Created in COMSOL Multiphysics 6.2

Squeeze-Film Gas Damping of a Vibrating Disc

Introduction

This benchmark model computes the damping force acting on a vibrating disc. The disc is in close proximity to a stationary surface and the damping results from the squeezing of a thin film of gas between the two surfaces. The squeezing action forces out the gas from between the two plates resulting in a damping force that acts to prevent mechanical contact between the two surfaces. The opposite effect takes place when the surfaces move away from each other as gas is drawn back into the bearing.

This model examines the effect of the periodic motion of the disc on the flow developed, including the pressure in the gas and the resulting damping forces. Small amplitude motion is analyzed using a linear frequency domain simulation. A nonlinear transient analysis is performed for small to large amplitude motion. The calculated film pressure and load carrying capacity are compared with analytical results.

Model Definition

The Thin-Film Flow, Edge interface is used to model the gas film on a flat circular plate. The model is 1D axisymmetric since the film pressure only varies radially. When Thin-Film Flow is assigned to a boundary, this 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. Figure 1 shows the configuration of the base and the wall in an arbitrary problem, and defines a number of terms used in the interface.

In this example, the model geometry is 1D axisymmetric and consists of a single line, with length set to the radius of the circular disc. The line is located at the origin and aligned with the r-axis. The base is coincident with the reference surface. A pressure is generated in the bearing by a periodic velocity of the wall in a direction normal to the wall.

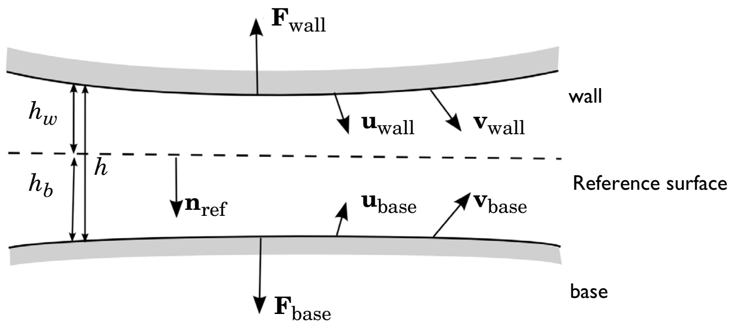

Figure 1: An example illustrating a typical configuration for thin-film flow.

For nonslip boundary conditions at the wall and the base, the modified Reynolds equation takes the following form for a general frequency domain problem:

$$\begin{aligned} p_{tot}(\mathbf{v}_b \cdot \mathbf{n}_{ref} - \mathbf{v}_w \cdot \mathbf{n}_{ref}) + i\omega h_0 p_f + \nabla_t \cdot (h_0 p_{tot} \mathbf{v}_{av}) \\ - p_{tot}(\mathbf{v}_w \cdot \nabla_t h_w + \mathbf{v}_b \cdot \nabla_t h_b) = 0 \end{aligned} \tag{1}$$

$$\mathbf{v}_{av} = \frac{1}{2}(\mathbf{I} - \mathbf{n}_r \mathbf{n}_r^T)(\mathbf{v}_w + \mathbf{v}_b) - \frac{h_0^2}{12\mu} \nabla_t p_f$$

where ρ is the fluid density, μ is its viscosity, h_0 is the mean film height, p_f is the pressure developed as a result of the flow (this is the dependent variable in COMSOL) and p_{tot} is the total pressure ($p_{tot} = p_A + p_f$, where p_A is the ambient pressure). Other terms are defined in Figure 1. A reference surface with normal $\mathbf{n}_{\text{ref}}$ sits in a narrow gap between a wall and base. In COMSOL the vector $\mathbf{n}_{\text{ref}}$ points into the base and out of the wall. The wall moves with a displacement from its initial position with displacement $\mathbf{u}_{\text{wall}}$ and velocity $\mathbf{v}_{\text{wall}}$. Similarly the base moves from its initial position with displacement $\mathbf{u}_{\text{base}}$ and velocity $\mathbf{v}_{\text{base}}$. The compression of the film results in an excess pressure, p_f, above the ambient pressure, p_A, and a gas velocity in the gap. At a point on the reference surface the average value of the film velocity along a line perpendicular to the surface is given by the in plane vector $\mathbf{v}_{\text{ave}}$. The motion of the gas results in forces on the wall ($\mathbf{F}_{\text{wall}}$) and the base ($\mathbf{F}_{\text{base}}$). The height of the wall above the reference surface is h_w whilst the base is a distance h_b below the reference surface. The total size of the gap is h=h_w+h_b. At a given point in time h_w=h_{w1}−$\mathbf{n}_{\text{ref}} \cdot \mathbf{n}_{\text{wall}}$ and h_b=h_{b1}−$\mathbf{n}_{\text{ref}} \cdot \mathbf{n}_{\text{wall}}$ where h_{w1} and h_{b1} are the initial heights of the wall and base, respectively.

Note that the frequency formulation assumes a small amplitude first order harmonic variation of film pressure, film height and wall velocity at the frequency of interest. The boundary conditions for this model are vanishing pressure due to the flow ($p_f = 0$) at $r = r_0$, where r_0 is the disc radius and symmetry/zero pressure gradient ($\mathrm{d}p_f/\mathrm{d}r = 0$) at $r = 0$.

For the case of a 1D axisymmetric problem Equation 1 can be greatly simplified to derive a simple closed form analytical solution. Note that these simplifications are not made in the simulation itself and consequently there are slight deviations from the analytic results in the model — these cannot be seen in the plots shown here and are not significant (strictly speaking the simulation is more accurate than the analytic results since no assumptions are made). The motion of the disc is in the vertical direction only, and the gap size is uniform across the disc, so a number of the terms in Equation 1 are zero. In this example, the term $i\omega h p_f$ is quite small compared to other terms and can be neglected for the purpose of deriving an analytical solution. The ambient pressure is 1 atmosphere and correspondingly $p_f \ll p_A$. so $p_{\mathrm{tot}} \approx p_A$. Making these assumptions the modified Reynolds equation reduces to

$$\frac{1}{r}\frac{d}{dr}\left(\frac{r{h_0}^3}{12\mu}\frac{dp_f}{dr}\right) = \mathrm{v}_w \tag{2}$$

where v_w is the velocity of the wall in the z direction. With the boundary conditions $p_f = 0$ at $r = r_0$, and $\mathrm{d}p_f/\mathrm{d}r = 0$ at $r = 0$, Equation 2 can be solved for p_f and is given by (see Ref. 1 for complete derivation)

$$p_{f,\mathrm{an}} = -\frac{3\mu\mathrm{v}_w}{{h_0}^3}({r_0}^2 - r^2) \tag{3}$$

The total analytical vertical load on the disc is then given by (see Ref. 1 for complete derivation)

$$F_{\mathrm{an}} = \int_0^{r_0} 2\pi r p_f dr = -\frac{3\pi\mu{r_0}^4\mathrm{v}_w}{2{h_0}^3}$$

A frequency-domain analysis is appropriate for small amplitude periodic motion of the bearing wall. For large amplitude periodic motion (where the amplitude of the motion becomes comparable to the gap size), a transient analysis is necessary, to capture the nonlinearities in the model. Again approximations are required to derive an analytic result for the transient simulation (but are not necessary for the model).

For a transient model the modified Reynolds equation takes the form

$$\frac{\partial}{\partial t}(p_{\mathrm{tot}}h) + \nabla_t \cdot (hp_{\mathrm{tot}}\mathbf{v}_{\mathrm{av}}) - p_{\mathrm{tot}}(\mathbf{v}_w \cdot \nabla_t h_w + \mathbf{v}_b \cdot \nabla_t h_b) = 0 \qquad (4)$$

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

For periodic motion of the wall, the total film height h is $h(t) = h_0 + \Delta h \sin(2\pi f t)$, where Δh is the amplitude and f is the frequency of wall periodic motion. The wall velocity v_{w} is then given by $v_{\mathrm{w}}(t) = (2\pi f \Delta h)\cos(2\pi f t)$. Once again making the assumption that $p_{tot} \approx p_A$ and noting that a number of these terms are zero for vertical motion of a parallel disc:

$$\frac{1}{r}\frac{d}{dr}\left(\frac{rh(t)^3}{12\mu}\frac{dp_f}{dr}\right) = \mathrm{v}_w(t)$$

Following the derivation of total analytical vertical load for the frequency domain analysis, the total vertical load on the disc for the transient model is given by

$$F_{\mathrm{an}}(t) = \int_0^{r_0} 2\pi r p_f(t) dr = -\frac{3\pi\mu r_0^{\ 4}\mathrm{v}_w(t)}{2h(t)^3}$$

The model compares the analytical values of film pressure and total vertical load against the values computed by using the Thin-Film Flow, Edge interface. The results are found to be in agreement with the analytical solutions.

Results and Discussion

The values of radial film pressure in the gas film for the frequency domain analysis are plotted in Figure 2. As expected, film pressure magnitude is maximum at the center of the circular disc and drops off to zero where the gas film exits the bearing geometry. Figure 2 also compares the numerical radial film pressure values with analytical values calculated using Equation 3. The calculated results agree well with the analytical results. Figure 3 shows an arrow plot of the fluid load per unit area on the circular disc. These values correspond to the film pressure since the vertical load is significantly larger than the radial load. Figure 4 shows the variation of film height (gap) with respect to time for different values of the amplitude of harmonic film height. The corresponding variation of film pressure and total vertical load on the circular disc is shown in Figure 5 and Figure 6, respectively. For larger amplitude of harmonic film height the response in terms of both film pressure and total load are nonlinear with respect to the applied harmonic motion of the circular plate. Such nonlinearity is appropriately calculated by performing a transient analysis. Figure 6 also compares analytical and calculated time dependent values of the total load on the circular disc. The plot indicates that the calculated values agree well with the analytical values.

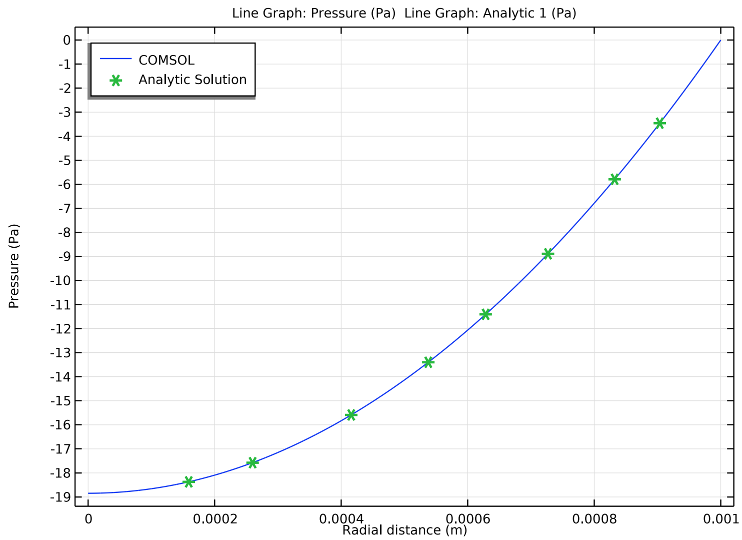

Figure 2: Film pressure vs radial distance from the center of the circular disc. The results computed by COMSOL are shown as the continuous curve and the analytical result is shown with green symbols.

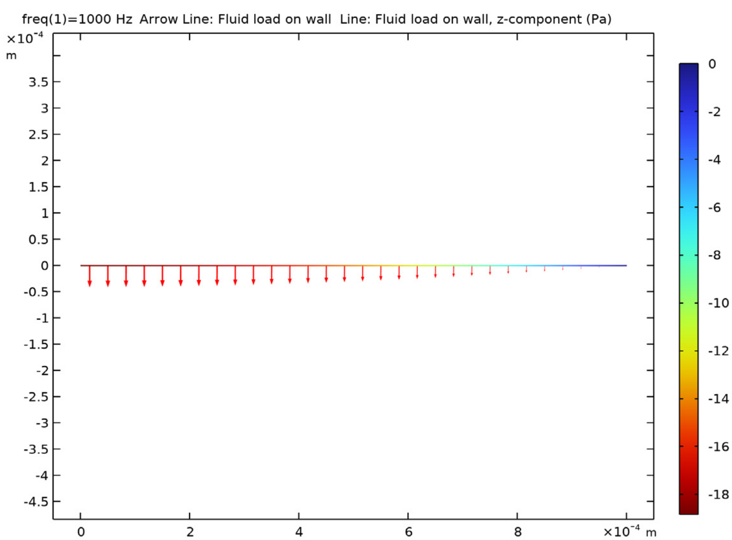

Figure 3: Radial distribution of the film load on the wall.

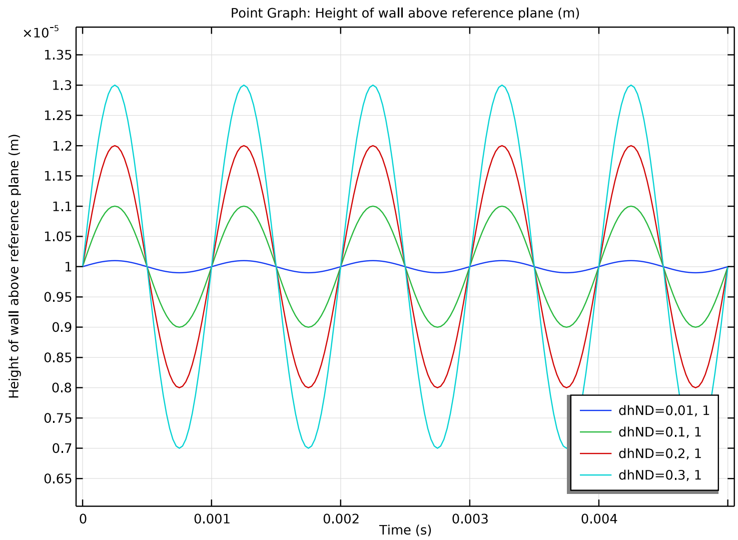

Figure 4: Film height variation vs time for different values of amplitude of harmonic film height.

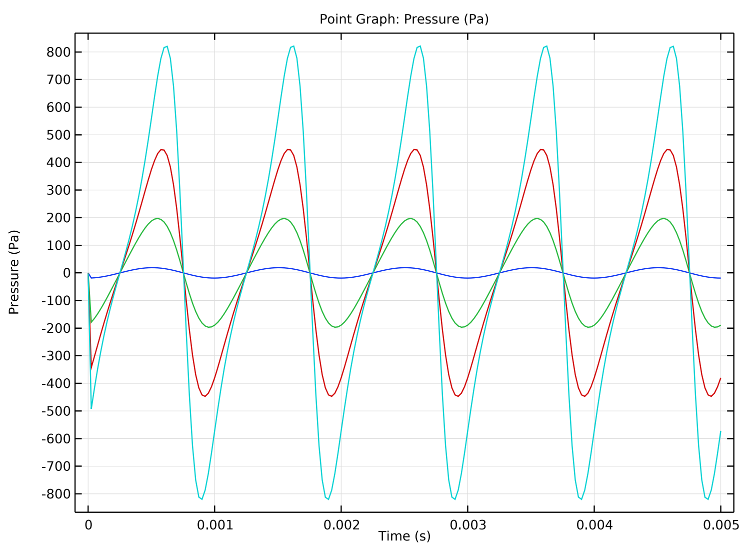

Figure 5: Film pressure vs time for different values of amplitude of harmonic film height. For higher values of film height amplitude the film pressure varies nonlinearly with respect to the film height.

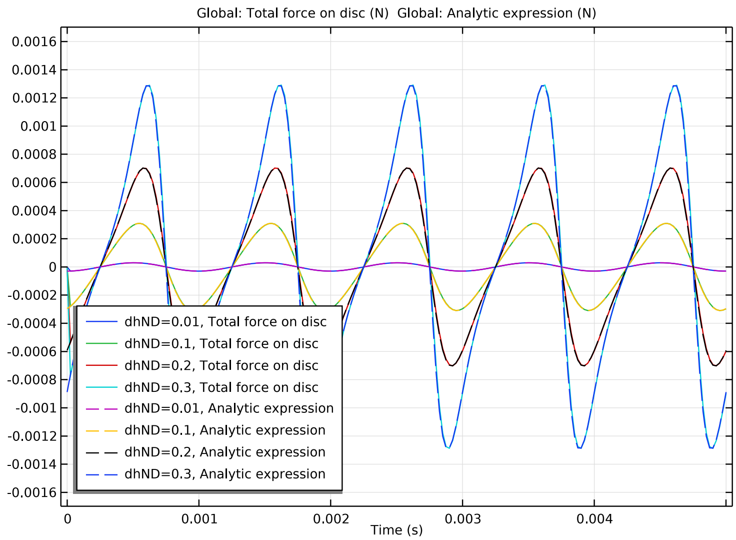

Figure 6: Total vertical load on the circular disc vs time. for different values of amplitude of harmonic film height. The results computed by COMSOL are shown by solid continuous curve and the analytical result is shown by dashed continuous curves. Similar to the film pressure, for higher values of film height amplitude the total load varies nonlinearly with respect to the film height.

Reference

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

This model is based on the discussion entitled *Parallel-Surface Bearing of infinite width* in section 12.2 of the above reference.

Application Library path: `MEMS_Module/Sensors/squeeze_film_disc`

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 Axisymmetric**.
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>Frequency Domain**.
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
r0	1[mm]	0.001 m	Disc radius
h0	10[um]	1E-5 m	Gap height
dh	dhND*h0	1E-7 m	Change in gap height
dhND	0.01	0.01	Fractional gap height change.
mu0	1e-5[Pa*s]	1E-5 Pa·s	Gas viscosity
f0	1000[Hz]	1000 Hz	Vibration frequency

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 **r** text field, type `r0`.
6 Click **Build All Objects**.
7 Click the **Zoom Extents** button in the **Graphics** toolbar.

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.

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
Ftot	intop1(tff.fwallz)	N	Total force on disc
Ftotan	-3*pi*mu0*vf*r0^4/(2*h0^3)	N	Analytic expression
vf	2*pi*f0*dh	m/s	Disc velocity
Ftotantime	-6*pi^2*mu0*f0*r0^4*dh* cos(2*pi*f0*t)/(2*(h0+dh* sin(2*pi*f0*t))^3)	N	Analytic expression

Analytic 1 (an1)

1 In the **Definitions** toolbar, click f(x) **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*pi*mu0*f0*dh*(r0^2-rf^2)/h0^3.

4 In the **Arguments** text field, type rf.

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

Argument	Unit
rf	m

6 In the **Function** text field, type Pa.

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

The density is not required because the modified Reynolds equation is solved.

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 **Physical Model** section.

3 From the **Fluid type** list, choose **Gas (modified Reynolds equation)**.

Fluid-Film Properties 1

Reverse the direction of the geometry normal. Note that this is necessary to ensure that the wall load acts in the vertical direction. See Figure 1 for more details on the orientation of the wall and base with respect to the geometric and reference surface normals. To view the orientation of the reference normal, show the default solver settings and then compute only to the dependent variables stage in the study sequence. Then plot the reference surface normal.

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**, click to expand the **Reference Surface Properties** section.

3 From the **Reference normal orientation** list, choose **Opposite direction to geometry normal**.

4 Locate the **Wall Properties** section. In the h_{w1} text field, type h0.

5 From the $\mathbf{u}_w$ list, choose **None**.

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

0	r
vf	z

7 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 From the $\mathbf{u}_w$ list, choose **User defined**. Specify the **associated** vector as

0	r
dh*sin(2*pi*f0*t)	z

5 From the $\mathbf{v}_w$ list, choose **Calculate from wall displacement**.

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 **Extra fine**.

4 Click **Build All**.

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

STUDY 1

Step 1: Frequency Domain

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

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

3 In the **Frequencies** text field, type `1000`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the tree, select **Component 1 (comp1)>Thin-Film Flow (tff)>Fluid-Film Properties 2**.

6 Right-click and choose **Disable**.

7 In the tree, select **Component 1 (comp1)**.

8 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 **Plot Settings** section.

3 Select the **x-axis label** check box. In the associated text field, type `Radial distance (m)`.

4 Select the **y-axis label** check box. In the associated text field, type `Pressure (Pa)`.

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**, click to expand the **Legends** section.

4 Select the **Show legends** check box.

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

6 In the table, enter the following settings:

Legends
`COMSOL`

Line Graph 2

1 In the **Model Builder** window, 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 `Pan(r)`.

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

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

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

8 Locate 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 Solution`

11 In the **1D Plot Group 2** toolbar, click **Plot**.

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

Radial Pressure

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

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

3 Click **OK**.

2D Plot Group 3

In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

Arrow Line 1

1 Right-click **2D Plot Group 3** and choose **Arrow Line**.

2 In the **Settings** window for **Arrow Line**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Thin-Film Flow>Fluid loads>tff.fwallr,tff.fwallz - Fluid load on wall**.

3 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `30`.

4 Locate the **Coloring and Style** section. Select the **Scale factor** check box.

5 Use the slider to adjust the arrow length.

6 In the **2D Plot Group 3** toolbar, click **Plot**.

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

Line 1

1 In the **Model Builder** window, right-click **2D Plot Group 3** and choose **Line**.

2 In the **Settings** window for **Line**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Thin-Film Flow>Fluid loads>Fluid load on wall - N/m²>tff.fwallz - Fluid load on wall, z-component**.

3 Locate the **Coloring and Style** section. From the **Color table transformation** list, choose **Reverse**.

4 In the **2D Plot Group 3** toolbar, click **Plot**.

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

Wall Load

1 Right-click **2D Plot Group 3** and choose **Rename**.

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

3 Click **OK**.

Global Evaluation 1

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`abs(Ftot)`	N	

4 Click **Evaluate**.

Global Evaluation 2

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)> Definitions>Variables>Ftotan - Analytic expression - N**.

3 Click next to **Evaluate**, then choose **Table 1 - Global Evaluation 1**.

ROOT

The agreement between the total force computed by COMSOL and the analytic expression is excellent.

Next add a study to solve the problem in the time domain.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Time Dependent**.

4 Click **Add Study** in the window toolbar.

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

STUDY 2

Step 1: Time Dependent

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

2 Click **Range**.

3 In the **Range** dialog box, type `1/(40*f0)` in the **Step** text field.

4 In the **Stop** text field, type `5/f0`.

5 Click **Replace**.

6 In the **Settings** window for **Time Dependent**, click to expand the **Study Extensions** section.

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

8 Click Add.

9 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
dhND (Fractional gap height change.)	0.01 0.1 0.2 0.3	

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

RESULTS

1D Plot Group 5

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 2/Solution 2 (sol2)**.

Point Graph 1

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

2 Select Point 1 only.

3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Thin-Film Flow>Wall and base properties>tff.hw - Height of wall above reference plane - m**.

4 In the **1D Plot Group 5** toolbar, click **Plot**.

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

Wall height vs. time

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

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

3 Select the **Manual axis limits** check box.

4 In the **y minimum** text field, type `0`.

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

6 In the **1D Plot Group 5** toolbar, click **Plot**.

7 In the **Label** text field, type `Wall height vs. time`.

8 In the **Wall height vs. time** toolbar, click **Plot**.

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

The wall height varies sinusoidally, as expected. Except in the case when `dhND` is 0.01, the height varies by a significant fraction of the gap. This means that the equation system is nonlinear.

1D Plot Group 6

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 2/Solution 2 (sol2)**.

Point Graph 1

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

2 Select Point 1 only.

3 In the **1D Plot Group 6** toolbar, click **Plot**.

Pressure vs. time

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

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

3 In the **Pressure vs. time** toolbar, click **Plot**.

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

Once the wall height varies by a significant fraction of the gap nonlinear effects become important. The pressure does not vary sinusoidally any more and it is no longer possible to model the equation straightforwardly in the frequency domain.

When the height variations are small direct comparison with the frequency domain solution is possible. Do this by comparing the maximum value of the pressure in the time domain with the amplitude of the pressure computed in the frequency domain.

Point Evaluation 1

1 In the **Results** toolbar, click **Point Evaluation**.

2 Select Point 1 only.

3 In the **Settings** window for **Point Evaluation**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
abs(pfilm)	Pa	

5 Click **Evaluate**.

Point Evaluation 2

1 In the **Results** toolbar, click 8.85 e-12 **Point Evaluation**.

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

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 From the **Parameter selection (dhND)** list, choose **From list**.

5 In the **Parameter values (dhND)** list, select **0.01**.

6 Select Point 1 only.

7 Locate the **Data Series Operation** section. From the **Transformation** list, choose **Minimum**.

8 Click **Evaluate**.

TABLE 3

1 Go to the **Table 3** window.

In this case the time and frequency domain analyses are in good agreement.

RESULTS

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 2/Solution 2 (sol2)**.

Global 1

1 Right-click **1D Plot Group 7** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>Ftot - Total force on disc - N**.

Global 2

1 In the **Model Builder** window, right-click **1D Plot Group 7** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>Ftotantime - Analytic expression - N**.

3 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

Total force on disc vs. time

1 In the **Model Builder** window, click **1D Plot Group 7**.

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

3 Select the **Manual axis limits** check box.

4 In the **x minimum** text field, type `0.003`.

5 In the **x maximum** text field, type `0.005`.

6 Locate the **Legend** section. From the **Position** list, choose **Lower left**.

7 In the **Label** text field, type `Total force on disc vs. time`.

8 In the **Total force on disc vs. time** toolbar, click **Plot**.

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

The agreement between the COMSOL result and the analytic expression is good. The force acting on the disc remains periodic but has higher harmonic components at large displacements relative to the gap size.