log scale** check box.

Function 1

1 In the **Load Capacity for Various psi Values** toolbar, click **More Plots** and choose **Function**.

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

3 From the **Dataset** list, choose **Grid 1D 2**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `768*sum(sum(W_summand_psi(m, n, a, 1, k, 0.02), n, 1, 15), m, 1, 15)/(pi^5)`.

5 Locate the **x-Axis Data** section. In the **Expression** text field, type `k`.

6 In the **Lower bound** text field, type `1`.

7 In the **Upper bound** text field, type `8`.

8 Locate the **Coloring and Style** section. From the **Color** list, choose **Cycle (reset)**.

9 Locate the **Legends** section. Select the **Show legends** check box.

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

11 In the table, enter the following settings:

Legends
`Analytic: \psi=1`

12 Right-click **Function 1** and choose **Duplicate**.

Function 2

1. In the **Model Builder** window, click **Function 2**.
2. In the **Settings** window for **Function**, locate the **y-Axis Data** section.
3. In the **Expression** text field, type `768*sum(sum(W_summand_psi(m, n, a, 0.1, k, 0.02), n, 1, 15), m, 1, 15)/(pi^5)`.
4. Locate the **Coloring and Style** section. From the **Color** list, choose **Cycle**.
5. Locate the **Legends** section. In the table, enter the following settings:

Legends
`Analytic: \psi=0.1`

6. Right-click **Function 2** and choose **Duplicate**.

Function 3

1. In the **Model Builder** window, click **Function 3**.
2. In the **Settings** window for **Function**, locate the **y-Axis Data** section.
3. In the **Expression** text field, type `768*sum(sum(W_summand_psi(m, n, a, 0.01, k, 0.02), n, 1, 15), m, 1, 15)/(pi^5)`.
4. Locate the **Legends** section. In the table, enter the following settings:

Legends
`Analytic: \psi=0.01`

5. Right-click **Function 3** and choose **Duplicate**.

Function 4

1. In the **Model Builder** window, click **Function 4**.
2. In the **Settings** window for **Function**, locate the **y-Axis Data** section.
3. In the **Expression** text field, type `768*sum(sum(W_summand_psi(m, n, a, 0.001, k, 0.02), n, 1, 15), m, 1, 15)/(pi^5)`.

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

Legends
`Analytic: \psi=0.001`

Table Graph 1

1 In the **Model Builder** window, right-click **Load Capacity for Various psi Values** and choose **Table Graph**.

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

3 From the **Table** list, choose **Table 5**.

4 From the **x-axis data** list, choose **k**.

5 From the **Plot columns** list, choose **Manual**.

6 In the **Columns** list, select **p*ht^3/(eta*a^3*b*sqvel)**.

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

8 From the **Color** list, choose **Cycle (reset)**.

9 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle (reset)**.

10 Locate the **Legends** section. Select the **Show legends** check box.

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

12 In the table, enter the following settings:

Legends
`Numeric: \psi=1`

13 Right-click **Table Graph 1** and choose **Duplicate**.

Table Graph 2

1 In the **Model Builder** window, click **Table Graph 2**.

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

3 From the **Table** list, choose **Table 6**.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **Cycle**.

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

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

Legends
`Numeric: \psi=0.1`

7 Right-click **Table Graph 2** and choose **Duplicate**.

Table Graph 3

1 In the **Model Builder** window, click **Table Graph 3**.

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

3 From the **Table** list, choose **Table 7**.

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

Legends
`Numeric: \psi=0.01`

5 Right-click **Table Graph 3** and choose **Duplicate**.

Table Graph 4

1 In the **Model Builder** window, click **Table Graph 4**.

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

3 From the **Table** list, choose **Table 8**.

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

Legends
`Numeric: \psi=0.001`

Annotation 1

1 In the **Model Builder** window, right-click **Load Capacity for Various psi Values** and choose **Annotation**.

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

3 From the **Dataset** list, choose **Mirror 3D 2**.

4 From the **Parameter value (k,psi,Hbar)** list, choose **1: k=8, psi=1, Hbar=0.02**.

5 Locate the **Annotation** section. In the **Text** text field, type `a=1 \\ H = 0.02`.

6 Select the **LaTeX markup** check box.

7 Locate the **Position** section. In the **x** text field, type `3`.

8 In the **y** text field, type `0.4`.

9 Locate the **Coloring and Style** section. Clear the **Show point** check box.

10 Select the **Show frame** check box.

Load Capacity for Various psi Values

1 In the **Model Builder** window, click **Load Capacity for Various psi Values**.

2 In the **Load Capacity for Various psi Values** toolbar, click **Plot**.

Load Capacity for Various phi Values

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

2 In the **Settings** window for **1D Plot Group**, type Load Capacity for Various phi Values in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **None**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type Load capacity for various \phi values.

6 Locate the **Plot Settings** section.

7 Select the **x-axis label** check box. In the associated text field, type \phi.

8 Select the **y-axis label** check box. In the associated text field, type W h³_t/(\eta a³b v_{sq}).

9 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Function 1

1 In the **Load Capacity for Various phi Values** toolbar, click **More Plots** and choose **Function**.

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

3 From the **Dataset** list, choose **Grid 1D 3**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type 768*sum(sum(W_summand_phi(m, n, a, 5e-9, 1, H), n, 1, 15), m, 1, 15)/(pi^5).

5 Locate the **x-Axis Data** section. In the **Expression** text field, type H.

6 In the **Lower bound** text field, type 0.02.

7 In the **Upper bound** text field, type 0.1.

8 Locate the **Coloring and Style** section. From the **Color** list, choose **Cycle (reset)**.

9 Locate the **Legends** section. Select the **Show legends** check box.

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

11 In the table, enter the following settings:

Legends
Analytic: \phi=5E-9

12 Right-click **Function 1** and choose **Duplicate**.

Function 2

1 In the **Model Builder** window, click **Function 2**.

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

3 In the **Expression** text field, type `768*sum(sum(W_summand_phi(m, n, a, 5e-10, 1, H), n, 1, 15), m, 1, 15)/(pi^5)`.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **Cycle**.

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

Legends
`Analytic: \phi=5E-10`

6 Right-click **Function 2** and choose **Duplicate**.

Function 3

1 In the **Model Builder** window, click **Function 3**.

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

3 In the **Expression** text field, type `768*sum(sum(W_summand_phi(m, n, a, 5e-11, 1, H), n, 1, 15), m, 1, 15)/(pi^5)`.

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

Legends
`Analytic: \phi=5E-11`

Table Graph 1

1 In the **Model Builder** window, right-click **Load Capacity for Various phi Values** and choose **Table Graph**.

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

3 From the **Table** list, choose **Table 9**.

4 From the **x-axis data** list, choose **Hbar**.

5 From the **Plot columns** list, choose **Manual**.

6 In the **Columns** list, select **p*ht^3/(eta*a^3*b*sqvel)**.

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

8 From the **Color** list, choose **Cycle (reset)**.

9 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle (reset)**.

10 Locate the **Legends** section. Select the **Show legends** check box.

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

12 In the table, enter the following settings:

Legends
`Numeric: \phi=5E-9`

13 Right-click **Table Graph 1** and choose **Duplicate**.

Table Graph 2

1 In the **Model Builder** window, click **Table Graph 2**.

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

3 From the **Table** list, choose **Table 10**.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **Cycle**.

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

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

Legends
`Numeric: \phi=5E-10`

7 Right-click **Table Graph 2** and choose **Duplicate**.

Table Graph 3

1 In the **Model Builder** window, click **Table Graph 3**.

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

3 From the **Table** list, choose **Table 11**.

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

Legends
`Numeric: \phi=5E-11`

Load Capacity for Various phi Values

1 In the **Model Builder** window, click **Load Capacity for Various phi Values**.

2 In the **Load Capacity for Various phi Values** toolbar, click **Plot**.

Darcy Pressure and Velocity Fields

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

2 In the **Settings** window for **3D Plot Group**, type `Darcy Pressure and Velocity Fields` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D 2**.

4 From the **Parameter value (k,psi,Hbar)** list, choose **18: k=1, psi=0.01, Hbar=0.02**.

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

6 In the **Title** text area, type `Surface: Darcy pressure, Streamline: Darcy velocity fields`.

7 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

8 Locate the **Color Legend** section. From the **Position** list, choose **Right double**.

Surface 1

1 Right-click **Darcy Pressure and Velocity Fields** and choose **Surface**.

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

3 In the **Expression** text field, type `p*ht^3/(eta*a^2*sqvel)`.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Linear>GrayScale** in the tree.

6 Click **OK**.

7 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

8 From the **Color table transformation** list, choose **Reverse**.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

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

3 From the **Selection** list, choose **Thin-Film Boundary**.

Streamline 1

1 In the **Model Builder** window, right-click **Darcy Pressure and Velocity Fields** and choose **Streamline**.

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

3 In the **x-component** text field, type `dl.u*dl.rho*a/eta`.

4 In the **y-component** text field, type `dl.v*dl.rho*a/eta`.

5 In the **z-component** text field, type `dl.w*dl.rho*a/eta`.

6 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Uniform density**.

7 In the **Separating distance** text field, type `0.05`.

8 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

Color Expression 1

1. Right-click **Streamline 1** and choose **Color Expression**.
2. In the **Settings** window for **Color Expression**, locate the **Expression** section.
3. In the **Expression** text field, type `dl.U*dl.rho*a/eta`.

Darcy Pressure and Velocity Fields

1. In the **Model Builder** window, under **Results** click **Darcy Pressure and Velocity Fields**.
2. In the **Darcy Pressure and Velocity Fields** toolbar, click **Plot**.
3. Click the **Zoom Extents** button in the **Graphics** toolbar.