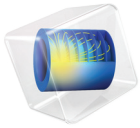


COMSOL Multiphysics

Application Library Manual

5.3a

COMSOL

COMSOL Multiphysics Application Library Manual

Version: COMSOL 5.3a

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries, contact Technical Support, or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located at the COMSOL Access page at www.comsol.com/support/case. Other useful links include:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/community
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM020004

Automotive Muffler

Introduction

This example describes the pressure wave propagation in a muffler for an explosion engine. The approach is general for analysis of damping of harmonic pressure waves.

The purpose of the application is to show how to treat 3D acoustics in a fairly complex geometry consisting of several separate sections and pipes divided by thin perfectly rigid walls. The analysis gives the transmission loss in the frequency range 100 Hz–1000 Hz.

Model Definition

The model geometry consists of three separate resonator chambers divided by thin walls. The inlet and the outlet correspond to the connection in the direction of the engine and of free air, respectively.

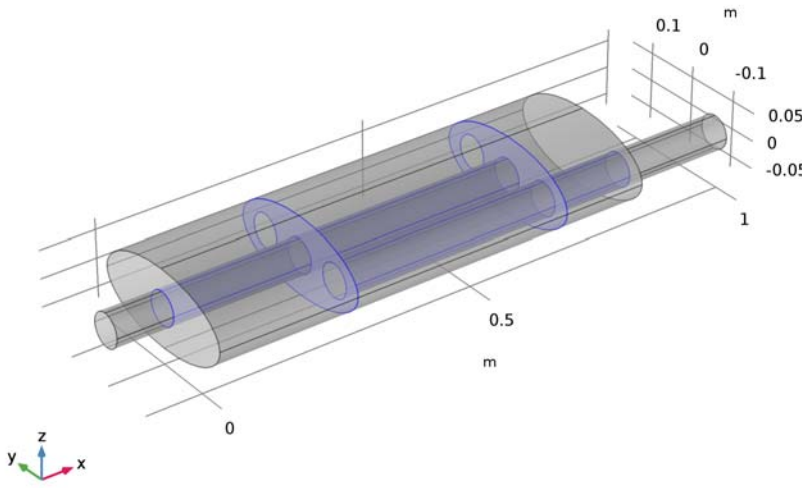


Figure 1: The geometry of a muffler. The exhaust fumes enter through the left pipe, pass the three resonator chambers, and exit through the right pipe.

DOMAIN EQUATIONS

You solve the problem in the frequency domain using the Pressure Acoustics, Frequency Domain interface. The model equation is a slightly modified Helmholtz equation for the acoustic pressure p:

$$\nabla \cdot \left(-\frac{\nabla p}{\rho}\right) - \frac{\omega^2 p}{c^2 \rho} = 0$$

where ρ is the density, c is the speed of sound, and ω is the angular frequency. The density needs to be included in the equation in cases where variations in density in different materials exist. The model assumes that in the low-frequency range, reactive damping prevails. Resistive damping is therefore not included.

BOUNDARY CONDITIONS

The boundary conditions are of three different types. At all the solid boundaries, which include the outer walls of the muffler, the dividing walls between the resonator chambers, and the walls of the pipes, sound hard (wall) boundary conditions are used:

$$\left(-\frac{\nabla p}{\rho}\right) \cdot \mathbf{n} = 0$$

At the inlet boundary, a combination of an incoming and an outgoing plane waves is assumed:

$$\left(-\frac{\nabla p}{\rho}\right) \cdot \mathbf{n} = \frac{i\omega}{\rho c} p - \frac{2\, i\omega}{\rho c} p_0$$

In this equation p_0 denotes the applied outer pressure and i the imaginary unit. At the outlet boundary, an outgoing plane wave is set:

$$\left(-\frac{\nabla p}{\rho}\right) \cdot \mathbf{n} = \frac{i\omega}{\rho c} p$$

Results and Discussion

Figure 2 visualizes the pressure field in the muffler at a frequency of 490 Hz using a boundary plot of the absolute value of the pressure and an isosurface plot of the pressure.

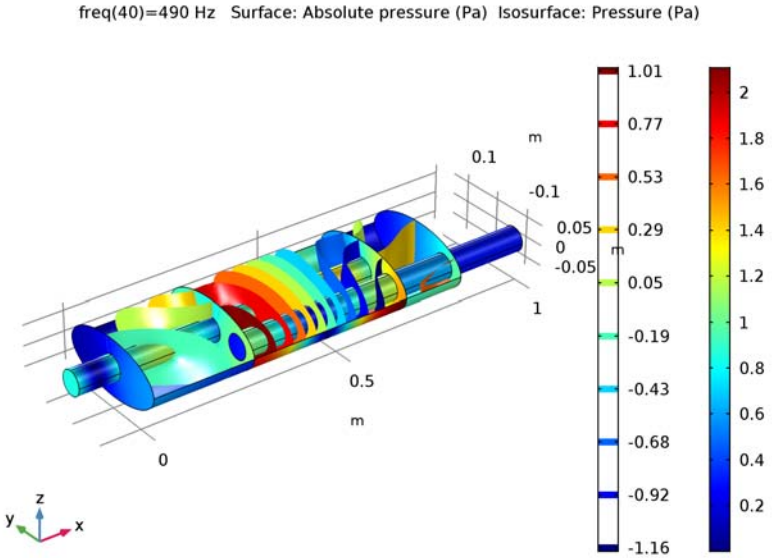


Figure 2: The solution at 490 Hz. The real value of the pressure is plotted as isosurfaces, and the absolute value of the pressure is displayed as a boundary plot on the inner walls of the muffler.

The following equation defines the transmission loss in the muffler:

$$\mathrm{TL} = 10\log\left(\frac{P_{\mathrm{in}}}{P_{\mathrm{out}}}\right)$$

Here, P_{in} and P_{out} denote the acoustic effect at the inlet and outlet, respectively. The acoustic effect is calculated using the following equations:

$$P_{\mathrm{in}} = \int_{\partial\Omega} \frac{p_0^{\,2}}{2\rho c} dA$$

$$P_{\mathrm{out}} = \int_{\partial\Omega} \frac{|p_c|^2}{2\rho c} dA$$

Figure 3 shows the result of a parametric frequency study. This plot reveals that the damping is better at higher frequencies, with the exception of several deep dips

throughout the frequency range. The dips correspond to the resonance frequencies for different parts of the muffler system.

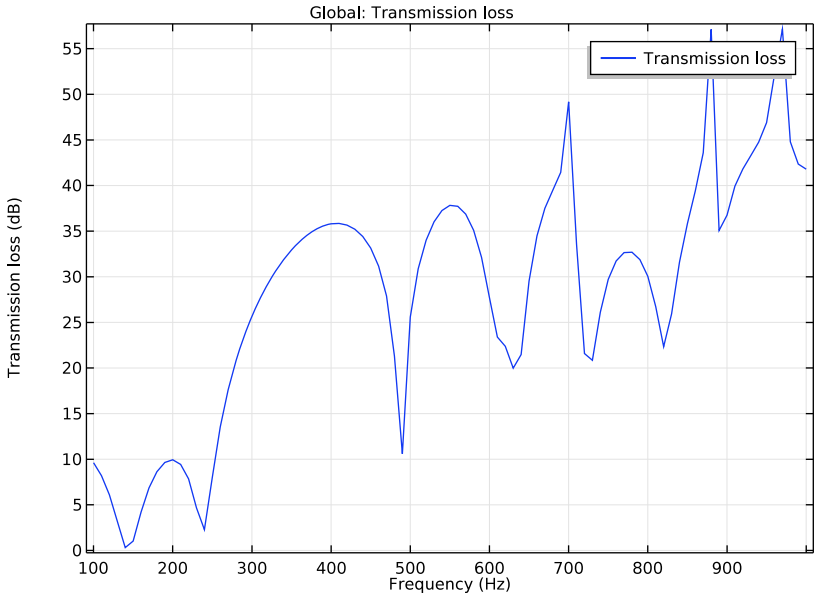


Figure 3: The damping (dB) in the muffler as a function of the frequency (Hz).

Application Library path: `COMSOL_Multiphysics/Acoustics/automotive_muffler`

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 **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Frequency Domain**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
p0	1[Pa]	1 Pa	Inlet pressure amplitude

GEOMETRY 1

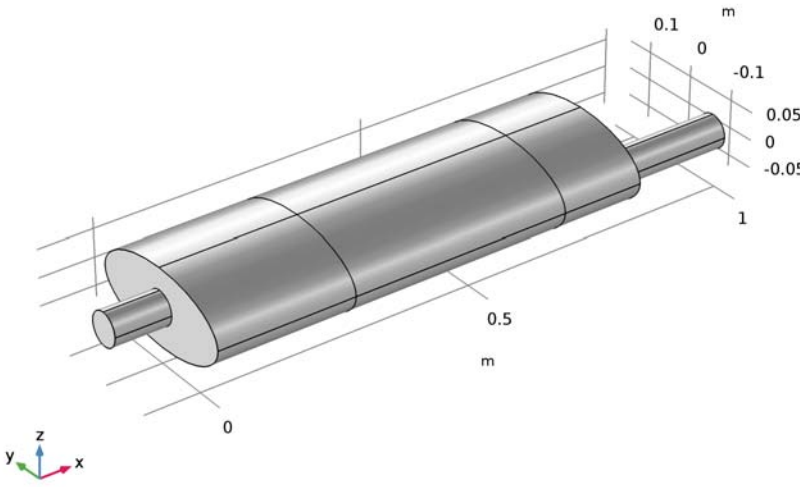
Create the geometry. To simplify this step, insert a prepared geometry sequence.

1 On the **Geometry** toolbar, click **Insert Sequence**.

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

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

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



DEFINITIONS

Define integration operators for the inlet and outlet, then use these to calculate the attenuation.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component 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.

5 In the **Operator name** text field, type `intop_inlet`.

6 In the **Label** text field, type `inlet`.

Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component 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 50 only.

5 In the **Operator name** text field, type `intop_outlet`.

6 In the **Label** text field, type `outlet`.

Variables 1

1 On the **Definitions** toolbar, click **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
`P_in`	`intop_inlet(p0^2/(2*acpr.rho*acpr.c))`	W	`Incoming power`
`P_out`	`intop_outlet(p*conj(p)/(2*acpr.rho*acpr.c))`	W	`Outgoing power`
`TL`	`10*log10(P_in/P_out)`		`Transmission loss`

Note that you need to define the incoming power using `p0` rather than the pressure variable, `p`, because `p` is the sum of incident and reflected pressure waves.

Explicit 1

Create a selection to simplify setting up the interior boundary conditions.

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

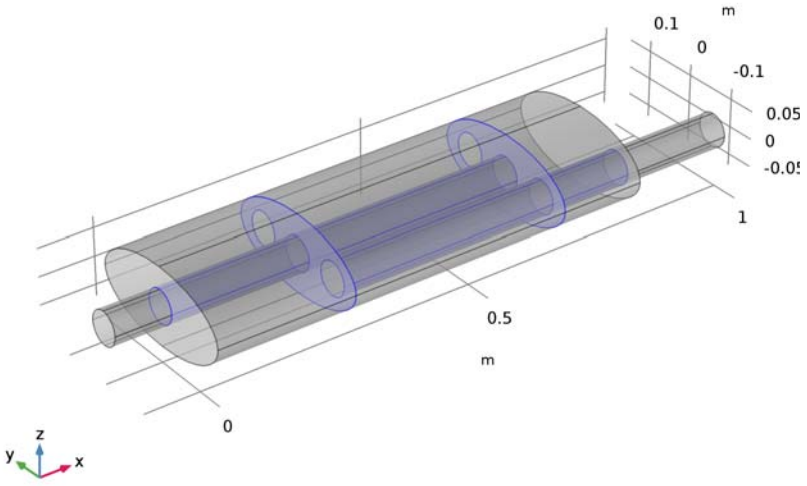
2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Click the **Transparency** button on the **Graphics** toolbar.

5 Select Boundaries 10, 11, 13, 14, 16, 20–23, 25, 26, 28, 29, 32, and 36–39 only.

To do this, click the **Paste Selection** button next to the **Selection** box, paste the text: 10, 11, 13, 14, 16, 20–23, 25, 26, 28, 29, 32, 36–39 in the text field of the dialog box that opens, and finally click **OK**.



6 In the **Label** text field, type `interior boundaries`.

ADD MATERIAL

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

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

3 In the tree, select **Built-In>Air**.

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

MATERIALS

By default, the first material you add applies for all domains. In the **Material Contents** table you can see which material properties the physics interfaces use, in this case the density andthe speed of sound. Notice that these quantities are functions of the temperature and (in the case of the density) the ambient pressure.

1 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Pressure Acoustics 1

In the **Model Inputs** section you can read off and, if desired, modify the temperature and absolute pressure at which the expressions for the air density and speed of sound are calculated. For this model, use the default settings.

Plane Wave Radiation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Plane Wave Radiation**.

2 Select Boundary 1 only.

Incident Pressure Field 1

1 On the **Physics** toolbar, click **Attributes** and choose **Incident Pressure Field**.

2 In the **Settings** window for **Incident Pressure Field**, locate the **Incident Pressure Field** section.

3 In the p_0 text field, type `p0`.

4 From the c list, choose **From material**.

5 From the **Material** list, choose **Air (mat1)**.

Plane Wave Radiation 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Plane Wave Radiation**.

2 Select Boundary 50 only.

3 Click the **Transparency** button on the **Graphics** toolbar to return to the default transparency state.

Interior Sound Hard Boundary (Wall) 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Interior Sound Hard Boundary (Wall)**.

2 In the **Settings** window for **Interior Sound Hard Boundary (Wall)**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **interior boundaries**.

MESH 1

Use the default physics-controlled mesh as a starting point and then modify the maximum element size so that you get 10 elements per wavelength for the highest frequency in the sweep, that is 1 kHz.

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

2 In the **Settings** window for **Mesh**, click **Build All**.

3 Right-click **Component 1 (comp1)>Mesh 1** and choose **Edit Physics-Induced Sequence**.

Size

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

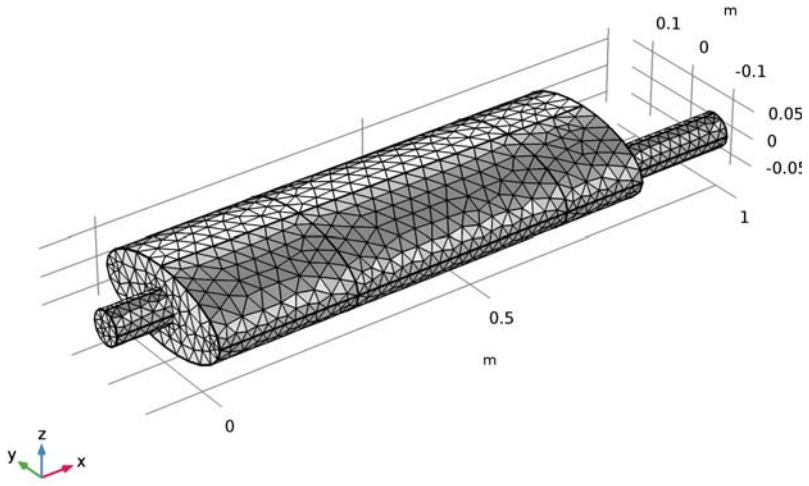
2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `343[m/s]/1[kHz]/10`.

This corresponds to one 10th of the shortest wavelength.

5 Click **Build All**.



STUDY 1

Step 1: Frequency Domain

1 In the **Model Builder** window, expand the **Study 1** node, then 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 `range(100,10,1000)`.

This computes the solution for 91 equally spaced frequencies from 100 Hz to 1000 Hz. If you want to run a faster analysis, try the same frequency range but with a step of 100 Hz instead (to do so, type `range(100,100,1000)`).

4 On the **Home** toolbar, click **Compute**.

RESULTS

Acoustic Pressure (acpr)

Before visualizing the acoustic pressure field, add a selection to the default solution data set that filters out the upper muffler boundaries for a better view.

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/ Solution 1 (sol1)**.

Selection

1 On the **Results** toolbar, click **Selection**.

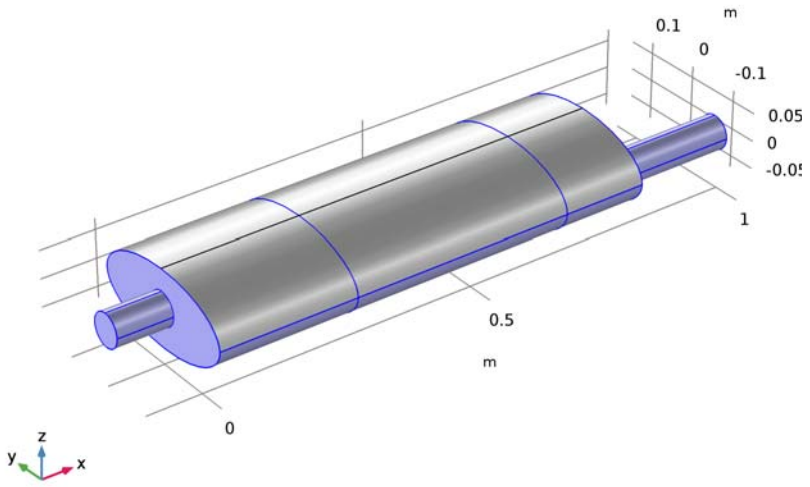
2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

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

4 Choose **All boundaries** from the **Selection** list.

5 Ctrl+click to highlight the six top faces of the muffler, then right-click to remove them from the selection.

Alternatively, you can click the **Paste Selection** button next to the **Selection** box and then paste the text "1–7, 9–14, 16, 17, 19–29, 31–33, 35–41, and 43–50" in the text field of the dialog box that opens before clicking **OK**.



Acoustic Pressure (acpr)

Reproduce the plot in Figure 2 by following these steps.

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (acpr)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Parameter value (freq (Hz))** list, choose **490**.

Surface 1

1 In the **Model Builder** window, expand the **Acoustic Pressure (acpr)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>acpr.absp - Absolute pressure**.

3 On the **Acoustic Pressure (acpr)** toolbar, click **Plot**.

Isosurface 1

1 In the **Model Builder** window, under **Results** right-click **Acoustic Pressure (acpr)** and choose **Isosurface**.

2 In the **Settings** window for **Isosurface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>p - Pressure**.

3 Locate the **Levels** section. In the **Total levels** text field, type `10`.

What is a suitable number of isosurface levels for the isosurface plot varies with the frequency. At frequencies with low damping many of the isosurfaces tend to congregate inside the pipe.

4 On the **Acoustic Pressure (acpr)** toolbar, click **Plot**.

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

Finally, reproduce the plot of attenuation versus frequency shown in Figure 3.

Global 1

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

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

3 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>Definitions>Variables>TL - Transmission loss**.

1D Plot Group 4

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

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

3 Locate the **Plot Settings** section. Select the **x-axis label** check box.

4 In the associated text field, type `Frequency (Hz)`.

5 Select the **y-axis label** check box.

6 In the associated text field, type `Transmission loss (dB)`.

7 On the **Transmission Loss** toolbar, click **Plot**.

Notice the deep dip in the damping around 490 Hz caused by the resonance in the second chamber. If you plot the pressure in the muffler at other dips, resonances in the other chambers appear.

The Black-Scholes Equation

Introduction

There are different types of stock options:

- A *call option* is the right to buy a security at a specified price (called the exercise or strike price) during a specified period of time.
- A *put option* is the right to sell a security at a specified price during a specified period of time.

American options can be exercised at any time up to and including the day the option expires. European options can be exercised only on the day the option expires.

The famous Black-Scholes equation computes the cost u of a European stock option

$$\frac{\partial u}{\partial t} + \frac{1}{2}\sigma^2 x^2 \frac{\partial^2 u}{\partial x^2} + rx\frac{\partial u}{\partial x} = ru$$

with the following parameters:

- x, the underlying asset price
- r, the continuous compounding rate of interest
- σ, the standard deviation of the asset's rate of return (also known as volatility)

A put option's value on the exercise day is

$$u(T, x) = \max(K - x, 0)$$

where K is the strike price. The problem domain is infinite and consists of the entire real axis across the time domain $0 \le t \le T$.

The assumptions made in deriving the Black-Scholes equation are:

- The underlying stock pays no dividends.
- The price of the stock, one period ahead, has a log-normal distribution with mean and standard deviation that are constant over the life of the option.
- The existence of a risk-free interest rate which is constant over the life of the option.
- You can lend and borrow at the risk-free interest rate.

Black and Scholes derived an analytical expression for the solution to the above problem. However, the formula works only for certain cases. For instance, you cannot use it when σ and r are functions of x and t. Using the PDE formulation, you can determine the price for such cases.

Model Definition

Because you work within a finite domain $0 \le x \le X$, it is necessary to specify not only the boundary conditions for $t = T$ but also for $x = 0$ and $x = X$. It is therefore necessary to analyze the problem's characteristics to determine the location of the input and output boundaries.

EQUATION DEFINITION

To put the equation in coefficient form, rewrite the equation as

$$\frac{\partial u}{\partial t} + \frac{\partial}{\partial x}\left(\frac{1}{2}\sigma^2 x^2 \frac{\partial u}{\partial x}\right) + \left(rx - \frac{\partial}{\partial x}\left(\frac{1}{2}\sigma^2 x^2\right)\right)\frac{\partial u}{\partial x} - rc = 0.$$

In the following, denote

$$\bar{r} = rx - \frac{\partial}{\partial x}\left(\frac{1}{2}\sigma^2 x^2\right).$$

To reduce the problems with inflow boundaries, start by considering a put option: Study the value of a put option at a strike price $K = 40$ with $\sigma = 0.3$ and $r = 0.12$.

BOUNDARY CONDITIONS AND INITIAL CONDITIONS

Make the domain be $0 \le x \le 80$ with time running from 12 to 0. Then the initial condition at $t = 12$ and $x = 80$ is 0 based on the put option's value. The initial condition in the region $0 \le x \le 40$ varies linearly from 0 to 40. At the end of the simulation domain, the boundary is free (use a homogeneous boundary condition).

Notes About the COMSOL Implementation

Model the Black-Scholes equation using the following approach:

- Create a 1D time-dependent model, using the time-stepping algorithm to solve for c as a function of x and t, the time. The time steps go backward in time. Using a variable substitution to reverse the sign of the time, the d_a coefficient becomes –1.
- To model the initial condition, use the logical expression `(x<40)*(40-x)`. This means that in the areas where $x > 40$, the initial value is zero.

Application Library path: `COMSOL_Multiphysics/Equation_Based/black_scholes_put`

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1. In the **Model Wizard** window, click **1D**.
2. In the **Select Physics** tree, select **Mathematics>PDE Interfaces>Coefficient Form PDE (c)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Time Dependent**.
6. Click **Done**.

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**.

   Keeping track of units is not important in this model; by turning off unit support, you avoid the need to specify dimensions for equation coefficients and coordinates to get rid of unit warnings.

GEOMETRY 1

Interval 1 (i1)

1. On the **Geometry** toolbar, click **Interval**.
2. In the **Settings** window for **Interval**, locate the **Interval** section.
3. In the **Right endpoint** text field, type `80`.
4. Right-click **Interval 1 (i1)** and choose **Build Selected**.

GLOBAL DEFINITIONS

Parameters

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
r	0.12	0.12	Continuous compounding interest rate
sigma	0.3	0.3	Volatility

COEFFICIENT FORM PDE (C)

Coefficient Form PDE 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Coefficient Form PDE (c)** click **Coefficient Form PDE 1**.

2 In the **Settings** window for **Coefficient Form PDE**, locate the **Diffusion Coefficient** section.

3 In the *c* text field, type `1/2*sigma^2*x^2`.

4 Locate the **Absorption Coefficient** section. In the *a* text field, type `r`.

5 Locate the **Source Term** section. In the *f* text field, type `0`.

6 Locate the **Damping or Mass Coefficient** section. In the d_a text field, type `-1`.

7 Click to expand the **Convection coefficient** section. Locate the **Convection Coefficient** section. In the β text field, type `(-r+sigma^2)*x`.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Coefficient Form PDE (c)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the *u* text field, type `(x<40)*(40-x)`.

Flux/Source 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Flux/Source**.

2 Select Boundary 1 only.

Dirichlet Boundary Condition 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Dirichlet Boundary Condition**.

2 Select Boundary 2 only.

MESH 1

Size

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

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `2`.

5 Click **Build All**.

The mesh consists of 40 elements.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.

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

3 In the **Times** text field, type `range(12,-0.5,0)`.

4 On the **Home** toolbar, click **Compute**.

RESULTS

1D Plot Group 1

To see the plot of `u` as a Line Graph at time = 0, follow the steps given below.

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

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

3 From the **Time selection** list, choose **From list**.

4 In the **Times (s)** list, select **0**.

5 Click to expand the **Title** section. Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `x`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `u`.

Line Graph 1

1 In the **Model Builder** window, expand the **1D Plot Group 1** node, then click **Line Graph 1**.

2 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Component 1>Geometry>Coordinate> x - x-coordinate**.

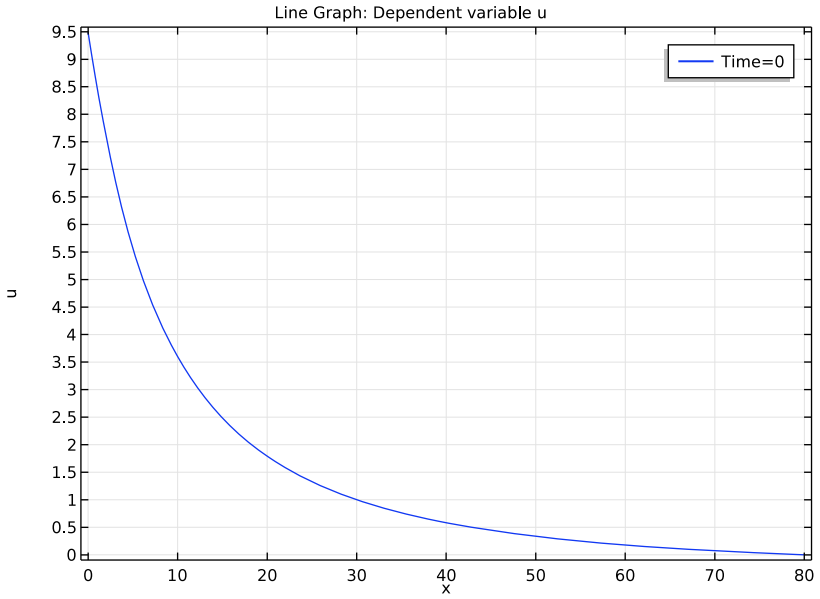
3 Click to expand the **Legends** section. Select the **Show legends** check box.

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

5 In the table, enter the following settings:

Legends
Time=0

6 On the **1D Plot Group 1** toolbar, click **Plot**.



The Blasius Boundary Layer

Introduction

The incompressible boundary layer on a flat plate in the absence of a pressure gradient is usually referred to as the Blasius boundary layer (Ref. 1). The steady, laminar boundary layer developing downstream of the leading edge eventually becomes unstable to Tollmien-Schlichting waves and finally transitions to a fully turbulent boundary layer. Due to its fundamental importance, this type of flow has become the subject of numerous studies on boundary-layer flow, stability, transition, and turbulence. This application considers the first section of the plate where the boundary layer remains steady and laminar, and compares results from incompressible, two-dimensional, single-phase-flow simulations obtained in COMSOL Multiphysics to the Blasius similarity solution. The solutions converge ideally with respect to both mesh refinement and discretization order.

Model Definition

Consider a homogeneous free-stream flow with speed U_0 parallel to an infinitely thin, flat plate located along the positive x-axis. The flow is assumed to be steady, symmetric with respect to y, and homogeneous in the z direction. Due to friction, the flow adjacent to the plate is retarded and a thin boundary layer, where the velocity gradually grows from zero to the free-stream value, develops downstream of the leading edge (see Figure 1).

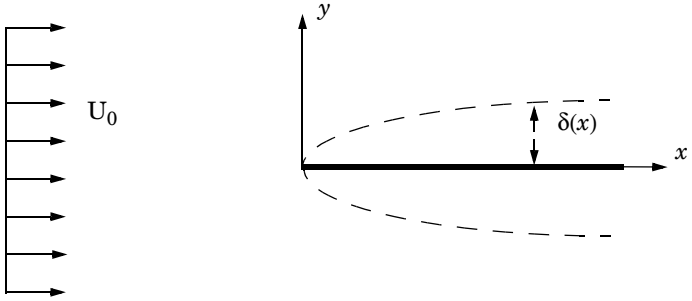


Figure 1: The boundary layer on a flat plate. $\delta(x)$ is the boundary-layer thickness, such that $u(x, \delta(x)) = U_0$.

A reasonably accurate solution for the flow field can be found by considering the boundary-layer approximation to the steady, incompressible Navier-Stokes equations

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = \upsilon\frac{\partial^2 u}{\partial y^2} \tag{1}$$

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{2}$$

Introducing a stream function,

$$u = \frac{\partial \Psi}{\partial y}, \quad v = -\frac{\partial \Psi}{\partial x}$$

and the similarity transformation,

$$\psi = \sqrt{\upsilon x U_0} f(\eta), \quad \eta = \frac{y}{\sqrt{\upsilon x / U_0}}$$

Equation 1 and Equation 2 reduce to the ODE

$$2f''' + ff'' = 0 \tag{3}$$

COMSOL solves Equation 3 on the interval $\eta \in [0, 10]$ with the boundary conditions

$$f(0) = 0, \quad f'(0) = 0$$
$$\lim_{\eta \to \infty} f'(\eta) = 1$$

by rewriting the equation as a system of two equations,

$$\begin{cases} f' = f_{\text{prime}} \\ f''_{\text{prime}} = -\frac{1}{2} f f_{\text{prime}} \end{cases}$$

and implementing the system within the Coefficient Form PDE interface.

Using the Laminar Flow interface for single-phase flow, the model solves the steady, incompressible Navier-Stokes equation in a domain $(x, y) \in ([-1, 2.1], [0, 0.5])$ m with the leading edge of the plate located at $x = 0$ m. The working fluid is air at a temperature of $T = 20$ °C and $U_0 = 0.75$ m/s. The simulations uses discretizations with linear basis functions for velocity and pressure (P1+P1) on three different meshes.

Results and Discussion

Figure 2 shows the similarity solution $u/U_0 = f'(\eta)$. At $\eta = 4.99$, the deviation from the free-stream value is 1%. This value can be used to define the boundary-layer thickness,

$$\delta_{99}(x) = 4.99 \sqrt{\frac{\upsilon x}{U_0}}$$

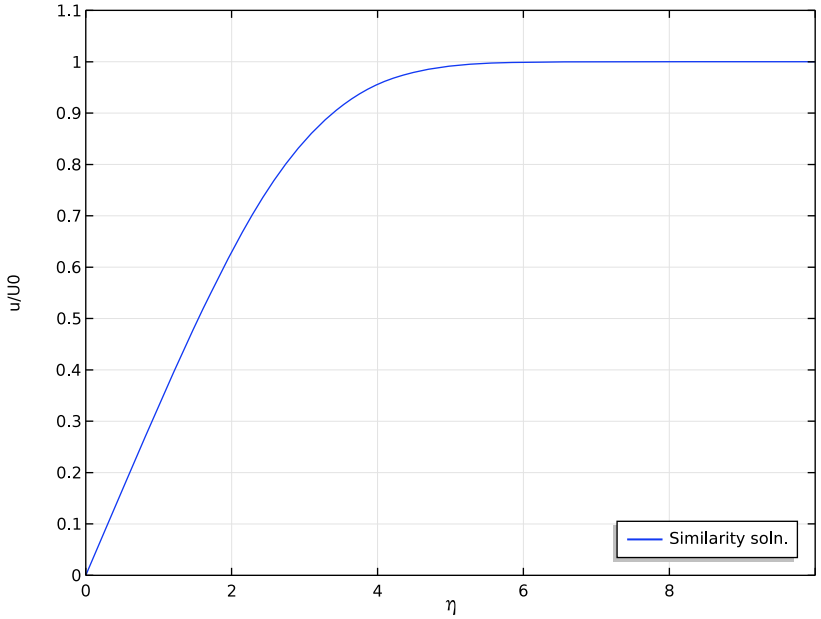


Figure 2: Similarity solution for the streamwise velocity component.

Figure 3 shows a comparison between the Blasius similarity solution and the results from the two-dimensional simulations at $x_E = 2$ m, corresponding to a Reynolds number of $\mathrm{Re}_x = 1.0 \cdot 10^5$. Only the results from the P1+P1 simulation on the coarse mesh show a significant deviation from the similarity solution. To quantify differences in the results, define the following measure,

$$\varepsilon = \sqrt{\int_0^{\eta_\infty} \left(\frac{u}{U_0} - f' \right)^2 d\eta}$$

Here, $\eta_\infty = 10$, for which the similarity solution has converged to its asymptotic value to within the numerical precision in the computations.

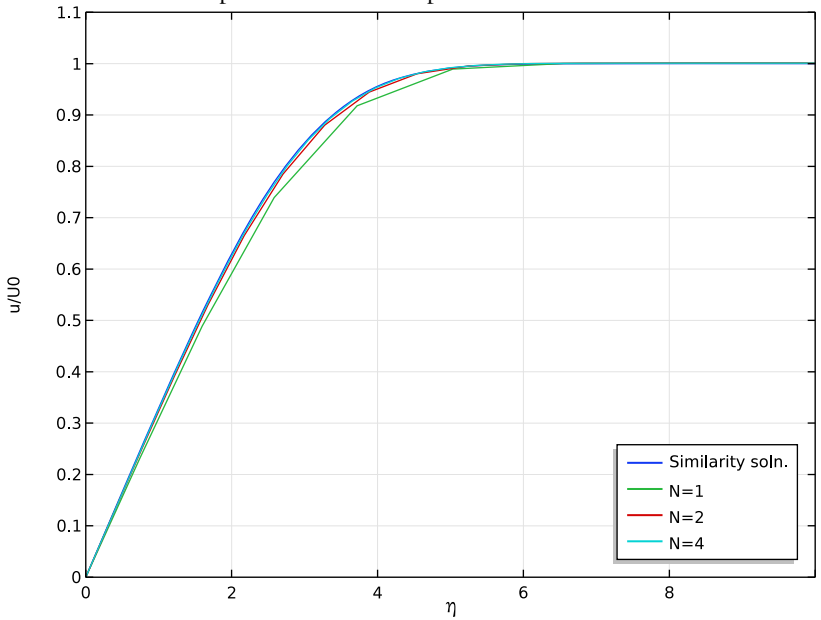


Figure 3: Comparison between the similarity solution and the two-dimensional simulations.

Table 1 displays deviations from the similarity solution together with the number of degrees of freedom (DOF) for the three simulations. The convergence is displayed in Figure 4 where the mesh size h is calculated as the maximum cell side in the mesh. The curve is close to straight line, which means that the model is in a mesh convergence regime; that is, the solution converges toward the correct solution when the mesh is refined.

TABLE 1: DEVIATION FROM THE BLASIUS SOLUTION

l/H	10	20	40
ε	$6.10 \cdot 10^{-2}$	$3.33 \cdot 10^{-2}$	$1.68 \cdot 10^{-2}$
DOF	2016	7749	30375

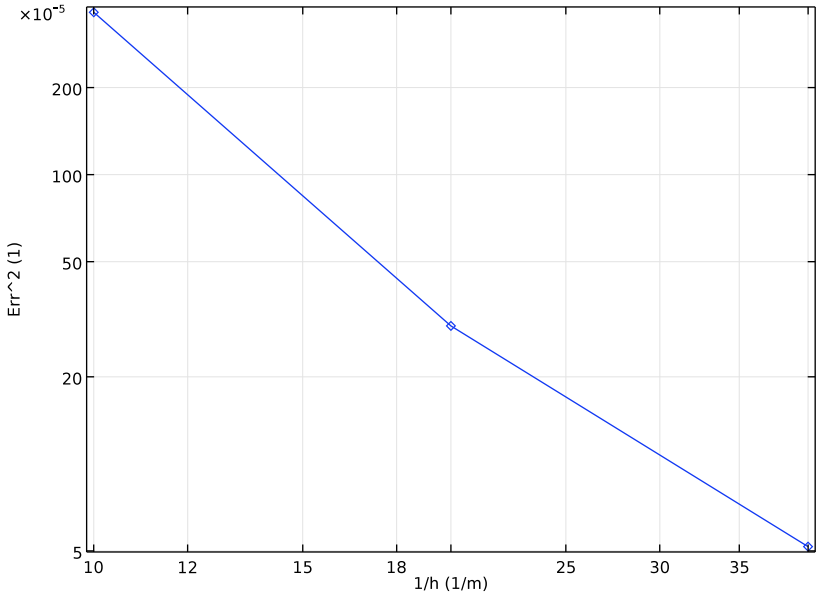


Figure 4: Convergence rate as a function of inverse maximum cell side.

Notes About the COMSOL Implementation

The relative tolerance is set to 10^{-4} in all the solvers to ensure that the equation systems become well converged. All meshes have monotonically increasing element sizes away from the plate, with distributions employing geometric sequences. A component coupling is set up to enable evaluation of the similarity solution in the two-dimensional model.

Reference

1. H. Blasius, "Grenzschichten in Flüssigkeiten mit kleiner Reibung," *Z. Math. Phys.*, vol. 56, pp. 1–37, 1908 (Engl. transl. in *NACA TM* 1256).

Application Library path: `COMSOL_Multiphysics/Fluid_Dynamics/blasius_boundary_layer`

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **1D**.

2 In the **Select Physics** tree, select **Mathematics>PDE Interfaces>Coefficient Form PDE (c)**.

3 Click **Add**.

4 In the **Field name** text field, type `f`.

5 Click **Add Dependent Variable**.

6 In the **Dependent variables** table, enter the following settings:

`f`
`fprime`

7 Click **Study**.

8 In the **Select Study** tree, select **Preset Studies>Stationary**.

9 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
U0	`0.75[m/s]`	0.75 m/s	`Inlet velocity`
nu	`1.506137e-5[m^2/s]`	1.5061E-5 m²/s	`Kinematic viscosity`
xE	`2[m]`	2 m	`Evaluation location`
b0	`nu/U0`	2.0082E-5 m	`B-L scale`
N	`1`	1	`Mesh refinement factor`

GEOMETRY 1

Interval 1 (i1)

1. On the **Geometry** toolbar, click **Interval**.
2. In the **Settings** window for **Interval**, locate the **Interval** section.
3. In the **Right endpoint** text field, type `10`.

COEFFICIENT FORM PDE (C)

Coefficient Form PDE 1

1. In the **Model Builder** window, expand the **Coefficient Form PDE (c)** node, then click **Coefficient Form PDE 1**.
2. In the **Settings** window for **Coefficient Form PDE**, locate the **Diffusion Coefficient** section.
3. In the c text-field array, type `0` in the first column of the first row.
4. In the c text-field array, type `-2` in the second column of the second row.
5. Locate the **Absorption Coefficient** section. In the a text-field array, type `1` in the second column of the first row.
6. Locate the **Source Term** section. In the f text-field array, type `0` on the first row.
7. In the f text-field array, type `0` on the second row.
8. Locate the **Damping or Mass Coefficient** section. In the d_a text-field array, type `0` in the first column of the first row.
9. In the d_a text-field array, type `0` in the second column of the second row.
10. Click to expand the **Convection coefficient** section. Locate the **Convection Coefficient** section. In the β text-field array, type `-1` in the first column of the first row.
11. In the β text-field array, type `f` in the second column of the second row.

Dirichlet Boundary Condition 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Dirichlet Boundary Condition**.
2. Select Boundary 1 only.

Dirichlet Boundary Condition 2

1. On the **Physics** toolbar, click **Boundaries** and choose **Dirichlet Boundary Condition**.
2. Click the **Zoom Extents** button on the **Graphics** toolbar.
3. Select Boundary 2 only.
4. In the **Settings** window for **Dirichlet Boundary Condition**, locate the **Dirichlet Boundary Condition** section.

5 Clear the **Prescribed value of f** check box.

6 In the r_2 text field, type `1`.

MESH 1

Edge 1

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

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

3 From the **Geometric entity level** list, choose **Entire geometry**.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Edge 1** and choose **Distribution**.

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

3 From the **Distribution properties** list, choose **Predefined distribution type**.

4 In the **Number of elements** text field, type `10000`.

5 In the **Element ratio** text field, type `100`.

6 From the **Distribution method** list, choose **Geometric sequence**.

7 Click **Build Selected**.

STUDY 1

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Stationary Solver 1**.

3 In the **Settings** window for **Stationary Solver**, locate the **General** section.

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

Allocate more memory than the default suggestion to avoid a warning message. The solver will automatically increase the allocation factor when needed, but changing it manually is more computationally efficient.

5 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1** node, then click **Direct**.

6 In the **Settings** window for **Direct**, locate the **General** section.

7 In the **Memory allocation factor** text field, type `1.5`.

8 On the **Study** toolbar, click **Compute**.

STUDY 1

1 In the **Model Builder** window, right-click **Study 1** and choose **Rename**.

2 In the **Rename Study** dialog box, type `Similarity soln.` in the **New label** text field.

3 Click **OK**.

RESULTS

Line Graph 1

1 In the **Model Builder** window, expand the **1D Plot Group 1** node, then click **Line Graph 1**.

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

3 In the **Expression** text field, type `fprime`.

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

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

6 In the table, enter the following settings:

Legends
Similarity soln.

1D Plot Group 1

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

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated text field, type `\eta`.

6 Select the **y-axis label** check box.

7 In the associated text field, type `u/U0`.

8 Click to expand the **Axis** section. Select the **Manual axis limits** check box.

9 In the **x minimum** text field, type `0`.

10 In the **x maximum** text field, type `10`.

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

12 In the **y maximum** text field, type `1.1`.

13 Click to expand the **Legend** section. From the **Position** list, choose **Lower right**.

14 On the **1D Plot Group 1** toolbar, click **Plot**.

DEFINITIONS

Set up a component coupling to be able to evaluate the similarity solution in the upcoming 2D model.

General Extrusion 1 (genext1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **General Extrusion**.

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

3 In the **Settings** window for **General Extrusion**, locate the **Destination Map** section.

4 In the **x-expression** text field, type `root.y/sqrt(b0*root.x)`.

ROOT

On the **Home** toolbar, click **Component** and choose **Add Component>2D**.

GEOMETRY 2

In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

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

3 In the tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Similarity soln.**

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

6 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ADD STUDY

1 On 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 **Preset Studies>Stationary**.

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

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

GEOMETRY 2

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `3.1`.

4 In the **Height** text field, type `0.5`.

5 Locate the **Position** section. In the **x** text field, type `-1`.

6 On the **Geometry** toolbar, click **Build All**.

Point 1 (pt1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Point**.

2 Click **Build All**.

Point 2 (pt2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Point**.

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

3 In the **y** text field, type `0.5`.

4 On the **Geometry** toolbar, click **Build All**.

ADD MATERIAL

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

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

3 In the tree, select **Built-In>Air**.

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

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

LAMINAR FLOW (SPF)

Inlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Inlet**, locate the **Velocity** section.

4 In the U_0 text field, type `U0`.

Open Boundary 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Open Boundary**.

2 Select Boundaries 3 and 5 only.

Outlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 Select Boundary 6 only.

Symmetry 1

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

2 Select Boundary 2 only.

MESH 2

Distribution 1

1 In the **Model Builder** window, under **Component 2 (comp2)** right-click **Mesh 2** and choose **Mapped**.

2 Right-click **Mapped 1** and choose **Distribution**.

3 Select Boundaries 4 and 5 only.

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

5 In the **Number of elements** text field, type `21*N`.

Distribution 2

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

2 Select Boundaries 2 and 3 only.

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

4 In the **Number of elements** text field, type `10*N`.

Distribution 3

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

2 Select Boundaries 1 and 6 only.

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

4 From the **Distribution properties** list, choose **Predefined distribution type**.

5 In the **Number of elements** text field, type `20*N`.

6 In the **Element ratio** text field, type `15`.

7 From the **Distribution method** list, choose **Geometric sequence**.

8 Click **Build Selected**.

STUDY 2

Parametric Sweep

1 On 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
N (Mesh refinement factor)	1 2 4

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 2** node, then click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Values of dependent variables** section.

3 Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

4 From the **Method** list, choose **Solution**.

5 From the **Study** list, choose **Similarity soln., Stationary**.

Solution 2 (sol2)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Stationary Solver 1**.

3 In the **Settings** window for **Stationary Solver**, locate the **General** section.

4 In the **Relative tolerance** text field, type `1e-5`.

5 In the **Model Builder** window, expand the **Study 2>Solver Configurations> Solution 2 (sol2)>Stationary Solver 1** node, then click **Fully Coupled 1**.

6 In the **Settings** window for **Fully Coupled**, click to expand the **Method and termination** section.

7 Locate the **Method and Termination** section. In the **Maximum number of iterations** text field, type `50`.

8 On the **Study** toolbar, click **Compute**.

RESULTS

Cut Line 2D 1

1 On the **Results** toolbar, click **Cut Line 2D**.

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

3 From the **Data set** list, choose **Study 2/Parametric Solutions 1 (5) (sol3)**.

4 Locate the **Line Data** section. In row **Point 1**, set **x** to `xE`.

5 In row **Point 2**, set **x** to `xE` and **y** to `10*sqrt(b0*xE)`.

1D Plot Group 1

In the **Model Builder** window, expand the **Results>1D Plot Group 1** node.

Line Graph 1

In the **Model Builder** window, expand the **1D Plot Group 1** node.

Line Graph 2

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

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

3 In the **Expression** text field, type `u/U0`.

4 Locate the **Data** section. From the **Data set** list, choose **Cut Line 2D 1**.

5 Locate the **x-Axis Data** section. In the **Expression** text field, type `y/sqrt(b0*x)`.

6 Click to expand the **Legends** section. In the table, enter the following settings:

Legends
N=1
N=2
N=4

7 On the **1D Plot Group 1** toolbar, click **Plot**.

Derived Values

In the **Model Builder** window, under **Results** click **Derived Values**.

Line Integration 1

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration> Line Integration**.

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

3 From the **Data set** list, choose **Cut Line 2D 1**.

4 Locate the **Expressions** section. Click **Clear Table**.

5 In the table, enter the following settings:

Expression	Unit	Description
`(u/U0-comp1.genext1(fprime))^2/sqrt(b0*x)`	1	`Err^2`

6 Click **Evaluate**.

Derived Values

In the **Model Builder** window, under **Results** click **Derived Values**.

Surface Minimum 1

1. On the **Results** toolbar, click **More Derived Values** and choose **Minimum>Surface Minimum**.
2. Click in the **Graphics** window and then press Ctrl+A to select all domains.
3. In the **Settings** window for **Surface Minimum**, locate the **Expressions** section.
4. In the table, enter the following settings:

Expression	Unit
1/h	1/m

5. Locate the **Data** section. From the **Data set** list, choose **Study 2/ Parametric Solutions 1 (5) (sol3)**.
6. Click the arrow next to the **Evaluate** button and choose **Table 1 - Line Integration 1 ((u/ U0-comp1.genext1(fprime))^2/sqrt(b0*x))**.

TABLE

1. Go to the **Table** window.
2. Click **Table Graph** in the window toolbar.

RESULTS

Table Graph 1

1. In the **Model Builder** window, under **Results>1D Plot Group 5** click **Table Graph 1**.
2. In the **Settings** window for **Table Graph**, locate the **Data** section.
3. From the **x-axis data** list, choose **1/h (1/m)**.
4. From the **Plot columns** list, choose **Manual**.
5. In the **Columns** list, select **Err^2 (1)**.
6. Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Diamond**.
7. From the **Positioning** list, choose **In data points**.
8. Click the **y-Axis Log Scale** button on the **Graphics** toolbar.
9. Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

Electrical Heating in a Busbar

For a description of this application, including detailed step-by-step instructions showing how to build it, see the book *Introduction to COMSOL Multiphysics*.

Application Library path: `COMSOL_Multiphysics/Multiphysics/busbar`

Convective Cooling of a Busbar

This is a template MPH-file that you can use as the starting point for extending the model Electrical Heating in a Busbar with a study of the convective cooling of the busbar by an air stream. For a description of that model, including detailed step-by-step instructions showing how to build it, see the book *Introduction to COMSOL Multiphysics*.

Application Library path: `COMSOL_Multiphysics/Multiphysics/busbar_box`

Parameterized Busbar Geometry

This is a template MPH-file containing the physics interfaces and the parameterized geometry for the model Electrical Heating in a Busbar. For a description of that model, including detailed step-by-step instructions showing how to build it, see the book *Introduction to COMSOL Multiphysics*.

Application Library path: `COMSOL_Multiphysics/Multiphysics/busbar_geom`

Parameterized Woven Carbon Fibers Geometry

This is a template MPH-file containing the geometry for the model Anisotropic Heat Transfer through Woven Carbon Fibers. For a description of that model, including detailed step-by-step instructions showing how to build it, see Anisotropic Heat Transfer through Woven Carbon Fibers.

Application Library path: `COMSOL_Multiphysics/Heat_Transfer/carbon_fibers_geom`

Conical Quantum Dot

This application computes the electronic states for a quantum-dot/wetting-layer system. It was inspired largely by the work of Dr. M. Willatzen and Dr. R. Melnik (Ref. 1) as well as B. Lassen.

Introduction

Quantum dots are nanoscale or microscale devices created by confining free electrons in a 3D semiconducting matrix. The tiny islands or droplets of confined "free electrons" (those with no potential energy) present many interesting electronic properties. They are of potential importance for applications in quantum computing, biological labeling, and lasers, to name only a few.

Scientists can create such structures experimentally using the Stranski-Krastanow molecular beam-epitaxy technique. In that way they obtain 3D confinement regions (the quantum dots) by growth of a thin layer of material (the wetting layer) onto a semiconducting matrix. Quantum dots can have many geometries including cylindrical, conical, or pyramidal. This application studies the electronic states of a conical InAs quantum dot grown on a GaAs substrate.

To compute the electronic states taken on by the quantum dot/wetting layer assembly embedded in the GaAs surrounding matrix, you must solve the 1-band Schrödinger equation in the effective mass approximation:

$$-\frac{h^2}{8\pi^2}\left(\nabla\cdot\left(\frac{1}{m_e(r)}\nabla\Psi(r)\right)\right)+V(r)\Psi(r)=E\Psi(r)$$

where h is Planck's constant, Ψ is the wave function, E is the eigenvalue (energy), and m_e is the effective electron mass (to account for screening effects).

Model Definition

The model works with the 1-particle stationary Schrödinger equation

$$-\nabla\cdot\left(\frac{h^2}{8\pi^2 m}\nabla\Psi\right)+V\Psi=E\Psi$$

It solves this eigenvalue problem for the quantum-dot/wetting-layer system using the following step potential barrier and effective-mass approximations:

- $V = 0$ for the InAs quantum dot/wetting layer and $V = 0.697$ eV for the GaAs substrate.
- $m_e = 0.023m$ for InAs and $m_e = 0.067m$ for GaAs.

Assume the quantum dot has perfect cylindrical symmetry. In that case you can model the overall structure in 2D as shown in the following figure.

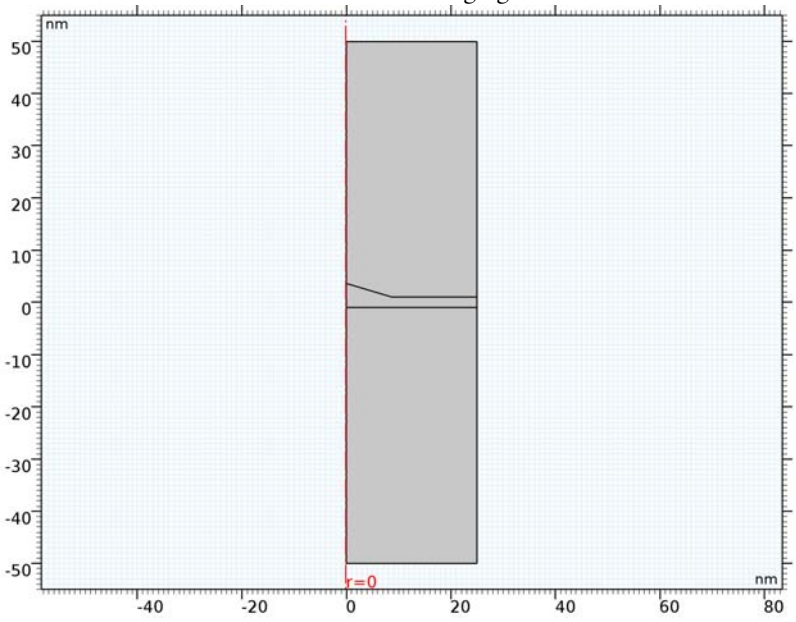


Figure 1: 2D geometry of a perfectly cylindrical quantum dot and wetting layer.

You can now separate the total wave function Ψ into

$$\Psi = \chi(z, r)\Theta(\varphi)$$

where φ is the azimuthal angle. Then rewrite the Schrödinger equation in cylindrical coordinates as

$$-\frac{h^2}{8\pi^2}\left[\frac{\partial}{\partial z}\left(\frac{1}{m_e}\frac{\partial \chi}{\partial z}\right) + \frac{1}{r}\frac{\partial}{\partial r}\left(\frac{r}{m_e}\frac{\partial \chi}{\partial r}\right)\right]\Theta - \frac{h^2}{8\pi^2}\frac{\chi}{m_e r^2}\frac{d^2\Theta}{d\varphi^2} + V\chi\Theta = E\chi\Theta$$

Dividing this equation by

$$\frac{\chi(z,r)}{m_e r^2}\Theta(\varphi)$$

and rearranging its terms lead to the two independent equations

$$\frac{1}{\Theta}\frac{d^2\Theta}{d\varphi^2} = -l^2 \tag{1}$$

and

$$-m_e r^2 \frac{h^2}{8\pi^2}\left[\frac{\partial}{\partial z}\left(\frac{1}{m_e}\frac{\partial\chi_l}{\partial z}\right)\frac{1}{\chi_l} + \frac{1}{r}\frac{\partial}{\partial r}\left(\frac{r}{m_e}\frac{\partial\chi_l}{\partial r}\right)\frac{1}{\chi_l}\right] + m_e r^2[V-E] = -\frac{h^2}{8\pi}l^2 \tag{2}$$

Equation 1 has obvious solutions of the form

$$\Theta = \exp[il\varphi]$$

where the periodicity condition $\Theta(\varphi + 2\pi) = \Theta(\varphi)$ implies that l, the principal quantum number, must be an integer. It remains to solve Equation 2, which you can rewrite as

$$-\frac{h^2}{8\pi^2}\left[\frac{\partial}{\partial z}\left(\frac{1}{m_e}\frac{\partial\chi_l}{\partial z}\right) + \frac{1}{r}\frac{\partial}{\partial r}\left(\frac{r}{m_e}\frac{\partial\chi_l}{\partial r}\right)\right] + \left(\frac{h^2}{8\pi^2 m_e}\frac{l^2}{r^2} + V\right)\chi_l = E_l\chi_l, \qquad l \in \mathbf{Z}$$

Note that this is an instance of a PDE on coefficient form,

$$\nabla\cdot(-c\nabla u - \alpha u + \gamma) + au + \beta\cdot\nabla u = d_a\lambda u$$

where the nonzero coefficients are

$$c = \frac{h^2}{8\pi^2 m_e}, \qquad a = \frac{h^2}{8\pi^2 m_e}\frac{l^2}{r^2} + V, \qquad \beta_r = -\frac{h^2}{8\pi^2 m_e}\frac{1}{r}, \qquad d_a = 1$$

and $\lambda = E_l$.

Results

This exercise models the eigenvalues for the four lowest electronic energy levels for the principal quantum number $l = 0$. The plots in Figure 2 show the eigenwave functions for those four states.

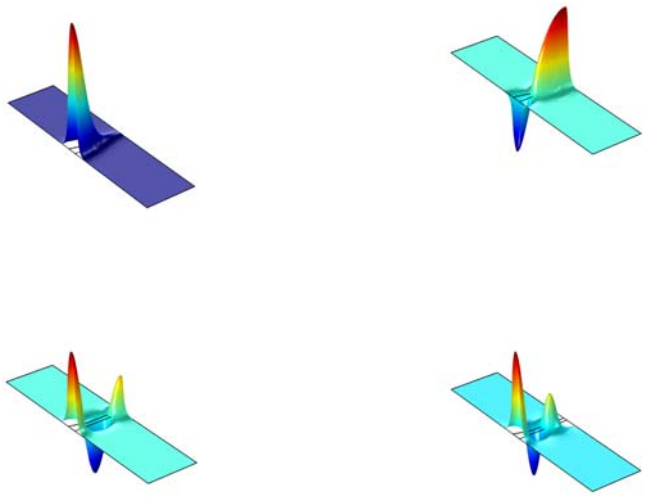


Figure 2: The four lowest electronic-energy levels for the case l = 0.

Notes About the COMSOL Implementation

To solve this problem, use the Coefficient Form PDE interface. The model solves for an eigenvalue/eigenfunction, for which you must input appropriate physical data and constants. Use electronvolts as the energy unit and nanometers as the length unit for the geometry.

Reference

1. R. Melnik and M. Willatzen, "Band structure of conical quantum dots with wetting layers," *Nanotechnology*, vol. 15, pp. 1–8, 2004.

Application Library path: `COMSOL_Multiphysics/Equation_Based/conical_quantum_dot`

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 **Mathematics>PDE Interfaces>Coefficient Form PDE (c)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Eigenvalue**.

6 Click **Done**.

GLOBAL DEFINITIONS

Define dimensionless parameters for the electron mass and the reduced Planck constant expressed in electronvolt units. You can obtain these values by dividing the SI-unit values of the corresponding predefined COMSOL Multiphysics constants, `me_const` and `hbar_const`, by the value of the elementary charge `e_const` in coulombs.

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
m	me_const[1/kg]/e_const[1/C]	5.6856E-12	Electron mass (eV/c^2)
hbar	hbar_const[1/(J*s)]/e_const[1/C]	6.582IE-16	Reduced Planck constant (eV*s)
V_In	0	0	Potential barrier, InAs (eV)

Name	Expression	Value	Description
V_Ga	0.697	0.697	Potential barrier, GaAs (eV)
c_In	hbar^2/(2*0.023*m)	1.6565E-18	c coefficient, InAs
c_Ga	hbar^2/(2*0.067*m)	5.6865E-19	c coefficient, GaAs
l	0	0	Principal quantum number

GEOMETRY 1

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

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **nm**.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type 25.

4 In the **Height** text field, type 100.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 In the **r** text field, type 12.5.

7 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type 25.

4 In the **Height** text field, type 2.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 In the **r** text field, type 12.5.

7 Right-click **Rectangle 2 (r2)** and choose **Build Selected**.

Polygon 1 (pol1)

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

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

3 In the **r** text field, type 0 12 0.

4 In the **z** text field, type 0 0 3.6.

5 Right-click **Polygon 1 (pol1)** and choose **Build Selected**.

Compose 1 (co1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Compose**.

2 Select the objects **pol1** and **r2** only.

3 In the **Settings** window for **Compose**, locate the **Compose** section.

4 In the **Set formula** text field, type `r2+pol1`.

5 Clear the **Keep interior boundaries** check box.

6 Right-click **Compose 1 (co1)** and choose **Build Selected**.

Compose 2 (co2)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Compose**.

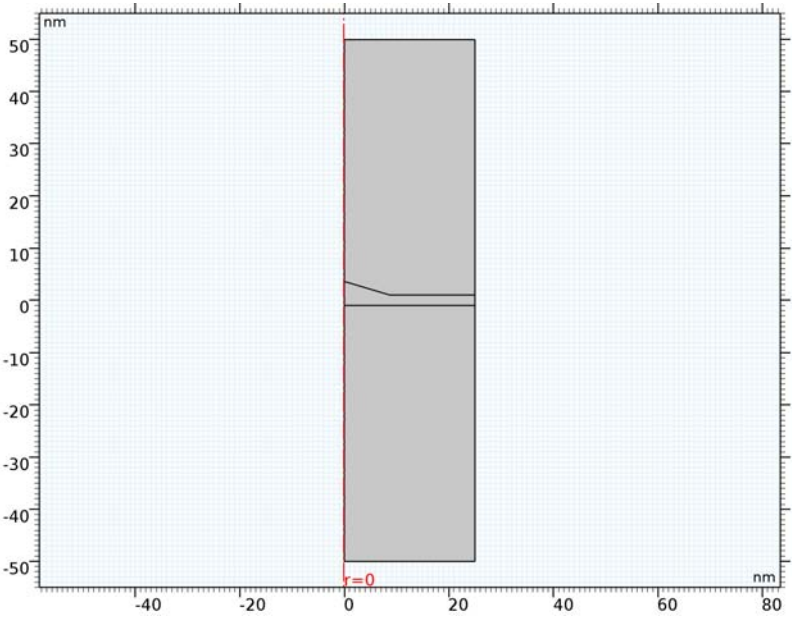
2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 In the **Settings** window for **Compose**, locate the **Compose** section.

4 In the **Set formula** text field, type `r1+co1`.

5 Right-click **Compose 2 (co2)** and choose **Build Selected**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.



COEFFICIENT FORM PDE (C)

Coefficient Form PDE 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Coefficient Form PDE (c)** node, then click **Coefficient Form PDE 1**.

2 In the **Settings** window for **Coefficient Form PDE**, locate the **Diffusion Coefficient** section.

3 In the *c* text field, type `c_In`.

4 Locate the **Absorption Coefficient** section. In the *a* text field, type `c_In*(1/r)^2+V_In`.

5 Click to expand the **Convection coefficient** section. Locate the **Convection Coefficient** section. Specify the β vector as

`-c_In/r`	`r`
`0`	`z`

Coefficient Form PDE 2

1 On the **Physics** toolbar, click **Domains** and choose **Coefficient Form PDE**.

2 Select Domains 1 and 3 only.

3 In the **Settings** window for **Coefficient Form PDE**, locate the **Diffusion Coefficient** section.

4 In the *c* text field, type `c_Ga`.

5 Locate the **Absorption Coefficient** section. In the *a* text field, type `c_Ga*(1/r)^2+V_Ga`.

6 Click to expand the **Convection coefficient** section. Locate the **Convection Coefficient** section. Specify the β vector as

`-c_Ga/r`	`r`
`0`	`z`

Dirichlet Boundary Condition 1

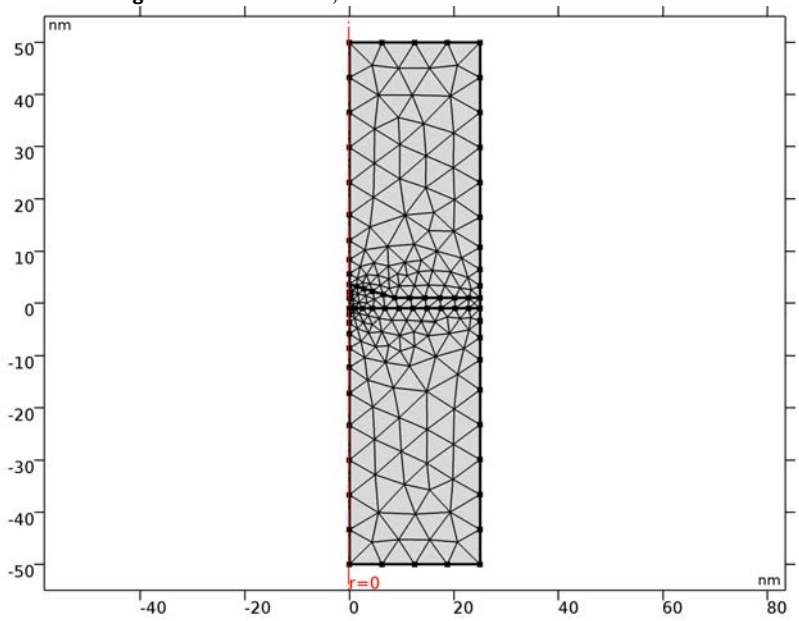
1 On the **Physics** toolbar, click **Boundaries** and choose **Dirichlet Boundary Condition**.

2 Select Boundaries 2 and 9 only.

MESH 1

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

2 In the **Settings** window for **Mesh**, click **Build All**.



STUDY 1

Step 1: Eigenvalue

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Eigenvalue**.

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

3 Select the **Desired number of eigenvalues** check box.

4 In the associated text field, type 4.

5 On the **Home** toolbar, click **Compute**.

RESULTS

2D Plot Group 1

Follow the instructions below to reproduce the series of plots in Figure 2.

Surface 1

In the **Model Builder** window, expand the **2D Plot Group 1** node.

Height Expression 1

1 Right-click **Surface 1** and choose **Height Expression**.

2 Click the **Zoom Extents** button on the **Graphics** toolbar.

Compare the result to the upper-left plot in Figure 2.

2D Plot Group 1

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

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

3 From the **Eigenvalue (rad/s)** list, choose **0.39332**.

4 On the **2D Plot Group 1** toolbar, click **Plot**.

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

Compare the result to the upper-right plot in Figure 2.

6 From the **Eigenvalue (rad/s)** list, choose **0.45893**.

7 On the **2D Plot Group 1** toolbar, click **Plot**.

8 Click the **Zoom Extents** button on the **Graphics** toolbar.

Compare the result to the lower-left plot in Figure 2.

9 From the **Eigenvalue (rad/s)** list, choose **0.56534**.

10 On the **2D Plot Group 1** toolbar, click **Plot**.

11 Click the **Zoom Extents** button on the **Graphics** toolbar.

Compare the result to the lower-right plot in Figure 2.

Flow Past a Cylinder

Introduction

The flow of fluid behind a blunt body such as an automobile is difficult to compute due to the unsteady flows. The wake behind such a body consists of unordered eddies of all sizes that create large drag on the body. In contrast, the turbulence in the thin boundary layers next to the streamlined bodies of aircraft and fish create only weak disturbances of flow.

An exception to this occurs when you place a slender body at right angles to a slow flow because the eddies organize. A von Kármán vortex street appears with a predictable frequency and involves the shedding of eddies from alternating sides. Everyday examples of this phenomenon include singing telephone wires and an automobile radio antenna vibrating in an air stream.

From an engineering standpoint, it is important to predict the frequency of vibrations at various fluid speeds and thereby avoid undesirable resonances between the vibrations of the solid structures and the vortex shedding. To help reduce such effects, plant engineers put a spiral on the upper part of high smokestacks; the resulting variation in shape prohibits the constructive interference of the vortex elements that the structure sheds from different positions.

Model Definition

To illustrate how you can study such effects, the following model examines unsteady, incompressible flow past a long cylinder placed in a channel at right angle to the oncoming fluid. With a symmetric inlet velocity profile, the flow needs some kind of asymmetry to trigger the vortex production. This can be achieved by placing the cylinder with a small offset from the center of the flow. In this case, an unstructured mesh is used, and the small asymmetry in the mesh proves to be enough to trigger the vortex production.

The simulation time necessary for a periodic flow pattern to appear is difficult to predict. A key predictor is the Reynolds number, which is based on cylinder diameter. For low values (below 100) the flow is steady. In this simulation, the Reynolds number equals 100, which gives a developed von Kármán vortex street, but the flow still is not fully turbulent.

The frequency and amplitude of oscillations are stable features, but flow details are extremely sensitive to perturbations. To gain an appreciation for this sensitivity, you can compare flow images taken at the same time but with such minor differences as are created by different tolerances for the time stepping. It is important to note that this sensitivity is a physical reality and not simply a numerical artifact.

Before calculating the time-varying forces on the cylinder, you can validate the method of computation at a lower Reynolds number using the direct nonlinear solver. This saves time because you can find and correct simple errors and mistakes before the final time-dependent simulation, which requires considerable time.

The viscous forces on the cylinder are proportional to the gradient of the velocity field at the cylinder surface. Evaluating the velocity gradient on the boundary by directly differentiating the FEM solution is possible but not very accurate. The differentiation produces 1st-order polynomials when second-order elements are used for the velocity field. A far better approach is to use a pair of reaction force operators to compute the integrals of the viscous forces, comparable to second-order accurate integrals of the viscous forces. An alternative approach would be to use a pair of weak-constraint variables to enforce the no slip condition. Preferably use the reaction force operator instead of weak constraints when computing integrals of reaction forces or fluxes in postprocessing.

The drag and lift forces themselves are not as interesting as the dimensionless drag and lift coefficients. These depend only on the Reynolds number and an object's shape, not its size. The coefficients are defined as

$$C_{\mathrm{D}} = \frac{2F_{\mathrm{D}}}{\rho U_{\mathrm{mean}}^2 A}$$

$$C_{\mathrm{L}} = \frac{2F_{\mathrm{L}}}{\rho U_{\mathrm{mean}}^2 A}$$

using the following parameters:

- F_{D} and F_{L} are the drag and lift forces
- ρ is the fluid's density
- U_{mean} is the mean velocity
- A is the projected area (product of thickness and diameter of cylinder)

Results and Discussion

Figure 1 shows the flow pattern resulting from the geometry.

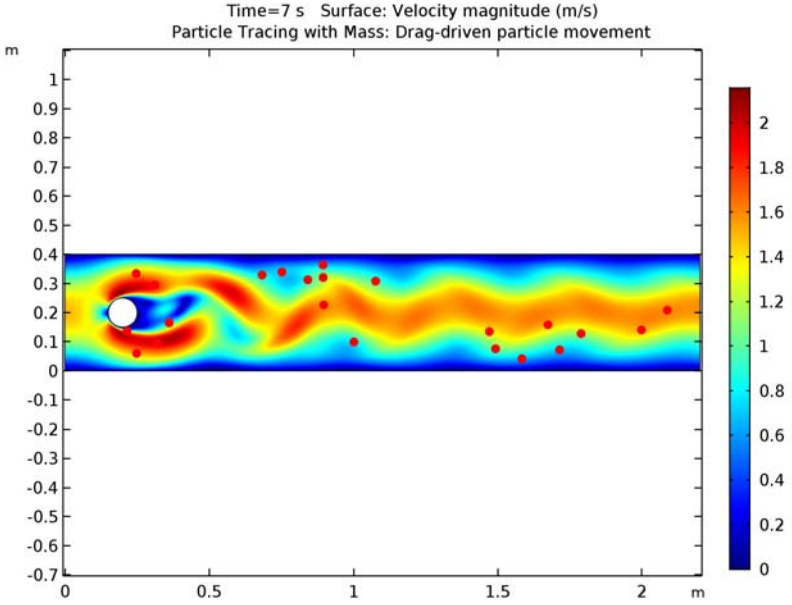


Figure 1: A plot of the last time step clearly shows the von Kármán path.

The flow around a cylinder is a common benchmark test for CFD algorithms. Various research teams have tried their strengths on this problem using different techniques. Results from some of these experiments have been collected by Schäfer and Turek (Ref. 1), who also used them to compute a probable value for the "real" answer.

Figure 2 shows how the lift coefficient develops a periodic variation as the von Kármán vortex structure is formed.

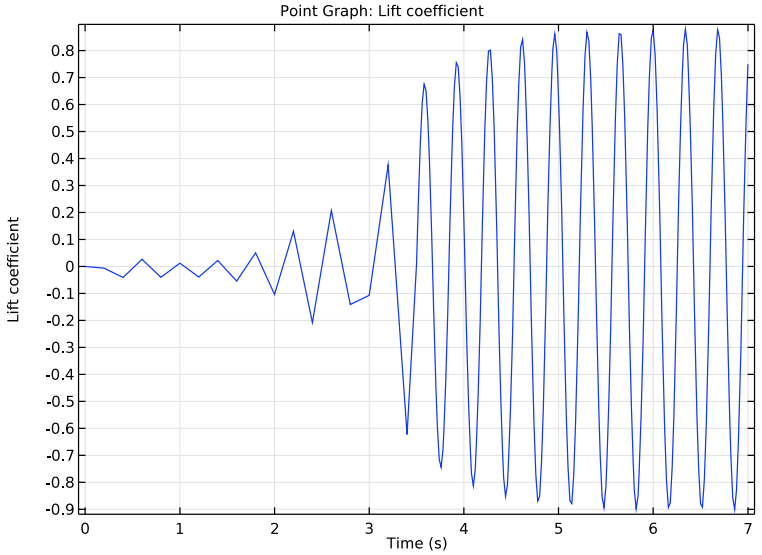


Figure 2: Lift coefficient, C_L, as a function of time.

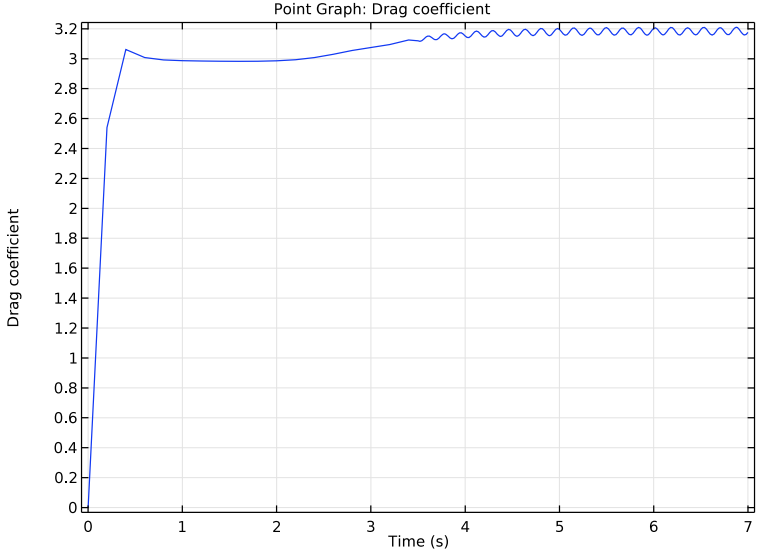


Figure 3: Drag coefficient, C_D, as a function of time.

Reference

1. M. Schäfer and S. Turek, "Benchmark Computations of Laminar Flow Around Cylinder," E.H. Hirschel ed., *Flow Simulation with High-Performance Computers II, Volume 52 of Notes on Numerical Fluid Mechanics*, Vieweg, pp. 547–566, 1996.

Application Library path: `COMSOL_Multiphysics/Fluid_Dynamics/cylinder_flow`

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>Single-Phase Flow>Laminar Flow (spf)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Time Dependent**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
U_mean	1[m/s]	1 m/s	Mean inflow velocity

Next, create a smoothed step function feature that you will use for ramping up the inflow velocity.

Step 1 (step1)

1 On the **Home** toolbar, click **Functions** and choose **Global>Step**.

2 In the **Settings** window for **Step**, locate the **Parameters** section.

3 In the **Location** text field, type `0.1`.

GEOMETRY 1

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `2.2`.

4 In the **Height** text field, type `0.4`.

5 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Position** section.

3 In the **x** text field, type `0.2`.

4 In the **y** text field, type `0.2`.

5 Locate the **Size and Shape** section. In the **Radius** text field, type `0.05`.

6 Right-click **Circle 1 (c1)** and choose **Build Selected**.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **r1** only.

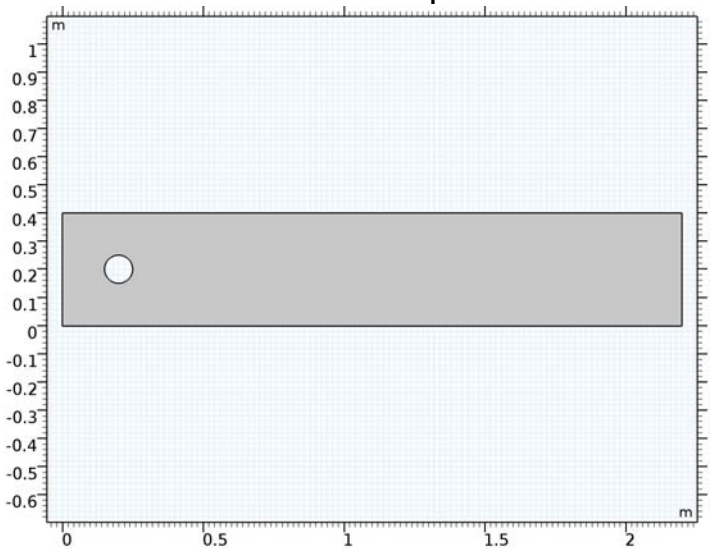
3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the object **c1** only.

6 On the **Geometry** toolbar, click **Build All**.

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



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
Density	rho	1	kg/m³	Basic
Dynamic viscosity	mu	1e-3	Pa·s	Basic

LAMINAR FLOW (SPF)

Inlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 1 only.

To define a parabolic velocity profile use the predefined local curve parameter `s`. Ramp up the velocity using the previously defined step function. Append the inverse unit bracket `[1/s]` to the time variable `t` because the step function expects a dimensionless argument.

3 In the **Settings** window for **Inlet**, locate the **Velocity** section.

4 In the U_0 text field, type `U_mean*6*s*(1-s)*step1(t[1/s])`.

Outlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 Select Boundary 4 only.

MESH 1

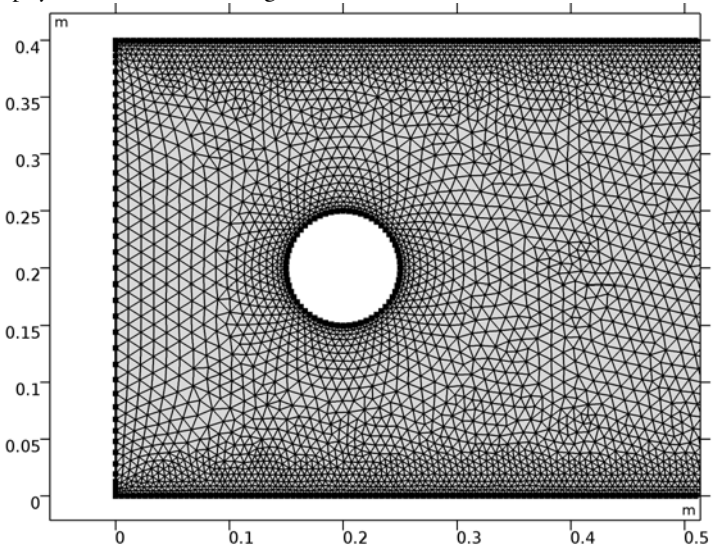
1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

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

3 From the **Element size** list, choose **Finer**.

4 Click **Build All**.

If you zoom in on the inlet and the cylinder you can see the boundary layers that the physics-controlled mesh gives.



STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.

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

3 In the **Times** text field, type `range(0,0.2,3.4) range(3.5,0.02,7)`.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

3 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

4 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time stepping** section.

5 Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Intermediate**.

6 On the **Study** toolbar, click **Compute**.

RESULTS

Velocity (spf)

Add a **Particle Tracing with Mass** node to the first default plot group to reproduce the plot in Figure 1.

1 In the **Model Builder** window, click **Velocity (spf)**.

Particle Tracing with Mass 1

1 On the **Velocity (spf)** toolbar, click **More Plots** and choose **Particle Tracing with Mass**.

2 In the **Settings** window for **Particle Tracing with Mass**, click to expand the **Mass and velocity** section.

3 Locate the **Mass and Velocity** section. In the **Mass** text field, type `4*pi/3*1e-9`.

4 Find the **Initial velocity** subsection. In the **x component** text field, type `u`.

5 In the **y component** text field, type `v`.

6 Locate the **Particle Positioning** section. In the **y** text field, type `range(0.1,0.05,0.3)`.

7 In the **x** text field, type `0`.

8 Click to expand the **Release** section. From the **Release particles** list, choose **At intervals**.

9 Select the **Start time** check box.

10 In the associated text field, type `3.6`.

11 In the **Time between releases** text field, type `0.4`.

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

13 Find the **Point style** subsection. From the **Type** list, choose **Point**.

14 Find the **Point motion** subsection. From the **When particle leaves domain** list, choose **Disappear**.

15 On the **Velocity (spf)** toolbar, click **Plot**.

To reproduce Figure 2 and Figure 3 of the lift and drag coefficients, first add an **Integral** data set for computing the total reaction force on the cylinder.

Selection

1 On the **Results** toolbar, click **More Data Sets** and choose **Evaluation>Integral**.

2 On the **Results** toolbar, click **Selection**.

3 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

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

5 Select Boundaries 5–8 only.

1D Plot Group 3

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

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

3 From the **Data set** list, choose **Integral 1**.

Point Graph 1

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

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

3 In the **Expression** text field, type `(-reacf(v)[N]*2/(spf.rho*U_mean^2*0.1[m^2]))[1/m]`.

4 Select the **Description** check box.

5 In the associated text field, type `Lift coefficient`.

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

Compare the graph with that shown in Figure 2.

1D Plot Group 4

Finally, visualize the drag coefficient using the following steps:

1 On 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 **Data set** list, choose **Integral 1**.

Point Graph 1

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

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

3 In the **Expression** text field, type `(-reacf(u)[N]*2/(spf.rho*U_mean^2*0.1[m^2]))[1/m]`.

4 Select the **Description** check box.

5 In the associated text field, type `Drag coefficient`.

6 On the **1D Plot Group 4** toolbar, click **Plot**.

Compare with Figure 3.

Diffraction Patterns

Introduction

This example simulates a two-slit interference experiment with water waves or sound. The model mimics the plane-wave excitation with two thin waveguides leading to slits in a screen and computes the diffraction pattern on the other side of the screen.

Model Definition

Theory predicts amplitude minima along rays where the difference in travel distance is an odd multiple of half the wavelength, and maxima at even multiples. For $n = 0, \pm 1, \pm 2, \ldots$:

$$\begin{cases} \text{min}, & \sin\theta = \left(n + \frac{1}{2}\right)\frac{\lambda}{D} \\ \text{max}, & \sin\theta = n\frac{\lambda}{D} \end{cases}$$

In this example, the distance D between the slits is 2λ. Maxima should then be present at $\theta = 0°$ and $30°$, while minima should appear at $\theta = 14.48°$ and $48.59°$.

Equation

For time-harmonic propagation, the wave equation turns into the Helmholtz equation:

$$-\nabla \cdot (\nabla u) - k^2 u = 0, \qquad k = \frac{2\pi}{\lambda}$$

Boundary Conditions

The absorbing boundary conditions have the form

$$\begin{aligned} \mathbf{n} \cdot (\nabla u) + iku &= 2ik, \quad \text{inflow} \\ \mathbf{n} \cdot (\nabla u) + iku &= 0, \quad \text{outflow} \end{aligned}$$

where $\mathbf{n}$ is the outward boundary normal vector. This follows directly from the assumption that the total wave consists of an incident plane wave plus an outgoing plane wave, both at normal incidence:

$$u = u_1 e^{ikx} + u_2 e^{-ikx}$$

where the first term represents the outgoing wave and the second term the incident wave. For example, at the left boundary of a computational domain along the x axis, the normal derivative becomes

$$\frac{\partial u}{\partial n} = -\frac{\partial u}{\partial x} = -iku_1 e^{ikx} + iku_2 e^{-ikx} = -ik(u - 2u_2 e^{-ikx})$$

which means that

$$\frac{\partial u}{\partial n} + iku = 2iku_2$$

and for $u_2 = 1$, you get the absorbing boundary condition at the inflow.

Results

The plot in Figure 1 shows the diffraction pattern clearly. The effect of quantization is that the numerical wavelength differs from λ, which results in a shift of the angles. You can correct for this effect by adjusting the value of k in the Helmholtz equation to the element size. These practices are important for modeling the interference effects of monochromatic waves.

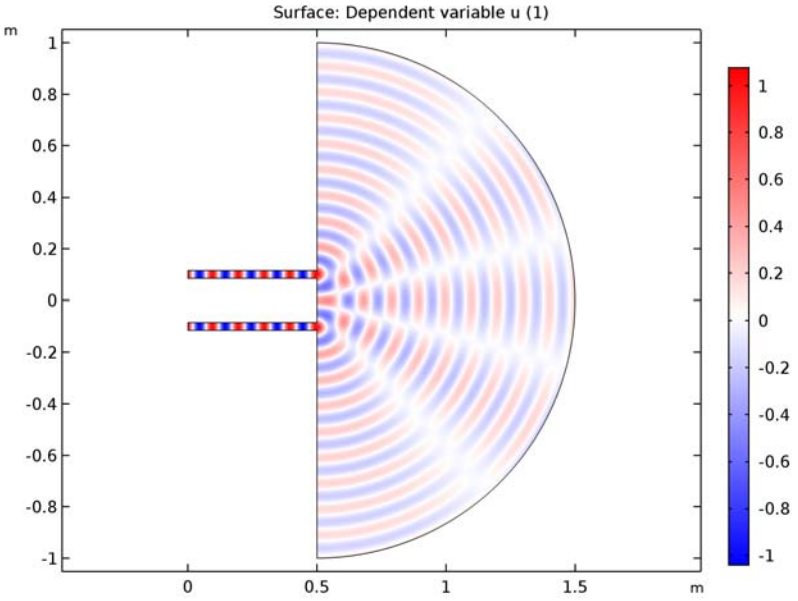


Figure 1: Diffraction pattern in the simulated two-slit experiment.

Application Library path: COMSOL_Multiphysics/Equation_Based/diffraction_patterns

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 **Mathematics>PDE Interfaces>Coefficient Form PDE (c)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
l	0.1[m]	0.1 m	Wavelength
k	2*pi[rad]/l	62.832 rad/m	Wave number

GEOMETRY 1

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Sector angle** text field, type 180.

4 Locate the **Position** section. In the **x** text field, type `0.5`.

5 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.

6 Right-click **Circle 1 (c1)** and choose **Build Selected**.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `0.5`.

4 In the **Height** text field, type `0.03`.

5 Locate the **Position** section. In the **y** text field, type `-0.015-0.1`.

6 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

Copy 1 (copy1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **r1** only.

3 In the **Settings** window for **Copy**, locate the **Displacement** section.

4 In the **y** text field, type `0.2`.

5 Right-click **Copy 1 (copy1)** and choose **Build Selected**.

Union 1 (uni1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

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

3 In the **Settings** window for **Union**, locate the **Union** section.

4 Clear the **Keep interior boundaries** check box.

5 On the **Geometry** toolbar, click **Build All**.

The model geometry is now complete. Next, turn to the physics settings.

COEFFICIENT FORM PDE (C)

Coefficient Form PDE 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Coefficient Form PDE (c)** node, then click **Coefficient Form PDE 1**.

2 In the **Settings** window for **Coefficient Form PDE**, locate the **Absorption Coefficient** section.

3 In the a text field, type `-k^2`.

4 Locate the **Source Term** section. In the f text field, type `0`.

Flux/Source 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Flux/Source**.

2 Select Boundaries 1 and 4 only.

3 In the **Settings** window for **Flux/Source**, locate the **Boundary Flux/Source** section.

4 In the *g* text field, type `2*i*k-i*k*u`.

Flux/Source 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Flux/Source**.

2 Select Boundaries 11 and 12 only.

3 In the **Settings** window for **Flux/Source**, locate the **Boundary Flux/Source** section.

4 In the *g* text field, type `-i*k*u`.

MESH 1

Create a mesh with a maximum element size determined by the wavelength. As a rule of thumb, you need 5 elements per wavelength for quadratic elements (the default for the PDE interface) to fully resolve the wave.

Size

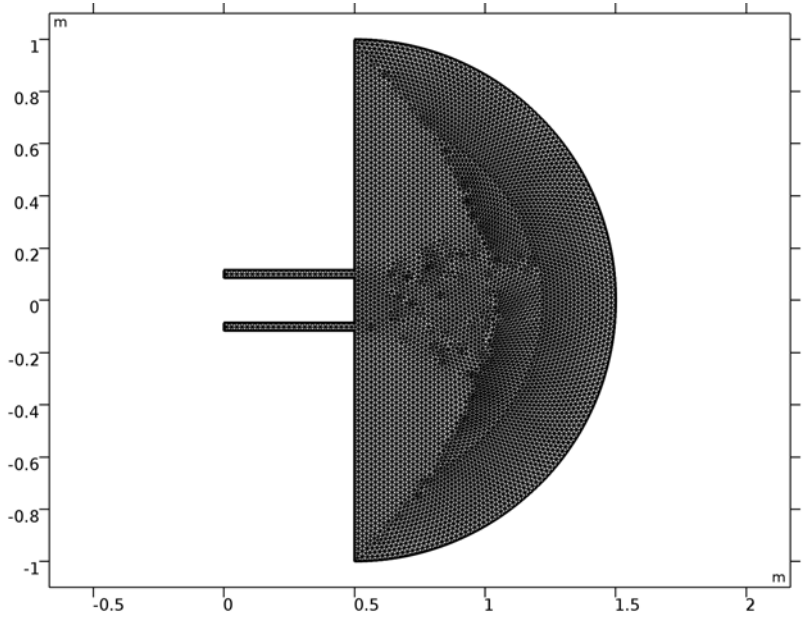
1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `1/5`.

5 Click **Build All**.



STUDY 1

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

RESULTS

2D Plot Group 1

To reproduce the plot shown in Figure 1, just change the color table.

Surface 1

1 In the **Model Builder** window, expand the **2D Plot Group 1** node, then click **Surface 1**.

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

3 From the **Color table** list, choose **WaveLight**.

4 On the **2D Plot Group 1** toolbar, click **Plot**.

Effective Diffusivity in Porous Materials

This example introduces the concept of effective diffusivity in porous media by comparing the transport through an artificial porous structure described in a detailed model with a simplified homogeneous porous media approach using effective transport properties.

The exercise consists of two parts. The first part describes how to create the model with a detailed geometry. The second part shows how to define a homogeneous model for porous media using an effective diffusivity calculated using the detailed model from the first part.

Introduction

Transport through porous structures is usually treated using simplified homogeneous models with effective transport properties. This is in most cases a necessity, since the typical dimensions of the pores and particles making up the porous structure are several orders of magnitude smaller than the size of the domain that is to be modeled.

However, it might be interesting to investigate the assumptions and simplifications done when homogenizing a porous structure by comparing a homogeneous model with a model defined using the detailed structure.

The artificial porous structure used in this example is shown in Figure 1 below.

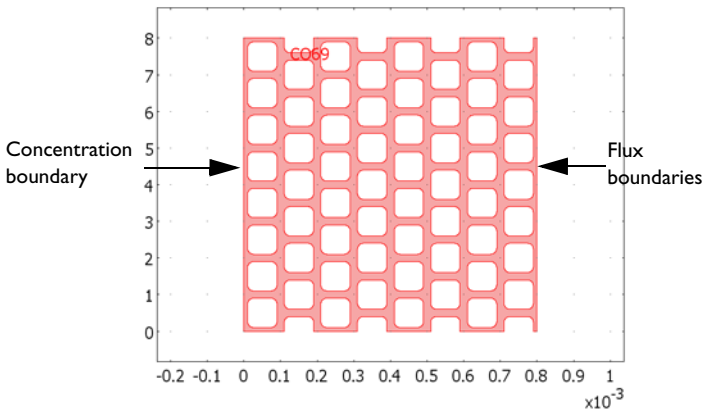


Figure 1: Artificial porous structure. The domain colored in red is accessible for diffusion.

Model Definition

The model equation in the modeled domain shown in Figure 1 is the time-dependent equation

$$\frac{\partial c}{\partial t} + \nabla \cdot (-D\nabla c) = 0$$

where c denotes concentration (mol/m^3 using SI units) and D the diffusion coefficient (m^2/s) of the solute.

The boundary conditions are of three different types. A concentration boundary condition applies at the left vertical boundary in Figure 1. It is expressed as

$$c = c_0$$

where c_0 is a given concentration.

The right vertical boundary in Figure 1 is set according to

$$(-D\nabla c) \cdot \mathbf{n} = k_m(c - c_1)$$

where k_m is the mass transfer coefficient (m/s), and c_1 is the concentration in a bulk solution outside of the porous structure.

All other boundaries are insulating boundaries according to

$$(-D\nabla c) \cdot \mathbf{n} = 0$$

The initial condition is given by a bell-shaped profile along the x-axis with its maximum at $x = 0$ and a corresponding value of $c = c_0$:

$$c(t_0) = c_0 \exp(-ax^2)$$

Assume a gaseous solution with a solute content of 3 mol/m^3 at the concentration boundary. The diffusion coefficient is set to $1{\cdot}10^{-5}$ m^2/s.

The second part of this exercise uses a homogenized 1D model geometry with effective transport properties and an average porosity. The model equation then becomes:

$$\varepsilon\frac{\partial c}{\partial t} + \nabla \cdot (-D^{\text{eff}}\nabla c) = 0$$

where ε denotes the average porosity and D^{eff} is the effective diffusivity. These properties are calculated from the results of the detailed structure; see the next section. At the boundaries, the concentration and flux conditions described above apply.

The simulations are run for t = 0 to 0.1 s, when the simulation reaches steady state. Figure 2 below shows the concentration profile after 0.05 s in the porous structure. Already at this stage the concentration has almost reached steady state, which is visible by the nearly linear concentration profile across the structure.

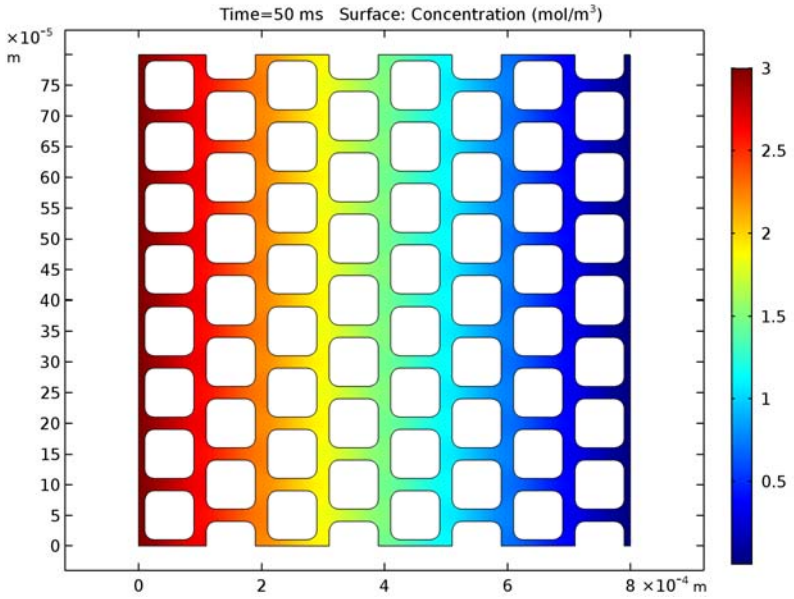


Figure 2: Concentration profile in the modeled artificial porous structure at t = 0.05 s.

When modeling porous media, the exact concentration in the pore structure is not the most important issue because the description of the structure is homogenized and not detailed as in Figure 2. The most interesting issue is then the description of the flux. To calculate the average flux, integrate over the flux boundary and divide by its length, L_0, which yields the following expression:

$$N_{\text{average}} = \frac{1}{L_0}\int_0^{L_0} k_m(c - c_1)\,dS$$

Figure 3 shows the value of this integral as a function of time. If you let the process reach steady-state, the average flux becomes $8.051 \cdot 10^{-3}$ mol/(m^2/s). Considering the almost linear profile across the structure, it is natural to replace the porous structure with a 1D

homogenized structure along the x-axis. It is then possible to calculate the effective diffusivity according to the following:

$$D^{\text{eff}}\frac{(c_0 - c_{\text{out}})}{L_1} = N_{\text{average}}$$

where c_{out} is the average concentration (mol/m^3) at the flux boundary, and L_1 is the length of the geometry along the x-axis. The average concentration is obtained by integrating according to the expression below:

$$c_{\text{out}} = \frac{1}{L_0}\int_0^{L_0} c\,dy$$

This gives an average concentration $c_{\text{out}} = 1.63 \cdot 10^{-3}$ mol/m^3. Using $L_1 = 8 \cdot 10^{-4}$ m, the effective diffusivity is:

$$D^{\text{eff}} = \frac{8.051 \cdot 10^{-3} \times 8.0 \cdot 10^{-4}}{(3 - 1.63 \cdot 10^{-3})}$$

which yields a value for the effective diffusivity of $2.15 \cdot 10^{-6}$ m^2/s compared to the "free" diffusivity of $1 \cdot 10^{-5}$ m^2/s. The effective and "free" diffusivities are usually related according to the equation

$$D^{\text{eff}} = D\frac{\varepsilon}{\tau}$$

where ε is the porosity of the structure and τ is the tortuosity, which is a measure of the actual length per unit effective length a molecule has to diffuse in a porous structure. To calculate the porosity of the modeled structure, you integrate the value 1 over the structure and then divide this by the length and width of the structure:

$$\varepsilon = \frac{1}{L_0 L_1}\int_0^{L_1}\int_0^{L_0} 1\,dx\,dy$$

resulting in a value of 0.383. The value of τ can then be calculated to 1.62. In addition, the tortuosity is usually expressed as a power of the porosity, resulting in an expression for the effective diffusivity according to

$$D^{\text{eff}} = D\varepsilon^p$$

If you use the calculated values for porosity and effective diffusivity, the value for p is 1.60. The experimental values for p for porous structures used for transport in catalysts, soils, and other porous structures is usually in the range 1.5–2.

Using the value of the effective diffusivity, a simple homogenized 1D model provides the possibility to compare the value of the flux using a homogenized model to the value using the detailed 2D structure. Figure 3 shows that there is an excellent agreement between the model using a detailed geometry and the homogenized model. The difference in the time-dependent flux is hardly visible between the two cases in the graph.

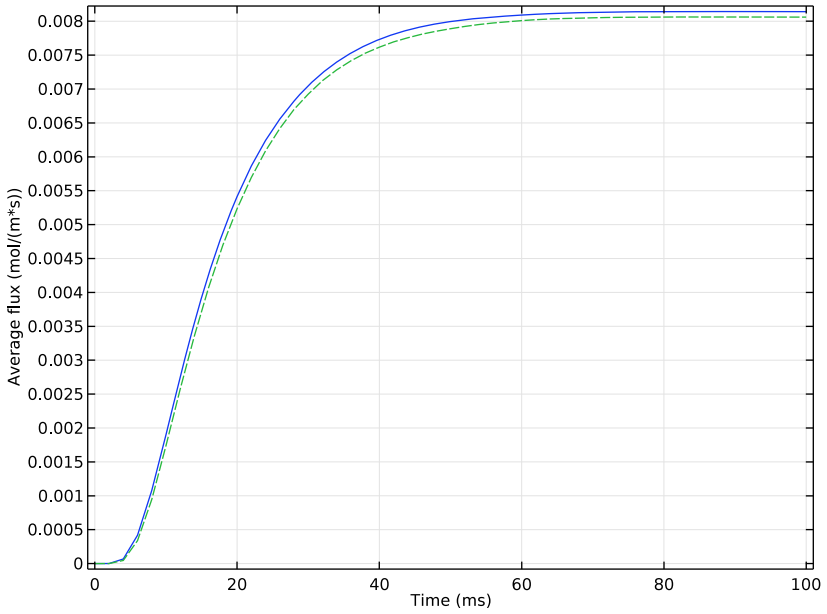


Figure 3: Average flux at the flux boundary in the detailed 2D model (solid blue line) and the 1D homogenized approximation (dashed green line).

Notes About the COMSOL Implementation

Both models described above are straightforward to define in COMSOL Multiphysics. One feature that is of great use in this example is the ability to define integration coupling operators to generate the values of the integrals needed to evaluate the results. The definition of these integrals is described in detail in the step-by-step instructions below.

Application Library path: `COMSOL_Multiphysics/Diffusion/effective_diffusivity`

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 **Chemical Species Transport> Transport of Diluted Species (tds)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Time Dependent**.
6. Click **Done**.

GEOMETRY 1

Import 1 (imp1)

1. On the **Home** toolbar, click **Import**.
2. In the **Settings** window for **Import**, locate the **Import** section.
3. From the **Source** list, choose **COMSOL Multiphysics file**.
4. Click **Browse**.
5. Browse to the model's Application Libraries folder and double-click the file `effective_diffusivity.mphbin`.
6. Click **Import**.

GLOBAL DEFINITIONS

Parameters

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
D2	1e-5[m^2/s]	1E-5 m²/s	Diffusion coefficient
c_max	3[mol/m^3]	3 mol/m³	Peak initial concentration
k_f	5[m/s]	5 m/s	Mass transfer coefficient
a	1000	1000	Dimensionless constant

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Global Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
c0	c_max*exp(a*(-(x/0.4[mm])^2))	mol/m³	Initial concentration

DEFINITIONS

Next, define selections that will be useful when defining the boundary conditions and an integration coupling operator, and also during postprocessing.

Explicit 1

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

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

3 In the **Rename Explicit** dialog box, type `Left boundary` in the **New label** text field.

4 Click **OK**.

5 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

7 Select Boundary 1 only.

Explicit 2

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

2 In the **Model Builder** window, right-click **Explicit 2** and choose **Rename**.

3 In the **Rename Explicit** dialog box, type `Right boundary` in the **New label** text field.

4 Click **OK**.

5 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

7 Select Boundary 276 only.

Explicit 3

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

2 In the **Model Builder** window, right-click **Explicit 3** and choose **Rename**.

3 In the **Rename Explicit** dialog box, type `Top-right vertex` in the **New label** text field.

4 Click **OK**.

5 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

7 Select Point 532 only.

TRANSPORT OF DILUTED SPECIES (TDS)

1 In the **Settings** window for **Transport of Diluted Species**, locate the **Transport Mechanisms** section.

2 Clear the **Convection** check box.

This setting gives a pure diffusion interface.

Transport Properties 1

1 In the **Model Builder** window, expand the **Transport of Diluted Species (tds)** node, then click **Transport Properties 1**.

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

3 In the D_c text field, type `D2`.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>** **Transport of Diluted Species (tds)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the c text field, type `c0`.

Concentration 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Concentration**.

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

3 From the **Selection** list, choose **Left boundary**.

4 Locate the **Concentration** section. Select the **Species c** check box.

5 In the $c_{0,c}$ text field, type `c_max`.

Flux 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Flux**.

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

3 From the **Selection** list, choose **Right boundary**.

4 Locate the **Inward Flux** section. From the **Flux type** list, choose **External convection**.

5 Select the **Species c** check box.

6 In the $k_{c,\,c}$ text field, type `k_f`.

DEFINITIONS

Proceed to define a variable for the average flux through the porous structure. Begin by defining an **Average** coupling operator on the rightmost boundary.

Average 1 (aveop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.

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

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

4 From the **Selection** list, choose **Right boundary**.

Variables 2

1 On the **Definitions** toolbar, click **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
`flux_avg`	`aveop1(k_f*c)`	mol/(m²·s)	`Average flux`

MESH 1

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

2 In the **Settings** window for **Mesh**, click **Build All**.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.

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

3 From the **Time unit** list, choose **ms**.

4 In the **Times** text field, type `range(0,2,100)`.

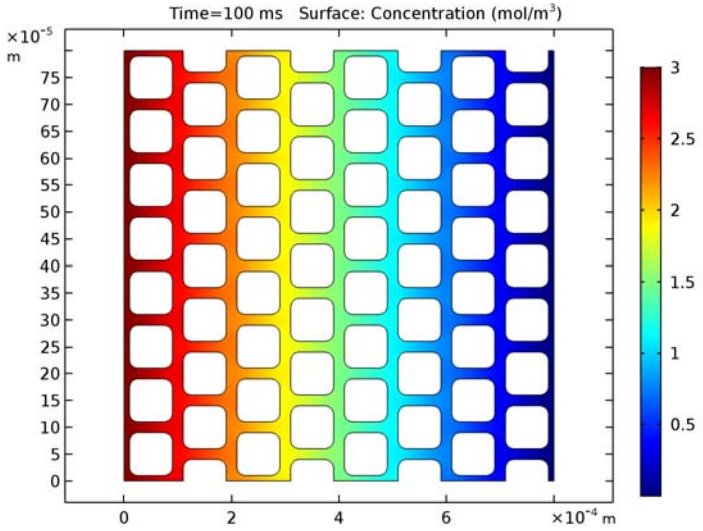
5 On the **Home** toolbar, click **Compute**.

RESULTS

Concentration (tds)

1 Click the **Zoom Extents** button on the **Graphics** toolbar.

The default plot shows concentration at the end time, that is, 0.1 seconds.



You can plot the concentration at different time steps.

2 In the **Model Builder** window, click **Concentration (tds)**.

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

4 From the **Time (ms)** list, choose **50**.

5 On the **Concentration (tds)** toolbar, click **Plot**.

Compare this plot to the one shown in Figure 2.

1D Plot Group 2

Now, plot the average flux.

Point Graph 1

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

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

3 In the **Settings** window for **Point Graph**, locate the **Selection** section.

4 From the **Selection** list, choose **Top-right vertex**.

5 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Definitions>Variables>flux_avg - Average flux**.

1D Plot Group 2

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

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

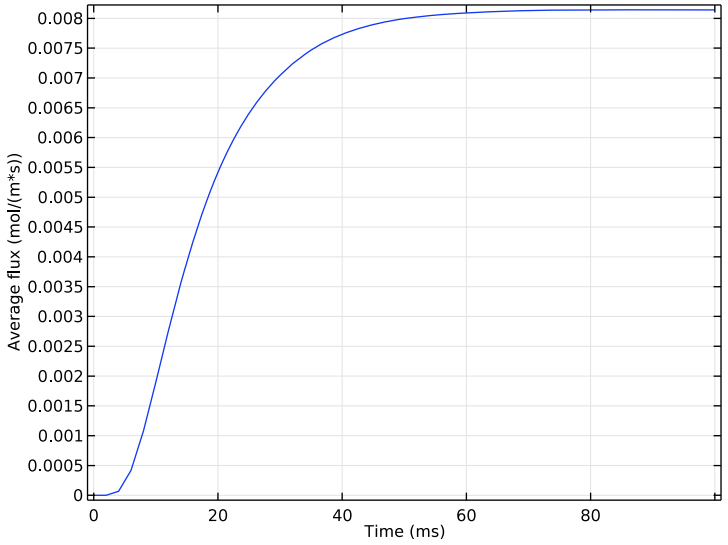
3 In the **Label** text field, type `Molar fluxes`.

4 Locate the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Plot Settings** section. Select the **y-axis label** check box.

6 In the associated text field, type `Average flux (mol/(m*s))`.

7 On the **Molar fluxes** toolbar, click **Plot**.



To get the porosity of the domain for the 1D model, perform a surface integration.

Surface Integration 1

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

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

3 From the **Selection** list, choose **All domains**.

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

Expression	Unit
1/(0.8[mm])^2	1

The denominator in this expression represents the product of the length and the width of the 2D model structure.

5 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The evaluated value of the integral should be close to 0.383.

ROOT

Now turn to the 1D model.

On the **Home** toolbar, click **Component** and choose **Add Component>1D**.

GEOMETRY 2

In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

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

3 In the tree, select **Chemical Species Transport>Transport of Diluted Species (tds)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.

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

6 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ADD STUDY

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

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

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Transport of Diluted Species (tds)** interface.

4 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies> Time Dependent**.

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

6 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

GEOMETRY 2

Interval 1 (i1)

1 On the **Geometry** toolbar, click **Interval**.

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

3 In the **Right endpoint** text field, type `8e-4`.

4 Right-click **Interval 1 (i1)** and choose **Build Selected**.

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

GLOBAL DEFINITIONS

Parameters

Add the following parameters to those you already defined.

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
`epsilon`	`0.383`	0.383	`Porosity`
`D1`	`2.15e-6[m^2/s]`	2.15E-6 m²/s	`Diffusion coefficient, 1D`

TRANSPORT OF DILUTED SPECIES 2 (TDS2)

On the **Physics** toolbar, click **Transport of Diluted Species (tds)** and choose **Transport of Diluted Species 2 (tds2)**.

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Transport of Diluted Species 2 (tds2)**.

2 In the **Settings** window for **Transport of Diluted Species**, locate the **Transport Mechanisms** section.

3 Clear the **Convection** check box.

Transport Properties 1

1 In the **Model Builder** window, expand the **Transport of Diluted Species 2 (tds2)** node, then click **Transport Properties 1**.

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

3 In the D_{c2} text field, type `D1/epsilon`.

Initial Values 1

1 In the **Model Builder** window, under **Component 2 (comp2)> Transport of Diluted Species 2 (tds2)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the $c2$ text field, type `c0`.

Concentration 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Concentration**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Concentration**, locate the **Concentration** section.

4 Select the **Species c2** check box.

5 In the $c_{0,c2}$ text field, type `c_max`.

Flux 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Flux**.

2 Select Boundary 2 only.

3 In the **Settings** window for **Flux**, locate the **Inward Flux** section.

4 From the **Flux type** list, choose **External convection**.

5 Select the **Species c2** check box.

6 In the $k_{c,\,c2}$ text field, type `k_f/epsilon`.

DEFINITIONS

Create a variable for the flux in the homogenized 1D model.

Variables 3

1 On the **Home** toolbar, click **Variables** and choose **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
`flux_hom`	`k_f*c2`	mol/(m²·s)	`Flux, 1D model`

MESH 2

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Mesh 2**.

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

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

4 Click **Build All**.

STUDY 2

Step 1: Time Dependent

1 In the **Model Builder** window, expand the **Study 2** node, then click **Step 1: Time Dependent**.

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

3 From the **Time unit** list, choose **ms**.

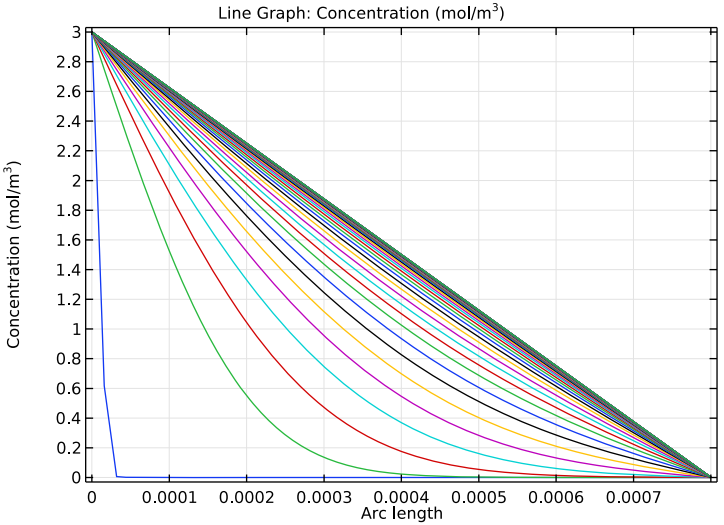
4 In the **Times** text field, type `range(0,2,100)`.

5 On the **Home** toolbar, click **Compute**.

RESULTS

Concentration (tds2)

The default plot for the 1D model shows the concentration for all time steps.



Finally, plot the result for the flux at the flux boundary in the homogenized 1D model in the same plot as the 2D result for comparison.

Point Graph 2

1. In the **Model Builder** window, under **Results** right-click **Molar fluxes** and choose **Point Graph**.
2. In the **Settings** window for **Point Graph**, locate the **Data** section.
3. From the **Data set** list, choose **Study 2/Solution 2 (3) (sol2)**.
4. Select Boundary 2 only.
5. Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 2>Definitions>Variables>flux_hom - Flux, 1D model**.
6. Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.
7. On the **Molar fluxes** toolbar, click **Plot**.

   Compare the result with that shown in Figure 3.

Appendix — Geometry Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Blank Model**.

ROOT

On the **Home** toolbar, click **Add Component** and choose **2D**.

GEOMETRY 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2. In the **Settings** window for **Geometry**, locate the **Units** section.
3. From the **Length unit** list, choose **mm**.

Rectangle 1 (r1)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `0.1`.
4. In the **Height** text field, type `0.8`.

Square 1 (sq1)

1. On the **Geometry** toolbar, click **Primitives** and choose **Square**.
2. In the **Settings** window for **Square**, locate the **Size** section.

3 In the **Side length** text field, type `0.08`.

4 Locate the **Position** section. In the **x** text field, type `0.01`.

5 In the **y** text field, type `0.01`.

6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

7 From the **Show in physics** list, choose **Off**.

Fillet 1 (fil1)

1 On the **Geometry** toolbar, click **Fillet**.

2 In the **Settings** window for **Fillet**, locate the **Points** section.

3 From the **Vertices to fillet** list, choose **Square 1**.

4 Locate the **Radius** section. In the **Radius** text field, type `0.016`.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `0.1`.

4 In the **Height** text field, type `0.9`.

5 Locate the **Position** section. In the **x** text field, type `0.1`.

6 Right-click **Rectangle 2 (r2)** and choose **Build Selected**.

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

Square 2 (sq2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Square**.

2 In the **Settings** window for **Square**, locate the **Size** section.

3 In the **Side length** text field, type `0.08`.

4 Locate the **Position** section. In the **x** text field, type `0.11`.

5 In the **y** text field, type `0.01`.

6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

7 From the **Show in physics** list, choose **Off**.

Fillet 2 (fil2)

1 On the **Geometry** toolbar, click **Fillet**.

2 In the **Settings** window for **Fillet**, locate the **Points** section.

3 From the **Vertices to fillet** list, choose **Square 2**.

4 Locate the **Radius** section. In the **Radius** text field, type `0.016`.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

6 From the **Show in physics** list, choose **Off**.

Array 1 (arr1)

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

2 Select the object **fil1** only.

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

4 In the **y size** text field, type `8`.

5 Locate the **Displacement** section. In the **y** text field, type `0.1`.

6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

7 From the **Show in physics** list, choose **Off**.

Array 2 (arr2)

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

2 In the **Settings** window for **Array**, locate the **Input** section.

3 From the **Input objects** list, choose **Fillet 2**.

4 Locate the **Size** section. In the **y size** text field, type `9`.

5 Locate the **Displacement** section. In the **y** text field, type `0.1`.

6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

7 From the **Show in physics** list, choose **Off**.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **r1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 From the **Objects to subtract** list, choose **Array 1**.

Difference 2 (dif2)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **r2** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 From the **Objects to subtract** list, choose **Array 2**.

5 Right-click **Difference 2 (dif2)** and choose **Build Selected**.

Move 1 (mov1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Move**.

2 Select the object **dif2** only.

3 In the **Settings** window for **Move**, locate the **Displacement** section.

4 In the **y** text field, type `-0.05`.

Array 3 (arr3)

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

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

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

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

5 Locate the **Displacement** section. In the **x** text field, type `0.2`.

6 Right-click **Array 3 (arr3)** and choose **Build Selected**.

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

Union 1 (uni1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

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

3 Clear the **Keep interior boundaries** check box.

4 Click the **Select All** button on the **Graphics** toolbar.

Partition Domains 1 (pard1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Domains**.

2 On the object **uni1**, select Domain 1 only.

3 In the **Settings** window for **Partition Domains**, locate the **Partition Domains** section.

4 From the **Partition with** list, choose **Extended edges**.

5 On the object **uni1**, select Boundaries 2 and 3 only.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 Click the **Select All** button on the **Graphics** toolbar.

5 On the object **pard1**, select Domains 2–9 only.

Form Union (fin)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

Explicit Selection 1 (sel1)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Top right vertex` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Point**.

4 On the object **fin**, select Point 546 only.

5 Right-click **Top right vertex** and choose **Build Selected**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

Explicit Selection 2 (sel2)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Left boundary` in the **Label** text field.

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

4 On the object **fin**, select Boundary 1 only.

Explicit Selection 3 (sel3)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Right boundary` in the **Label** text field.

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

4 On the object **fin**, select Boundary 290 only.

5 Right-click **Right boundary** and choose **Build Selected**.

Eigenmodes of a Room

Introduction

Resonance can at times be a problem in everyday life. The low bass notes from the music system or home theater in the living room can shake the windows and make the floor vibrate. This happens only for certain frequencies — the eigenfrequencies of the room.

It is only in the low-frequency range that the eigenfrequencies are well separated. In the mid- and high-frequency ranges, the eigenfrequencies are packed so closely, with less than a halftone between them, that the individual resonances are insignificant for music and other natural sounds. Nevertheless, the music experience is affected by the acoustics of the room.

When designing a concert hall, it is extremely important to take the resonances into account. For a clear and neutral sound, the eigenfrequencies should be evenly spaced. For the home theater or music system owner, who cannot change the shape of the living room, another question is more relevant: Where should the speakers be located for the best sound?

Model Definition

For example, take a room with the dimensions 5 by 4 by 2.6 meters equipped with a flat-screen TV, a sideboard, two speakers, and a couch. To illustrate the effects on the music, compute a few resonance frequencies in the vicinity of 90 Hz together with the corresponding eigenmodes. The eigenmode shows the sound intensity pattern for its associated eigenfrequency. From the characteristics of the eigenmodes, you can draw some conclusions as to where the speakers should be placed.

DOMAIN EQUATIONS

Sound propagating in free air is described by the wave equation:

$$-\Delta p + \frac{1}{c^2}\frac{\partial^2 p}{\partial t^2} = 0$$

where p is the pressure, and c is the speed of sound. If the air is brought into motion by a harmonically oscillating source, for example, a loudspeaker, only one frequency f exists in the room. For that reason it makes sense to look for a time-harmonic solution on the form

$$p = \hat{p} e^{i\omega t}$$

The wave equation then simplifies to the Helmholtz equation for p, the amplitude of the acoustic disturbances:

$$\Delta\hat{p} + \frac{\omega^2}{c^2}\hat{p} = 0$$

BOUNDARY CONDITIONS

This model assumes that all boundaries — walls, floor, ceiling, and furniture — are perfectly rigid (sound hard boundaries). This means that it returns no information of the damping properties of the room, but the distribution of the pressure should still be reasonably correct.

ANALYTIC COMPARISON

It is possible to solve the simpler case of an empty room analytically. Each eigenfrequency corresponds to an integer triple (i, l, m):

$$f_{i,l,m} = \frac{c}{2}\sqrt{\left(\frac{i}{L_x}\right)^2 + \left(\frac{l}{L_y}\right)^2 + \left(\frac{m}{L_z}\right)^2}$$

The eigenmodes can be divided into three distinct classes:

- Eigenfrequencies with only one index different from zero give rise to axial modes, that is, plane standing waves between two opposite walls.
- If one index is zero, the mode is tangential.
- If all indices are different from zero, the mode is oblique.

Theoretical resonance frequencies below 100 Hz for a room without furniture are found in the following table.

MODE INDEX	FREQUENCY	MODE INDEX	FREQUENCY
0,0,0	0	0,1,1	78.7
1,0,0	34.3	2,1,0	80.9
0,1,0	42.9	0,2,0	85.8
1,1,0	54.9	1,1,1	85.8
0,0,1	66.0	1,2,0	92.4
2,0,0	68.6	2,0,1	95.2
1,0,1	74.3	3,0,0	103

Results and Discussion

The relevant quantity when it comes to placing the loudspeakers is the amplitude of the standing pressure wave. A sound source excites an eigenmode the most if it is placed in one of the pressure antinodes for the mode. Conversely, with the source in a pressure node, the eigenmode remains silent.

All modes have local maxima in the corners of an empty room so speakers in the corners excite all eigenfrequencies. This simulation predicts eigenmodes that strongly resemble those of the corresponding empty room. The higher the frequency, the more the placing of the furniture matters. For instance, some of the high-frequency eigenmodes are located behind the couch.

In the strictest sense, the results of this simulation only apply to a room with perfectly rigid walls and nonabsorbing furniture. The prediction that speakers placed in the corners of the room excite many eigenmodes and give a fuller and more neutral sound, however, holds for real-life rooms.

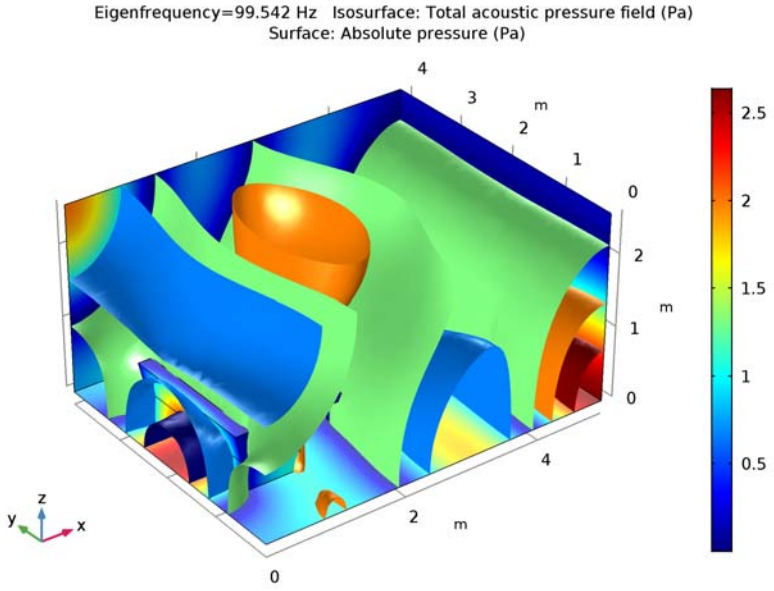


Figure 1: The sound pressure distribution for an eigenfrequency of 99.5 Hz. The real part of the pressure is visualized as an isosurface plot, and the absolute value of the pressure as a boundary plot. Note that this mode does not correspond to any of the analytical modes listed above.

Notes About the COMSOL Implementation

When simulating acoustics, or in general any wave phenomenon, it is important to resolve the expected wavelength properly. As a rule of thumb, 6 second-order element per wavelength is a reasonable trade-off between computational effort and accuracy. For this example, where wavelengths of interest are below 100 Hz, this implies a maximum element size of

$$h_{\max} = \frac{\lambda}{6} = \frac{c}{6f} = \frac{343}{6 \cdot 100}$$

Therefore, an element size of 57 cm or less, corresponding to at least five elements between floor and ceiling, provides sufficient accuracy. The default mesh for this geometry meets the requirement. However, in general it is advisable to specify a maximum element size explicitly in the mesh settings.

Application Library path: `COMSOL_Multiphysics/Acoustics/eigenmodes_of_room`

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 **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Eigenfrequency**.
6. Click **Done**.

GEOMETRY 1

The geometry in this model can be created within COMSOL Multiphysics. Here it is imported for convenience.

Import 1 (imp1)

1 On the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

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

5 Click **Import**.

6 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

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**, type `Air` in the **Label** text field.

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

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

5 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	`1.25`	kg/m³	Basic
Speed of sound	c	343	m/s	Basic

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

The boundary conditions are all **Sound Hard Boundary (Wall)** by default.

STUDY 1

Step 1: Eigenfrequency

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Eigenfrequency**.

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

3 In the **Search for eigenfrequencies around** text field, type 90.

This setting will get you at least 6 solutions with eigenfrequencies in the vicinity of 90 Hz. For this model you typically get additional eigensolutions that the eigenvalue solver finds.

4 On the **Home** toolbar, click **Compute**.

RESULTS

Acoustic Pressure (acpr)

The first default plot shows the pressure distribution on the exterior boundaries of the geometry. To see what goes on inside the room, you need to suppress some of the boundaries. This is most conveniently done by first selecting all boundaries and then removing a few of them (number 1, 2, and 4) from the selection.

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/ Solution 1 (sol1)**.

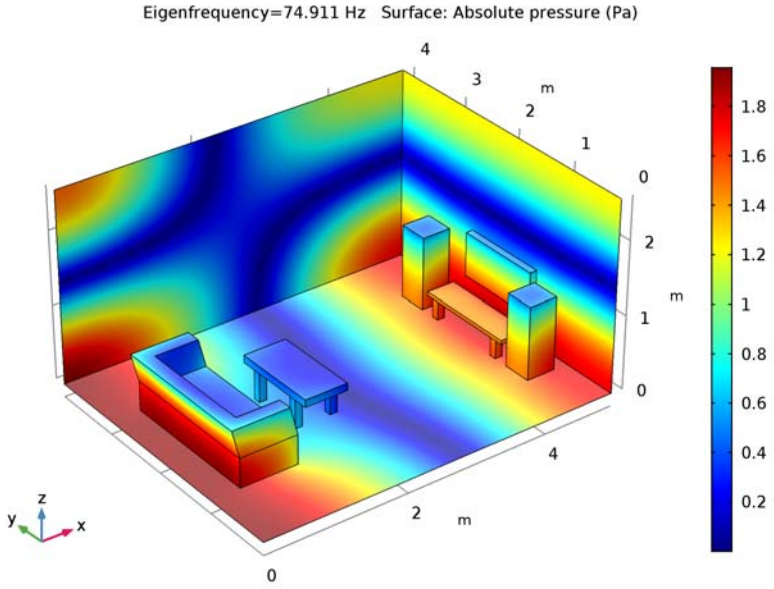
Selection

1. On the **Results** toolbar, click **Selection**.
2. In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Boundary**.
4. From the **Selection** list, choose **All boundaries**.
5. Select Boundaries 3 and 5–79 only.

Surface 1

1. In the **Model Builder** window, expand the **Results>Acoustic Pressure (acpr)** node, then click **Surface 1**.
2. In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>acpr.absp - Absolute pressure**.
3. On the **Acoustic Pressure (acpr)** toolbar, click **Plot**.

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



The first plot group now shows the distribution of the absolute value of the pressure for the lowest one of the eigenfrequencies, 74.9 Hz. This appears to be the (1,0,1) mode.

Acoustic Pressure, Isosurfaces (acpr)

The third default plot shows the isosurfaces for the same frequency.

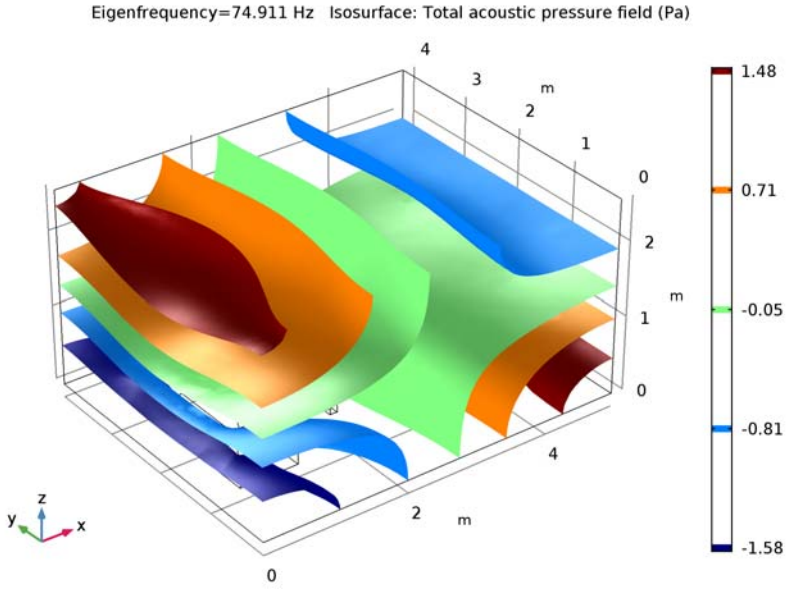
Isosurface 1

1 In the **Model Builder** window, expand the **Acoustic Pressure, Isosurfaces (acpr)** node, then click **Isosurface 1**.

2 In the **Settings** window for **Isosurface**, locate the **Levels** section.

3 In the **Total levels** text field, type 5.

4 On the **Acoustic Pressure, Isosurfaces (acpr)** toolbar, click **Plot**.



Modify the third default plot to show the isosurfaces and pressure distribution at 85 Hz; this is the (1,1,1) mode.

Acoustic Pressure, Isosurfaces (acpr)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure, Isosurfaces (acpr)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Eigenfrequency (Hz)** list, choose **85.093**.

Surface 1

1 Right-click **Results>Acoustic Pressure, Isosurfaces (acpr)** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>acpr.absp - Absolute pressure**.

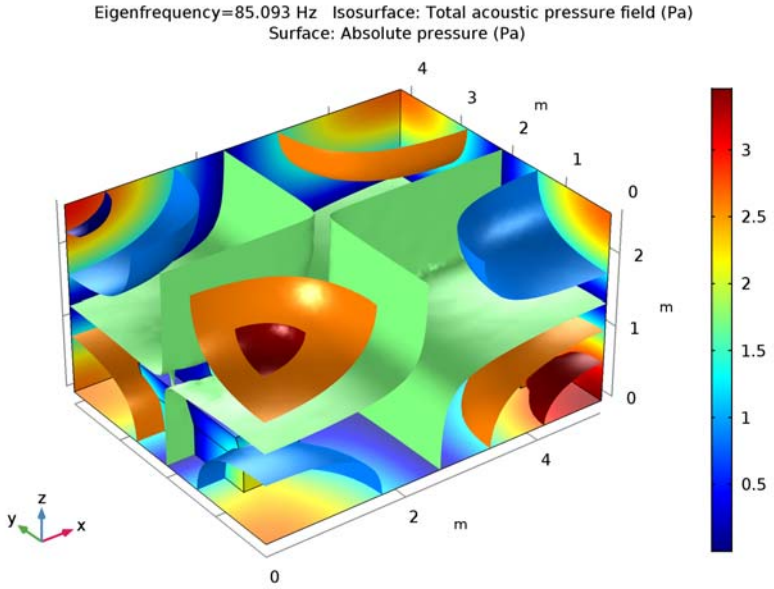
3 On the **Acoustic Pressure, Isosurfaces (acpr)** toolbar, click **Plot**.

Isosurface 1

1 In the **Model Builder** window, under **Results>Acoustic Pressure, Isosurfaces (acpr)** click **Isosurface 1**.

2 In the **Settings** window for **Isosurface**, locate the **Coloring and Style** section.

3 Clear the **Color legend** check box.



Acoustic Pressure, Isosurfaces (acpr)

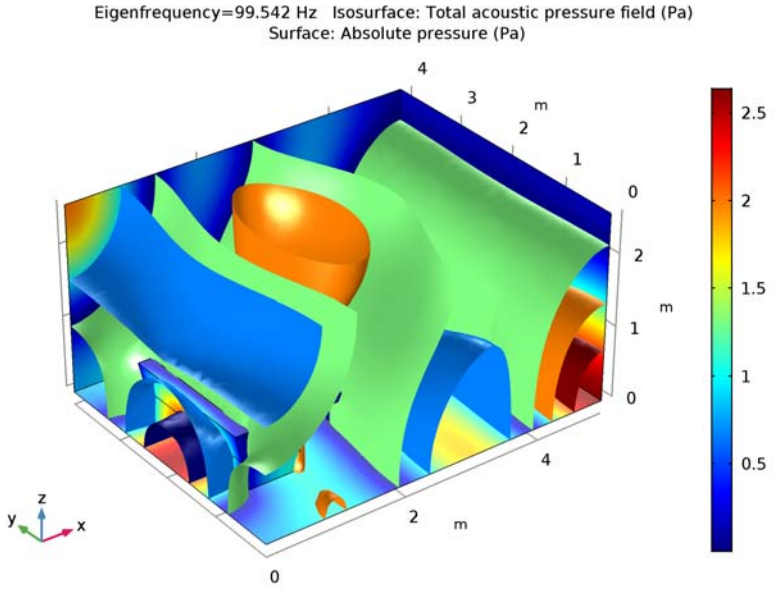
Finally, reproduce Figure 1.

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure, Isosurfaces (acpr)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Eigenfrequency (Hz)** list, choose **99.542**.

4 On the **Acoustic Pressure, Isosurfaces (acpr)** toolbar, click **Plot**.



This mode looks a little different from what you would find in an empty room. It is focused behind the couch.

Electric Sensor

Introduction

This example illustrates — in a simplified electrostatic setting — electric impedance tomography (EIT), a method used for imaging of the distribution of electrical permittivity in an object. The tomography is performed by measuring currents and voltages at the object's surface.

An application of this technique is medical diagnosis. Due to the different electrical properties of various organs and parts of the body, it is possible to obtain information on their position and movement in a non-invasive way.

This model shows how to determine from the outside the shape and the placement of small objects with different material properties inside a box. Applying a potential difference on the boundaries of the box creates a surface charge density that varies depending on the permittivity distribution inside the box.

Model Definition

This model solves Gauss' law with $\rho = 0$:

$$-\nabla \cdot (\varepsilon_0 \varepsilon_r \nabla V) = \rho$$

The box contains air with ε_r equal to 1. The different objects are made of materials with different values of the relative permittivity, ε_r: 1, 2, and 3.

To get a voltage difference, a ground condition ($V = 0$) is set on the bottom while the condition $V = 1$ is applied on the top of the box. On the side, the boundary condition used is electric insulation: $\mathbf{n} \cdot \mathbf{D} = 0$.

Results and Discussion

As seen in Figure 1, the surface charge density is higher in correspondence of materials with higher permittivity, as expected. An imaging of the figures inside the box is reproduced in the surface charge density plot.

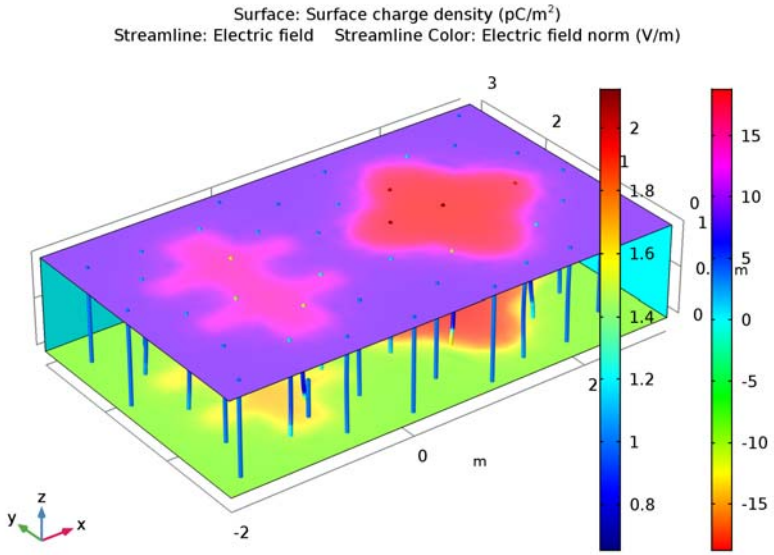


Figure 1: Surface charge density (boundary), electric field (streamline density), and electric potential (streamline color).

Inside the geometry the streamlines show how the electric field varies. The electric field is lower in media with larger value of the permittivity.

Application Library path: `COMSOL_Multiphysics/Electromagnetics/electric_sensor`

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 **AC/DC>Electrostatics (es)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

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

3 In the **z-coordinate** text field, type `0.1`.

4 Click **Show Work Plane**.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `0.5`.

4 In the **Height** text field, type `2`.

5 Locate the **Position** section. In the **xw** text field, type `-1`.

6 In the **yw** text field, type `0.5`.

7 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

Rectangle 2 (r2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `1.5`.

4 In the **Height** text field, type `0.25`.

5 Locate the **Position** section. In the **xw** text field, type `-1.5`.

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

7 Right-click **Rectangle 2 (r2)** and choose **Build Selected**.

Rectangle 3 (r3)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `1.5`.

4 In the **Height** text field, type `0.25`.

5 Locate the **Position** section. In the **xw** text field, type `-1.5`.

6 In the **yw** text field, type `1.75`.

7 Right-click **Rectangle 3 (r3)** and choose **Build Selected**.

Union 1 (uni1)

1 On the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Union**.

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

3 In the **Settings** window for **Union**, locate the **Union** section.

4 Clear the **Keep interior boundaries** check box.

5 Right-click **Union 1 (uni1)** and choose **Build Selected**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

Ellipse 1 (e1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Ellipse**.

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

3 In the **a-semiaxis** text field, type `0.5`.

4 Locate the **Position** section. In the **xw** text field, type `1.5`.

5 In the **yw** text field, type `1.5`.

6 Right-click **Ellipse 1 (e1)** and choose **Build Selected**.

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

Ellipse 2 (e2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Ellipse**.

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

3 In the **b-semiaxis** text field, type `0.5`.

4 Locate the **Position** section. In the **xw** text field, type `1.5`.

5 In the **yw** text field, type `1.5`.

6 Right-click **Ellipse 2 (e2)** and choose **Build Selected**.

Compose 1 (co1)

1 On the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Compose**.

2 Select the objects **e1** and **e2** only.

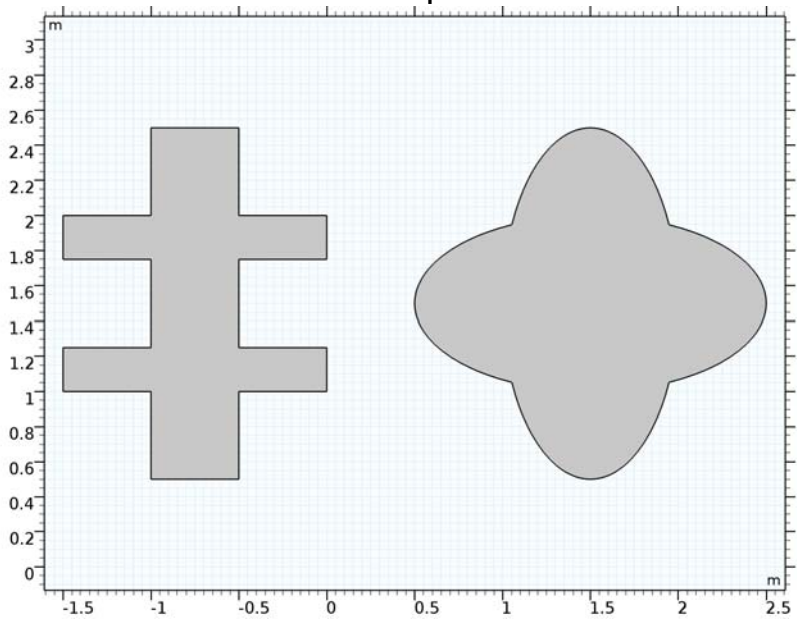
3 In the **Settings** window for **Compose**, locate the **Compose** section.

4 Clear the **Keep interior boundaries** check box.

5 In the **Set formula** text field, type `e1+e2`.

6 Right-click **Compose 1 (co1)** and choose **Build Selected**.

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



Work Plane 1 (wp1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 1 (wp1)**.

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (m)
0.8

4 Right-click **Extrude 1 (ext1)** and choose **Build Selected**.

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

Block 1 (blk1)

1 On 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 `5`.

4 In the **Depth** text field, type `3`.

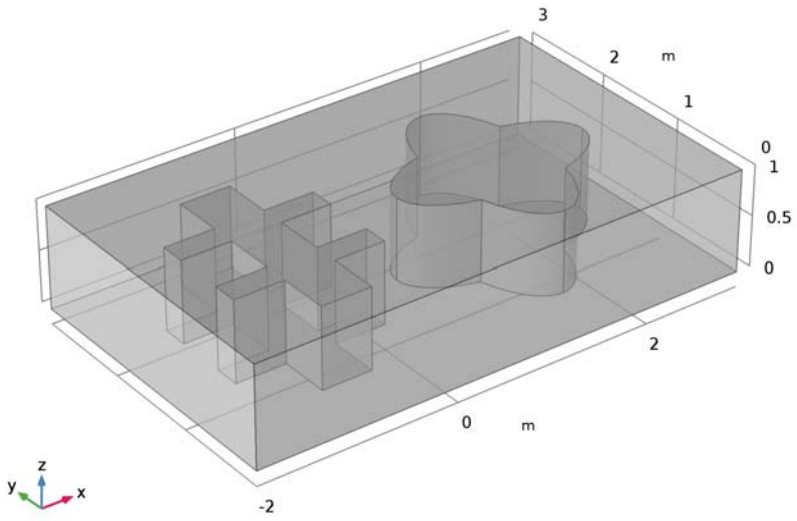
5 Locate the **Position** section. In the **x** text field, type `-2`.

6 Click **Build All Objects**.

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

8 Click the **Transparency** button on the **Graphics** toolbar.

This completes the model geometry.



ELECTROSTATICS (ES)

Ground 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.

2 Select Boundary 3 only.

Electric Potential 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.

4 In the V_0 text field, type 1.

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
Relative permittivity	epsilonr	1	1	Basic

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

2 Select Domain 2 only.

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

4 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr	2	1	Basic

Material 3 (mat3)

1 Right-click **Materials** and choose **Blank Material**.

2 Select Domain 3 only.

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

4 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr	3	1	Basic

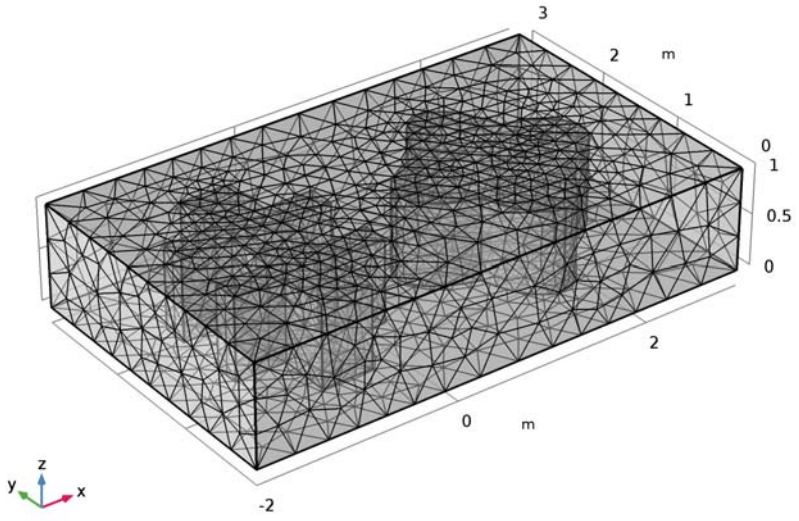
MESH 1

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

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

3 From the **Element size** list, choose **Fine**.

4 Click **Build All**.



STUDY 1

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

RESULTS

Electric Potential (es)

To reproduce the plot shown in Figure 1, begin by suppressing some boundaries so that the inside of the box becomes visible.

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/ Solution 1 (sol1)**.

Selection

1 On the **Results** toolbar, click **Selection**.

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

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

4 From the **Selection** list, choose **All boundaries**.

5 Select Boundaries 3–5 and 38 only.

6 Click the **Transparency** button on the **Graphics** toolbar.

Electric Potential (es)

Remove the default slice plot of the potential.

1 In the **Model Builder** window, under **Results** right-click **Electric Potential (es)** and choose **Delete**.

2 Click **Yes** to confirm.

Surface 1

1 On the **Results** toolbar, click **3D Plot Group**.

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

3 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electrostatics> Currents and charge>es.nD - Surface charge density**.

4 Locate the **Expression** section. In the **Unit** field, type `pC/m^2`.

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

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

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

Streamline 1

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

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

3 From the **Positioning** list, choose **Magnitude controlled**.

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

Color Expression 1

1 Right-click **Results>3D Plot Group 1>Streamline 1** and choose **Color Expression**.

2 In the **Settings** window for **Color Expression**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electrostatics> Electric>es.normE - Electric field norm**.

3 Click to expand the **Title** section. From the **Title type** list, choose **Automatic**.

Compare the resulting plot with that in Figure 1.

Electrochemical Polishing

Introduction

This example illustrates the principle of electrochemical polishing. The simplified 2D model geometry consists of two electrodes and an intermediate electrolyte domain. The positive electrode has a protrusion, representing a surface defect. The purpose of the application is to examine how this protrusion and the surrounding electrode material are depleted over a period of time.

Model Definition

The potential drop over the electrodes is 30 V, and the electrolyte has a conductivity of 10 S/m.

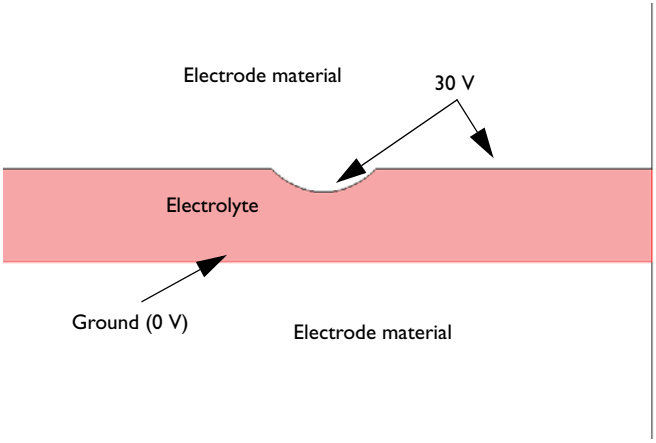


Figure 1: Model geometry.

Modeling the depletion of the positive electrode requires a moving boundary because the geometry changes and the current density distribution with it. A simple model for the depletion is based on the assumption that the depletion rate is proportional to the normal current density at the electrode surface. The velocity, U, normal to the mesh at the electrode surface then becomes

$$U = -KJ_n \tag{1}$$

where K is the coefficient of proportionality, and J_n is the normal current density. In this example, $K = 10^{-11}$ m^3/As.

The part of the electrode and electrolyte that the model includes is about 3 mm wide and the distance between the electrodes is 0.4 mm.

Results and Discussion

After a period of 10 s, the protrusion is somewhat smoothed out, and a significant portion of the positive electrode has been depleted.

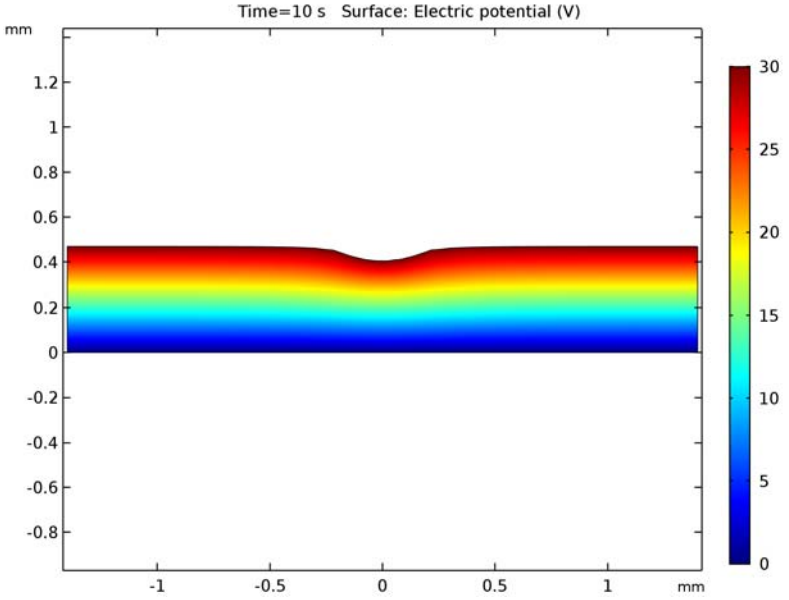


Figure 2: Potential distribution and electrode depletion after 10 s.

Using Equation 1, the expected total depletion increment, $d(\Delta t = 10\ \text{s})$, over the simulated time interval can be estimated as

$$d(\Delta t) = |U|\Delta t = K|J_n|\Delta t = \left(10^{-11}\ \frac{\text{m}^3}{\text{As}}\right)\cdot\left(10^{6}\frac{\text{A}}{\text{m}}\right)\cdot(10^{1}\text{s}) = 10^{-4}\ \text{m} \qquad (2)$$

This estimate agrees with the maximum value for the y-displacement obtained for the model, showing that the approximate formula (which does not take effects from the curved boundary into account) is in fact very accurate.

Notes About the COMSOL Implementation

This application uses the Electric Currents and Deformed Geometry interfaces. The variable for the normal current density defines the mesh velocity. The dynamics in this

example is quasi-static in nature, and the time dependence only enters in the depletion (removal of material) of the electrode.

Application Library path: `COMSOL_Multiphysics/Electromagnetics/electrochemical_polishing`

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 **Mathematics>Deformed Mesh>Deformed Geometry (dg)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **AC/DC>Electric Currents (ec)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Time Dependent**.

8 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Description
K	1e-11[m^3/(A*s)]	Coefficient of proportionality

GEOMETRY 1

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

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `2.8`.

4 In the **Height** text field, type `0.4`.

5 Locate the **Position** section. In the **x** text field, type `-1.4`.

6 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `0.3`.

4 Locate the **Position** section. In the **y** text field, type `0.6`.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **r1** only to add it to the **Objects to add** list.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

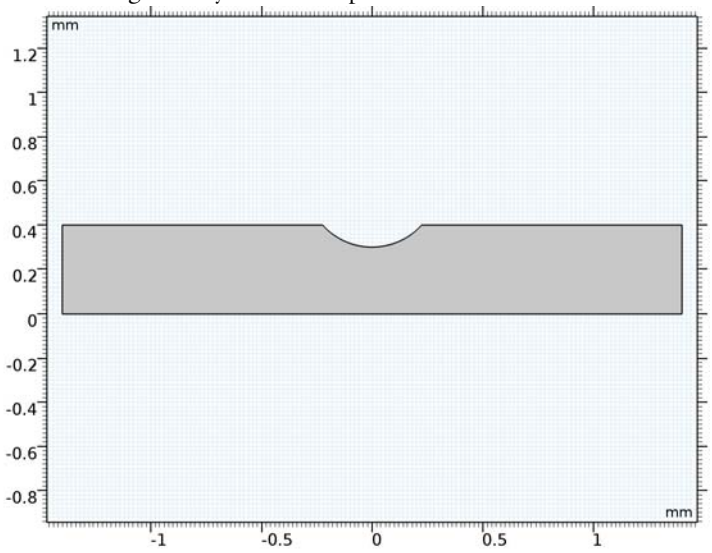
4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the object **c1** only.

6 Right-click **Difference 1 (dif1)** and choose **Build Selected**.

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

The model geometry is now complete.



Before turning to the **Deformed Geometry** interface settings, define variables for the local displacement components.

DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **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
dx	x-Xg	m	x-displacement
dy	y-Yg	m	y-displacement

Here, `Xg` and `Yg` are geometry-frame coordinates corresponding to x and y.

DEFORMED GEOMETRY (DG)

Free Deformation 1

1 On the **Physics** toolbar, click **Domains** and choose **Free Deformation**.

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

Prescribed Mesh Velocity 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Mesh Velocity**.

2 Select Boundaries 1 and 5 only.

3 In the **Settings** window for **Prescribed Mesh Velocity**, locate the **Prescribed Mesh Velocity** section.

4 Clear the **Prescribed Y velocity** check box.

Prescribed Normal Mesh Velocity 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Normal Mesh Velocity**.

2 Select Boundaries 3, 4, 6, and 7 only.

3 In the **Settings** window for **Prescribed Normal Mesh Velocity**, locate the **Normal Mesh Velocity** section.

4 In the v_n text field, type `-K*(-ec.nJ)`.

ELECTRIC CURRENTS (EC)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electric Currents (ec)**.

2 In the **Settings** window for **Electric Currents**, click to expand the **Equation** section.

3 From the **Equation form** list, choose **Stationary**.

With this setting you specify that the current distribution can be regarded as stationary on the time scale determined by the depletion rate.

Current Conservation 1

1 In the **Model Builder** window, expand the **Electric Currents (ec)** node, then click **Current Conservation 1**.

2 In the **Settings** window for **Current Conservation**, locate the **Conduction Current** section.

3 From the σ list, choose **User defined**. In the associated text field, type `10`.

4 In the **Model Builder** window, click **Electric Currents (ec)**.

Electric Potential 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

2 Select Boundaries 3, 4, 6, and 7 only.

3 In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.

4 In the V_0 text field, type `30`.

Ground 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.

2 Select Boundary 2 only.

Electric Insulation 1

For the left and right boundaries, the default boundary condition is a good approximation if you want to simulate that the electrodes are extended indefinitely in both directions.

MESH 1

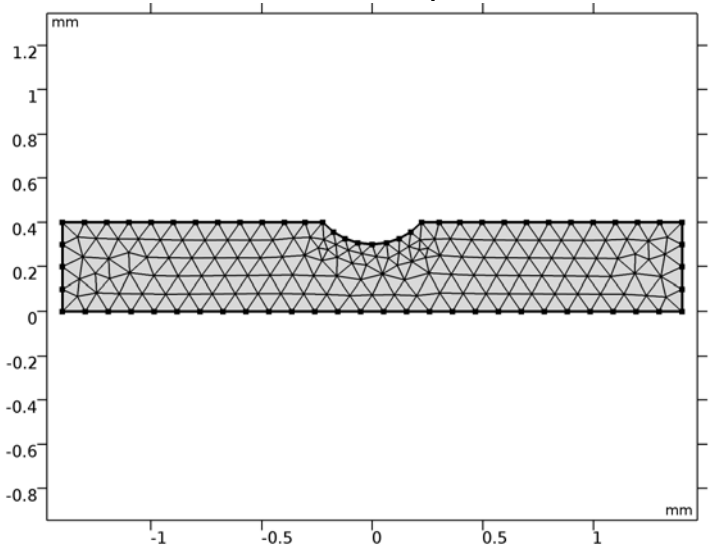
1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

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

3 From the **Element size** list, choose **Finer**.

4 Click **Build All**.

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



STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.

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

3 In the **Times** text field, type `range(0,10)`.

4 On the **Home** toolbar, click **Compute**.

RESULTS

Electric Potential (ec)

1 Click the **Zoom Extents** button on the **Graphics** toolbar.

The default plot shows the potential field at the end of the simulation interval; compare with Figure 2.

2D Plot Group 2

Next, plot the current distribution.

Surface 1

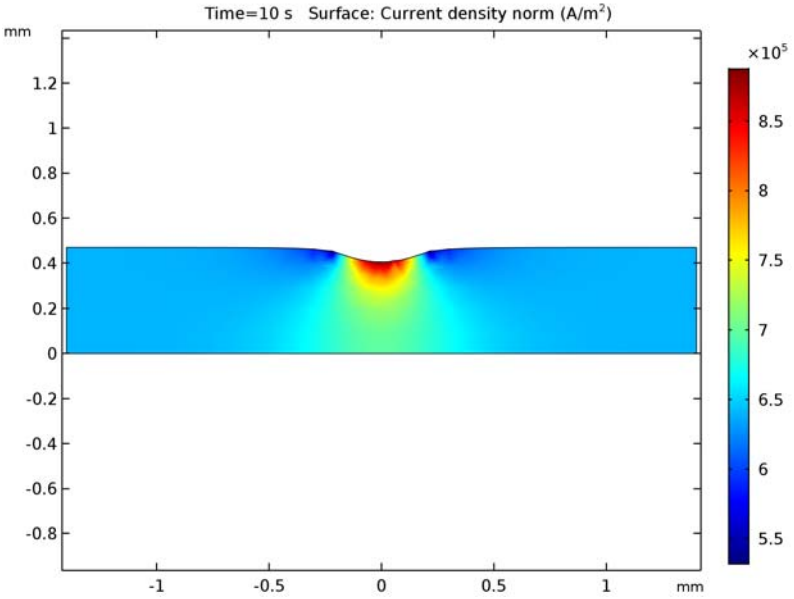
1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

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

3 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electric Currents> Currents and charge>ec.normJ - Current density norm**.

4 On the **2D Plot Group 2** toolbar, click **Plot**.

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

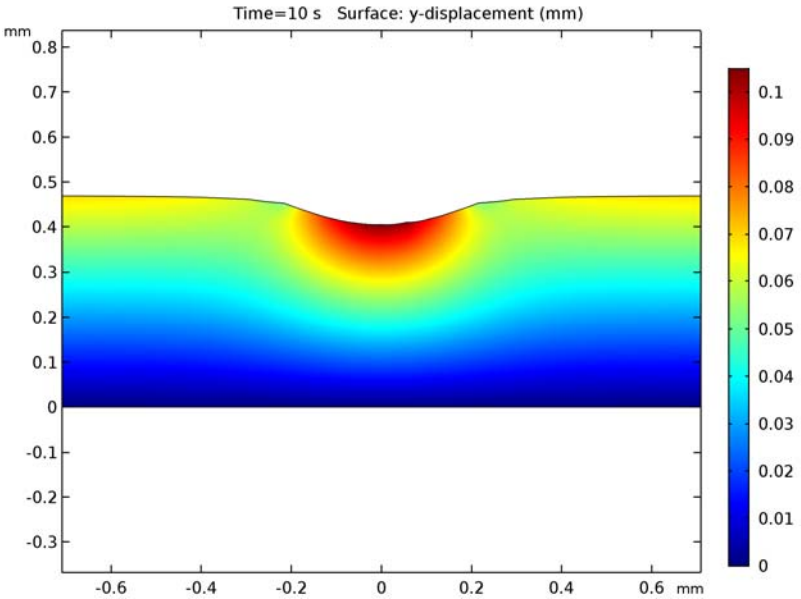


The maximum current density appears to be of the order of 10^6 A/m^2.

To see the magnitude of the depletion in the y-direction more easily, plot the y-component of the mesh displacement.

Surface 1

1. On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.
2. In the **Model Builder** window, right-click **2D Plot Group 3** and choose **Surface**.
3. In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Definitions>Variables>dy - y-displacement**.
4. On the **2D Plot Group 3** toolbar, click **Plot**.
5. Click the **Zoom In** button on the **Graphics** toolbar.



The maximum value for the y-displacement is approximately 0.1 mm, which agrees with the value calculated in Equation 2.

Steady-State 2D Heat Transfer with Conduction

Introduction

This example shows a 2D steady-state thermal analysis including convection to a prescribed external (ambient) temperature. The example is taken from a NAFEMS benchmark collection (see Ref. 1).

Model Definition

This example considers 0.6 m-by-1.0 m domain. For the boundary conditions:

- The left boundary is insulated.
- The lower boundary is kept at 100 °C.
- The upper and right boundaries are convecting to 0 °C with a heat transfer coefficient of 750 W/(m^2·°C).

In the domain use the following material property:

- The thermal conductivity is 52 W/(m·°C).

Results

The plot in Figure 1 shows the temperature field in the modeling domain.

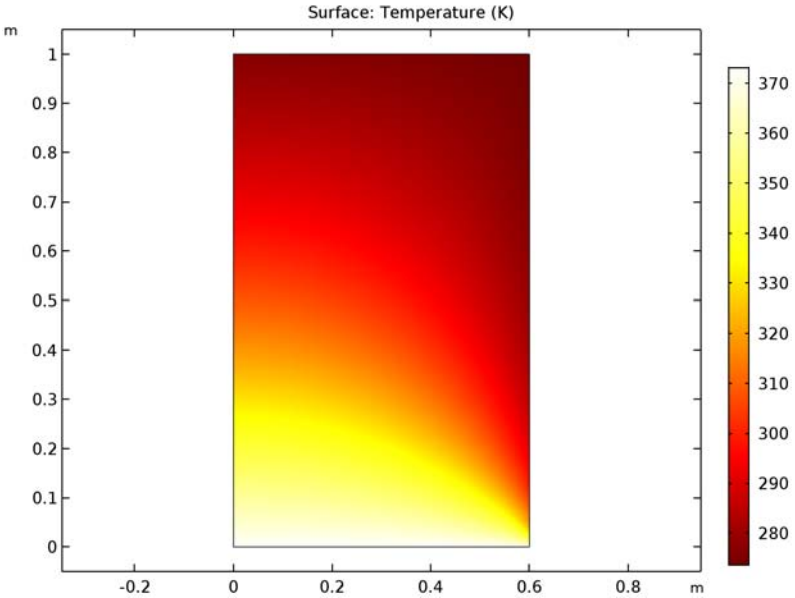


Figure 1: Temperature distribution resulting from convection to a prescribed external temperature.

The benchmark result for the target location (x = 0.6 m and y = 0.2 m) is a temperature of 18.25 °C. The COMSOL Multiphysics model, using a mapped mesh with 9×15 quadratic elements, gives a temperature of 18.265 °C.

Reference

1. A.D. Cameron, J.A. Casey, and G.B. Simpson, *NAFEMS Benchmark Tests for Thermal Analysis (Summary)*, NAFEMS, Glasgow, 1986.

Application Library path: `COMSOL_Multiphysics/Heat_Transfer/heat_convection_2d`

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 **Heat Transfer>Heat Transfer in Solids (ht)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `0.6`.

4 Click **Build All Objects**.

HEAT TRANSFER IN SOLIDS (HT)

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 2 only.

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

4 In the T_0 text field, type `100[degC]`.

Heat Flux 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 Select Boundaries 3 and 4 only.

3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

4 Click the **Convective heat flux** button.

5 In the h text field, type `750`.

6 In the T_{ext} text field, type `0[degC]`.

Solid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids (ht)** click **Solid 1**.

2 In the **Settings** window for **Solid**, locate the **Heat Conduction, Solid** section.

3 From the k list, choose **User defined**. In the associated text field, type `52`.

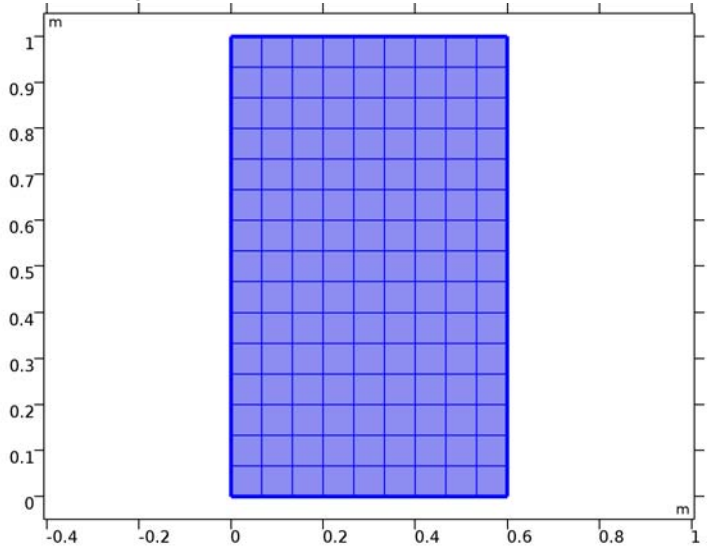
No other material properties enter into the domain equations for this stationary model.

MESH 1

Mapped 1

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

2 In the **Settings** window for **Mapped**, click **Build All**.



STUDY 1

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

RESULTS

Temperature (ht)

1 Click the **Zoom Extents** button on the **Graphics** toolbar.

The first default plot group shows the temperature field; compare with Figure 1.

Data Sets

The benchmark value for the temperature at $x = 0.6$ m and $y = 0.2$ m is 18.25 °C. To compare this value with that from the simulation, evaluate the temperature in this position.

Cut Point 2D 1

1 On the **Results** toolbar, click **Cut Point 2D**.

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

3 In the **X** text field, type `0.6`.

4 In the **Y** text field, type `0.2`.

Point Evaluation 1

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

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

3 From the **Data set** list, choose **Cut Point 2D 1**.

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

Expression	Unit	Description
`T`	`degC`	`Temperature`

5 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The result should be close to 18.265 °C.

Steady-State 1D Heat Transfer with Radiation

Introduction

This example shows a 1D steady-state thermal analysis including radiation to a prescribed ambient temperature. The example is taken from a NAFEMS benchmark collection (Ref. 1).

Model Definition

This 1D model has a domain of length 0.1 m. The left end is kept at 1000 K, and at the right end there is radiation to 300 K. The model uses the following material properties:

- For the radiation, the emissivity, ε, is 0.98.
- The thermal conductivity is 55.563 W/(m·K).

Results

The following plot shows the temperature as a function of position:

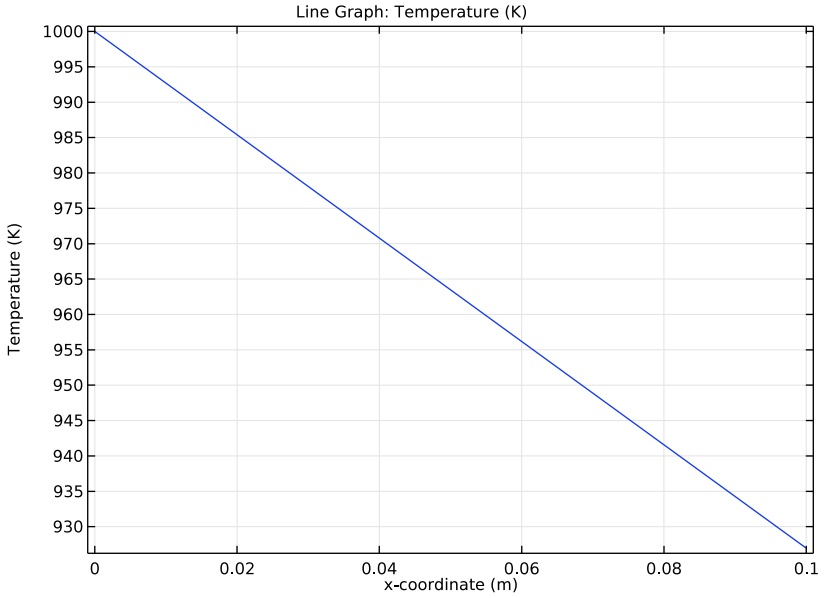


Figure 1: Temperature as a function of position.

The benchmark result for the right end is a temperature of 927.0 K. The COMSOL Multiphysics model, using a default mesh with 15 elements, gives a temperature at the end as 926.97 K, which is the exact benchmark value to four significant digits.

Reference

1. A.D. Cameron, J.A. Casey, and G.B. Simpson, *NAFEMS Benchmark Tests for Thermal Analysis (Summary)*, NAFEMS, Glasgow, 1986.

Application Library path: `COMSOL_Multiphysics/Heat_Transfer/heat_radiation_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 **1D**.

2 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

Interval 1 (i1)

1 On the **Geometry** toolbar, click **Interval**.

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

3 In the **Right endpoint** text field, type `0.1`.

4 Click **Build All Objects**.

HEAT TRANSFER IN SOLIDS (HT)

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 1 only.

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

4 In the T_0 text field, type `1000`.

Diffuse Surface 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Diffuse Surface**.

2 Select Boundary 2 only.

3 In the **Settings** window for **Diffuse Surface**, locate the **Surface Emissivity** section.

4 From the ε list, choose **User defined**. In the associated text field, type `0.98`.

5 Locate the **Ambient** section. In the T_{amb} text field, type `300`.

Solid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids (ht)** click **Solid 1**.

2 In the **Settings** window for **Solid**, locate the **Heat Conduction, Solid** section.

3 From the k list, choose **User defined**. In the associated text field, type `55.563`.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids (ht)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, type `1000` in the T text field.

MESH 1

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

2 In the **Settings** window for **Mesh**, click **Build All**.

STUDY 1

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

RESULTS

Derived Values

The benchmark value for the temperature at the right end is 927.0 K. To compare the value from the simulation, evaluate the temperature in that position.

Point Evaluation 1

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

2 Select Boundary 2 only.

3 In the **Settings** window for **Point Evaluation**, click **Evaluate**.

Axisymmetric Transient Heat Transfer

Introduction

This example shows an axisymmetric transient thermal analysis with a step change to 1000 °C at time 0. The example is taken from a NAFEMS benchmark collection (Ref. 1).

Model Definition

This example considers the 0.3 m-by-0.4 m domain. For the boundary conditions, assume the following:

- The left boundary is the symmetry axis.
- The other boundaries have a temperature of 1000 °C. The entire domain is at 0 °C at the start, which represents a step change in temperature at the boundaries.

In the domain use the following material properties:

- The density, ρ, is 7850 kg/m^3
- The heat capacity is 460 J/(kg·°C)
- The thermal conductivity is 52 W/(m·°C)

Results

The following revolved surface plot shows the temperature distribution inside the cylinder after 190 seconds:

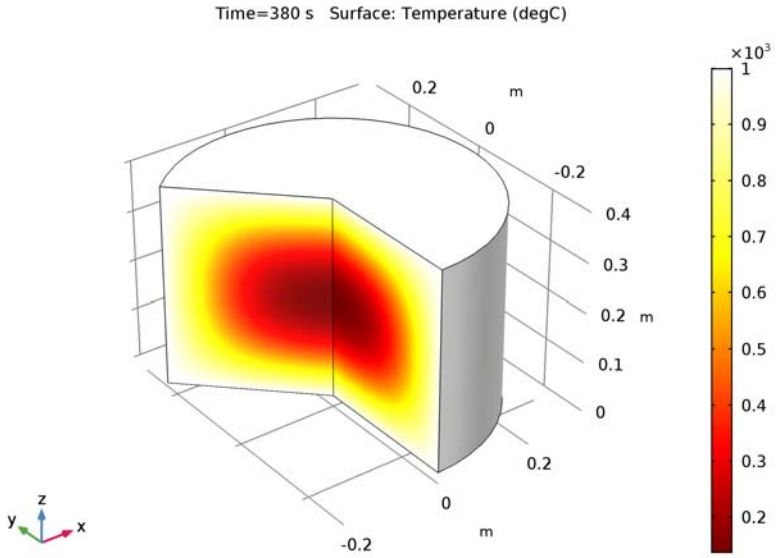


Figure 1: Temperature distribution after 190 seconds.

The benchmark result for the target location ($r = 0.1$ m and $z = 0.3$ m) is a temperature of 186.5 °C. The COMSOL Multiphysics model, using a default mesh with about 425 elements, gives a temperature of roughly 186.3 °C.

Reference

1. A.D. Cameron, J.A. Casey, and G.B. Simpson, *NAFEMS Benchmark Tests for Thermal Analysis (Summary)*, NAFEMS, Glasgow, 1986.

Application Library path: `COMSOL_Multiphysics/Heat_Transfer/heat_transient_axi`

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 **Heat Transfer>Heat Transfer in Solids (ht)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Time Dependent**.
6. Click **Done**.

GEOMETRY 1

Rectangle 1 (r1)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `0.3`.
4. In the **Height** text field, type `0.4`.
5. Click **Build All Objects**.

HEAT TRANSFER IN SOLIDS (HT)

Temperature 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.
2. In the **Settings** window for **Temperature**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **All boundaries**.
4. Locate the **Temperature** section. In the T_0 text field, type `1000[degC]`.

Solid 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids (ht)** click **Solid 1**.
2. In the **Settings** window for **Solid**, locate the **Heat Conduction, Solid** section.
3. From the k list, choose **User defined**. In the associated text field, type `52`.

4 Locate the **Thermodynamics, Solid** section. From the C_p list, choose **User defined**. In the associated text field, type `460`.

5 From the ρ list, choose **User defined**. In the associated text field, type `7850`.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids (ht)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, type `0[degC]` in the T text field.

MESH 1

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

2 In the **Settings** window for **Mesh**, click **Build All**.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, type `Time Dependent - Continuous Simulation (with Heating)` in the **Label** text field.

3 Locate the **Study Settings** section. In the **Times** text field, type `range(0,10,380)`.

4 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

5 On the **Home** toolbar, click **Compute**.

RESULTS

Temperature, 3D (ht)

To get the plot shown in Figure 1, just change the unit as follows:

Surface

1 In the **Model Builder** window, expand the **Temperature, 3D (ht)** node, then click **Surface**.

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

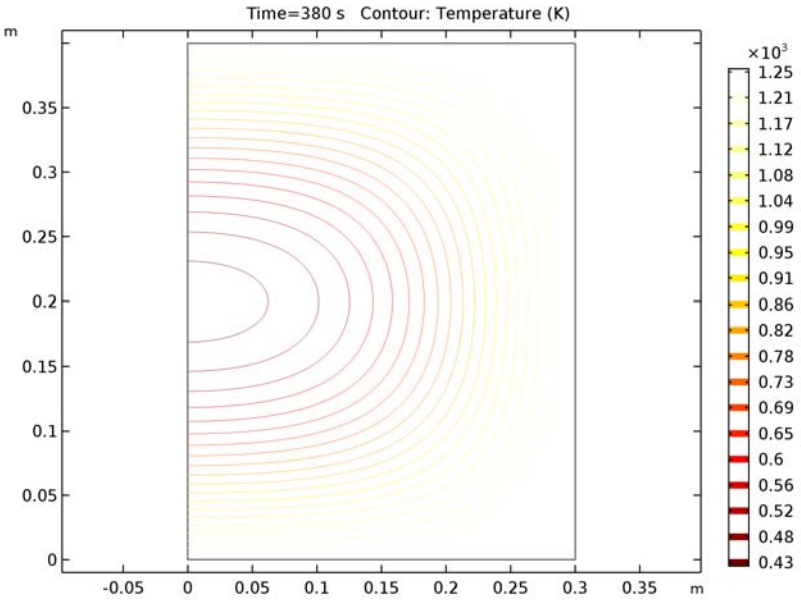
3 From the **Unit** list, choose **degC**.

4 On the **Temperature, 3D (ht)** toolbar, click **Plot**.

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

Isothermal Contours (ht)

The second default plot group visualizes the temperature field using a contour plot.



Data Sets

The benchmark value for the temperature at $r = 0.1$ m and $z = 0.3$ m is 186.5 °C. To compare the value from the simulation, evaluate the temperature in that position.

Cut Point 2D 1

1 On the **Results** toolbar, click **Cut Point 2D**.

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

3 In the **R** text field, type `0.1`.

4 In the **Z** text field, type `0.3`.

Point Evaluation 1

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

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

3 From the **Data set** list, choose **Cut Point 2D 1**.

4 From the **Time selection** list, choose **Last**.

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

Expression	Unit	Description
T	degC	Temperature

6 Click **Evaluate**.

As an optional extension of the model, you can add a study sequence where, starting from 190 s, the boundaries are thermally insulated.

ADD STUDY

1 On 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 **Preset Studies> Time Dependent**.

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

STUDY 2

Step 1: Time Dependent

1 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

2 In the **Settings** window for **Time Dependent**, type `Time Dependent - First Part (with Heating)` in the **Label** text field.

3 Locate the **Study Settings** section. In the **Times** text field, type `range(0,10,190)`.

Step 2: Time Dependent 2

1 On the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Time Dependent**.

2 In the **Settings** window for **Time Dependent**, type `Time Dependent - Second Part (with Insulation)` in the **Label** text field.

3 Locate the **Study Settings** section. In the **Times** text field, type `range(190,10,380)`.

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

5 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Heat Transfer in Solids (ht)>Temperature 1**.

6 Click **Disable**.

7 On the **Study** toolbar, click **Compute**.

To combine the two time-dependent simulations, add a **Combine Solutions** study step. This concatenates the two solutions and makes it possible to treat the output as a single continuous time-dependent solution.

STUDY 2

Combine Solutions

1 On the **Study** toolbar, click **Combine Solutions**.

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

3 From the **First solution** list, choose **Study 2/Solution Store 1 (sol3)**.

4 On the **Study** toolbar, click **Compute**.

RESULTS

Surface

1 In the **Model Builder** window, expand the **Temperature, 3D (ht) 1** node, then click **Surface**.

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

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

4 On the **Temperature, 3D (ht) 1** toolbar, click **Plot**.

Cut Point 2D 1

1 In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Cut Point 2D 1**.

2 In the **Settings** window for **Cut Point 2D**, type `Cut Point 2D - Continuous Heating` in the **Label** text field.

Cut Point 2D 2

1 On the **Results** toolbar, click **Cut Point 2D**.

2 In the **Settings** window for **Cut Point 2D**, type `Cut Point 2D - Combined Solutions` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **Point Data** section. In the **R** text field, type `0.1`.

5 In the **Z** text field, type `0.3`.

Join 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Join**.

2 In the **Settings** window for **Join**, type `Join - Temperature Difference` in the **Label** text field.

3 Locate the **Data 1** section. From the **Data** list, choose **Cut Point 2D - Continuous Heating**.

4 Locate the **Data 2** section. From the **Data** list, choose **Cut Point 2D - Combined Solutions**.

1D Plot Group 5

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

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

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

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

5 In the **Title** text area, type `Temperature vs. Time for Continuous and Concatenated Solutions`.

Point Graph 1

1 Right-click **Temperature, 1D** and choose **Point Graph**.

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

3 From the **Data set** list, choose **Cut Point 2D - Continuous Heating**.

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

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

Point Graph 2

1 In the **Model Builder** window, under **Results** right-click **Temperature, 1D** and choose **Point Graph**.

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

3 From the **Data set** list, choose **Cut Point 2D - Combined Solutions**.

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

Point Graph 1

1 In the **Model Builder** window, under **Results>Temperature, 1D** click **Point Graph 1**.

2 In the **Settings** window for **Point Graph**, click to expand the **Legends** section.

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

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

5 In the table, enter the following settings:

Legends
`Continuous heating`

Point Graph 2

1 In the **Model Builder** window, under **Results>Temperature, 1D** click **Point Graph 2**.

2 In the **Settings** window for **Point Graph**, locate the **Legends** section.

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

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

5 In the table, enter the following settings:

Legends
Thermal insulation applied after 190 s

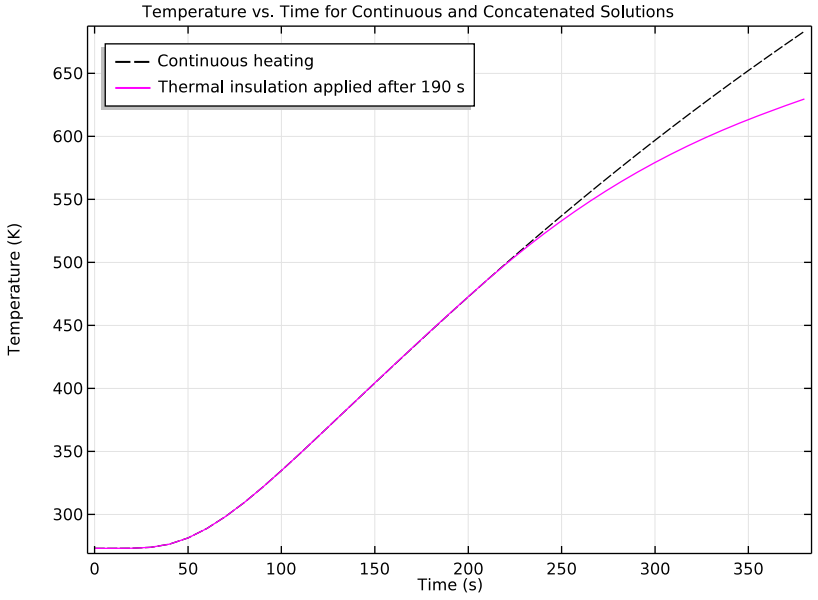
Temperature, 1D

1 In the **Model Builder** window, under **Results** click **Temperature, 1D**.

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

3 From the **Position** list, choose **Upper left**.

4 On the **Temperature, 1D** toolbar, click **Plot**.



1D Plot Group 6

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

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

3 Locate the **Data** section. From the **Data set** list, choose **Join - Temperature Difference**.

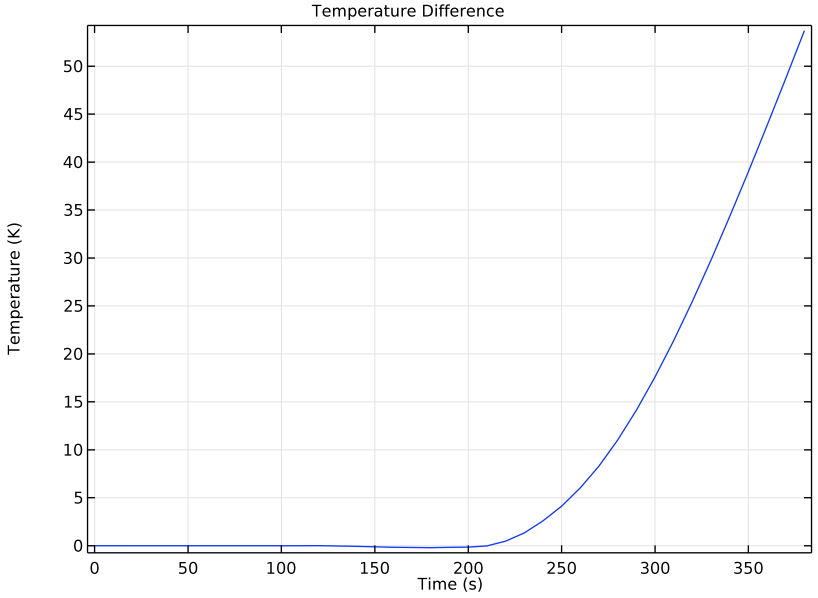
4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Temperature Difference`.

Point Graph 1

1 Right-click **Temperature Difference, 1D** and choose **Point Graph**.

2 On the **Temperature Difference, 1D** toolbar, click **Plot**.



The KdV Equation and Solitons

Introduction

The Korteweg-de Vries (KdV) equation, formulated in 1895 by Korteweg and de Vries, models water waves. It contrasts sharply to the Burgers equation because it introduces no dissipation and the waves travel seemingly forever. In 1965, Zabusky and Kruskal named such waves *solitons*.

The KdV equation with boundary conditions and initial value is formulated as

$$\begin{aligned} &u_t + u_{xxx} = 6uu_x && \text{in } \Omega = [-8, 8] \\ &u(-8, t) = u(8, t), && \text{periodic} \\ &u(x, 0) = -6\,\mathrm{sech}^2(x) \end{aligned}$$

The equation models the steepening and dispersion of wavefronts but does not support a train of simple harmonic waves. Such trains comprise the wave crests normally associated with the ocean: simply a momentary constructive interference of contributing waves moving at different speeds. However, the equation does support solitons, single "humps" that travel without changing shape or speed for unexpectedly long distances.

Indeed, Perry and Schimke (Ref. 2) concluded from shipboard oceanographic measurements that bands of choppy water in the Andaman Sea, which lies east of the Bay of Bengal and west of Burma (Union of Myanmar) and Thailand, are associated with large-amplitude oceanic internal waves. Satellite images have since clarified that these waves originate on shallow banks on a layer between warm and cool water. Further, Osborne and Burch (Ref. 1) analyzed oceanographic data in an effort to assess the forces of underwater current fluctuations associated with such waves on offshore drilling rigs. They concluded that the visually observed roughness bands are caused by internal solitons that follow the KdV equation (Ref. 3).

A more recent development is the application of the KdV equation to another type of waves — light waves. Today solitons have their primary practical application in optical fibers. Specifically, a fiber's linear dispersion properties level out a wave while the nonlinear properties give a focusing effect. The result is a very stable, long-lived pulse (Ref. 3). It is amazing that researchers have discovered a formula for such waves:

$$u = \frac{v}{\left[2\cosh^2\left(\frac{1}{2}\sqrt{v}\right)(x - vt - f)\right]}$$

This equation says that the pulse speed is what determines the pulse amplitude and the pulse width. The following simulation illustrates this effect. An initial pulse, which does not conform to the formula, immediately breaks down into two pulses of different

amplitudes and speeds. The two new pulses follow the formula and thus can travel forever. While the formula does not reveal how solitons interact, the simulation shows that they can collide and reappear, seemingly unchanged, just as linear waves do, another counterintuitive observation that is difficult to observe without predictions by computing.

Model Definition

In the model, the term uu_x describes the focusing of a wave and u_{xxx} refers to its dispersion. The balancing of these two terms permits waves to travel with their shape unchanged.

Because COMSOL Multiphysics does not evaluate third derivatives directly, you rewrite the original equation above as a system of two variables to solve it:

$$u_{1t} + u_{2x} = 6u_1u_{1x}$$
$$u_{1xx} = u_2$$

Using the General Form PDE interface, you need to define two dependent variables, `u1` and `u2`, and identify the d_a, Γ, and F coefficients in the following equation:

$$d_a\frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot \Gamma = F$$

- Only the first equation has a time derivative, and it is with respect to u_1, so only d_a(1, 1) is 1; the other three components are zero.
- The divergence is a space derivative with respect to x. This means that the Γ component from the first equation is u_2, which you type as `u2`. The Γ component from the second equation is u_{1x}, which you express using COMSOL Multiphysics syntax as `u1x`.
- The F term components are the right-hand side of the equations: F_1 is $6u_1u_{1x}$ (type `6*u1*u1x`), and F_2 is u_2 (type `u2`).

The initial condition for u_1 uses a hyperbolic cosine function to provide an interesting wave form to start with. For u_2, you must provide the second space derivative of this function to provide consistent initial conditions.

The boundary conditions are periodic boundary conditions: the solution at one end is always identical to the one at the other end of the domain.

Results

The following plot shows how solitons collide and reappear with their shape intact.

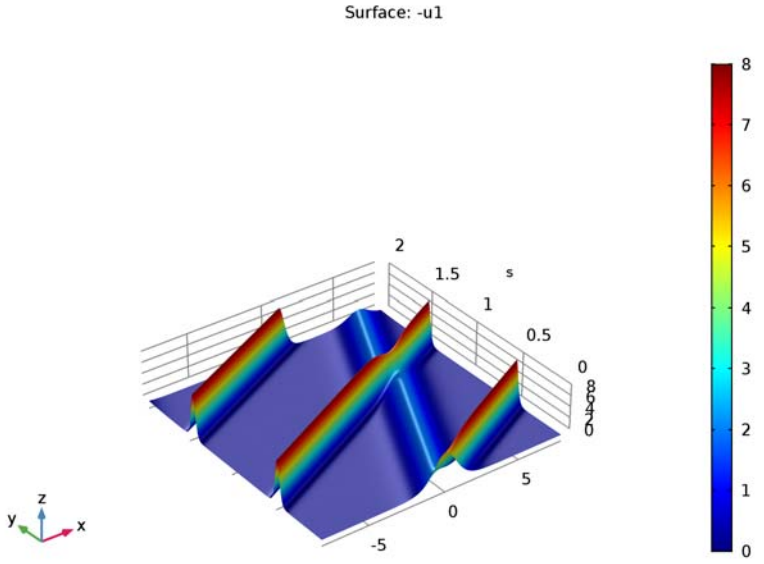


Figure 1: Solution visualizing a soliton collision.

References

1. A.R. Osborne and T.L. Burch, "Internal Solitons in the Andaman Sea," *Science*, vol. 208, no. 4443, pp. 451–460, 1980.

2. R.B. Perry and G.R. Schimke, "Large-Amplitude Internal Waves Observed Off the Northwest Coast of Sumatra," *J. Geophys. Res.*, vol. 70, no. 10, pp. 2319–2324, 1965.

3. G. Strang, *Applied Mathematics*, Wellesley-Cambridge, 1986.

Application Library path: `COMSOL_Multiphysics/Equation_Based/kdv_equation`

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 **1D**.

2 In the **Select Physics** tree, select **Mathematics>PDE Interfaces>General Form PDE (g)**.

3 Click **Add**.

4 In the **Number of dependent variables** text field, type 2.

5 Click **Study**.

6 In the **Select Study** tree, select **Preset Studies>Time Dependent**.

7 Click **Done**.

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**.

Keeping track of units is not important in this model; by turning off unit support, you avoid the need to specify dimensions for equation coefficients and coordinates to get rid of unit warnings.

GEOMETRY 1

Interval 1 (i1)

1 On the **Geometry** toolbar, click **Interval**.

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

3 In the **Left endpoint** text field, type -8.

4 In the **Right endpoint** text field, type 8.

GENERAL FORM PDE (G)

Periodic Condition 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Periodic Condition**.

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

3 From the **Selection** list, choose **All boundaries**.

General Form PDE 1

1 In the **Model Builder** window, under **Component 1 (comp1)>General Form PDE (g)** click **General Form PDE 1**.

2 In the **Settings** window for **General Form PDE**, locate the **Conservative Flux** section.

3 In the Γ text-field array, type `u2` on the first row.

4 In the Γ text-field array, type `u1x` on the second row.

5 Locate the **Source Term** section. In the *f* text-field array, type `6*u1*u1x` on the first row.

6 In the *f* text-field array, type `u2` on the second row.

7 Locate the **Damping or Mass Coefficient** section. In the d_a text-field array, type `0` in the second column of the second row.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>General Form PDE (g)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the $u1$ text field, type `-6*sech(x)^2`.

4 In the $u2$ text field, type `-24*sech(x)^2*tanh(x)^2+12*sech(x)^2*(1-tanh(x)^2)`.

MESH 1

Size

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

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.1`.

5 Click **Build All**.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.

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

3 In the **Times** text field, type `range(0,0.0025,2)`.

4 From the **Tolerance** list, choose **User controlled**.

5 In the **Relative tolerance** text field, type `3e-6`.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

3 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

4 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time stepping** section.

5 Locate the **Time Stepping** section. From the **Method** list, choose **Generalized alpha**.

The **Generalized alpha** time stepper is well suited for wave problems. For an accurate solution, use tighter tolerance settings.

6 Click to expand the **Absolute tolerance** section. Locate the **Absolute Tolerance** section. From the **Tolerance method** list, choose **Manual**.

7 In the **Absolute tolerance** text field, type `3e-7`.

8 Click to expand the **Time stepping** section. Locate the **Time Stepping** section. In the **Amplification for high frequency** text field, type `0.98`.

9 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Time-Dependent Solver 1** node, then click **Fully Coupled 1**.

10 In the **Settings** window for **Fully Coupled**, click to expand the **Method and termination** section.

11 Locate the **Method and Termination** section. In the **Tolerance factor** text field, type `0.1`.

12 On the **Study** toolbar, click **Compute**.

RESULTS

1D Plot Group 1

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

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

3 From the **Time selection** list, choose **From list**.

4 In the **Times (s)** list, select **0.025**.

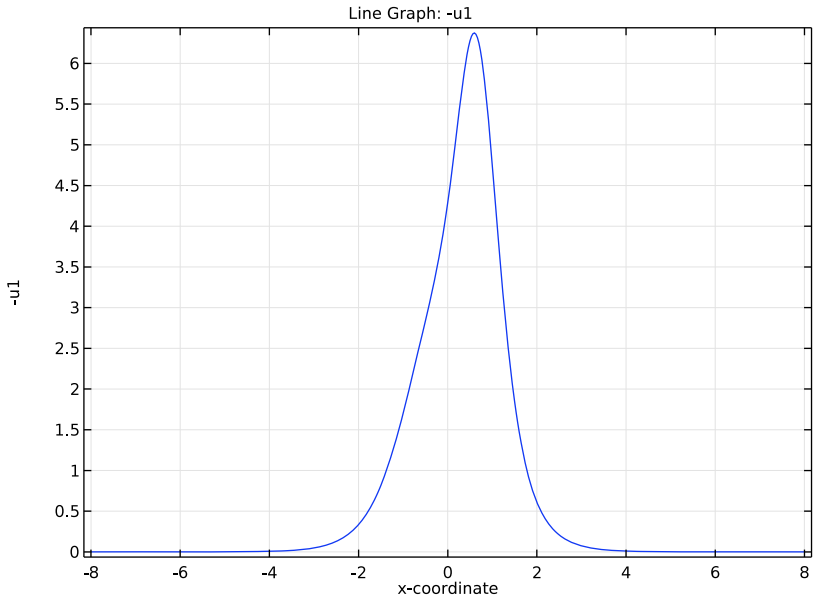
Line Graph 1

1 In the **Model Builder** window, expand the **1D Plot Group 1** node, then click **Line Graph 1**.

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

3 In the **Expression** text field, type `-u1`.

4 On the **1D Plot Group 1** toolbar, click **Plot**.



The solution to the KdV equation at 0.25 s.

Data Sets

To visualize the solution, extrude results along the time.

2D Plot Group 2

On the **Results** toolbar, click **More Data Sets** and choose **Parametric Extrusion 1D**.

Surface 1

1 On the **Results** toolbar, click **2D Plot Group**.

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

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

4 In the **Expression** text field, type `-u1`.

Height Expression 1

1 Right-click **Results>2D Plot Group 2>Surface 1** and choose **Height Expression**.

2 Click the **Zoom Extents** button on the **Graphics** toolbar.

3 In the **Settings** window for **Height Expression**, locate the **Axis** section.

4 Select the **Scale factor** check box.

5 In the associated text field, type `0.2`.

Compare with the plot shown in Figure 1.

Laser Heating of a Silicon Wafer

Introduction

A silicon wafer is heated up by a laser that moves radially in and out over time while the wafer itself rotates on its stage. Modeling the incident heat flux from the laser as a spatially distributed heat source on the surface, the transient thermal response of the wafer is obtained. The average, maximum, and minimum temperatures, as well as the peak temperature difference across the wafer, are stored at every calculation step. The temperature distribution across the entire wafer is stored at a specified number of output timesteps.

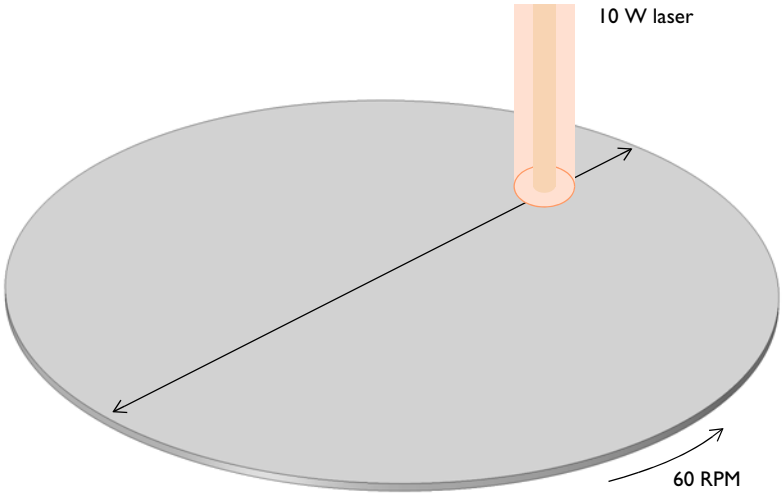


Figure 1: A silicon wafer is heated with a laser that moves back and forth. The wafer is also being rotated about its axis.

Model Definition

A 2-inch silicon wafer, as shown in Figure 1, is heated for one minute by a 10 W laser that moves radially inwards and outwards, while the wafer rotates on its stage. Assuming good thermal isolation from the environment, the only source of heat loss is from the top surface via radiation to the processing chamber walls, which are assumed to be at a fixed temperature of 20 °C.

The laser beam is modeled as a heat source in the plane with Gaussian profile. To set up the heating profile the model uses the built-in Gaussian Pulse functions, which enforce that the integral under the curve equals unity. The focal point is moved by using a triangular waveform to define its position along the x-axis over time. The wafer is assigned a bulk rotational velocity in the governing heat transfer equation.

The emissivity of the surface of the wafer is approximately 0.8. At the operating wavelength of the laser, it is assumed that absorptivity equals emissivity. The heat load due to the laser is thus multiplied by the emissivity. Assuming also that the laser is operating at a wavelength at which the wafer is opaque, no light is passing through the wafer. Therefore, all of the laser heat is deposited at the surface.

The wafer is meshed using a triangular swept mesh. Swept meshing allows for only a single thin element through the thickness, and still maintains reasonable element size in the plane. A finer mesh and tighter solver tolerances would give slightly more accurate predictions of the peak temperature, but predictions of average and minimum temperature would not be greatly affected.

Results and Discussion

Figure 2 shows the probe plots of the maximum, minimum, and average temperatures of the wafer, while Figure 3 shows the probe plot of the difference between the maximum and minimum temperature. The temperature distribution across the wafer is plotted in Figure 4.

The heating profile does introduce some significant temperature variations, because the laser deposits the same amount of heat over a larger total swept area when it is focused at the outside of the wafer. An interesting modification to this example would be to investigate alternative heating profiles for smoother heating.

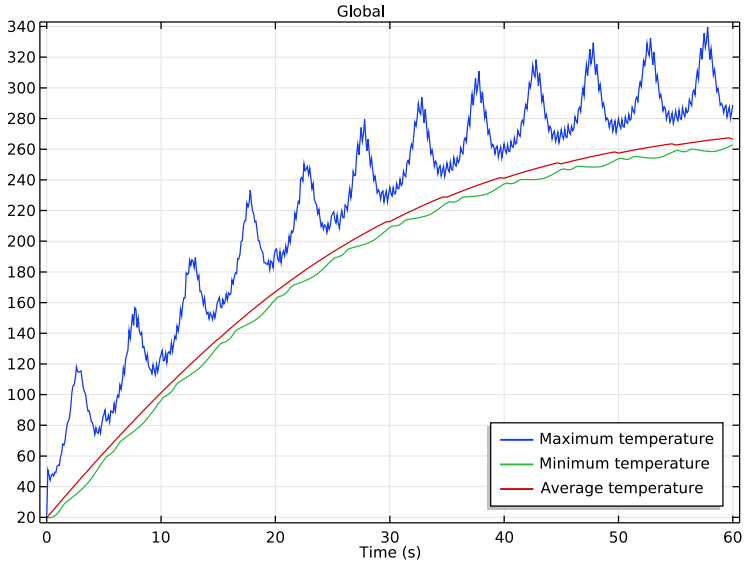


Figure 2: Maximum, minimum, and average temperatures of the wafer as functions of time.

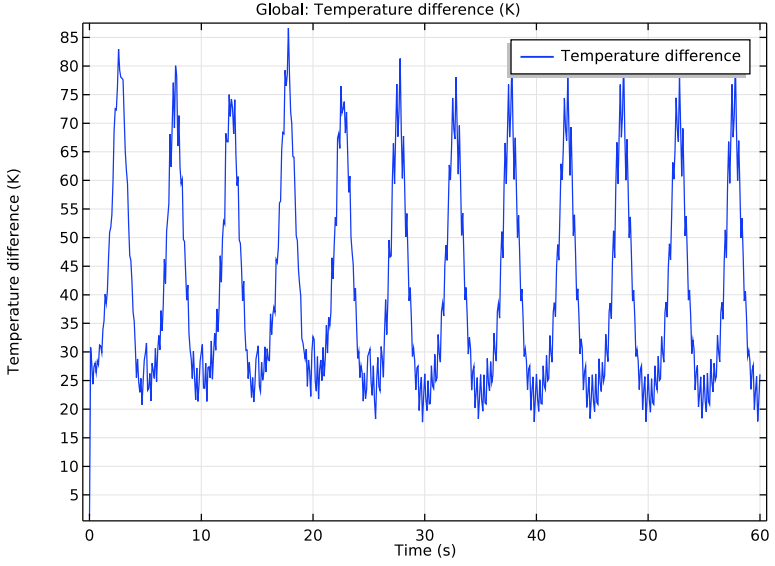


Figure 3: Difference between maximum and minimum temperatures on the wafer.

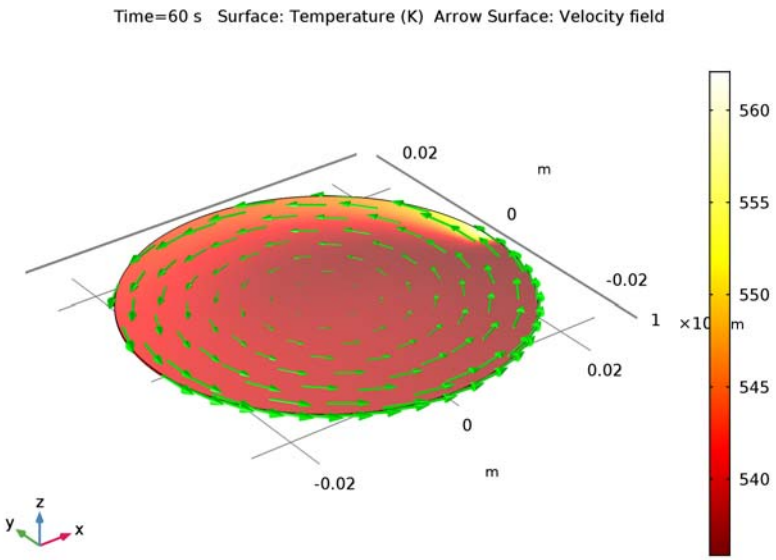


Figure 4: Temperature variation across the wafer. The arrow plot describes the velocity of the wafer.

Application Library path: `COMSOL_Multiphysics/Heat_Transfer/laser_heating_wafer`

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 **Heat Transfer>Heat Transfer in Solids (ht)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Time Dependent**.

6 Click **Done**.

GLOBAL DEFINITIONS

Start by defining parameters for use in the geometry, functions, and physics settings.

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
r_wafer	1[in]	0.0254 m	Wafer radius
thickness	275[um]	2.75E-4 m	Wafer thickness
v_rotation	60[rpm]	1 1/s	Rotational speed
angular_v	2*pi*v_rotation	6.2832 1/s	Angular velocity
period	10[s]	10 s	Time for laser to move back and forth
r_spot	2[mm]	0.002 m	Laser beam radius
emissivity	0.8	0.8	Surface emissivity of wafer
p_laser	10[W]	10 W	Laser power

Here, the unit 'rpm' is revolution per minute.

GEOMETRY 1

Create a cylinder for the silicon wafer.

Cylinder 1 (cyl1)

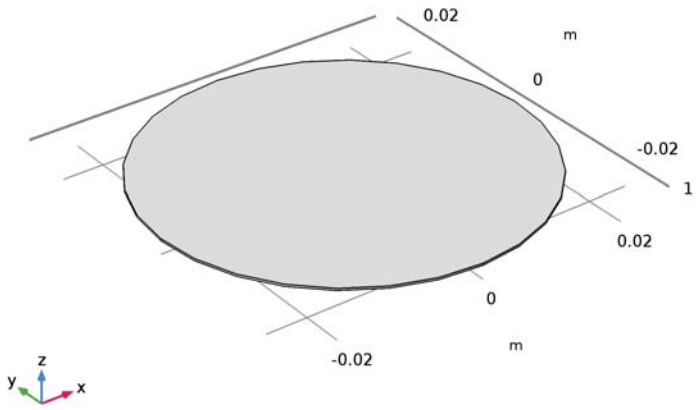
1 On the **Geometry** toolbar, click **Cylinder**.

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

3 In the **Radius** text field, type r_wafer.

4 In the **Height** text field, type thickness.

5 Click **Build All Objects**.



DEFINITIONS

Define functions for use before setting up the physics.

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
x_focus	r_wafer*Triangle(t/period)	m	x-location of laser focal point
y_focus	0[m]	m	y_location of laser focal point
r_focus	sqrt((x-x_focus)^2+(y-y_focus)^2)	m	distance from focal point
Flux	((2*p_laser)/(pi*r_spot^2))*exp(-(2*r_focus^2)/r_spot^2)	W/m²	laser heat flux, Gaussian profile

Waveform 1 (wv1)

1 On the **Home** toolbar, click **Functions** and choose **Local>Waveform**.

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

3 Locate the **Parameters** section. From the **Type** list, choose **Triangle**.

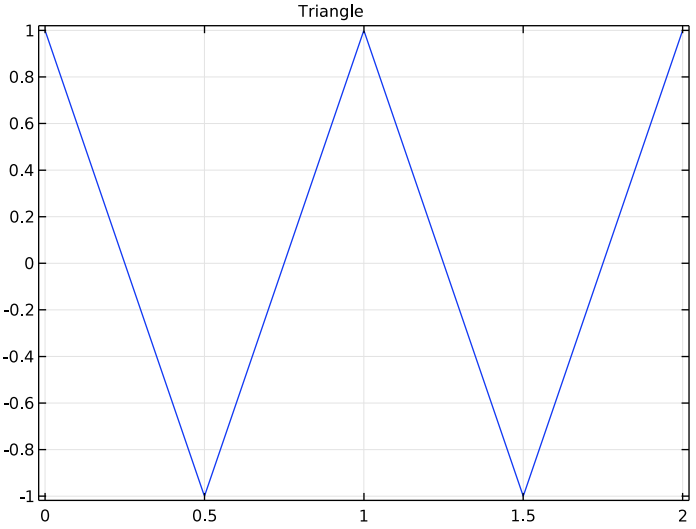
4 Clear the **Smoothing** check box.

5 In the **Angular frequency** text field, type `2*pi`.

6 In the **Phase** text field, type `pi/2`.

7 In the **Amplitude** text field, type `1`.

8 Click **Plot**.



Domain Probe 1 (dom1)

1 On the **Definitions** toolbar, click **Probes** and choose **Domain Probe**.

2 In the **Settings** window for **Domain Probe**, type `T_average` in the **Variable name** text field.

Domain Probe 2 (dom2)

1 On the **Definitions** toolbar, click **Probes** and choose **Domain Probe**.

2 In the **Settings** window for **Domain Probe**, type `T_max` in the **Variable name** text field.

3 Locate the **Probe Type** section. From the **Type** list, choose **Maximum**.

Domain Probe 3 (dom3)

1 On the **Definitions** toolbar, click **Probes** and choose **Domain Probe**.

2 In the **Settings** window for **Domain Probe**, type `T_min` in the **Variable name** text field.

3 Locate the **Probe Type** section. From the **Type** list, choose **Minimum**.

Global Variable Probe 1 (var1)

1 On the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.

2 In the **Settings** window for **Global Variable Probe**, type `T_diff` in the **Variable name** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `T_max-T_min`.

4 In the **Table and plot unit** field, type `K`.

HEAT TRANSFER IN SOLIDS (HT)

Set up the physics. First, include the wafer's rotational velocity in the governing heat transfer equation.

Solid 1

In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids (ht)** click **Solid 1**.

Translational Motion 1

1 On the **Physics** toolbar, click **Attributes** and choose **Translational Motion**.

2 In the **Settings** window for **Translational Motion**, locate the **Translational Motion** section.

3 Specify the $\mathbf{u}_{\text{trans}}$ vector as

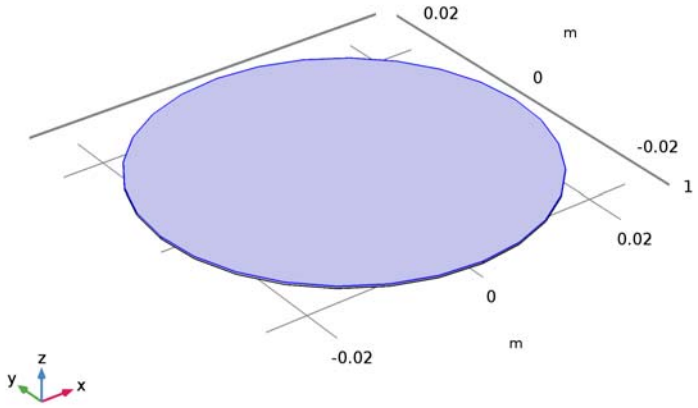
`-y*angular_v`	`x`
`x*angular_v`	`y`
`0`	`z`

Next, add heat flux and surface-to-ambient radiation on the wafer's top surface.

Heat Flux 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 Select Boundary 4 only.



3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

4 In the q_0 text field, type `emissivity*Flux`.

Diffuse Surface 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Diffuse Surface**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Diffuse Surface**, locate the **Surface Emissivity** section.

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

ADD MATERIAL

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

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

3 In the tree, select **Built-In>Silicon**.

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

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

MESH 1

Use a fine triangular swept mesh.

Swept 1

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

2 In the **Settings** window for **Swept**, click to expand the **Sweep method** section.

3 Locate the **Sweep Method** section. From the **Face meshing method** list, choose **Triangular (generate prisms)**.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>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 1.

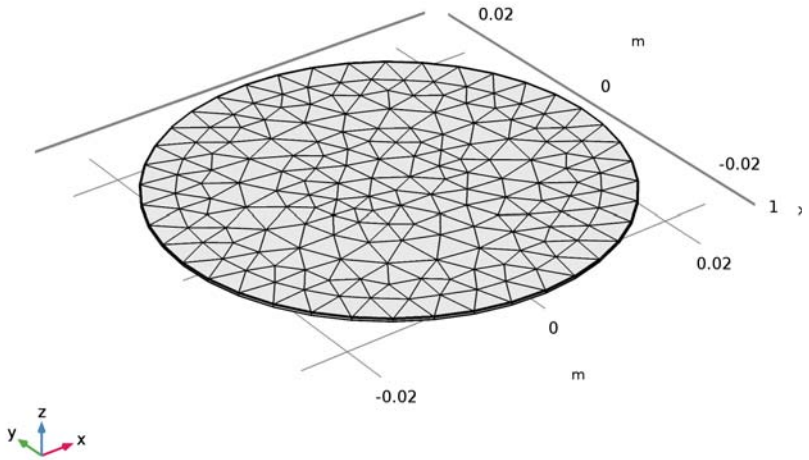
Size

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

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Fine**.

4 Click **Build All**.



STUDY 1

Step 1: Time Dependent

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

2 In the **Times** text field, type `range(0,1.25,60)`.

3 On the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht)

The first default plot shows the temperature on the wafer surface. Add an arrow plot for the velocity field describing the wafer rotation.

Arrow Surface 1

1 In the **Model Builder** window, under **Results** right-click **Temperature (ht)** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1> Heat Transfer in Solids>ht.ux,ht.uy,ht.uz - Velocity field**.

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

4 In the associated text field, type `0.036`.

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

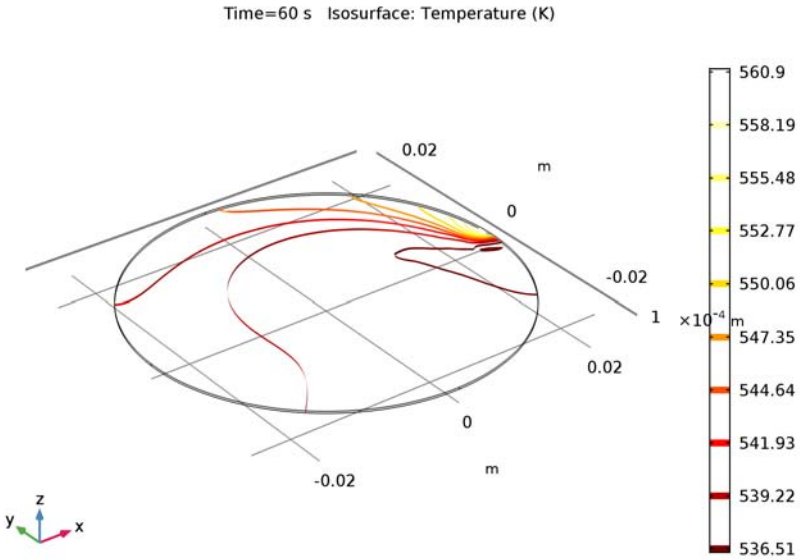
6 On the **Temperature (ht)** toolbar, click **Plot**.

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

Compare the temperature variation with that shown in Figure 4.

Isothermal Contours (ht)

The second default plot shows the isosurface temperature.



Probe Table Graph 1

1. In the **Model Builder** window, expand the **Probe Plot Group 3** node, then click **Probe Table Graph 1**.
2. In the **Settings** window for **Table Graph**, locate the **Data** section.
3. In the **Columns** list, choose **Temperature (K), Domain Probe 1**, **Temperature (K), Domain Probe 2**, and **Temperature (K), Domain Probe 3**.
4. On the **Probe Plot Group 3** toolbar, click **Plot**.

1D Plot Group 4

1. On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Probe Plot Group 4` in the **Label** text field.

Table Graph 1

1. Right-click **Probe Plot Group 4** and choose **Table Graph**.
2. In the **Settings** window for **Table Graph**, type `Probe Table Graph 1` in the **Label** text field.
3. Locate the **Data** section. From the **Plot columns** list, choose **Manual**.

4 In the **Columns** list, select **T_max-T_min (K)**.

5 On the **Probe Plot Group 4** toolbar, click **Plot**.

Loaded Spring — Using Global Equations to Satisfy Constraints

Introduction

In this tutorial example, which demonstrates a more generally applicable method, a structural mechanics model of a spring is augmented by a global equation that solves for the load required to achieve a desired total extension of the spring.

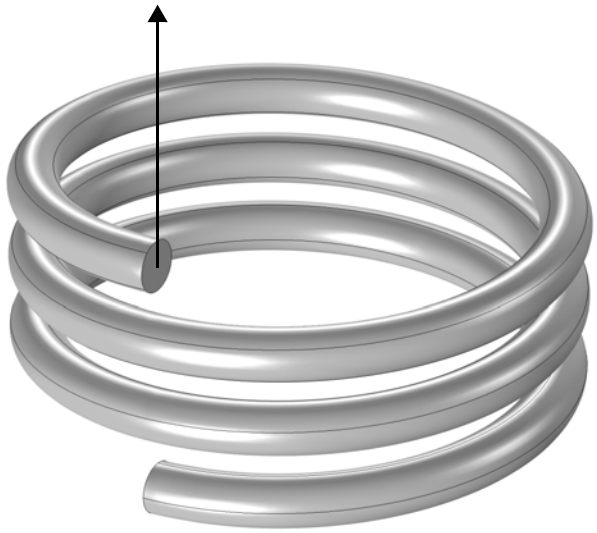


Figure 1: A three-turn steel spring is fixed at one end, and has a load applied at the other. The load is a variable which is solved for to achieve a total displacement.

Model Definition

Figure 1 shows the modeled three-turn steel spring. One end of the spring is fixed rigidly, and the other end has a distributed load applied to it, acting in the axial direction of the spring. Rather than an input to the model, this load is a variable being solved for; it is implicitly specified via a global equation in such a way as to give a total spring extension of 2 cm. The extension of the spring is computed by using an average operator on the moving end of the spring. The average operator evaluates the average z-displacement over the boundary at which the load is applied.

The global equation adds one additional degree of freedom to the model, the unknown load. Not all available equations solvers are suited for such problems, but the direct solver

used as default for structural mechanics can handle it. Because the structure has a uniform cross section, use a swept mesh.

Results and Discussion

Figure 2 shows the deformed shape of the spring. The average displacement of the end of the spring is 2 cm, as specified by the global equation. The force required to get this displacement is 705 N. Although this problem uses a linear elastic material model, this approach would work equally well if the material model was nonlinear or if geometric nonlinearity was taken into account.

Global equations do have certain restrictions upon their usage. The global equation must be continuous and differentiable with respect to all of the unknowns, and it must not over-constrain, nor under-constrain, the problem. Each global equation should add one constraint and one degree of freedom to the model. Under these conditions, the global equations can be used in a variety of ways beyond what is shown here.

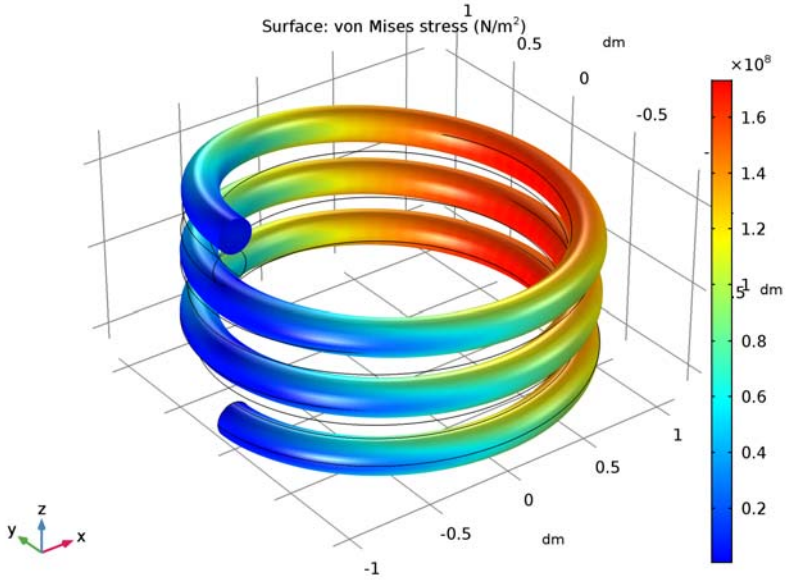


Figure 2: The deformed shape of the spring.

Application Library path: `COMSOL_Multiphysics/Structural_Mechanics/loaded_spring`

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 **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
dh	2[cm]	0.02 m	Prescribed extension

GEOMETRY 1

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

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **dm**.

Helix 1 (hel1)

Create a helix for the spring (Figure 1).

1 On the **Geometry** toolbar, click **Helix**.

2 In the **Settings** window for **Helix**, locate the **Rotation Angle** section.

3 In the **Rotation** text field, type 180.

4 Click **Build All Objects**.

DEFINITIONS

Next, add an **Average** operator that you will later use to average the z-directional displacement field on the end of the spring.

Average 1 (aveop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.

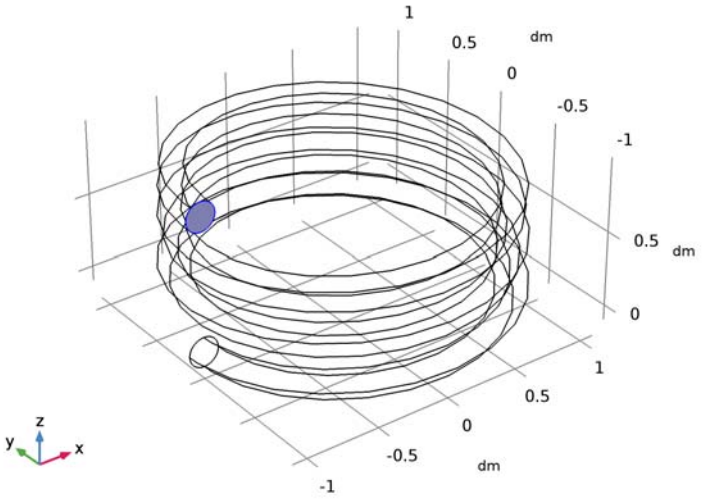
Choose wireframe rendering to get a better view on some boundaries where you will assign boundary conditions.

2 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

3 In the **Settings** window for **Average**, locate the **Source Selection** section.

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

5 Select Boundary 4 only.



SOLID MECHANICS (SOLID)

Next, set up the physics. Add a global equation to compute the appropriate load for the prescribed extension. As an advanced feature, the **Global Equations** entry is not available by default in the context menu.

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

Global Equations 1

1 On the **Physics** toolbar, click **Global** and choose **Global Equations**.

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

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)
Force	aveop1(w)-dh	0	0

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, In the associated text field, type `id:force`.

6 Click **Filter**.

7 In the tree, select **Solid>Force load (N)**.

8 Click **OK**.

9 In the **Settings** window for **Global Equations**, locate the **Units** section.

10 Click **Select Source Term Quantity**.

11 In the **Physical Quantity** dialog box, In the associated text field, type `id:displacement`.

12 Click **Filter**.

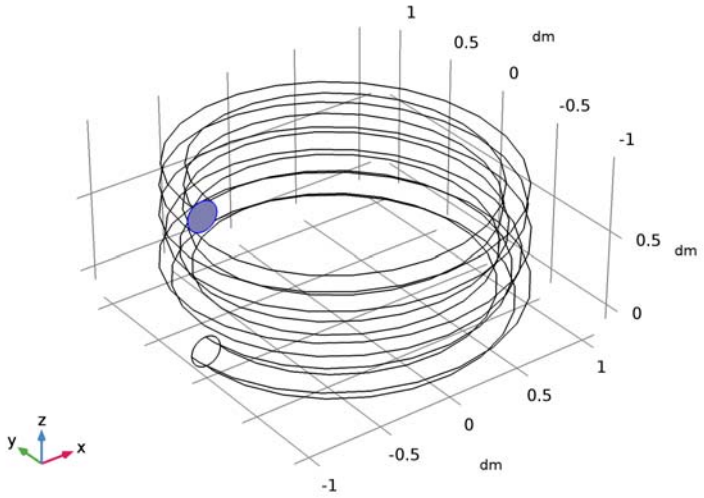
13 In the tree, select **Solid>Displacement field (m)**.

14 Click **OK**.

Boundary Load 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundary 4 only.



3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 From the **Load type** list, choose **Total force**.

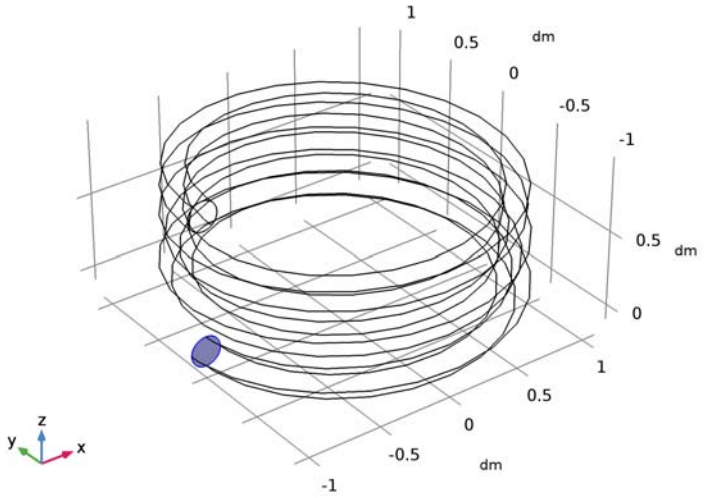
5 Specify the $\mathbf{F}_{\text{tot}}$ vector as

0	x
0	y
Force	z

Fixed Constraint 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Select Boundary 3 only.



MATERIALS

Assign material properties. Use **Steel AISI 4340** for all domains.

ADD MATERIAL

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

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

3 In the tree, select **Built-In>Steel AISI 4340**.

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

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

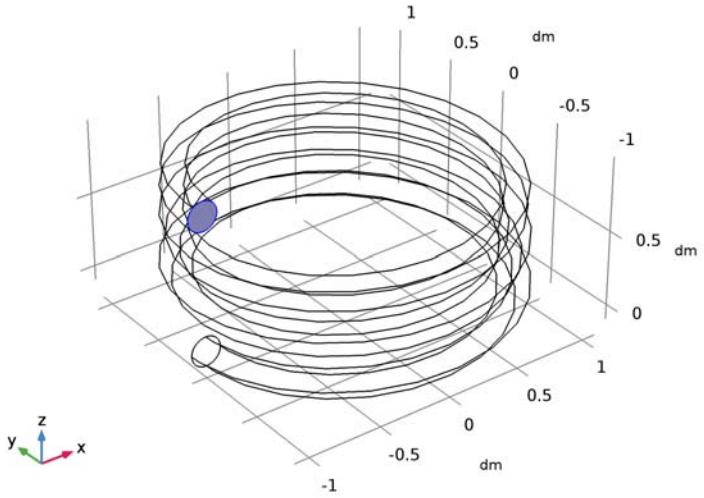
MESH 1

Use swept mesh to generate a uniform mesh over the spring domain. Start by specifying the mesh on one end face of the spring.

Free Triangular 1

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

2 Select Boundary 4 only.



Size

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

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Coarser**.

Distribution 1

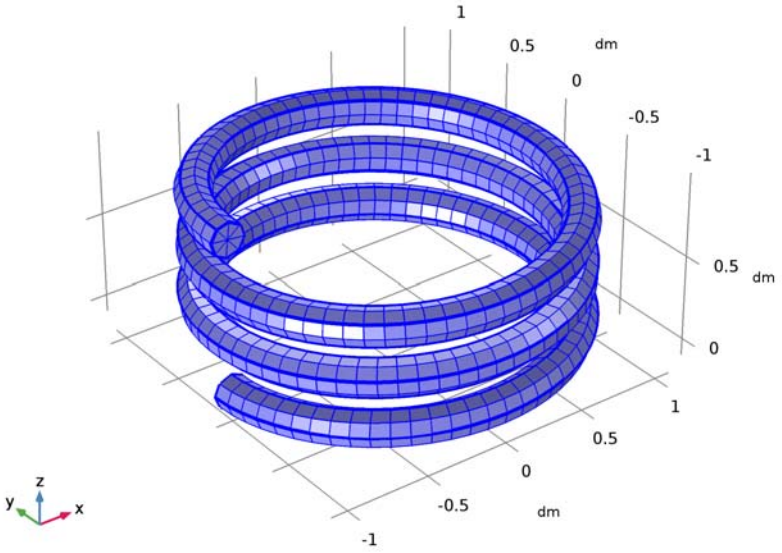
1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

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

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

4 In the **Number of elements** text field, type 200.

5 Click **Build All**.



STUDY 1

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

RESULTS

Stress (solid)

The default plot shows the von Mises stress on the surface of the spring. Compare the plot with Figure 2.

Derived Values

Evaluate the force required to get the displacement specified in the global equations.

Global Evaluation 1

1 On 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
`Force`	`N`	`State variable Force`

4 Click **Evaluate**.

Derived Values

Finish the result analysis by evaluating the average displacement of the end of the spring.

Global Evaluation 2

1 On 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
aveop1(w)	dm	Average 1

4 Click **Evaluate**.

Stiffness Analysis of a Communication Mast's Diagonal Mounting

Introduction

Communication masts usually have a framework with a bolted triangular lattice design as illustrated in Figure 1. The diagonals of the framework are assembled from several parts and welded together.

When operating under a given wind load at a specific location, the antenna's total rotation angle should stay below a certain limit to ensure uninterrupted communications. For the type of mast used in this example, the engineers have determined that its torsional stiffness is too low, and this effect is due to the geometry of the diagonal mountings. The goal is to increase the stiffness of such a diagonal mounting by first analyzing a parameterized geometry followed by an update of the geometry and a new analysis.

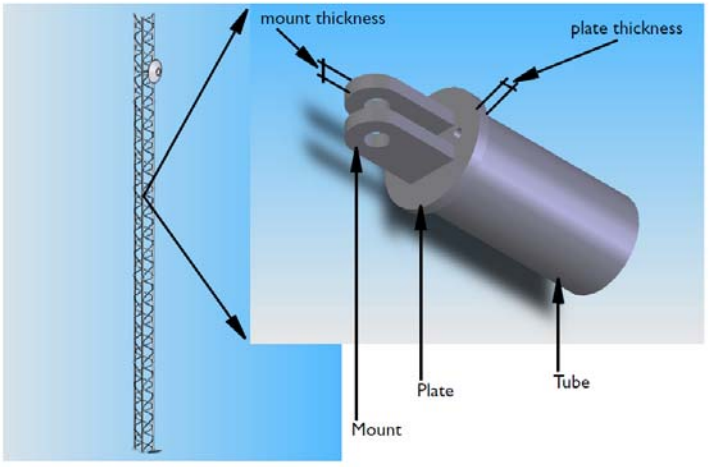


Figure 1: Mounting detail of a mast diagonal.

Model Definition

The model geometry includes only a short section of the diagonal tubing together with the other parts of the mounting as illustrated in Figure 1. Although a symmetry exists in both the geometry and load for this problem, this example models the entire assembly for illustrative purposes.

After obtaining the stiffness of the diagonal mounting, assume that the geometry has been updated to improve the stiffness. Originally 10 mm, the plate thickness and mount thickness (see Figure 1) have been changed to 12 mm and 15 mm, respectively.

MATERIAL PROPERTIES

Assume that the material is a structural steel. This is the default material with a Young's modulus of $2.0 \cdot 10^{11}$ N/m^2 and a Poisson's ratio of 0.33.

BOUNDARY CONDITIONS

Figure 2 shows the boundaries with an applied load and constrained displacements. Assume that the diagonal is loaded in tension by a force, F = 30 kN, which is transferred through the bolt to the mounting.

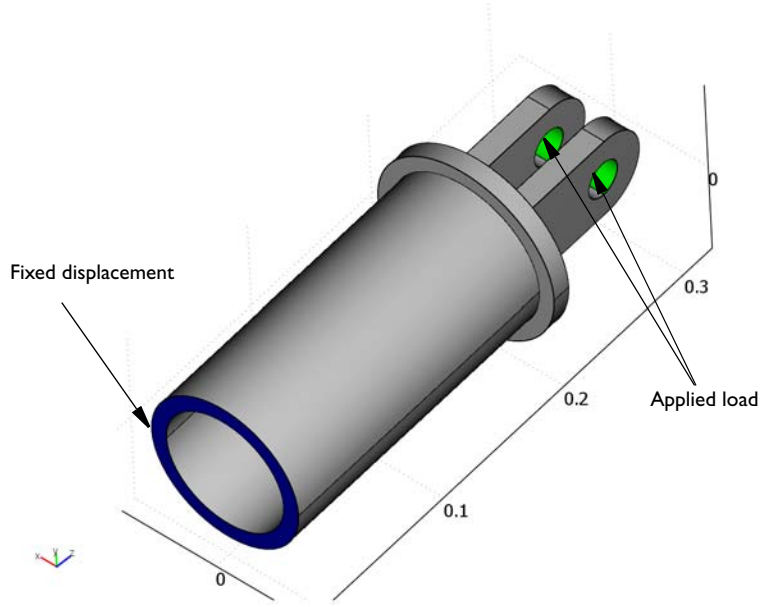


Figure 2: Boundaries with constrained displacements and applied loads.

Neglect contact conditions between the bolt and the mounting hole, and also neglect the constraint imposed on the mount by the bolt. Assume that the bolt fills out the entire hole volume. The load is distributed on the appropriate halves of the hole surfaces by allying a pressure, p, according to

$$p = \frac{F}{2 \cdot A_{\mathrm{mh}}^{\mathrm{xy}}} \cdot \frac{3}{2}\left(1 - \left(\frac{y}{r_{\mathrm{mh}}}\right)^2\right)$$

$$A_{\mathrm{mh}}^{\mathrm{xy}} = 2 \cdot r_{\mathrm{mh}} t_{\mathrm{m}}$$

where r_{mh} is the mount hole radius, t_{m} is the thickness of the mount, y is the y-coordinate and $A_{\mathrm{mh}}^{\mathrm{xy}}$ is the mount hole cross section area projected on the xy-plane.

In current analysis design engineers want to increase the stiffness of the assembly. Since the load is transferred through the mount holes it is crucial to compute the displacement of the mount holes once the external load is applied. By denoting the average z-displacement of the middle plane of the mount holes by δ_{mh}, the real stiffness of the assembly is given by

$$S = \frac{F}{\delta_{\mathrm{mh}}}$$

In an assembly with a constant cross section the relation between the applied force and resulting displacement is given by

$$\frac{F}{A} = E\frac{\delta}{L}$$

where δ is the displacement, A is the cross section area and L is the total length. This relation can be rearrange to

$$\frac{F}{\delta} = E\frac{A}{L}$$

Since the stiffness is defined as applied force divided by resulting displacement, which is the left hand side in the above expression, the right hand side can be seen as an ideal stiffness, S_{I}, of a body with a constant cross section. In current example that would be a tube stretching as far as the mounting hole, to be welded together with the rest of the mast. The ideal stiffness can thus be expressed with

$$S_{\mathrm{I}} = E\frac{A_{\mathrm{t}}}{L_{\mathrm{t}}}$$

$$L_{\mathrm{t}} = h_{\mathrm{t}} + t_{\mathrm{p}} + o_{\mathrm{mh}}$$

where L_{t} is the equivalent tube length if the entire assembly was made of a tube only, h_{t} is the tube height in the assembly with a mount, t_{p} is the plate thickness and o_{mh} is the mount hole center offset, measured from the plate surface. Observe that the tube height and the equivalent tube length are both measured along the same dimension of the tube. The difference is the fact that tube height is used to relate the tube used in the mount assembly and tube length is used to define the tube in an assembly consisting of a tube only.

The variable used as the evaluation parameter is the stiffness ratio between the real stiffness of the assembly with a mount and the ideal stiffness.

$$S_R = \frac{S}{S_I}$$

If this value equals one, then the stiffens of the assembly with a mount is exactly the same as the stiffens of a single tube.

Results and Discussion

In the original geometry where both the plate thickness and the mount thickness are 10 mm, the stiffness ratio is 0.38. Then the plate thickness is increased to 12 mm and the mount thickness to 15 mm, the stiffness ratio increases to 0.53. Figure 3 shows the z-component of the displacement when loading the stiffer geometry.

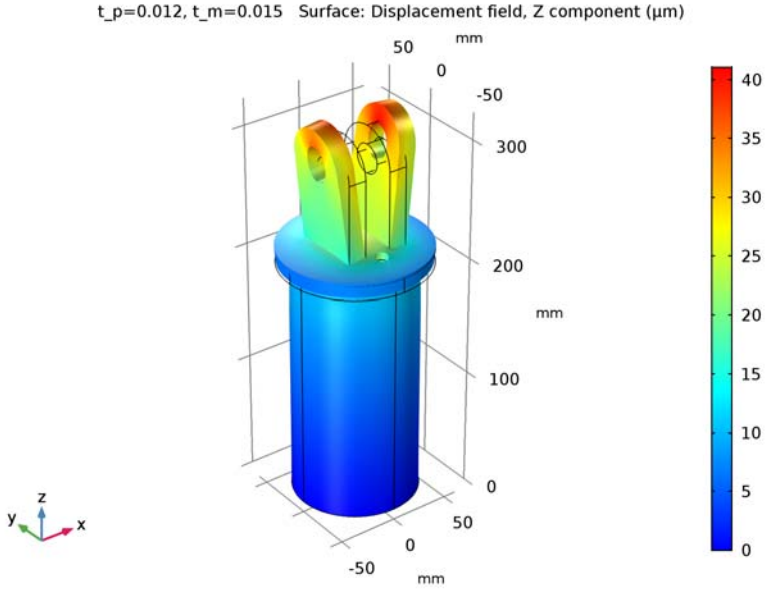


Figure 3: Deformed shape and boundary plot of the axial displacement for the mounting assembly with an end-plate thickness of 12 mm and mount thickness of 15 mm.

Application Library path: `COMSOL_Multiphysics/Structural_Mechanics/mast_diagonal_mounting`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

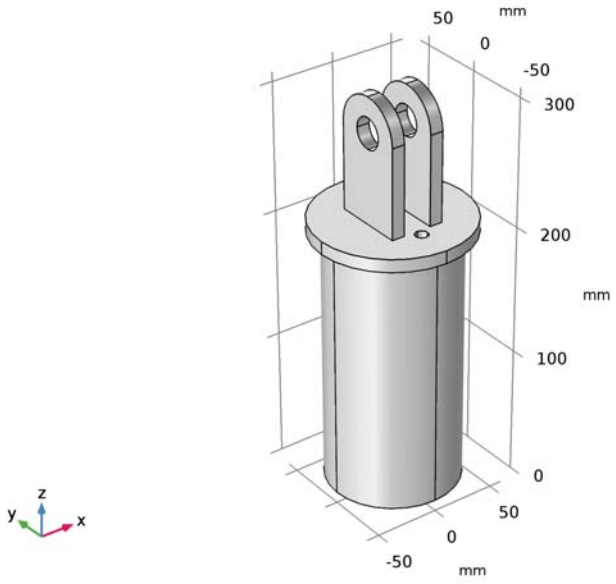
1. In the **Model Wizard** window, click **3D**.
2. In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Stationary**.
6. Click **Done**.

GEOMETRY 1

The model geometry is available as a parameterized geometry sequence in a separate MPH-file.

1. On the **Geometry** toolbar, click **Insert Sequence**.
2. Browse to the model's Application Libraries folder and double-click the file `mast_diagonal_mounting_geom_sequence.mph`.
3. On the **Geometry** toolbar, click **Build All**.

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



GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
F	30[kN]	30000 N	Applied force

DEFINITIONS

Create an average operator for evaluation of variables across the mid plane of both mount holes.

Average 1 (aveop1)

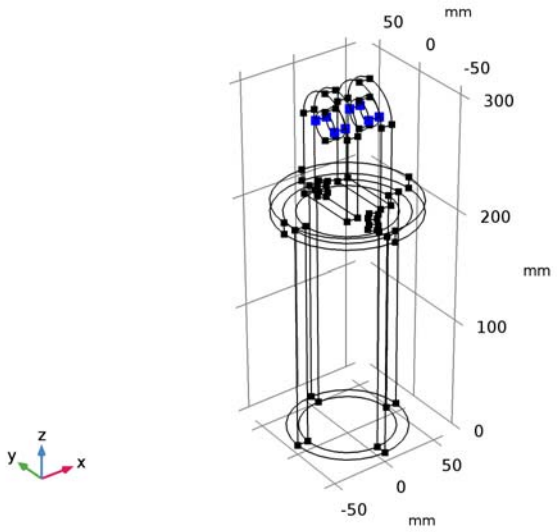
1 On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.

2 In the **Settings** window for **Average**, type `Mount, mid level` in the **Label** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Point**.

4 Select Points 9, 13, 18, 22, 55, 59, 64, and 68 only.

The points along the mid plane are shown in figure below.



5 Click **Create Selection**.

6 In the **Create Selection** dialog box, type `Mount, mid level` in the **Selection name** text field.

7 Click **OK**.

Variables 1

1 On the **Definitions** toolbar, click **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
p	`-(3/2)*F/(2*Axy_mh)*(1-(Y/r_mh)^2)`	N/m²	Mount hole stress
fy_mh	p*nY	N/m²	Mount hole force, y component
fz_mh	p*nZ	N/m²	Mount hole force, z component
d_mh	aveop1(w)	m	Mount hole displacement
L_t	h_t+t_p+o_mh	m	Equivalent tube length

Name	Expression	Unit	Description
Axy_mh	2*r_mh*t_m	m²	Mount hole xy projected area
A_t	pi*(r_t^2-(r_t-t_t)^2)	m²	Tube cross section area
S	F/d_mh	N/m	Stiffness of the assembly
S_i	200e9[Pa]*A_t/L_t	N/m	Ideal stiffness
S_R	S/S_i		Stiffness ratio

ADD MATERIAL

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

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

3 In the tree, select **Built-In>Structural steel**.

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

MATERIALS

Structural steel (mat1)

By default, the first material you add applies on all domains so you need not alter any settings.

1 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

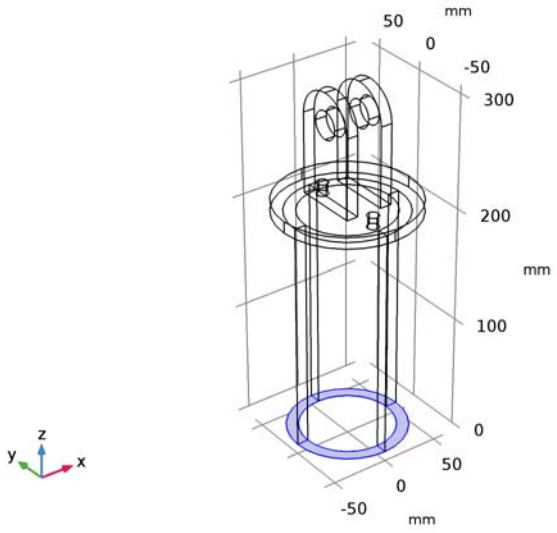
By default, the physics interface takes the required material model properties from the domain material.

Next, define the boundary conditions.

Fixed Constraint 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

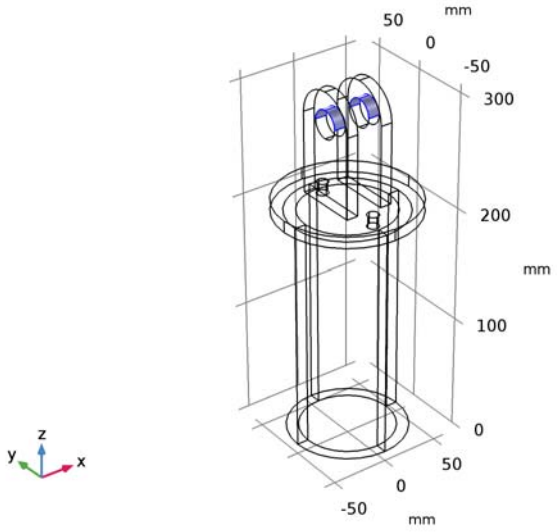
2 Select Boundaries 8, 9, 33, and 42 only.



Boundary Load 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundaries 20, 22, 51, and 53 only.



Use the force components you defined earlier.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

0	x
fy_mh	y
fz_mh	z

MESH 1

This section illustrates how you can mesh different parts of the model individually to get a suitable mesh.

Free Tetrahedral 1

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

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

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

4 From the **Selection** list, choose **Manual**.

5 Select Domains 4 and 7 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Tetrahedral 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Fine**.

Free Tetrahedral 1

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

2 In the **Settings** window for **Free Tetrahedral**, click **Build Selected**.

Free Tetrahedral 2

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Tetrahedral**.

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

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

4 From the **Selection** list, choose **Manual**.

5 Select Domain 1 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Tetrahedral 2** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Finer**.

Free Tetrahedral 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Tetrahedral 2**.

2 In the **Settings** window for **Free Tetrahedral**, click **Build Selected**.

Swept 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

2 In the **Settings** window for **Swept**, click to expand the **Source faces** section.

3 Locate the **Source Faces** section. Select the **Active** toggle button.

4 Select Boundary 4 only.

5 Click to expand the **Destination faces** section. Locate the **Destination Faces** section. Select the **Active** toggle button.

6 Select Boundary 3 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Swept 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

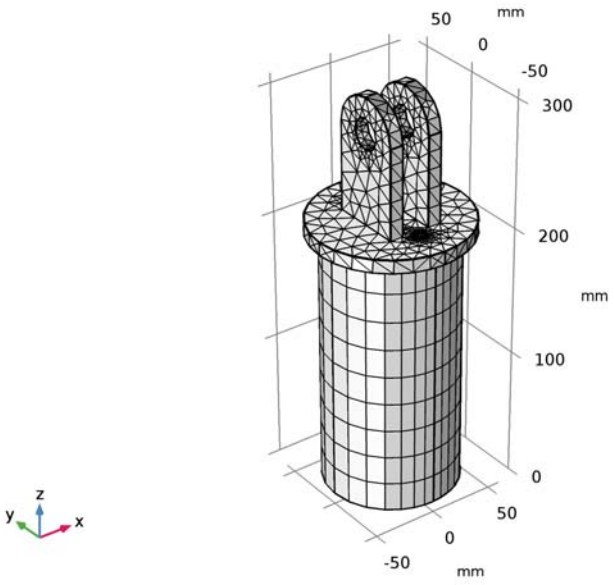
3 From the **Predefined** list, choose **Fine**.

Swept 1

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

2 In the **Settings** window for **Swept**, click **Build Selected**.

3 Click the **Wireframe Rendering** button on the **Graphics** toolbar to restore the default state.



STUDY 1

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

RESULTS

Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

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

Displacement

Now, plot the z-displacement and compare the stiffness ratio for the current model geometry dimensions with the ideal one.

Surface 1

1 In the **Model Builder** window, expand the **Displacement** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics> Displacement>Displacement field (material and geometry frames)>w - Displacement field, Z component**.

3 Locate the **Expression** section. From the **Unit** list, choose **µm**.

4 On the **Displacement** toolbar, click **Plot**.

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

Global Evaluation 1

1 On 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>Definitions> Variables>S_R - Stiffness ratio**.

3 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The stiffness ratio obtained is 0.38, which is less than the desired value.

STUDY 1

Proceed to add a parametric sweep feature that varies the mount thickness and plate thickness.

Parametric Sweep

1 On 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
t_p (Plate thickness)	10[mm] 12[mm]	

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
t_m (Mount thickness)	10[mm] 15[mm]	

7 Locate the **Output While Solving** section. Select the **Plot** check box.

8 On the **Study** toolbar, click **Compute**.

RESULTS

Displacement

To display the results from the parametric sweep, change the data set.

1. In the **Model Builder** window, under **Results** click **Displacement**.
2. In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3. From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

   The default parameter values correspond to those for the sweep's last parameter pair.
4. On the **Displacement** toolbar, click **Plot**.
5. Click the **Zoom Extents** button on the **Graphics** toolbar.

Finally, compare the updated stiffness value for the updated model geometry dimensions with the ideal one.

Global Evaluation 2

1. On the **Results** toolbar, click **Global Evaluation**.
2. In the **Settings** window for **Global Evaluation**, locate the **Data** section.
3. From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.
4. Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Definitions>Variables>S_R - Stiffness ratio**.
5. Click **Evaluate**.

TABLE

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

   Observe that the stiffness ratio obtained now is 0.53.

Sensitivity Analysis of a Communication Mast Detail

Introduction

The example Stiffness Analysis of a Communication Mast's Diagonal Mounting in the COMSOL Multiphysics Application Libraries shows how you can modify a 3D CAD model to improve its performance. In that case, the applied changes were based solely on the analyst's experience with similar structures. A senior design engineer can sometimes reach acceptable performance after analyzing only a handful of designs, while an unexperienced analyst may have to spend a lot of time on failed attempts.

Usually, you can indeed improve a design by trial and error, but it is difficult to ensure that the price you pay — in this example, added weight and material costs — is as low as possible. With sensitivity analysis, you can find the most cost-efficient direction for a small modification and estimate the effect it has before attempting an updated design.

Model Definition

The original model simulates the deformation of a part of a communication mast, shown in Figure 1, under loads in the linear regime. The ratio of the part's effective stiffness to the stiffness of an equal length of straight pipe is evaluated as a measure of its performance. Using sensitivity analysis together with a Deformed Geometry interface, you can predict what effect changing the dimensions of the end plate and the mount plates has on the part's relative stiffness.

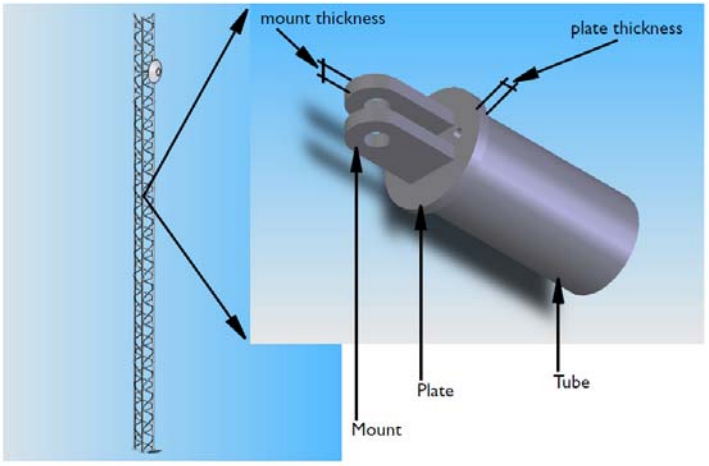


Figure 1: Mounting detail of a mast diagonal.

From the designer's point of view, the material thickness of the end plate, t_p, and of the mount plates, t_m, are the most relevant parameters because they are easy to change both in the CAD system and on the production line. These quantities are, however, not directly comparable to each other because a unit change of t_p incurs a different cost — added weight and material use — compared to a unit change in t_m.

For a fair analysis, it is therefore more convenient to parameterize the model in terms of the masses Δm_p and Δm_m added to the end plate and mount, respectively. The relation between added mass, Δm, and thickness change, Δt, is given by

$$\Delta m = \rho A \Delta t$$

where A is the area affected by the thickness change (m^2) and ρ is the density of the material (kg/m^3).

As output from a sensitivity analysis using the stiffness ratio, S_R, as objective function and the differential masses Δm_{pl} and Δm_{mt} as sensitivity variables, you get the partial derivatives

$$Q_p = \frac{\partial S_R}{\partial m_p} \qquad Q_m = \frac{\partial S_R}{\partial m_m}$$

For a modified geometry corresponding to small values of Δm_p and Δm_m you can therefore expect to see a change in the stiffness ratio equal to

$$\Delta S_R = Q_p \Delta m_p + Q_m \Delta m_m$$

Note that this relation holds only for a small incremental change from the current configuration because the stiffness ratio is clearly a nonlinear function of the thicknesses. Now suppose that you want to select the best possible design update for a given added mass $\Delta m = \Delta m_p + \Delta m_m$ with the added condition that both Δm_p and Δm_m are nonnegative. It is not too difficult to realize that the best option is to take $\Delta m_p = \Delta m$, $m_m = 0$ if $Q_p > Q_m$, and $\Delta m_{pl} = 0$, $m_{mt} = \Delta m$ otherwise.

The optimal stiffness for a given total mass of the structure can be sought by relaxing the nonnegativity condition for the updates and instead restricting the maximum change in Δm_{pl} and Δm_{mt} during one iteration. With the total mass as only constraint, you find the optimum design at a point where $Q_p = Q_m$. This follows strictly from the Karush-Kuhn-Tucker conditions but also from the simple fact that at such a point, the increased stiffness from adding mass to the plate is exactly canceled by the decrease in stiffness from removing the same mass from the mount.

PARAMETERIZING THE GEOMETRY

In the Sensitivity interface, you declare sensitivity variables, which can be used to parameterize the physics. The sensitivity variables can appear anywhere COMSOL Multiphysics accepts an expression containing the dependent variables. However, neither dependent variables nor sensitivity variables can be used directly to set dimensions in the geometry.

To evaluate the sensitivity of a model with respect to geometrical changes, the geometry must first be made an active part of the system of equations. You accomplish this by moving all physics onto a deformed configuration controlled by a Deformed Geometry interface, described in Deformed Geometry and Moving Mesh in the *COMSOL Multiphysics Reference Manual*. This interface sets up an equation governing the position of the mesh nodes inside the domains, while the outer shape of the domain is controlled by boundary conditions.

When doing sensitivity analysis, these boundary conditions are quite simple: on fixed surfaces, set the mesh displacement to zero; on surfaces that may be modified, specify the displacement in terms of the sensitivity variables. In this particular case, where the material thickness of the end plate and the mount can change, it is enough to set the normal displacement of these surfaces equal to the thickness change calculated from the corresponding added mass. For the latter calculation, assume the total undeformed length of the part to be fixed and define Δm_{pl} as the net mass added when t_{pl} increases.

On surfaces adjacent to a domain with a parameterized normal displacement, it is preferable to restrict the mesh displacement to zero only in the normal direction to avoid an inconsistent constraint on the common edge. However, any remaining inconsistencies do not invalidate your results completely but only effectively modify the parametrization. In cases like this, when the purpose of the analysis is a rough estimate and guidance for a manual redesign, such minor errors in the sensitivities are unimportant.

Another potential source of errors must be checked more carefully, though. Changing the material thickness of the mount also changes the area where the loading is applied. In the Solid Mechanics interface, you specify the load as a given force per area. If you keep this number fixed when the thickness of the part changes, the total applied load also changes; when evaluating the stiffness ratio of the composite part you must account for this effect. Alternatively, you can keep the total force fixed and make sure that the applied force per unit area is calculated using a hole area that follows the parametrization of the geometry. This is simply done by changing the expression for the mount hole xy-projected area to

$$A_{\mathrm{mh}}^{\mathrm{xy}} = 2 \cdot r_{\mathrm{mh}}(t_{\mathrm{m}} + \delta t_{\mathrm{m}})$$

where δt_m is the increase in the mount thickness.

CHOOSING FORWARD OR ADJOINT SENSITIVITY ANALYSIS

By default, the **Sensitivity** interface uses the adjoint method, which is more efficient than the forward method when the number of sensitivity variables is large. When there is only a handful of scalar parameters, as in this case, the forward method has the advantage that it returns the sensitivity of the entire solution with respect to the sensitivity variables in addition to the sensitivity of the objective function. This additional information can sometimes be important in itself, but more often it is useful for checking the model setup because it is easy to visualize.

Results and Discussion

The analysis shows that when the thickness of the end plate is 12 mm and the material thickness in the mount is 15 mm, the sensitivities to adding mass to the two details are $Q_p = 0.25$ and $Q_m = 0.23$, respectively. Because, apparently, adding mass to the end plate has more effect than adding it to the mount, the next redesign of the part should be fitted with a thicker plate. You might even consider decreasing the material thickness of the mount while adding to the plate to keep the weight of the part constant.

Note that these conclusions only hold for the current instance of the design. An experienced analyst quickly realizes that the stiffness contribution of the plate is due to bending action, while the stiffness contribution of the mount is almost pure tensile action. A plate's resistance to bending grows as its thickness cubed, while resistance to tension is proportional to the cross-sectional area. Therefore, as the thickness of the plate increases, its contribution to the stiffness of the composite part increases rapidly.

If you increase the thickness of the end plate too much, the stiffness of the mount takes over as dominant factor in the overall behavior. Using sensitivity analysis, you can easily detect when this happens, because it leads to $Q_m > Q_p$. As noted above, an optimum design for a given total mass is found when $Q_p = Q_m$.

Because the sensitivity analysis was performed using the forward method, the derivatives of the solution with respect to the parameters Δm_p and Δm_m have also been stored. You can access this data when processing the results by using the syntax `sens(`*`expr`*`,`*`var`*`)`. For example, by plotting the expression `sens(w,dm_mt)`, you can directly examine the local effect of a unit increase in the mount mass, see Figure 2.

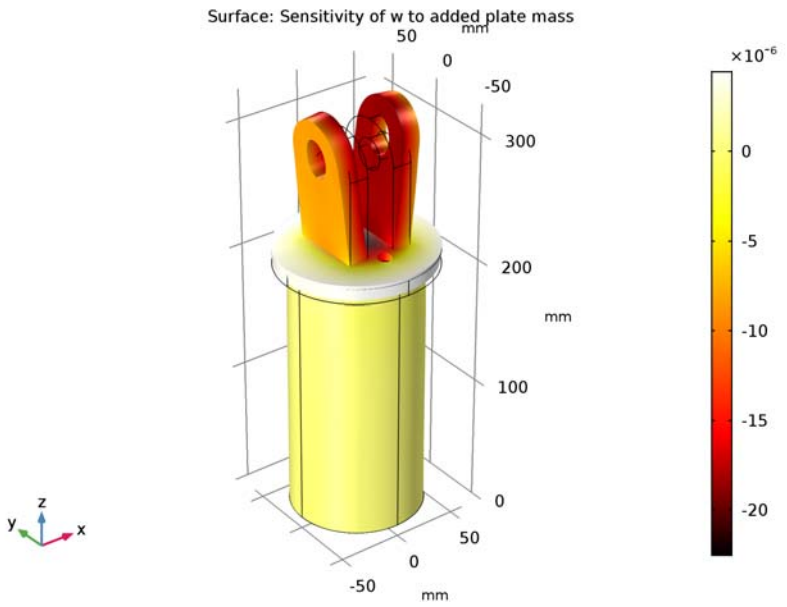


Figure 2: The influence of 1 mm mount thickness increase on the overall displacement.

Application Library path: `COMSOL_Multiphysics/Structural_Mechanics/mast_diagonal_mounting_sensitivity`

Modeling Instructions

From the **File** menu, choose **Open**.

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

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

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

3 In the tree, select **Mathematics>Deformed Mesh>Deformed Geometry (dg)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.

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

6 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ADD STUDY

1 On 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 **Preset Studies>Stationary**.

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

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

GLOBAL DEFINITIONS

Modify the parameters for the plate thickness and mount thickness to correspond to the updated configuration of the original model.

Parameters

1 In the **Model Builder** window, expand the **Global Definitions** node, then click **Parameters**.

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

3 In the table, modify the following settings:

Name	Expression	Value	Description
t_p	12[mm]	0.012 m	Plate thickness
t_m	15[mm]	0.015 m	Mount thickness

4 Next, define the masses added to the plate and mount that you will use later as global control variables.

Name	Expression	Value	Description
dm_p	0	0	Added mass, plate
dm_m	0	0	Added mass, mount

DEFINITIONS

Create a selection for the domains where the mesh will be allowed to deform.

Union 1

1 On the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, locate the **Input Entities** section.

3 Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, in the **Selections to add** list, choose **End plate** and **Mount**.

5 Click **OK**.

6 Right-click **Union 1** and choose **Rename**.

7 In the **Rename Union** dialog box, type `Deformed mesh domains` in the **New label** text field.

8 Click **OK**.

9 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

Integration 1 (intop1)

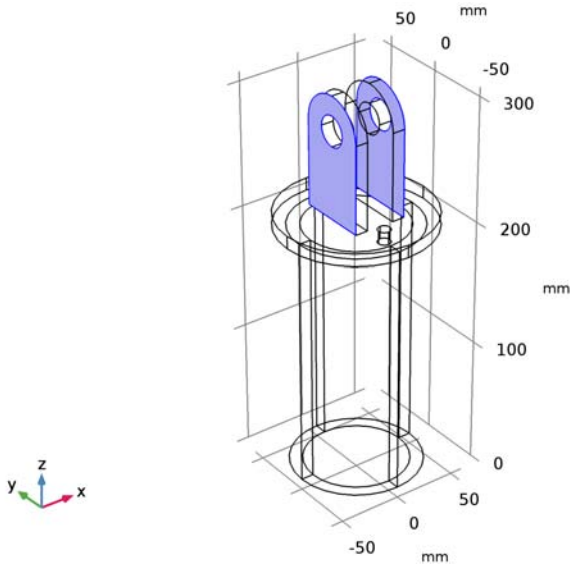
1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

Add component couplings for integrating over the flat surfaces of the plate and mount.

2 In the **Settings** window for **Integration**, type `dA_mt` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 15 and 56 only.



5 Click **Create Selection**.

6 In the **Create Selection** dialog box, type `Outer mount faces` in the **Selection name** text field.

7 Click **OK**.

8 Right-click **Integration 1 (intop1)** and choose **Rename**.

9 In the **Rename Integration** dialog box, type `Outer mount faces` in the **New label** text field.

10 Click **OK**.

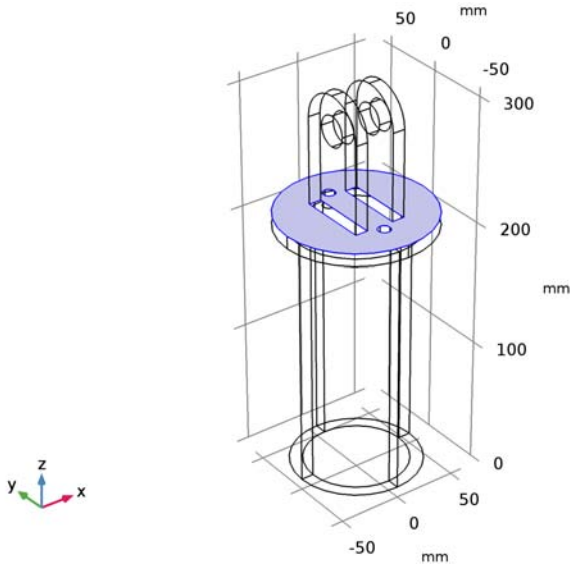
Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `dA_pl` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 4 only.



5 Click **Create Selection**.

6 In the **Create Selection** dialog box, type `Plate surface` in the **Selection name** text field.

7 Click **OK**.

8 Right-click **Integration 2 (intop2)** and choose **Rename**.

9 In the **Rename Integration** dialog box, type `Plate surface` in the **New label** text field.

10 Click **OK**.

11 Click the **Wireframe Rendering** button on the **Graphics** toolbar to return to the default state.

Next, define variables for the displacements as functions of the added masses.

Variables 2

1 On the **Definitions** toolbar, click **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
mA_pl	dA_pl(solid.rho)	kg/m	Mass per unit thickness, plate
mA_mt	dA_mt(solid.rho)	kg/m	Mass per unit thickness, mount
dt_p	dm_p[kg]/mA_pl	m	Displacement, plate end face
dt_m	dm_m[kg]/mA_mt	m	Displacement, outer mount faces

The mass per unit thickness for the plate refers to the area outside the surface where the plate connects to the mount; thus, it gives the net added mass when the thickness of the plate increases.

4 Right-click **Variables 2** and choose **Rename**.

5 In the **Rename Variables** dialog box, type `Sensitivity Variables` in the **New label** text field.

6 Click **OK**.

Variables 1

Modify the variables that are affected by change in plate and mount thickness.

1 In the **Model Builder** window, expand the **Definitions** node, then click **Variables 1**.

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

3 In the table, modify the following settings:

Name	Expression	Unit	Description
Axy_mh	2*r_mh*(t_m+dt_m)	m^2	Mount hole xy projected area

DEFORMED GEOMETRY (DG)

By default, the mesh is fixed in all domains. Remove this constraint for the mount and plates by adding a **Free Deformation** node:

Free Deformation 1

1. On the **Physics** toolbar, click **Domains** and choose **Free Deformation**.
2. In the **Settings** window for **Free Deformation**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Deformed mesh domains**.

The virtual boundary displacements must be described in a boundary system which does not rotate with the deformation.

DEFINITIONS

Boundary System 1 (sys1)

1. In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node, then click **Boundary System 1 (sys1)**.
2. In the **Settings** window for **Boundary System**, locate the **Settings** section.
3. From the **Frame** list, choose **Geometry configuration**.

DEFORMED GEOMETRY (DG)

Prescribed Mesh Displacement 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Deformed Geometry (dg)** click **Prescribed Mesh Displacement 1**.
2. In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Coordinate System Selection** section.
3. From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.
4. Locate the **Prescribed Mesh Displacement** section. Clear the **Prescribed t1 displacement** check box.
5. Clear the **Prescribed t2 displacement** check box.

   This default condition will apply to the boundaries where you do not prescribe nonzero displacements.

Prescribed Mesh Displacement 2

1. On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Mesh Displacement**.
2. In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Outer mount faces**.

4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.

5 Locate the **Prescribed Mesh Displacement** section. Clear the **Prescribed t1 displacement** check box.

6 Clear the **Prescribed t2 displacement** check box.

7 In the d_n text field, type `dt_m`.

Prescribed Mesh Displacement 3

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Mesh Displacement**.

2 In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Plate surface**.

4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.

5 Locate the **Prescribed Mesh Displacement** section. Clear the **Prescribed t1 displacement** check box.

6 Clear the **Prescribed t2 displacement** check box.

7 In the d_n text field, type `dt_p`.

STUDY 2

Now, set up the study. In particular, you need to add a **Sensitivity** node. But first, you must enable advanced study options.

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Study Options** in the menu.

Sensitivity

1 In the **Model Builder** window, right-click **Study 2** and choose **Sensitivity**.

By using the forward method, you will have access to the derivatives of the solution with respect to the sensitivity variables `dm_p` and `dm_m`.

2 In the **Settings** window for **Sensitivity**, locate the **Sensitivity Method** section.

3 From the **Gradient method** list, choose **Forward**.

The stiffness ratio computed in the original model serves as the objective function.

4 Click **Replace Expression** in the upper-right corner of the **Objective function** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>comp1.S_R - Stiffness ratio**.

5 Locate the **Control Variables and Parameters** section. Click **Add**.

6 Click **Add**.

7 In the table, enter the following settings:

Parameter name	Value	Scale	Value type
dm_p (Added mass, plate)	0	1	real
dm_m (Added mass, mount)	0	1	real

8 On the **Home** toolbar, click **Compute**.

RESULTS

Derived Values

Begin by evaluating the sensitivities of the stiffness ratio with respect to the two types of small additions.

Global Evaluation 3

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

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

3 From the **Data set** list, choose **Study 2/Solution 5 (sol5)**.

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

Expression	Unit	Description
`fsens(dm_m)`		

5 Click **Evaluate**.

Global Evaluation 4

1 Right-click **Global Evaluation 3** and choose **Duplicate**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`fsens(dm_p)`		

4 Click the arrow next to the **Evaluate** button and choose **Table 3 - Global Evaluation 3 (fsens(dm_m))**.

Create a plot for the sensitivity of axial displacement with respect to change in the added platemass.

3D Plot Group 4

1 On the **Results** toolbar, click **3D Plot Group**.

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

3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 5 (sol5)**.

Surface 1

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

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

3 In the **Expression** text field, type `sens(w,dm_p)`.

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

5 On the **Sensitivity** toolbar, click **Plot**.

Deformation 1

1 Right-click **Results>Sensitivity>Surface 1** and choose **Deformation**.

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

3 In the **X component** text field, type `u+1[mm]*mA_pl*sens(u,dm_p)`.

4 In the **Y component** text field, type `v+1[mm]*mA_pl*sens(v,dm_p)`.

5 In the **Z component** text field, type `w+1[mm]*mA_pl*sens(w,dm_p)`.

These deformations give a linar approximation of the deformation that would result from a 1 mm increase in the plate thickness.

6 On the **Sensitivity** toolbar, click **Plot**.

Sensitivity

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

2 In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Surface: Sensitivity of w to added plate mass`.

5 On the **Sensitivity** toolbar, click **Plot**.

Micromixer — Batch Version

Introduction

The purpose of this example is to demonstrate how to access the batch functionality in COMSOL from the COMSOL Desktop and use it on your computer. The model also demonstrates how you can measure the speedup of COMSOL on your computer. The speedup is defined as the quotient between the total runtime using only one core of the machine and the runtime using all cores of the machine.

For detailed information about the model, see Micromixer.

Application Library path: `COMSOL_Multiphysics/Tutorials/micromixer_batch`

Modeling Instructions

APPLICATION LIBRARIES

1 On the **Home** toolbar, click **Application Libraries**.

2 In the **Application Library** tree, select **COMSOL Multiphysics>Fluid Dynamics>micromixer**.

3 Click **Open Application**.

STUDY 1

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Study Options** in the menu.

With this setting active, Batch is available from the Study node's context menu.

Batch

1 In the **Model Builder** window, right-click **Study 1** and choose **Batch**.

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

In the Study Settings section of the Settings window for Batch, you specify where to store the model for the batch job:

Enter a suitable filename in the **Filename** text field, and specify a directory by clicking Browse and then making a choice in the Batch Directory dialog box or by typing in the directory path directly in the Directory text field.

3 On the **Home** toolbar, click **Compute**.

When the process has finished, you are automatically transferred to the External Process node for the batch job you just ran.

External Process 1

1 In the **External Process** settings window, locate the **General** section.

In the General section, click Open to open the file containing the model generated by the batch job associated with this external process in a new COMSOL session.

In the Process Status section, the log shows the total solution time. COMSOL automatically takes advantage of all cores; to measure the speedup, set the number of cores to 1 and run another batch job.

Batch 1

1 In the **Model Builder** window, click **Study 1>Job Configurations>Batch 1**.

2 In the **Settings** window for Batch, locate the **General** section.

3 From the **Defined by study step** list, choose **User defined**.

4 Select the **Number of cores** check box.

Verify that the setting in the associated text field is 1.

Next, change the filename to create a new External Process node for the run:

5 Locate the **Files** section. In the associated text field, enter a name of your choice.

6 Click the **Run** button.

When the process has finished, compare the total time in the log for the new external process with the previous value. The speedup is equal to the previous value divided by the new value. The speedup is dependent on the mesh size. To improve the numbers, try refining the mesh.

Micromixer — Cluster Version

Introduction

The purpose of this example is to demonstrate how to access the cluster computing functionality in COMSOL from the COMSOL Desktop and use it to submit a batch job to a cluster through a job scheduler. The model also demonstrates how you can measure the speedup of COMSOL on your computer. The speedup is defined as the quotient between the total runtime using only one physical node and one core of the cluster and the runtime using several physical nodes and all cores of each physical node of the cluster.

For detailed information about the model, see Micromixer. For information about how to set up a COMSOL for running on a cluster, see the *COMSOL Multiphysics Installation Guide*.

Note: This application requires a Floating Network License.

Application Library path: `COMSOL_Multiphysics/Tutorials/micromixer_cluster`

Modeling Instructions

APPLICATION LIBRARIES

1 On the **Home** toolbar, click **Application Libraries**.

2 In the **Application Library** tree, select **COMSOL Multiphysics>Fluid Dynamics>micromixer**.

3 Click **Open Application**.

STUDY 1

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Study Options** in the menu.

With this setting active, Cluster Computing is available from the Study node's context menu.

Cluster Computing

1 In the **Model Builder** window, right-click **Study 1** and choose **Cluster Computing**.

2 In the **Settings** window for Cluster Computing, locate the **Batch Settings** section.

Choose from one of the following settings the type of job scheduler to use:

General: Use the General scheduler if you intend to submit a job to a job scheduler that you have configured COMSOL to run on.

HPCS 2008: Use the HPCS 2008 scheduler if you intend to submit a job to a Windows HPC Server 2008 using the Windows HPC Cluster Manager.

WCCS 2003: Use the WCCS 2003 scheduler if you intend to submit a job to a Windows Compute Cluster Server 2003.

3 From the **Scheduler type** list, choose **Not distributed**.

Not Distributed: Use this setting when you have configured COMSOL to run on job scheduler but only intend to run on a single node of the cluster.

Make sure you configure the batch directories and COMSOL installation directories correctly. It is good practice to save these values as default once you have good settings.

4 Click the **Save as Default** button.

Only MUMPS and SPOOLES are distributed so you need to enable the direct solver to get distributed speedup.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1(sol1)> Stationary Solver 1** node.

2 Right-click **Direct** and choose **Enable**.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1(sol1)> Stationary Solver 2** node.

4 Right-click **Direct** and choose **Enable**.

5 On the **Home** toolbar, click **Compute**.

Batch 1

In the General section, click Open to open the file containing the model generated by the batch job associated with this external process in a new COMSOL session.

In the Process Status section, the log shows the total solution time. COMSOL automatically takes advantage of all cores; to measure the speedup, set the number of cores to 1 and run a new job.

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

2 In the **Settings** window for Batch, locate the **General** section.

3 From the **Defined by study step** list, choose **User defined**.

4 Select the **Number of cores** check box.

Verify that the setting in the associated text field is 1.

Next, change the filename to create a new External Process node for the run:

5 Locate the **Files** section. In the **Filename** text field, enter a name of your choice.

Cluster Computing 1

If you are not using the cluster type Not distributed, make sure to set the Number of nodes to 1.

1 In the **Model Builder** window, under **Study 1>Job Configurations** right-click **Cluster Computing 1** and choose **Run**.

When the process has finished, compare the total time in the log for the new external process with the previous value. The speedup is equal to the previous value divided by the new value. The speedup depends on the mesh size. To improve the numbers, try refining the mesh.

Pacemaker Electrode

Introduction

This example illustrates the use of COMSOL Multiphysics for modeling of ionic current distribution problems in electrolytes, in this case in human tissue. The problem is exemplified on a pacemaker electrode, but it can be applied in electrochemical cells like fuel cells, batteries, corrosion protection, or any other process where ionic conduction takes place in the absence of concentration gradients.

The modeled device is a pacemaker electrode that is placed inside the heart and helps the patient's heart to keep a normal rhythm. The device is referred to as an electrode, but it actually consists of two electrodes: a cathode and an anode.

Figure 1 shows a schematic drawing of two pair of electrodes placed inside the heart. The electrodes are supplied with current from the pulse generator unit, which is also implanted in the patient.

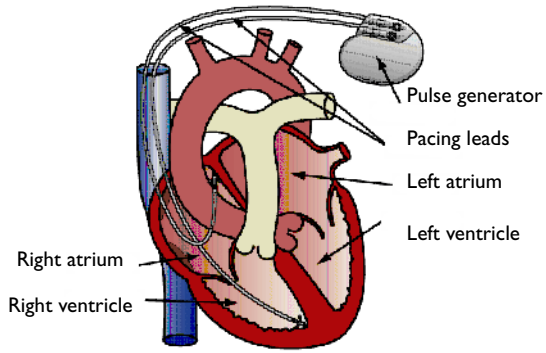


Figure 1: Schematic drawing of the heart with two pairs of pacemaker electrodes.

This example deals with the current and potential distribution around one pair of electrodes.

Model Definition

The model domain consists of the blood and tissue surrounding the electrode pair. The actual electrodes and the electrode support are boundaries to the modeled domain. Figure 2 shows the electrode in a darker shade, while the surrounding modeling domain

is shown in a lighter shade.

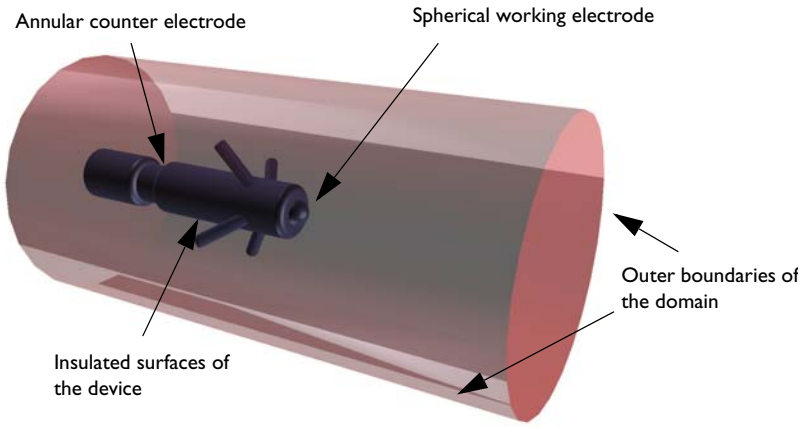


Figure 2: Modeling domain and boundaries.

The working electrode consists of a a hemisphere placed on the tip of the supporting cylindrical structure. The counter electrode is placed in the "waist" of this structure. All other surfaces of the supporting structure are insulated. The outer boundaries are placed far enough from the electrode to give a small impact on the current and potential distribution.

In COMSOL Multiphysics, use the Electric Currents interface for the analysis of the electrode. This physics interface is useful for modeling conductive materials where a current flows due to an applied electric field.

DOMAIN EQUATIONS

The current in the domain is controlled by the continuity equation, which follows from Maxwell's equations:

$$-\nabla \cdot (\sigma \nabla V) = 0$$

where σ is the conductivity of the human tissue. This equation uses the following relations between the electric potential and the fields.

$$\mathbf{E} = -\nabla V$$
$$\mathbf{J} = \sigma \mathbf{E}$$

BOUNDARY CONDITIONS

Ground potential boundary conditions are applied on the thinner waist of the electrode. The tip of the electrode has a fixed potential of 1 V. All other boundaries are electrically insulated.

$$\mathbf{n} \cdot \mathbf{J} = 0$$

Results and Discussion

This simulation gives the potential distribution on the electrode surface and streamlines of the current distribution inside the human heart; see Figure 3.

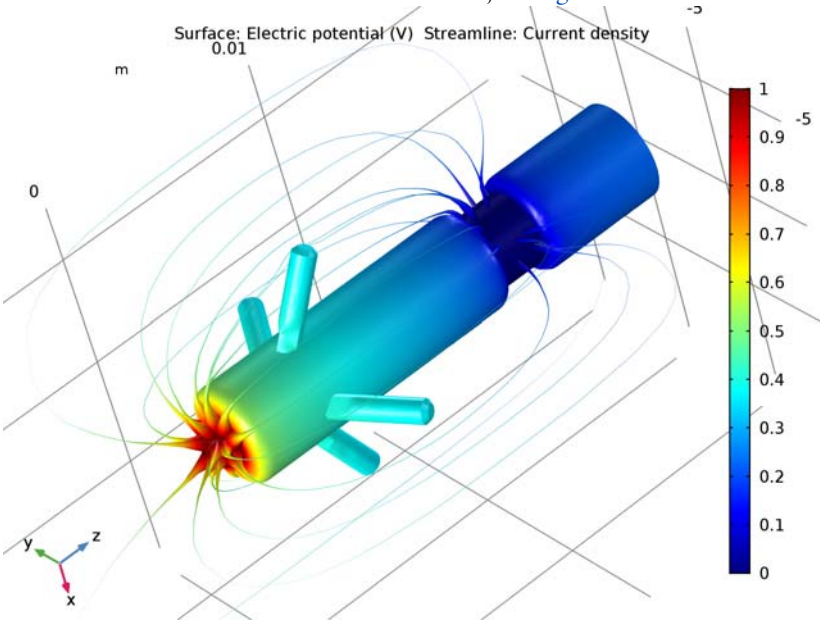


Figure 3: The plot shows the electrostatic potential distributed on the surface of the electrode. The total current density is shown as streamlines.

As expected, the current density is highest at the small hemisphere, which is the one that causes the excitation of the heart. The current density is fairly uniform on the working electrode. The counter electrode is larger and there are also larger variations in current density on its surface. Mainly, the current is lower with increasing distance from the working electrode. The model shows that the anchoring arms of the device have little influence on the current density distribution.

Application Library path: `COMSOL_Multiphysics/Electromagnetics/pacemaker_electrode`

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 **AC/DC>Electric Currents (ec)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Stationary**.
6. Click **Done**.

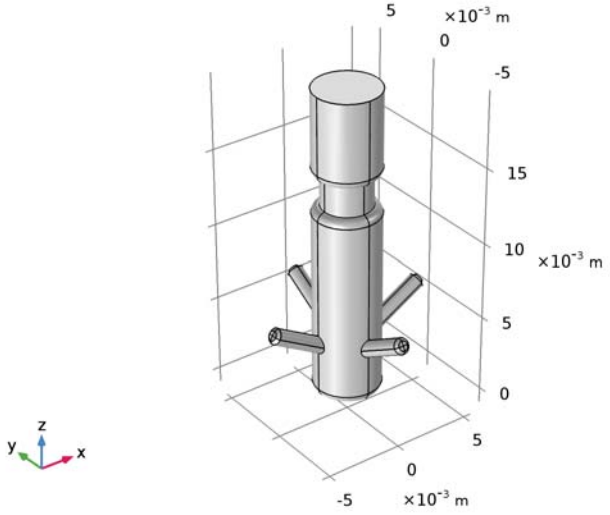
GEOMETRY 1

Although you can readily build the pacemaker geometry from geometric primitives using the CAD tools in COMSOL Multiphysics, it is supplied as a COMSOL Multiphysics geometry file for convenience.

Import 1 (imp1)

1. On the **Home** toolbar, click **Import**.
2. In the **Settings** window for **Import**, locate the **Import** section.
3. Click **Browse**.
4. Browse to the model's Application Libraries folder and double-click the file `pacemaker_electrode.mphbin`.

5 Click **Import**.



Next, define the volume surrounding the electrode. The simulation only takes place in this volume, where the boundaries of the electrode influence the result.

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

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

3 In the **Radius** text field, type `0.01`.

4 In the **Height** text field, type `0.04`.

5 Locate the **Position** section. In the **z** text field, type `-0.02`.

6 Right-click **Cylinder 1 (cyl1)** and choose **Build Selected**.

7 Click the **Go to Default View** button on the **Graphics** toolbar.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **cyl1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the object **imp1** only.

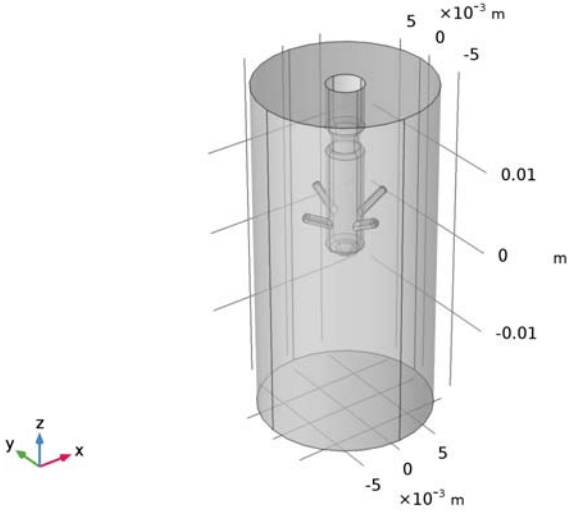
6 Right-click **Difference 1 (dif1)** and choose **Build Selected**.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

The model geometry is now complete.

2 Click the **Transparency** button on the **Graphics** toolbar.



MATERIALS

A convenient way to find out which material parameters you need to specify is to add a material. COMSOL Multiphysics then indicates any missing parameters for the physics interfaces you have added to the model.

Material 1 (mat1)

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

By default, the first material you add applies to all domains, so you do not need to modify the geometric scope.

The electrode is inserted into the human heart, so you must specify the conductivity for the heart tissue.

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
Electrical conductivity	sigma	5000	S/m	Basic
Relative permittivity	epsilonr	1	1	Basic

4 Right-click **Component 1 (comp1)>Materials>Material 1 (mat1)** and choose **Rename**.

5 In the **Rename Material** dialog box, type `Heart Tissue` in the **New label** text field.

6 Click **OK**.

ELECTRIC CURRENTS (EC)

The only physics settings that remain to specify are the electrode potentials.

Ground 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.

2 Select Boundaries 29, 30, 58, and 63 only.

The easiest way to do this is by entering these numbers in the dialog box that opens if you click the **Paste Selection** button in the **Boundaries** section.

Optionally, you can create a named selection for these boundaries as follows:

3 In the **Settings** window for **Ground**, locate the **Boundary Selection** section.

4 Click **Create Selection**.

5 In the **Create Selection** dialog box, type `Counter Electrode` in the **Selection name** text field.

6 Click **OK**.

Electric Potential 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

2 Select Boundaries 31, 32, 59, and 60 only.

3 In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.

4 In the V_0 text field, type `1`.

5 Locate the **Boundary Selection** section. Click **Create Selection**.

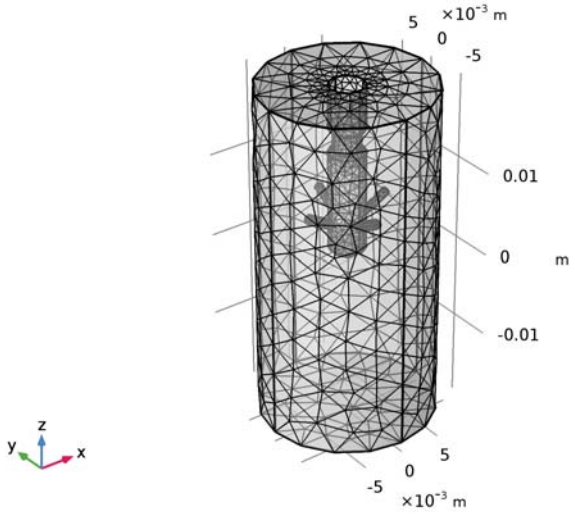
6 In the **Create Selection** dialog box, type `Spherical Electrode` in the **Selection name** text field.

7 Click **OK**.

MESH 1

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

2 In the **Settings** window for **Mesh**, click **Build All**.



STUDY 1

Use the default settings for the stationary solver, which gives the conjugate gradients iterative solver with algebraic multigrid as the preconditioner.

1 On the **Home** toolbar, click **Compute**.

RESULTS

Electric Potential (ec)

The default plot shows the slices of the electrical potential. To reproduce the plot shown in Figure 3, start by resetting the transparency state and instead hide the outer boundaries.

1 Click the **Transparency** button on the **Graphics** toolbar.

ELECTRIC CURRENTS (EC)

1 Click the **Click and Hide** button on the **Graphics** toolbar.

2 Click the **Select Boundaries** button on the **Graphics** toolbar.

3 In the **Model Builder** window, under **Component 1 (comp1)** click **Electric Currents (ec)**.

4 Select Boundaries 1–4, 45, and 74 only.

5 Click the **Click and Hide** button on the **Graphics** toolbar.

RESULTS

3D Plot Group 2

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

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 Clear the **Plot data set edges** check box.

Surface 1

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

2 On the **3D Plot Group 2** toolbar, click **Plot**.

3D Plot Group 2

Combine the surface plot of the potential with streamlines visualizing the total current density.

Streamline 1

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

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

3 Select the **Active** toggle button.

4 From the **Selection** list, choose **All boundaries**.

5 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Start point controlled**.

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

7 In the **Tube radius expression** text field, type `ec.normJ`. This is the variable for the current density norm.

To get suitably thick streamlines you need to adjust the scale factor.

8 Select the **Radius scale factor** check box.

9 In the associated text field, type `1e-10`.

10 On the **3D Plot Group 2** toolbar, click **Plot**.

Color Expression 1

1 Right-click **Results>3D Plot Group 2>Streamline 1** and choose **Color Expression**.

Use the default expression. Because it is the same as the surface expression, you can disable the color legend:

2 In the **Settings** window for **Color Expression**, locate the **Coloring and Style** section.

3 Clear the **Color legend** check box.

After proper rotation and zoom operations, you should see something similar to the plot in Figure 3.

Process Control Using a PID Controller

Introduction

In the chemical process industry it is often important to control a specific process. PID control (proportional-integral-derivative-control) is one way to achieve that, but it can be difficult to optimize the parameters in the PID algorithm. This example illustrates how you can implement a PID control algorithm to simulate a process control system and to find the optimal PID parameters.

This application is a generic example but could resemble the environment in a combustion chamber where the concentration at the ignition point is crucial. Two gas streams with different oxygen concentrations are mixed in the combustion chamber. The concentration is measured at the ignition point before complete mixing of the streams is reached. The control algorithm alters the inlet velocity of the gas with the lower oxygen content to achieve the desired total concentration at the ignition point.

Model Definition

The model geometry appears in Figure 1. At the upper inlet, a gas stream with high oxygen content enters the reactor at a velocity of 10 mm/s, while a gas with a lower oxygen level enters from the left. The oxygen concentration is measured at a measurement point, and the inlet velocity of the less concentrated stream is altered by the PID control algorithm to achieve the desired concentration at that point.

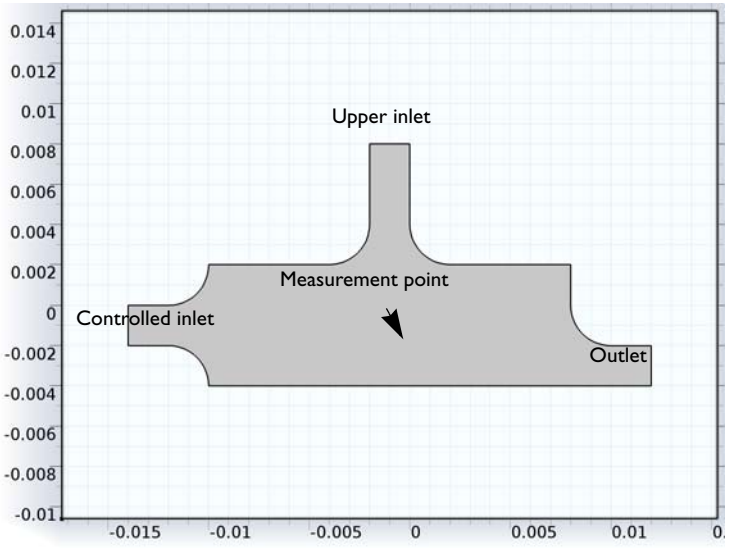


Figure 1: Model geometry.

The model uses the Laminar Flow interface to describe the fluid flow and the Transport of Diluted Species interface for the mass balance. The corresponding equations read (assuming incompressible flow and absence of reactions)

$$\rho\frac{\partial \mathbf{u}}{\partial t} - \nabla \cdot [\eta(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)] + \rho \mathbf{u} \cdot \nabla \mathbf{u} + \nabla p = \mathbf{0}$$

$$\nabla \cdot \mathbf{u} = 0$$

$$\frac{\partial c}{\partial t} + \nabla \cdot (-D\nabla c) = -\mathbf{u} \cdot \nabla c$$

To formulate the boundary conditions for the mass-transport equation, begin by assuming that you know the two inlet concentrations. In addition, assume that the reactant transport at the outlet is mainly driven by convection, that is, neglect diffusion in the main direction of the convective flow. A no-flux boundary condition describes all walls. The boundary conditions for the mass balance are:

BOUNDARY	CONSTRAINT
Upper inlet	$c = c_{\text{in,top}}$
Controlled inlet	$c = c_{\text{in,inlet}}$
Outlet	$\mathbf{n} \cdot (-D\nabla c) = 0$
Walls	$\mathbf{N} \cdot \mathbf{n} = 0$

Here c is the concentration; $c_{\text{in,top}}$ and $c_{\text{in,inlet}}$ are the inlet concentrations (mol/m^3) for the upper and controlled inlets, respectively; D is the applied diffusivity (m^2/s); and $\mathbf{N}$ is the molar flux (mol/(m^2·s)).

The model uses the following boundary conditions for the fluid flow:

BOUNDARY	CONSTRAINT
Upper inlet	$\mathbf{u} = (0, -v_{\text{in,top}})$
Controlled inlet	$\mathbf{u} = (u_{\text{in}}, 0)$
Outlet	$p_0 = 0$
Inlet sections	$\mathbf{n} \cdot \mathbf{u} = 0$
Walls	$\mathbf{u} = \mathbf{0}$

Here $\mathbf{u}$ is the velocity vector (m/s), $v_{\text{in,top}}$ is the inlet velocity at the top inlet, and u_{in} is the PID controlled velocity. At the outlet, set the pressure to 0. No Slip boundary conditions describe all walls except the inlet sections where slip conditions apply, allowing for a smooth transition to a laminar velocity profile.

The PID control algorithm used to calculate u_{in} is

$$u_{in} = k_P(c - c_{set}) + k_I\int_0^t (c - c_{set})dt + k_D\frac{\partial}{\partial t}(c - c_{set}) \quad (1)$$

with the following parameters:

PARAMETER	VALUE
c_{set}	0.5 mol/m^3
k_P	0.5 m^4/(mol·s)
k_I	1 m^4/(mol·s^2)
k_D	10^{-3} m^4/mol

In practice, the derivative constant, k_D, is set to 0 in most cases as this parameter can be difficult to determine. Moreover, the derivative term may increase the fluctuations in the system because it amplifies noise in the error $c - c_{set}$.

Results and Discussion

The two plots in Figure 2 show the oxygen concentration and the velocity stream lines in the chamber after 0.05 s and 2 s, respectively. The figures show that the measured concentration depends strongly on the flow field. At start-up, when the inlet velocity of the stream entering from the left is very low, the sensor is entirely exposed to the highly concentrated stream, and as the left inlet velocity increases the opposite relation occurs.

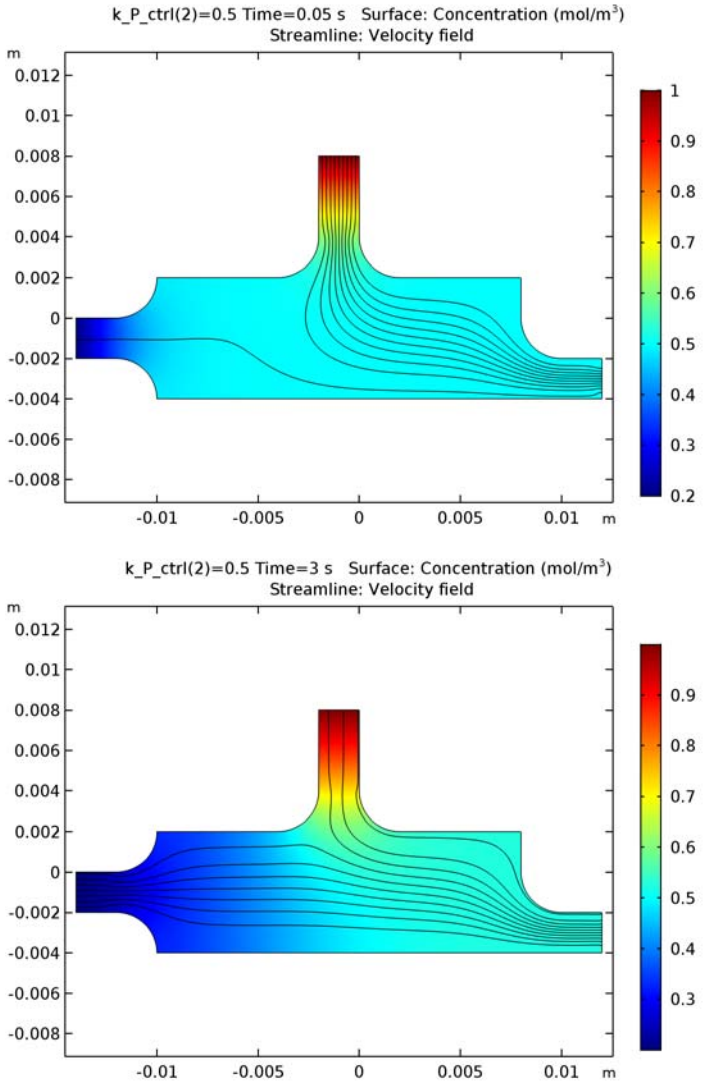


Figure 2: Oxygen concentration and velocity streamlines after 0.1 s (top) and 1.5 s (bottom).

Figure 3 shows the inlet velocity and concentration in the measurement point as a function of time for two different values for the k_P parameter. The solid line represents the results for a k_P value of 0.5 $m^4/(mol \cdot s)$ while the dashed line corresponds to k_P equal to 0.1 $m^4/$

(mol·s). The results evaluated for the smaller k_P value oscillate more before stabilizing. Thus, it is clear that for this case the higher k_P value yields a more stable process control.

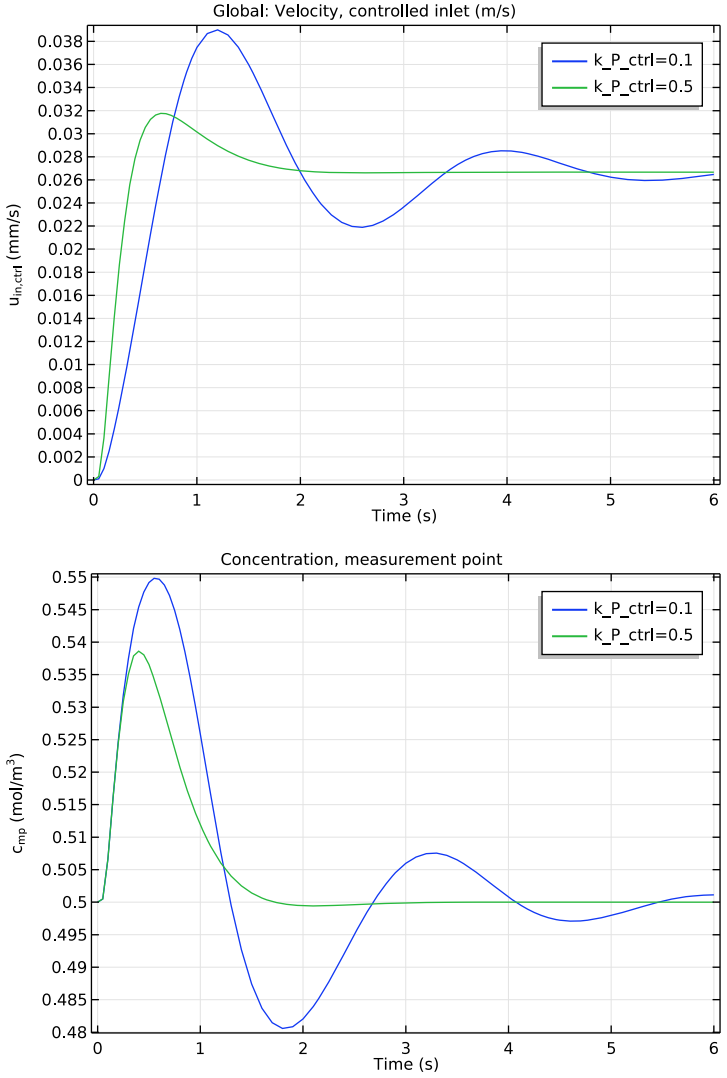


Figure 3: PID-controlled inlet velocity (top) and concentration in the measurement point (bottom) as a function of time for k_P = 0.5 $m^4/(mol·s)$ (blue) and k_P = 0.1 $m^4/(mol·s)$ (green).

Application Library path: `COMSOL_Multiphysics/Multiphysics/pid_control`

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>Single-Phase Flow>Laminar Flow (spf)**.
3. Click **Add**.
4. In the **Select Physics** tree, select **Chemical Species Transport> Transport of Diluted Species (tds)**.
5. Click **Add**.
6. In the **Select Physics** tree, select **Mathematics>ODE and DAE Interfaces> Global ODEs and DAEs (ge)**.
7. Click **Add**.
8. Click **Study**.
9. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Time Dependent**.
10. Click **Done**.

GLOBAL DEFINITIONS

Parameters

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
v_in_top	0.01[m/s]	0.01 m/s	Velocity, upper inlet
c_in_top	1[mol/m^3]	1 mol/m³	Concentration, upper inlet
c_in_inlet	0.2[mol/m^3]	0.2 mol/m³	Concentration, controlled inlet
c00	0.5[mol/m^3]	0.5 mol/m³	Initial concentration, chamber interior
D	1e-4[m^2/s]	1E-4 m²/s	Diffusivity
c_set	0.5[mol/m^3]	0.5 mol/m³	Setpoint concentration
k_P_ctrl	0.5[m^4/(mol*s)]	0.5 m^4/(s·mol)	Proportional parameter
k_I_ctrl	1[m^4/(mol*s^2)]	1 m^4/(s²·mol)	Integral parameter
k_D_ctrl	1e-3[m^4/mol]	0.001 m^4/mol	Derivative parameter

GEOMETRY 1

Create the geometry. To simplify this step, insert a prepared geometry sequence.

1 On the **Geometry** toolbar, click **Insert Sequence**.

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

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

MATERIALS

Material 1 (mat1)

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

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

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

4 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	`1.2[kg/m^3]`	kg/m³	Basic
Dynamic viscosity	mu	`3e-5`	Pa·s	Basic

LAMINAR FLOW (SPF)

Inlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Inlet**, locate the **Velocity** section.

4 In the U_0 text field, type `u_in_ctrl*(u_in_ctrl>0)`.

Inlet 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 7 only.

3 In the **Settings** window for **Inlet**, locate the **Velocity** section.

4 In the U_0 text field, type `v_in_top`.

Outlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 Select Boundary 13 only.

Wall 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Wall**.

2 Select Boundaries 2, 3, 6, and 8 only.

3 In the **Settings** window for **Wall**, locate the **Boundary Condition** section.

4 From the **Wall condition** list, choose **Slip**.

TRANSPORT OF DILUTED SPECIES (TDS)

Transport Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)> Transport of Diluted Species (tds)** click **Transport Properties 1**.

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

3 In the D_c text field, type `D`.

4 Locate the **Convection** section. From the **u** list, choose **Velocity field (spf)**.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)> Transport of Diluted Species (tds)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the *c* text field, type `c00`.

4 In the **Model Builder** window, click **Transport of Diluted Species (tds)**.

Inflow 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Inflow**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Inflow**, locate the **Concentration** section.

4 In the $c_{0,c}$ text field, type `c_in_inlet`.

Inflow 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Inflow**.

2 Select Boundary 7 only.

3 In the **Settings** window for **Inflow**, locate the **Concentration** section.

4 In the $c_{0,c}$ text field, type `c_in_top`.

Outflow 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outflow**.

2 Select Boundary 13 only.

DEFINITIONS

Next, add a probe to sample the concentration and its time derivative at the point $x = 0$, $y = -0.002$.

Domain Point Probe 1

1 On the **Definitions** toolbar, click **Probes** and choose **Domain Point Probe**.

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

3 In row **Coordinates**, set **y** to `-0.002`.

4 In the **Model Builder** window, expand the **Domain Point Probe 1** node, then click **Point Probe Expression 1 (ppb1)**.

5 In the **Settings** window for **Point Probe Expression**, type `c_mp` in the **Variable name** text field.

6 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Transport of Diluted Species>c - Concentration**.

7 In the **Model Builder** window, right-click **Domain Point Probe 1** and choose **Point Probe Expression**.

8 In the **Settings** window for **Point Probe Expression**, type `ct_mp` in the **Variable name** text field.

9 Locate the **Expression** section. In the **Expression** text field, type `ct`.

Variables 1

1 On the **Definitions** toolbar, click **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Description
`u_in_ctrl`	`nojac(k_P_ctrl*(c_mp-c_set)+k_I_ctrl*I[mol*s/m^3]+k_D_ctrl*ct_mp)`	`Velocity, controlled inlet`

The `nojac` operator ensures that the above expression gives no Jacobian contribution. In practice, this means that the control velocity will always be evaluated based on the previous time step. This is necessary to avoid evaluation of an implicit time derivative in the inlet condition, which is not supported in the time dependent solver.

Moreover, '`I`' refers to the time integral in Equation 1, which you define next.

GLOBAL ODES AND DAES (GE)

Global Equations 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)> Global ODEs and DAEs (ge)** node, then click **Global Equations 1**.

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

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
`I`	`It[s]-(c_mp[m^3/mol]-c_set[m^3/mol])`	`0`	`0`	`Time integral term`

MESH 1

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

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

3 From the **Element size** list, choose **Finer**.

4 Click **Build All**.

STUDY 1

Use a parametric sweep to solve for two different values of the proportional parameter, k_P.

Parametric Sweep

1 On 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
k_P_ctrl (Proportional parameter)	`0.1 0.5`

Step 1: Time Dependent

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

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

3 In the **Times** text field, type `range(0,0.05,1) range(1.1,0.1,6)`.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time stepping** section.

4 Locate the **Time Stepping** section. From the **Method** list, choose **Generalized alpha**.

5 From the **Steps taken by solver** list, choose **Intermediate**.

This forces the solver to take at least one step in each of the time intervals you specified.

6 Click to expand the **Advanced** section. Locate the **Time Stepping** section. Find the **Algebraic variable settings** subsection. From the **Error estimation** list, choose **Exclude algebraic**.

7 On the **Study** toolbar, click **Compute**.

RESULTS

Velocity (spf)

The presence of the derivative term leads to a warning message from the solver. As already mentioned in the introduction, this term is difficult to determine and also sensitive to noise, so it is often set to 0.

Concentration (tds)

Add a streamline plot of the velocity to the default surface plot that shows the concentration at the end of the simulated time span (Figure 2). Study the solution at t = 0.05 s and t = 2 s.

Streamline 1

1. In the **Model Builder** window, under **Results** right-click **Concentration (tds)** and choose **Streamline**.
2. In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.
3. From the **Positioning** list, choose **Magnitude controlled**.
4. In the **Density** text field, type 10.
5. On the **Concentration (tds)** toolbar, click **Plot**.

Concentration (tds)

1. In the **Model Builder** window, under **Results** click **Concentration (tds)**.
2. In the **Settings** window for **2D Plot Group**, locate the **Data** section.
3. From the **Time (s)** list, choose **0.05**.
4. On the **Concentration (tds)** toolbar, click **Plot**.
5. Click the **Zoom Extents** button on the **Graphics** toolbar.
6. From the **Time (s)** list, choose **3**.
7. On the **Concentration (tds)** toolbar, click **Plot**.

1D Plot Group 4

Plot the PID-controlled inlet velocity (Figure 3).

1. In the **Model Builder** window, under **Results** click **1D Plot Group 4**.
2. In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.
3. Select the **y-axis label** check box.
4. In the associated text field, type `u<sub>in,ctrl</sub> (mm/s)`.

Global 1

1. In the **Model Builder** window, expand the **1D Plot Group 4** node, then click **Global 1**.

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>Definitions>Variables> u_in_ctrl - Velocity, controlled inlet**.

3 Click to expand the **Legends** section. Find the **Include** subsection. Clear the **Description** check box.

4 On the **1D Plot Group 4** toolbar, click **Plot**.

Proceed to plot the concentration at the measurement point as a function of time (Figure 3).

1D Plot Group 6

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

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Concentration, measurement point`.

5 Locate the **Plot Settings** section. In the **y-axis label** text field, type `c<sub>mp</sub> (mol/m<sup>3</sup>)`.

Global 1

1 In the **Model Builder** window, expand the **1D Plot Group 6** node, then click **Global 1**.

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>Definitions>c_mp - Probe variable c_mp**.

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

The resulting plot should look like that in the lower panel of Figure 3.

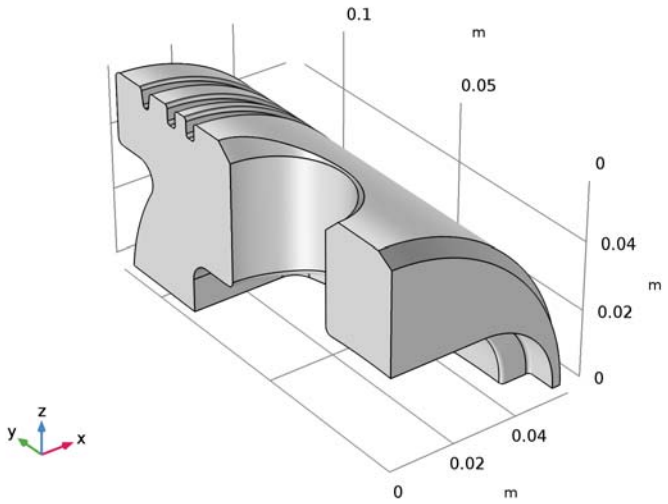
Free Tetrahedral Meshing of a Piston Geometry

Introduction

The free meshing algorithm using tetrahedral elements is the most general meshing technique, and does not pose any constraints on the structure of the geometry. Hence you can use it to mesh any object. There are nine predefined parameter sets for the mesher, ranging from "extremely fine" to "extremely coarse." These settings result in a good mesh for most geometries and simulation problems. In addition you can tune the mesh parameters individually, as demonstrated in this tutorial.

Model Definition

Create a tetrahedral mesh for the geometry of an engine piston as shown in the following figure.



As you can see the geometry contains small details such as fillets and chamfers. To better resolve these details with the mesh you will work with the following mesh parameters:

- Minimum element size
- Curvature factor
- Resolution of narrow regions
- Maximum element growth rate

You will also learn how to use the tools for assessing the mesh quality.

Application Library path: `COMSOL_Multiphysics/Meshing_Tutorials/piston_mesh`

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 Click **Done**.

GEOMETRY 1

Import 1 (imp1)

1 On the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

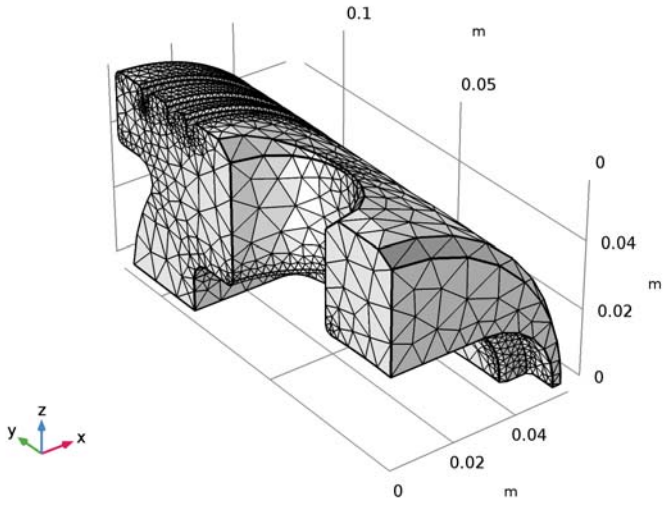
4 Browse to the model's Application Libraries folder and double-click the file `piston_quarter.mphbin`.

5 Click **Import**.

MESH 1

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

2 In the **Settings** window for **Mesh**, click **Build All**.

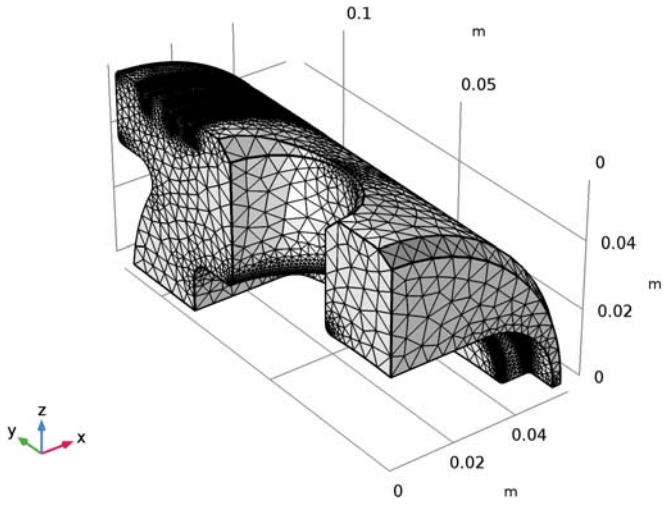


The **Messages** window indicates that there are roughly 22,000 tetrahedral elements in this mesh.

Assume that the current mesh does not resolve details such as fillets and chamfers sufficiently for your simulation needs and a finer parameter setting is required. This would be the case for a stress analysis of the part.

3 Locate the **Mesh Settings** section. From the **Element size** list, choose **Finer**.

4 Click **Build All**.



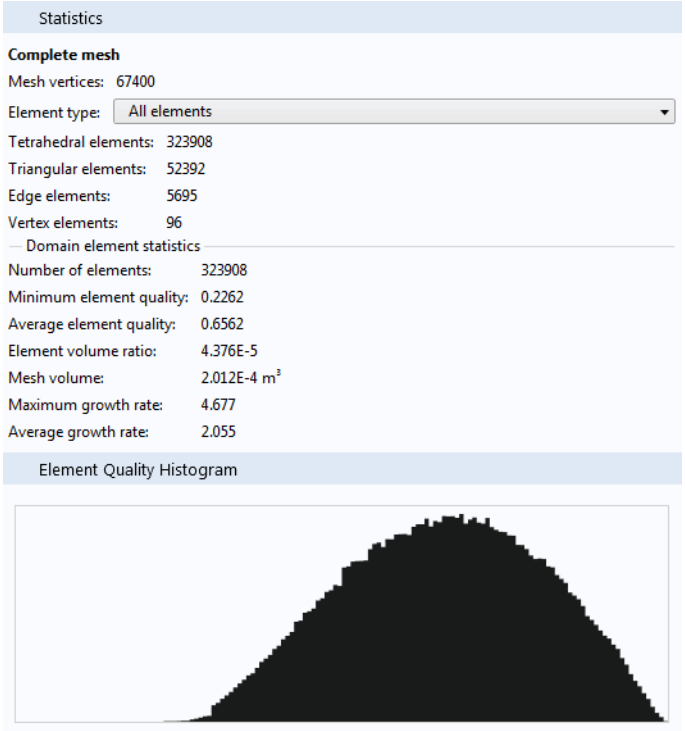
This mesh consists of approximately 320,000 elements. Many of the finer details of the geometry are adequately resolved, but there is a significant increase in the total number of elements compared to the Normal mesh setting.

In the following you will test how to tune mesh parameters to refine the mesh only on selected boundaries.

MESH STATISTICS

Continue with examining the quality of the mesh.

1 Right-click **Component 1 (comp1)>Mesh 1** and choose **Statistics**.



The **Statistics** window contains details about the mesh, including the number and type of elements, and a histogram of element quality.

The element quality has a value between 0 and 1, where 1 describes a perfectly symmetric element and 0 describes a degenerated, or completely flat, element. For 3D meshes in general a minimum quality of about 0.1 means a satisfactory mesh.

However, this depends on the type of geometry and physics application. Note also that the quality number is calculated based on the linear elements.

Meshing with predefined parameter sets usually results in a mesh with quite good quality. According to the information under the section **Domain element statistics**, the present mesh has an average quality of 0.72 with a minimum quality of 0.12.

The histogram reveals the element quality distribution. In this case, the elements with low quality, represented by the tail of the distribution plot, represent a very small fraction of the mesh.

Before adjusting individual mesh parameters start by restoring the mesh with the

normal size settings.

2 Right-click **Component 1 (comp1)>Mesh 1** and choose **Settings**.

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

4 In the **Settings** window for **Mesh**, locate the **Mesh Settings** section.

5 From the **Element size** list, choose **Normal**.

6 Click **Build All**.

MESHING SEQUENCE

1 Right-click **Mesh 1** and choose **Edit Physics-Induced Sequence**.

You can now access and modify the default meshing sequence that appears under the **Mesh 1** node.

The first **Size** feature node in the meshing sequence is a *global attribute node*, since it influences all subsequent *operation nodes* in the meshing sequence. This first **Size** node cannot be deleted from the sequence.

Instead of editing parameters of the global Size node add a Size node to the Free Tetrahedral 1 mesh operation.

Free Tetrahedral 1

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

The **Size 1** node is a *local attribute node* because it only applies to its parent mesh node.

RESOLUTION OF CURVATURE

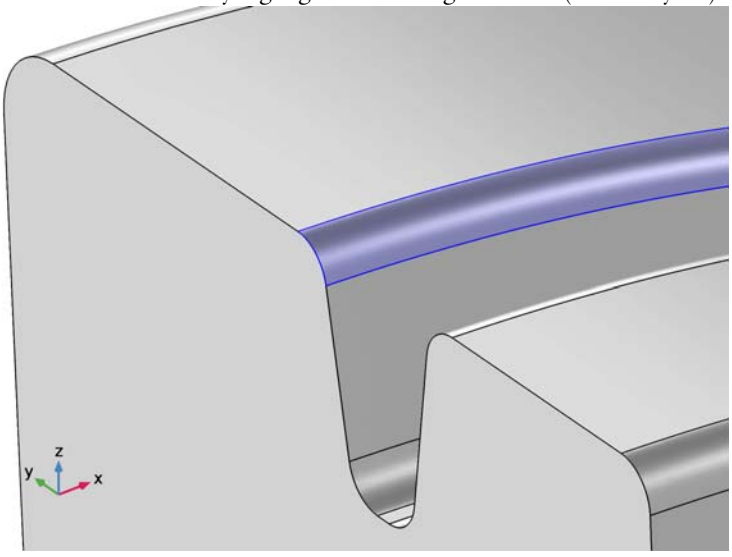
Size 1

1 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

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

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

4 Select the boundary highlighted in the figure below (Boundary 39).



By selecting only one of the fillets you can save time generating the mesh while testing parameter values. You will be able to change the selection of the **Size 1** node to all boundaries after you have found the right set of parameters.

5 In the **Settings** window for **Size**, locate the **Element Size** section.

6 Click the **Custom** button.

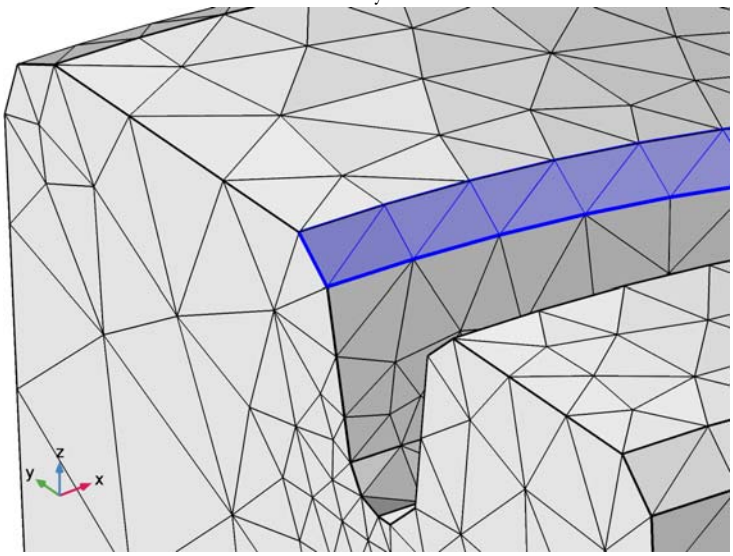
7 Locate the **Element Size Parameters** section. Select the **Curvature factor** check box.

8 In the associated text field, type `0.2`.

The **Curvature factor** parameter determines the size of boundary elements compared to the curvature of the geometric boundary. The curvature radius multiplied by the curvature factor gives the maximum allowed element size along the boundary. A lower value gives a finer mesh along curved boundaries.

9 Click **Build All**.

Zoom in on the selected boundary for a closer look at the mesh.



It seems that reducing the resolution of curvature had almost no effect on the number of mesh elements on the fillet. The reason is that another mesh parameter limits the minimum element size allowed in the mesh.

MINIMUM ELEMENT SIZE

1 In the **Model Builder** window, under **Component 1 (comp 1)> Mesh 1> Free Tetrahedral 1** click **Size 1**.

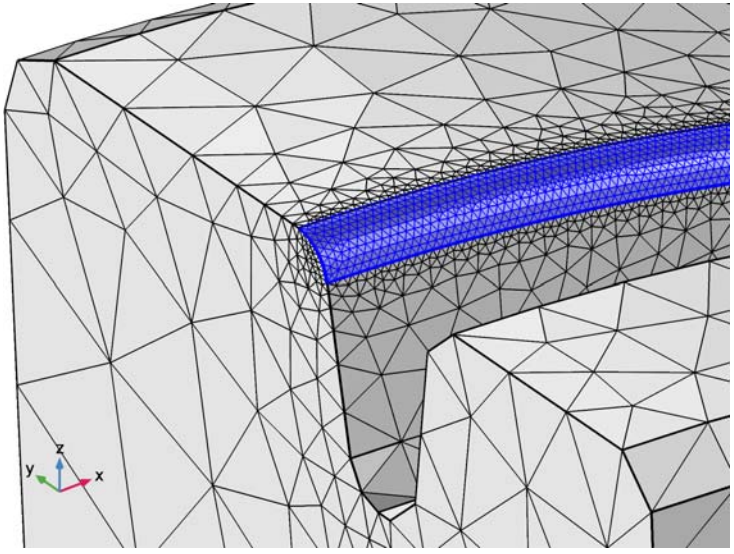
2 In the **Size** settings window, locate the **Element Size Parameters** section.

3 Select the **Minimum element size** check box.

4 In the associated text field, type `0.0002`.

The value in the **Minimum element size** field specifies the minimum allowed element size. You can use this value to, for example, prevent the generation of many elements around small curved parts of the geometry.

5 Click **Build All**.

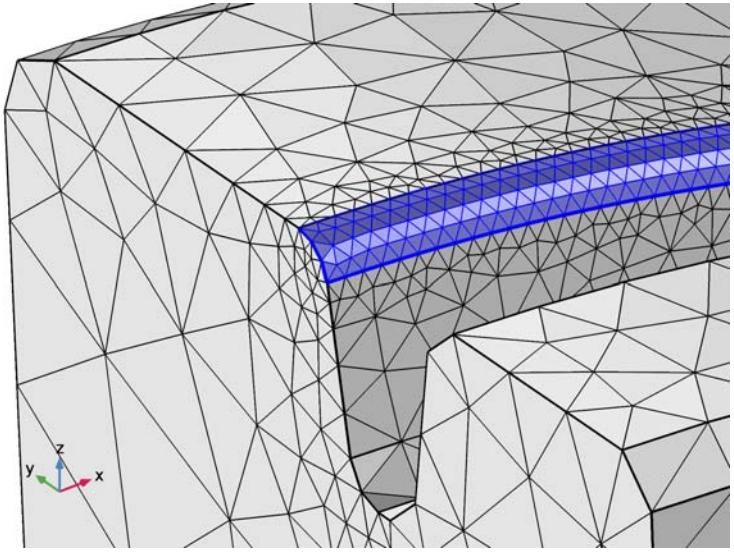


This time the selected boundary has a much finer mesh. Adjust the mesh again by increasing the resolution of curvature.

6 In the **Settings** window for Size, locate the **Element Size Parameters** section.

7 In the **Curvature factor** text field, type `0.45`.

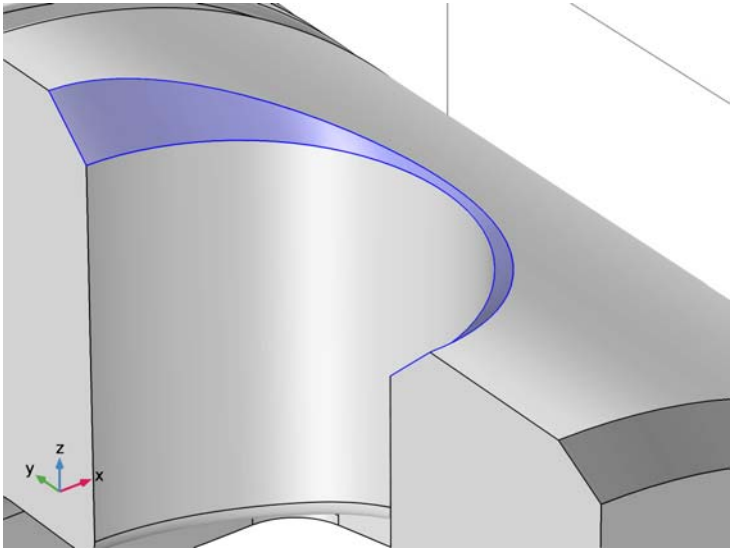
8 Click **Build All**.



Now assume that you also want a better resolution of narrow regions with no curvature such as chamfers.

RESOLUTION OF NARROW REGIONS

1 Add the face highlighted below to the selection. The selection should now contain both boundaries 8 and 39.



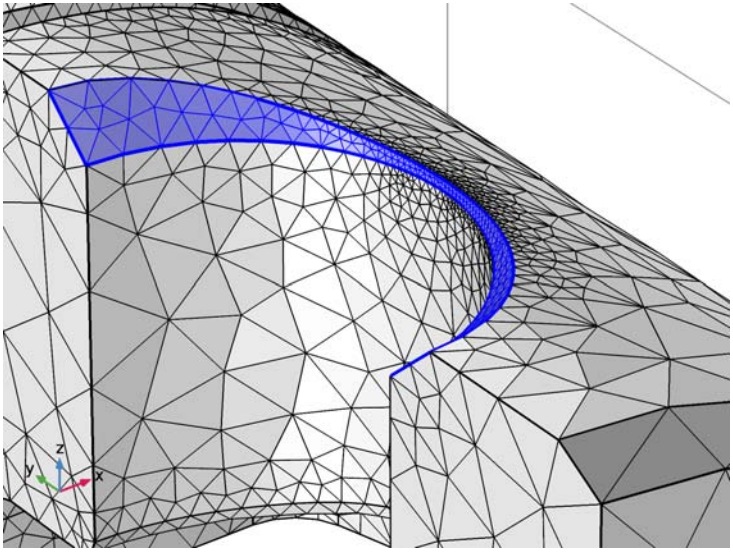
2 In the **Settings** window for **Size**, locate the **Element Size Parameters** section.

3 Select the **Resolution of narrow regions** check box.

4 In the associated text field, type 2.

The **Resolution of narrow regions** mesh parameter controls the number of element layers that are created in narrow regions (approximately). If the value of this parameter is less than one, the mesh generator might create elements that are anisotropic in size in narrow regions.

5 Click **Build All**.

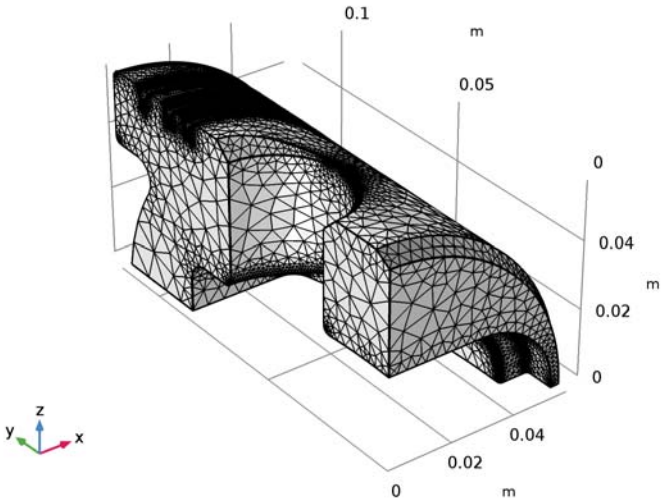


Assume that you are happy with the parameter settings for curved and narrow regions. Now apply these for all boundaries of the geometry.

6 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **All boundaries**.

7 Click **Build All**.

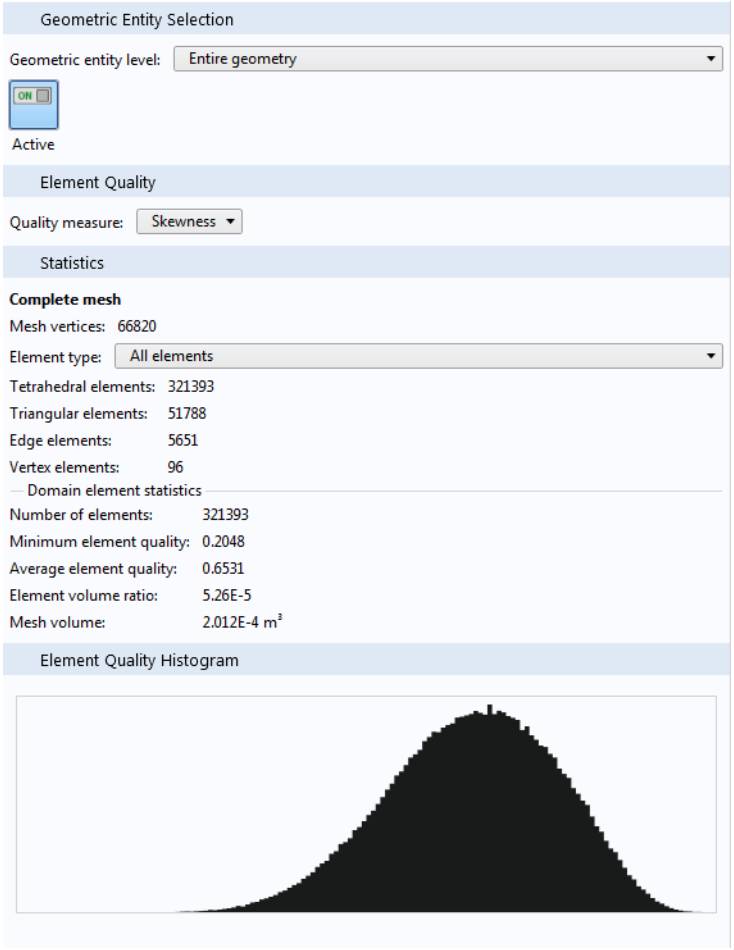
8 Click the **Go to Default View** button on the **Graphics** toolbar to get the view in the figure below.



The fine details of the geometry are resolved satisfactorily with this mesh of approximately 280,000 tetrahedral elements.

Continue with checking the mesh quality.

9 In the **Model Builder** window, right-click **Mesh 1** and choose **Statistics**.



Compared to the mesh with the **Finer** predefined mesh parameter set, the average quality is slightly less and the minimum quality is also lower. This is expected because the boundaries are finely meshed and the elements are growing toward the inner parts of the geometry according to the **Normal** parameter set specified in the global **Size** node. Allowing even higher element growth will reduce the number of elements further and will result in even lower element quality.

MAXIMUM ELEMENT GROWTH

Reduce the number of mesh elements further by specifying the rate of growth from the small elements on the surface to the larger elements inside.

Size

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

2 In the **Settings** window for **Size**, locate the **Element Size** section.

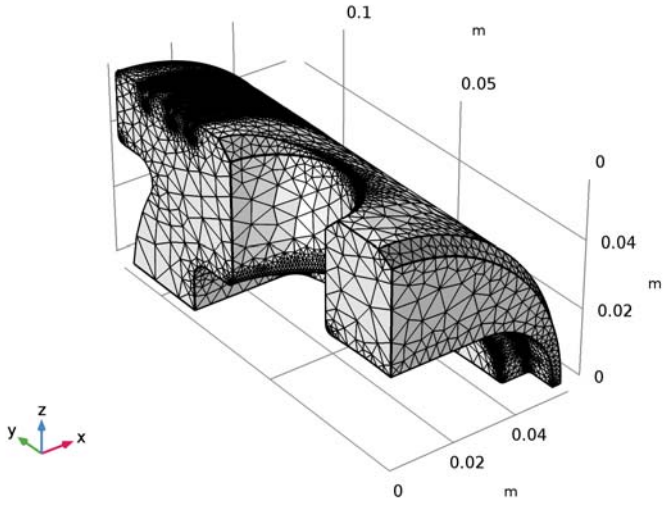
3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element growth rate** text field, type `1.8`.

The **Maximum element growth rate** parameter determines the maximum rate at which the element size can grow from a region with small elements to a region with larger elements.

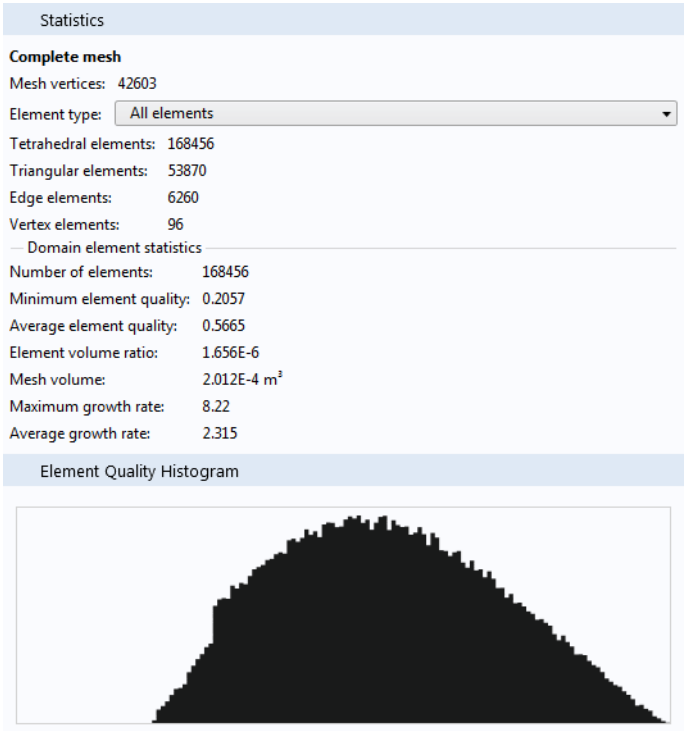
5 In the **Model Builder** window, click **Mesh 1**.

6 In the **Settings** window for **Mesh**, click **Build All**.



The mesh now consists of approximately 170,000 elements while keeping the fine mesh on curved and narrow boundaries.

7 Right-click **Mesh 1** and choose **Statistics**.



As expected the increase of the growth rate parameter results in even lower quality. The histogram reveals that the distribution has a ?thicker tail?, with the low quality elements making up a larger than before fraction of the total element number.

A mesh plot can help with localizing the worst quality elements.

8 On the **Mesh** toolbar, click **Plot**.

RESULTS

Mesh 1

The **Mesh 1** plot is added to the **3D Plot Group 1** under the **Results** section of the **Model Builder** window. The default mesh plot that appears in the **Graphics** window contains the surface elements colored according to quality.

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

2 In the **Settings** window for **Mesh**, click to expand the **Element filter** section.

3 Locate the **Element Filter** section. Select the **Enable filter** check box.

4 From the **Criterion** list, choose **Worst quality**.

5 In the **Fraction** text field, type `0.005`.

6 Locate the **Level** section. From the **Level** list, choose **Volume**.

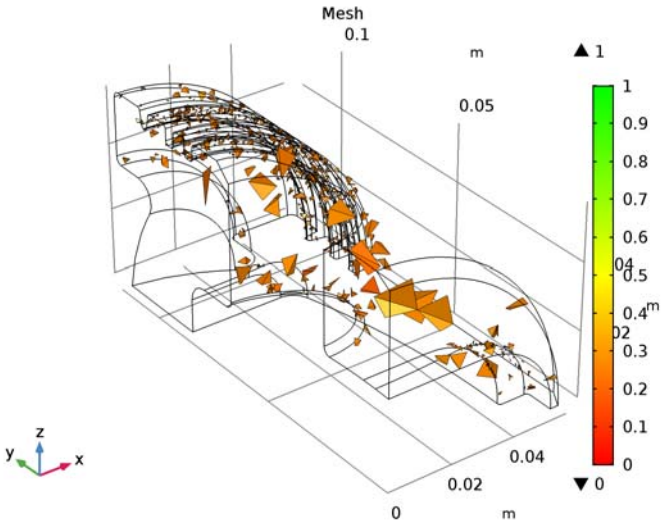
3D Plot Group 1

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

2 In the **Settings** window for **3D Plot Group**, locate the **Color Legend** section.

3 Select the **Show maximum and minimum values** check box.

4 On the **3D Plot Group 1** toolbar, click **Plot**.



You can now see 0.5% of the tetrahedral elements with the worst quality. These are mostly located in the regions where the elements are growing from the surfaces toward the inside of the geometry.

Quadrupole Lens

Introduction

Just like optical lenses focus light, electric and magnetic lenses can focus beams of charged particles. Systems of magnetic quadrupole lenses find a common use in focusing both ion and particle beams in accelerators at nuclear and particle physics centers such as CERN, SLAC, and ISIS. This COMSOL Multiphysics model shows the path of B^{5+} ions going through three consecutive magnetic quadrupole lenses. The model is set up in a cross section of the geometry.

Model Definition

The quadrupole consists of an assembly of four permanent magnets, as seen in Figure 1 below, where the magnets work together to give a good approximation of a quadrupole field. To strengthen the field and keep it contained within the system, the magnets are set in an iron cylinder.

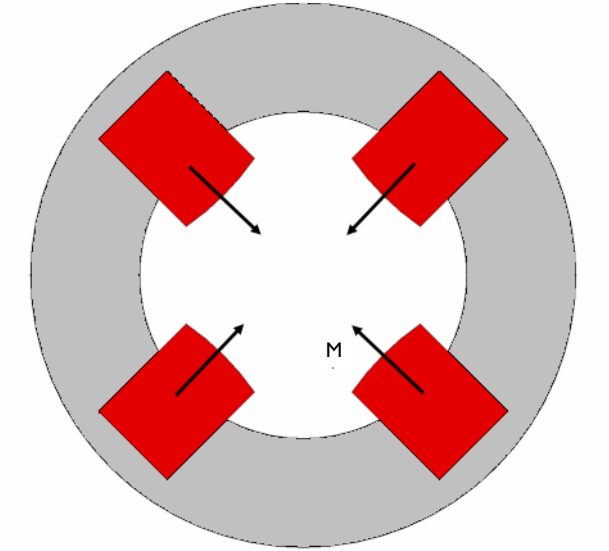


Figure 1: Cross-sectional view of one of the magnetic quadrupoles used in the lens.

The ions are sent through a system of three consecutive quadrupole assemblies. The middle one is twice as long as the other ones, and is rotated by 90 degrees around the central axis. This means the polarity of its magnets is reversed. Figure 2 gives a full view of

the magnetic quadrupole lens.

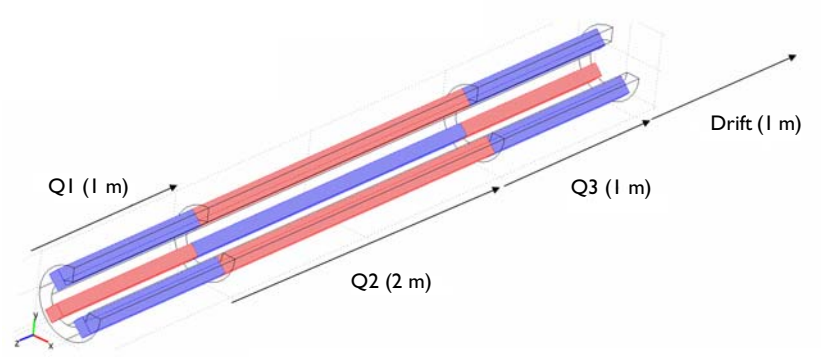


Figure 2: Cutout of the quadrupole lens. The second quadrupole (Q2) has its polarities reversed compared to Q1 and Q3. After traveling through the lens, the ions are left to drift 1 m.

An accelerator feeds the system with ions traveling with the velocity $0.01\ c$ along the central axis. To study the focusing effect of the quadrupoles, track a number of ions starting out from a distance of 3 cm from the central axis, evenly distributed along the circumference of a circle in the transverse plane. They are all assumed to have a zero initial transverse velocity. Each quadrupole focuses the ion beam along one of the transverse axes and defocuses it along the other one. The net effect after traveling through the system of the three quadrupoles and the drift length is focusing in all directions. As the ions exit the system, they are all contained within a 1 cm radius in the transverse plane.

The model is set up in a 2D cross section of any of the two identical quadrupoles Q1 and Q3. Neglecting fringe fields, the transverse magnetic field at a given point in a transverse plane in Q2 automatically has the same magnitude as the corresponding in Q1 and Q3, but point in the opposite direction. It is therefore sufficient to model the fields in one of the quadrupoles.

DOMAIN EQUATIONS

The magnetic field is described using the Magnetostatics equation, solving for the z component of the magnetic potential $\mathbf{A}$ (Wb/m):

$$\nabla \times (\mu_0^{-1}(\nabla \times A_z - \mathbf{M})) - \sigma \mathbf{v} \times (\nabla \times A_z) = J_z^{\mathrm{e}}$$

Here $\mu_0 = 4\pi \cdot 10^{-7}$ H/m denotes the permeability of vacuum, $\mathbf{M}$ is the magnetization (A/m), σ the conductivity (S/m), and $\mathbf{v}$ the velocity of the medium (m/s). In this example, the medium is not moving. The right-hand side of the equation holds an imposed current,

specified in terms of an external current density $J_z^{\,e}$ (A/m^2). No currents are imposed. The iron domain uses a slightly different formulation of the same equation:

$$\left(\nabla \times \frac{1}{\mu_0 \mu_r} \nabla \times A_z\right) - \sigma \mathbf{v} \times (\nabla \times A_z) = J_z^{\,e}$$

where $\mu_r = 4000$ is the relative permeability. The magnetic potential is everywhere defined so that $\mathbf{B} = \nabla \times \mathbf{A}$.

BOUNDARY CONDITIONS

The magnetic field is approximately parallel to the exterior boundary of the iron cylinder. To enforce this, use the *magnetic insulation* boundary condition, stating that $A_z = 0$.

Results and Discussion

The magnetic field density and streamlines in a cross section of Q1 or Q3 appear in Figure 3 below.

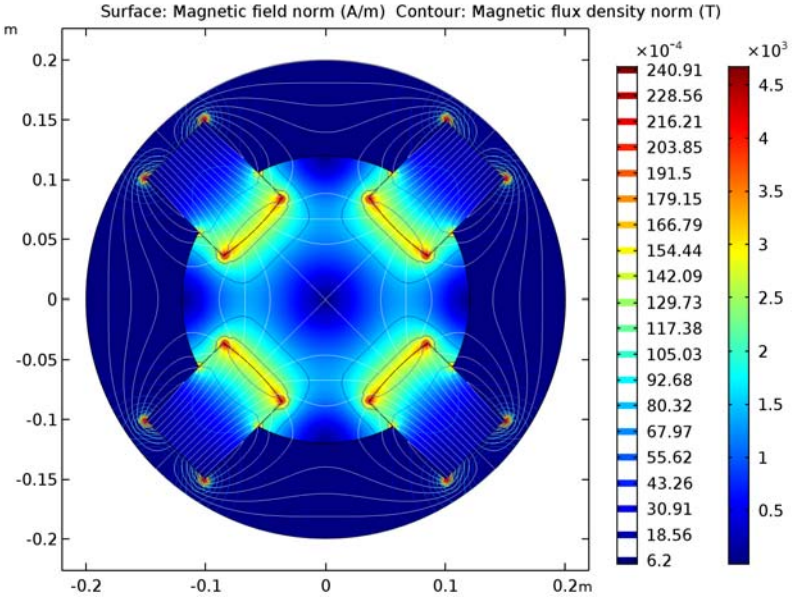


Figure 3: The magnetic field density and flow lines in the center of one of the quadrupole magnets.

Each ion passing through the assembly experiences a Maxwell force equal to $\mathbf{F} = q\mathbf{v} \times \mathbf{B}$, where $\mathbf{v}$ (m/s) is the ion's velocity. Next, assume that the z component of the velocity is

constant and much larger than the x and y (transverse) components. Thus consider only the force contributions from the z component of the velocity. To find the transverse position as a function of time, you need to solve Newton's second law for each ion, $q\mathbf{v} \times \mathbf{B} = m\mathbf{a}$, where m is the ion mass (kg), and $\mathbf{a}$ denotes its acceleration (m/s^2). If the computed magnetic flux density in Q1 equals $\mathbf{B}'$, and the length of quadrupole i is L_i (m), the flux density that the ion experiences is given by

$$\mathbf{B} = \begin{cases} \mathbf{B}' \text{ if } t < \dfrac{L_1}{v_z} \\ -\mathbf{B}' \text{ if } \dfrac{L_1}{v_z} < t < \dfrac{L_1 + L_2}{v_z} \\ \mathbf{B}' \text{ if } \dfrac{L_1 + L_2}{v_z} < t < \dfrac{L_1 + L_2 + L_3}{v_z} \\ 0 \text{ if } t > \dfrac{L_1 + L_2 + L_3}{v_z} \end{cases}$$

where t (s) is the time of flight. This dependency of the magnetic flux density on the time of flight is fed to the particle-tracing algorithm as a logical expression. Figure 4 below shows how the ions travel in the transverse plane.

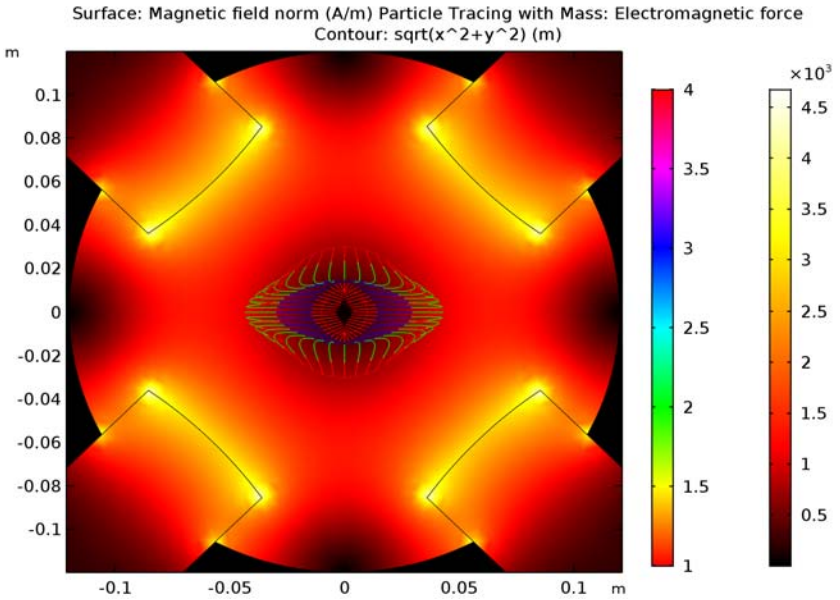


Figure 4: As the ions enter Q1, they start out evenly distributed around the larger circle, 3 cm from the z-axis. Q1 focuses along the x-axis and defocuses along the y-axis. The force on each ion is approximately proportional to its distance from the z-axis, so as the ions enter Q2, those that are far out on the x-axis rapidly turn around and move toward the center. Q3 stabilizes the motion and gets all ions on the right track. Finally the ions are left to drift toward a waist situated a little bit more than 1 m beyond Q3.

Application Library path: `COMSOL_Multiphysics/Electromagnetics/quadrupole`

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 **AC/DC>Magnetic Fields (mf)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
M	11	11	Ion mass number
Z	5	5	Ion charge number
L1	1[m]	1 m	Length of first quadrupole
L2	2[m]	2 m	Length of second quadrupole
L3	1[m]	1 m	Length of third quadrupole
vz	0.01*c_const	2.9979E6 m/s	Ion velocity
m	M*mp_const	1.8399E-26 kg	Ion mass
q	Z*e_const	8.0109E-19 C	Ion charge
MQ	5.8e3[A/m]	5800 A/m	Quadrupole magnetization

Here, `c_const`, `mp_const` and `e_const` are predefined constants for, respectively, the speed of light, the mass of the proton and the elementary charge.

GEOMETRY 1

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `0.177`.

4 In the **Height** text field, type `0.07`.

5 Locate the **Position** section. In the **x** text field, type `0`.

6 In the **y** text field, type `-0.035`.

7 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

Rotate 1 (rot1)

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

2 Select the object **r1** only.

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

4 In the **Rotation** text field, type `45`.

5 Right-click **Rotate 1 (rot1)** and choose **Build Selected**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `0.2`.

4 Locate the **Position** section. In the **x** text field, type `0.2`.

5 In the **y** text field, type `0.2`.

6 Right-click **Circle 1 (c1)** and choose **Build Selected**.

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

Intersection 1 (int1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Intersection**.

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

3 Right-click **Intersection 1 (int1)** and choose **Build Selected**.

Rotate 2 (rot2)

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

2 Select the object **int1** only.

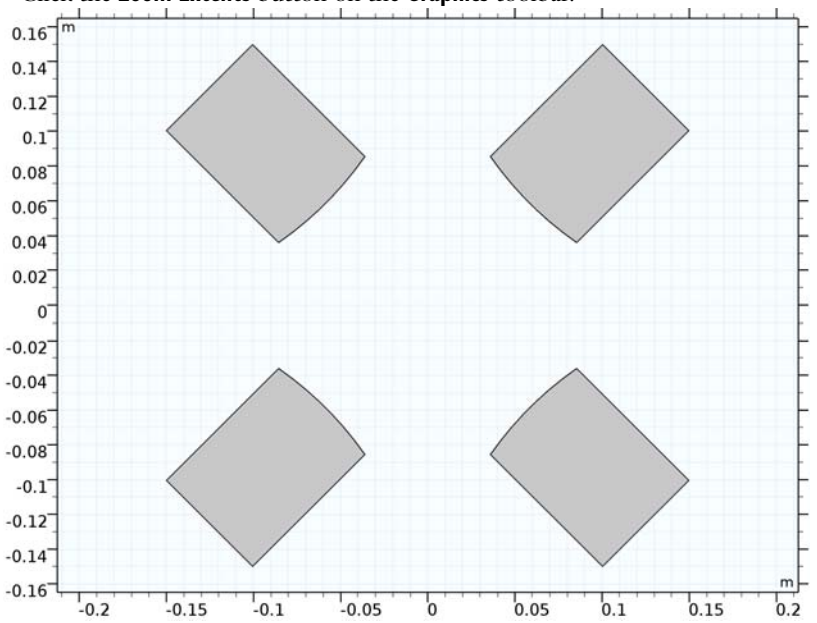
3 In the **Settings** window for **Rotate**, locate the **Rotation Angle** section.

4 In the **Rotation** text field, type `90, 180, 270`.

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

6 Right-click **Rotate 2 (rot2)** and choose **Build Selected**.

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



Now make two circles centered at the origin.

Circle 2 (c2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `0.2`.

4 Right-click **Circle 2 (c2)** and choose **Build Selected**.

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

Circle 3 (c3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `0.12`.

4 Right-click **Circle 3 (c3)** and choose **Build Selected**.

Create a composite object subtracting the magnets from the union of the two circles. This complex operation can be done in one step by using the Compose geometry operation and specifying the formula.

Compose 1 (co1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Compose**.

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

3 In the **Settings** window for **Compose**, locate the **Compose** section.

4 In the **Set formula** text field, type `c2+c3-(int1+rot2(1)+rot2(2)+rot2(3))`.

5 Right-click **Compose 1 (co1)** and choose **Build Selected**.

Make one more circle centered at the origin.

Circle 4 (c4)

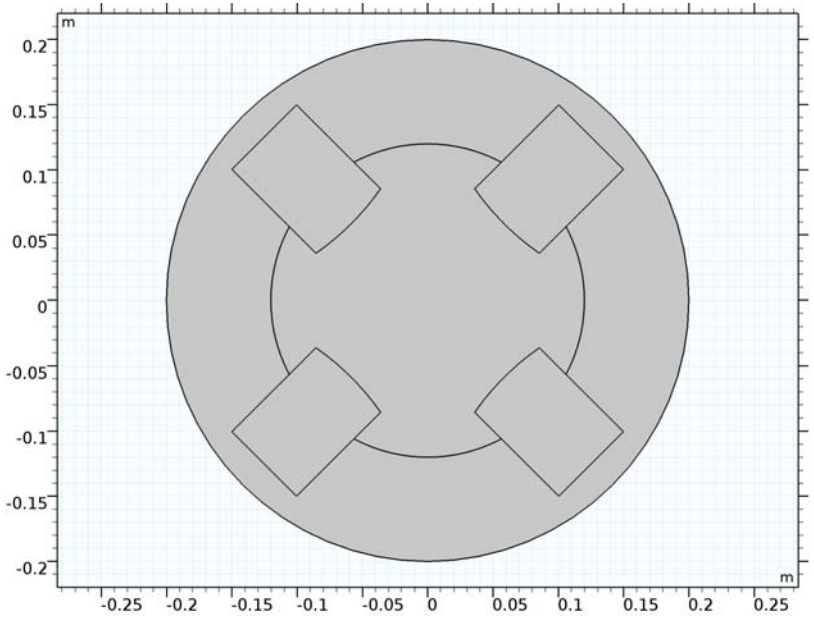
1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `0.2`.

4 Right-click **Circle 4 (c4)** and choose **Build Selected**.

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



ADD MATERIAL

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

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

3 In the tree, select **Built-In>Iron**.

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

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

MATERIALS

Iron (mat1)

Select Domain 2 only.

MAGNETIC FIELDS (MF)

Use the default condition, **Magnetic Insulation**, on all exterior boundaries.

Ampère's Law 2

1 On the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 Select Domain 6 only.

3 In the **Settings** window for **Ampère's Law**, locate the **Magnetic Field** section.

4 From the **Constitutive relation** list, choose **Magnetization**.

5 Specify the $\mathbf{M}$ vector as

-MQ/sqrt(2)	x
-MQ/sqrt(2)	y
0	z

6 Locate the **Conduction Current** section. From the σ list, choose **User defined**. Locate the **Electric Field** section. From the ε_r list, choose **User defined**.

Ampère's Law 3

1 On the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 Select Domain 3 only.

3 In the **Settings** window for **Ampère's Law**, locate the **Magnetic Field** section.

4 From the **Constitutive relation** list, choose **Magnetization**.

5 Specify the $\mathbf{M}$ vector as

-MQ/sqrt(2)	x
MQ/sqrt(2)	y
0	z

6 Locate the **Conduction Current** section. From the σ list, choose **User defined**. Locate the **Electric Field** section. From the ε_r list, choose **User defined**.

Ampère's Law 4

1 On the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 Select Domain 1 only.

3 In the **Settings** window for **Ampère's Law**, locate the **Magnetic Field** section.

4 From the **Constitutive relation** list, choose **Magnetization**.

5 Specify the $\mathbf{M}$ vector as

MQ/sqrt(2)	x
MQ/sqrt(2)	y
0	z

6 Locate the **Conduction Current** section. From the σ list, choose **User defined**. Locate the **Electric Field** section. From the ε_r list, choose **User defined**.

Ampère's Law 5

1 On the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 Select Domain 5 only.

3 In the **Settings** window for **Ampère's Law**, locate the **Magnetic Field** section.

4 From the **Constitutive relation** list, choose **Magnetization**.

5 Specify the $\mathbf{M}$ vector as

MQ/sqrt(2)	x
-MQ/sqrt(2)	y
0	z

6 Locate the **Conduction Current** section. From the σ list, choose **User defined**. Locate the **Electric Field** section. From the ε_r list, choose **User defined**.

Ampère's Law 6

1 On the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 Select Domain 4 only.

3 In the **Settings** window for **Ampère's Law**, locate the **Magnetic Field** section.

4 From the μ_r list, choose **User defined**. Locate the **Conduction Current** section. From the σ list, choose **User defined**. Locate the **Electric Field** section. From the ε_r list, choose **User defined**.

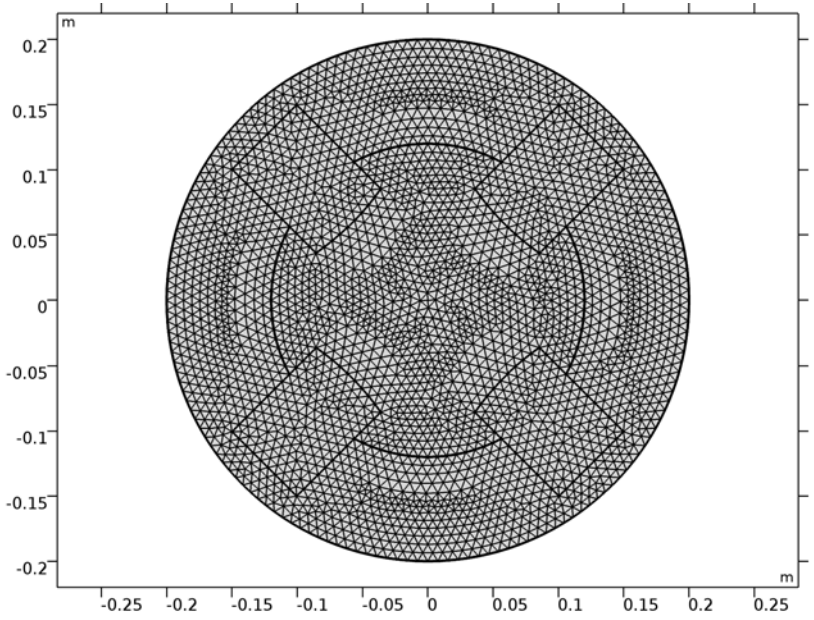
MESH 1

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

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

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

4 Click **Build All**.



STUDY 1

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

RESULTS

Magnetic Flux Density Norm (mf)

The default plot shows the norm of the magnetic flux density. Follow the instructions below to view the magnetic field.

Surface 1

1 In the **Model Builder** window, expand the **Magnetic Flux Density Norm (mf)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Magnetic Fields>Magnetic> mf.normH - Magnetic field norm**.

3 On the **Magnetic Flux Density Norm (mf)** toolbar, click **Plot**.

Contour 1

1 In the **Model Builder** window, under **Results** right-click **Magnetic Flux Density Norm (mf)** and choose **Contour**.

2 In the **Settings** window for **Contour**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Magnetic Fields>Magnetic> Magnetic vector potential>Az - Magnetic vector potential, z component**.

3 On the **Magnetic Flux Density Norm (mf)** toolbar, click **Plot**.

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

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

6 Clear the **Color legend** check box.

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

To see how the ions travel through the system of quadrupoles, do the following:

2D Plot Group 2

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

2 In the **Settings** window for **2D Plot Group**, type `Ions Trajectories` in the **Label** text field.

Surface 1

1 Right-click **Ions Trajectories** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Magnetic Fields>Magnetic> mf.normH - Magnetic field norm**.

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

Ions Trajectories

In the **Model Builder** window, under **Results** click **Ions Trajectories**.

Particle Tracing with Mass 1

1 On the **Ions Trajectories** toolbar, click **More Plots** and choose **Particle Tracing with Mass**.

Enter the expressions for the Lorentz force acting on the particles.

The position of the particle along the z-axis can be computed using the particle time variable `partt`.

2 In the **Settings** window for **Particle Tracing with Mass**, locate the **Total Force** section.

3 In the **FX** text field, type `-q*vz*mf.By*(1-2*(partt>L1/vz)+2*(partt>(L1+L2)/vz)-(partt>(L1+L2+L3)/vz))`.

4 In the **FY** text field, type `q*vz*mf.Bx*(1-2*(partt>L1/vz)+2*(partt>(L1+L2)/vz)-(partt>(L1+L2+L3)/vz))`.

5 Click to expand the **Mass and velocity** section. Locate the **Mass and Velocity** section. In the **Mass** text field, type `m`.

6 Locate the **Particle Positioning** section. In the **X** text field, type `0.03*cos(range(0,0.05*pi,2*pi))`.

7 In the **Y** text field, type `0.03*sin(range(0,0.05*pi,2*pi))`.

Color Expression 1

1 Right-click **Particle Tracing with Mass 1** and choose **Color Expression**.

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

3 In the **Expression** text field, type `1+(partt>L1/vz)+(partt>(L1+L2)/vz)+(partt>(L1+L2+L3)/vz)`.

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

Particle Tracing with Mass 1

1 In the **Model Builder** window, under **Results>Ions Trajectories** click **Particle Tracing with Mass 1**.

2 In the **Settings** window for **Particle Tracing with Mass**, click to expand the **Release** section.

3 Click to expand the **Coloring and style** section. Click to expand the **Quality** section. Find the **ODE solver settings** subsection. In the **Relative tolerance** text field, type `1e-6`.

4 Click to expand the **Advanced** section. Find the **Termination** subsection. Select the **Maximum number of steps** check box.

5 In the associated text field, type `1e5`.

6 Find the **Instantaneous flow field** subsection. Select the **End time** check box.

7 In the associated text field, type `5/3e6`.

8 On the **Ions Trajectories** toolbar, click **Plot**.

Contour 1

1 In the **Model Builder** window, under **Results** right-click **Ions Trajectories** and choose **Contour**.

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

3 In the **Expression** text field, type `sqrt(x^2+y^2)`.

4 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

5 In the **Levels** text field, type `0.01 0.03`.

6 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

7 From the **Color** list, choose **Custom**.

8 Clear the **Color legend** check box.

9 On the **Ions Trajectories** toolbar, click **Plot**.

Zoom in on the center of the model geometry.

Submodeling of a Shaft

Introduction

Submodeling can be used when it is not possible to resolve all details of a complex geometry in a global model.

You can cope with this type of problems with a technique known as *submodeling*. First you solve the complete model with a mesh which is sufficient to capture the stiffness of the structure. In a second analysis you create a local model (submodel) of the region around the stress concentration with a fine mesh, and solve it using the displacements from the global model as boundary conditions.

There are some underlying assumptions when using submodels:

- The global model is accurate enough to give correct displacements on the boundary to the submodel.
- The improvements introduced in the submodel are so small that they do not introduce significant changes in stiffness on the global level. Given this, it could still be possible to introduce a nonlinear material locally in the submodel.

In this example this technique is applied to perform an accurate stress evaluation in a structural mechanics model, but the same approach is applicable to many physical problems. The example geometry as such is not so complicated, so there is nothing to gain from a submodeling in this case. The purpose of the example is to show the technique.

Model Definition

The geometry consists of a shaft with a sudden change in diameter. At the location of the diameter change, there is a fillet with a small radius. In the fillet, stress concentrations will appear. There is also a central hole through the shaft. The geometry and mesh are shown in Figure 1.

The shaft is fixed at the thick end. On the thin end, a tensile force of 300 N and a shear force of 100 N are applied.

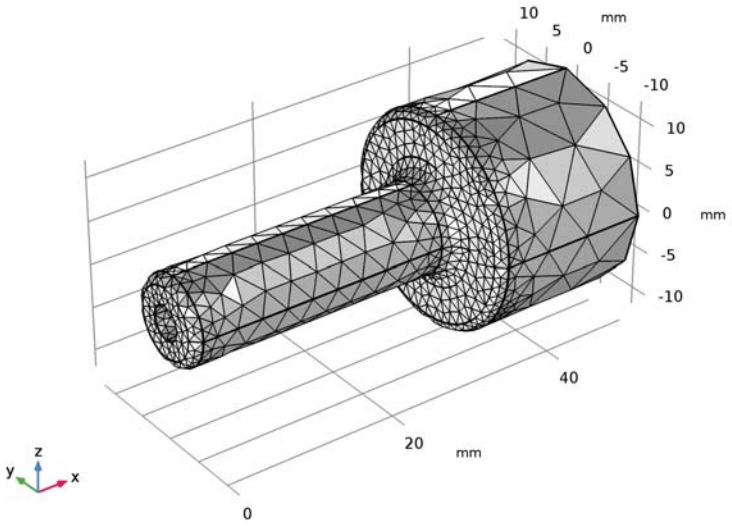


Figure 1: The full model.

As submodel, a region around the fillet at side giving the highest stress is chosen. The cuts where the boundary conditions are applied on the submodel should preferably be placed at locations where the stress field is fairly smooth.

The geometry and mesh of the submodel are shown in Figure 2. As can be seen, the fillet has a very good mesh resolution since the purpose is to obtain results with high accuracy there.

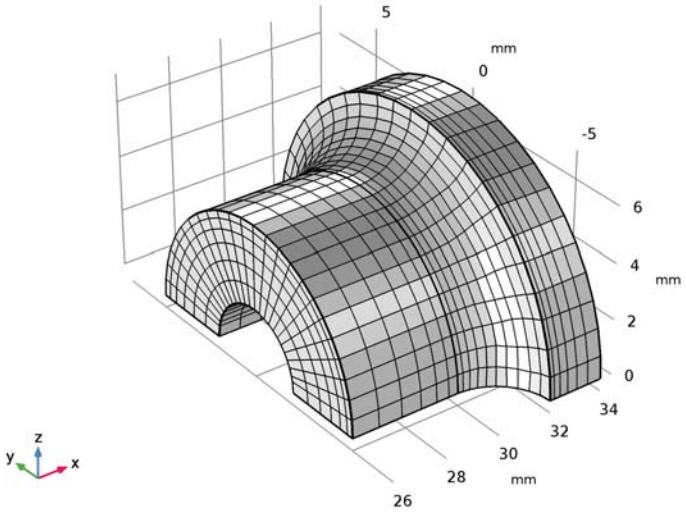


Figure 2: The submodel.

Results

The general stress distribution is shown in Figure 3. Since the load is unsymmetrical (there is both an axial and a bending component), the highest stress occurs at the side with positive Z coordinate.

The cut through the model in Figure 4 displays that the stresses are not well resolved. There are significant jumps between the neighboring elements.

In the corresponding figures from the submodel, Figure 5 and Figure 6, the stress field is smooth and well resolved.

The computed peak stress for the global model is about 9% different compared to that in the submodel, which is expected given the coarse mesh in the global model.

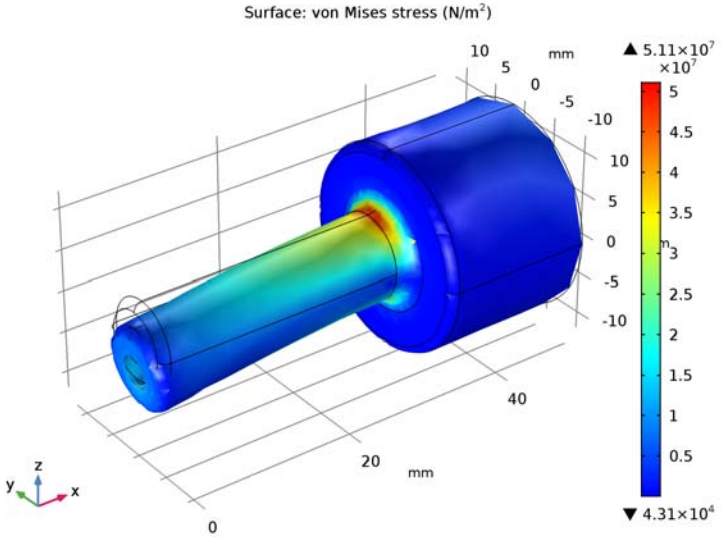


Figure 3: Stress distribution in the global model.

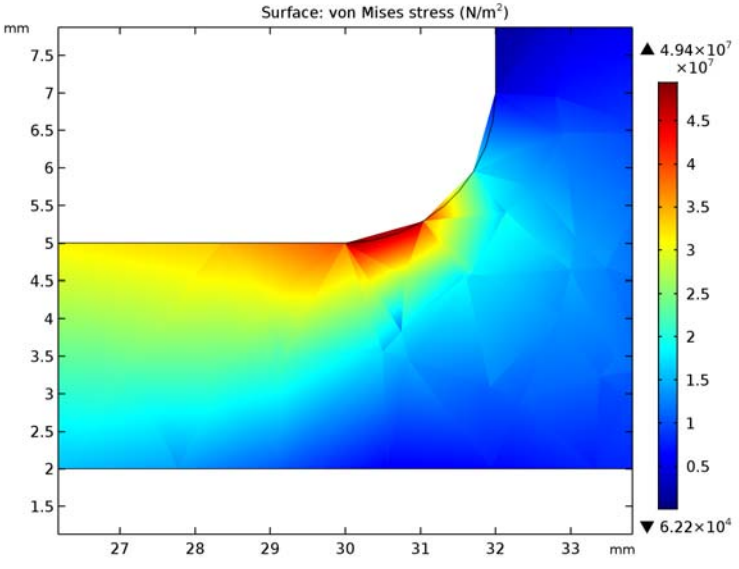


Figure 4: Stress within the full model (cut view).

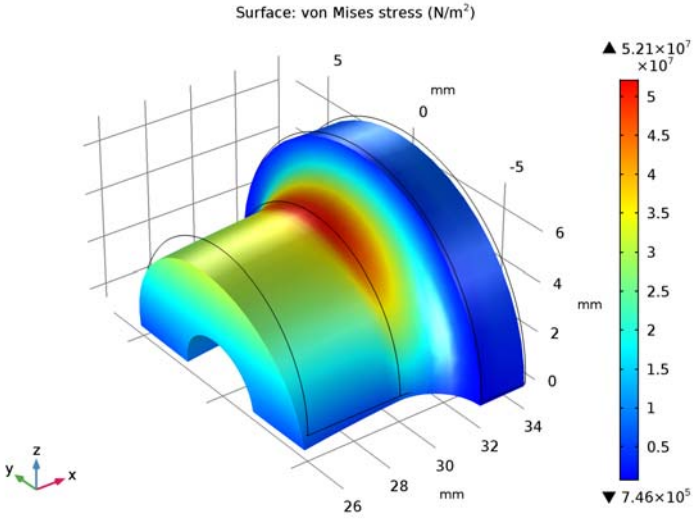


Figure 5: Stress distribution in the submodel.

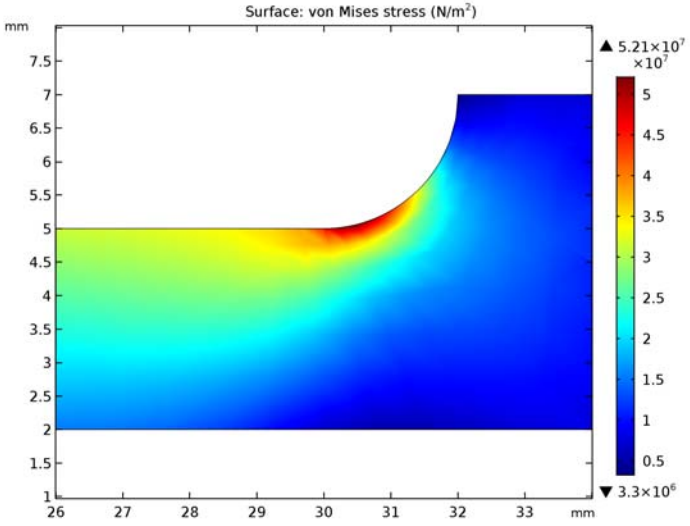


Figure 6: Stress within the submodel (cut view).

As a verification, it is a good habit to always check the stress state at the cuts where the submodel has displacements prescribed by the results in the global model. Such a comparison is shown in Figure 7. The results have an almost perfect match, which strongly indicates that the submodel has been set up correctly.

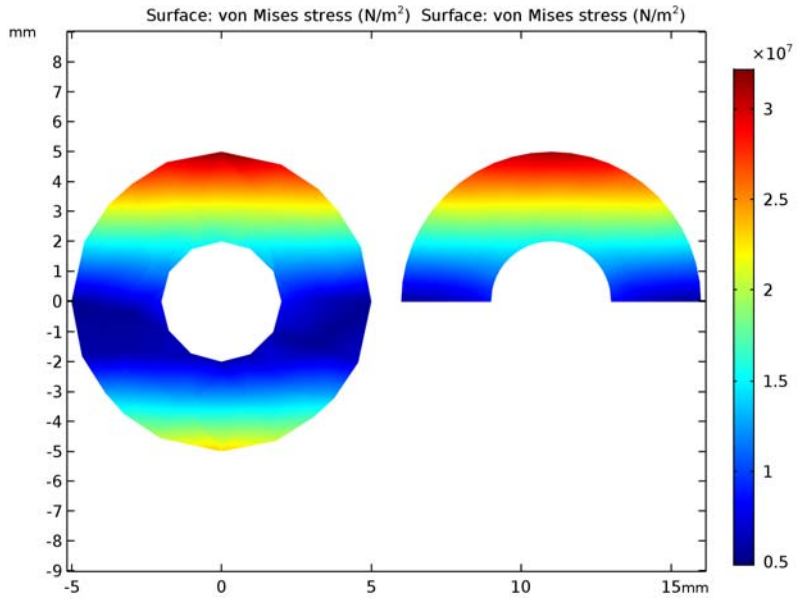


Figure 7: Stress distribution over one of the cuts. Global model (left) and submodel (right) are compared.

Notes About the COMSOL Implementation

Two different components are used within the same mph file. In the global model, a general extrusion feature is introduced in order to describe the mapping of results from the global model to the submodel. The prescribed displacements on the cut boundaries in the submodel reference the displacements in the global model through this mapping.

In this example, there are no volume forces. If there were, such forces must be applied also on the submodel.

Application Library path: `COMSOL_Multiphysics/Structural_Mechanics/shaft_submodeling`

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 **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

COMPONENT 1 (COMP1)

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

2 In the **Settings** window for **Component**, type `Full model` in the **Label** text field.

GEOMETRY 1

1 In the **Model Builder** window, under **Full model (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Import 1 (imp1)

1 On the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

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

5 Click **Import**.

ADD MATERIAL

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

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

3 In the tree, select **Built-In>Structural steel**.

4 Click **Add to Global Materials**.

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

The material is now defined as global material so that both the full model and the submodel can use the same material definition, defined at one location only in the model tree.

MATERIALS

In the **Model Builder** window, under **Full model (comp1)** right-click **Materials** and choose **Material Link**.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Select Boundary 27 only.

Boundary Load 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 From the **Load type** list, choose **Total force**.

5 Specify the $\mathbf{F}_{\text{tot}}$ vector as

-300	x
0	y
-100	z

MESH 1

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

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

3 From the **Element size** list, choose **Coarse**.

4 Click **Build All**.

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

STUDY 1

1 In the **Settings** window for **Study**, type `Full model` in the **Label** text field.

2 On the **Home** toolbar, click **Compute**.

RESULTS

Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 In the **Settings** window for **3D Plot Group**, type `Stress - Full model` in the **Label** text field.

3 Click to expand the **Color legend** section. Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Surface 1

1 In the **Model Builder** window, expand the **Results>Stress - Full model** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click to expand the **Quality** section.

3 From the **Resolution** list, choose **No refinement**.

4 From the **Smoothing** list, choose **None**.

5 On the **Stress - Full model** toolbar, click **Plot**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

Cut Plane 1

1 On the **Results** toolbar, click **Cut Plane**.

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

3 From the **Plane** list, choose **XZ-planes**.

2D Plot Group 2

1 On the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Cut Plane Stress -Full model` in the **Label** text field.

3 Click to expand the **Color legend** section. Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Surface 1

1 Right-click **Cut Plane Stress -Full model** and choose **Surface**.

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

3 In the **Expression** text field, type `solid.mises`.

4 Click to expand the **Quality** section. From the **Resolution** list, choose **No refinement**.

5 From the **Smoothing** list, choose **None**.

6 On the **Cut Plane Stress -Full model** toolbar, click **Plot**.

7 In the **Model Builder** window, expand the **Results>Views** node.

Axis

1 In the **Model Builder** window, expand the **Results>Views>View 2D 2** node, then click **Axis**.

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

3 In the **x minimum** text field, type `25.8`.

4 In the **x maximum** text field, type `34.2`.

5 In the **y minimum** text field, type `1.5`.

6 In the **y maximum** text field, type `7.5`.

7 Click **Update**.

ROOT

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

COMPONENT 2 (COMP2)

1 In the **Model Builder** window, click **Component 2 (comp2)**.

2 In the **Settings** window for **Component**, type `Submodel` in the **Label** text field.

GEOMETRY 2

1 In the **Model Builder** window, under **Submodel (comp2)** click **Geometry 2**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Import 1 (imp1)

1 On the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

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

5 Click **Import**.

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

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

3 In the **Radius** text field, type 7.

4 In the **Height** text field, type 8.

5 Locate the **Position** section. In the **x** text field, type 26.

6 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

Block 1 (blk1)

1 On 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 10.

4 In the **Depth** text field, type 14.

5 In the **Height** text field, type 10.

6 Locate the **Position** section. In the **x** text field, type 26.

7 In the **y** text field, type -7.

Intersection 1 (int1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Intersection**.

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

3 In the **Settings** window for **Intersection**, click **Build All Objects**.

Ignore Edges 1 (ige1)

1 On the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Edges**.

2 On the object **fin**, select Edges 7, 9, 16, and 21 only.

3 In the **Settings** window for **Ignore Edges**, click **Build Selected**.

MATERIALS

In the **Model Builder** window, under **Submodel (comp2)** right-click **Materials** and choose **Material Link**.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

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

3 In the tree, select **Structural Mechanics>Solid Mechanics (solid)**.

4 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Studies	Solve
Full model	√

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

6 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

DEFINITIONS

General Extrusion 1 (genext1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **General Extrusion**.

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

3 In the **Settings** window for **General Extrusion**, locate the **Destination Map** section.

4 In the **x-expression** text field, type `X`.

5 In the **y-expression** text field, type `Y`.

6 In the **z-expression** text field, type `Z`.

7 Locate the **Source** section. From the **Source frame** list, choose **Material (X, Y, Z)**.

You have now created the coupling operator which will be used for mapping the solution from the full model to the submodel.

SOLID MECHANICS 2 (SOLID2)

In the **Model Builder** window, under **Submodel (comp2)** click **Solid Mechanics 2 (solid2)**.

Prescribed Displacement 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

2 Select Boundaries 1, 2, 5, 7, and 8 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in x direction** check box.

5 In the u_{0x} text field, type `comp1.genext1(comp1.u)`.

6 Select the **Prescribed in y direction** check box.

7 In the u_{0y} text field, type `comp1.genext1(comp1.v)`.

8 Select the **Prescribed in z direction** check box.

9 In the u_{0z} text field, type `comp1.genext1(comp1.w)`.

FULL MODEL (COMP1)

In the **Model Builder** window, expand the **Full model (comp1)** node.

MESH 2

In the **Model Builder** window, expand the **Full model (comp1)>Definitions** node.

Mapped 1

1 Right-click **Submodel (comp2)>Mesh 2** and choose **More Operations>Mapped**.

2 Select Boundary 2 only.

Distribution 1

1 Right-click **Submodel (comp2)>Mesh 2>Mapped 1** and choose **Distribution**.

2 Select Edge 3 only.

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

4 From the **Distribution properties** list, choose **Predefined distribution type**.

5 In the **Number of elements** text field, type 6.

6 In the **Element ratio** text field, type 3.

7 Select the **Reverse direction** check box.

Distribution 2

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

2 Select Edge 1 only.

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

4 From the **Distribution properties** list, choose **Predefined distribution type**.

5 In the **Number of elements** text field, type 6.

6 In the **Element ratio** text field, type 3.

7 Select the **Reverse direction** check box.

Distribution 3

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

2 Select Edge 18 only.

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

4 In the **Number of elements** text field, type 6.

Distribution 4

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

2 Select Edge 10 only.

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

4 From the **Distribution properties** list, choose **Predefined distribution type**.

5 In the **Number of elements** text field, type 10.

6 In the **Element ratio** text field, type 2.

Distribution 5

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

2 Select Edge 13 only.

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

4 From the **Distribution properties** list, choose **Predefined distribution type**.

5 In the **Number of elements** text field, type 6.

6 In the **Element ratio** text field, type 3.

Swept 1

1 In the **Model Builder** window, right-click **Mesh 2** and choose **Swept**.

2 In the **Settings** window for **Swept**, click to expand the **Destination faces** section.

3 Select Boundary 2 only.

4 Locate the **Destination Faces** section. Click **Clear Selection**.

5 Select Boundary 5 only.

6 Click to expand the **Source faces** section. Select Boundary 2 only.

Distribution 1

1 In the **Model Builder** window, 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 24.

4 Click **Build All**.

5 In the **Settings** window for **Mesh**, click **Build All**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

ADD STUDY

1 On 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 **Preset Studies>Stationary**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Solid Mechanics (solid)** interface.

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

6 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

In the **Settings** window for **Study**, type `Submodel` in the **Label** text field.

SUBMODEL

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, click to expand the **Values of dependent variables** section.

3 Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

4 From the **Method** list, choose **Solution**.

5 From the **Study** list, choose **Full model, Stationary**.

6 On the **Home** toolbar, click **Compute**.

RESULTS

Stress (solid2)

1 In the **Model Builder** window, under **Results** click **Stress (solid2)**.

2 In the **Settings** window for **3D Plot Group**, type `Stress - Submodel` in the **Label** text field.

3 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Surface 1

1 In the **Model Builder** window, expand the **Results>Stress - Submodel** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click to expand the **Quality** section.

3 From the **Resolution** list, choose **No refinement**.

4 From the **Smoothing** list, choose **None**.

5 On the **Stress - Submodel** toolbar, click **Plot**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

Cut Plane 2

1 On the **Results** toolbar, click **Cut Plane**.

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

3 From the **Plane** list, choose **XZ-planes**.

4 Locate the **Data** section. From the **Data set** list, choose **Submodel/Solution 2 (3) (sol2)**.

2D Plot Group 4

1 On the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Cut Plane Stress - Submodel` in the **Label** text field.

3 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

4 Locate the **Data** section. From the **Data set** list, choose **Cut Plane 2**.

Surface 1

1 Right-click **Cut Plane Stress - Submodel** and choose **Surface**.

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

3 In the **Expression** text field, type `solid2.mises`.

4 Click to expand the **Quality** section. From the **Resolution** list, choose **No refinement**.

5 From the **Smoothing** list, choose **None**.

6 On the **Cut Plane Stress - Submodel** toolbar, click **Plot**.

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

Now verify that the differences in stress between the full model and the submodel are small where the displacements are mapped.

Cut Plane 3

1 On the **Results** toolbar, click **Cut Plane**.

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

3 In the **X-coordinate** text field, type `26`.

2D Plot Group 5

1 On the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Stress comparison` in the **Label** text field.

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

Surface 1

1 Right-click **Stress comparison** and choose **Surface**.

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

3 From the **Data set** list, choose **Cut Plane 3**.

4 Locate the **Expression** section. In the **Expression** text field, type `solid.mises`.

5 Locate the **Quality** section. From the **Resolution** list, choose **No refinement**.

6 From the **Smoothing** list, choose **None**.

7 On the **Stress comparison** toolbar, click **Plot**.

Data Sets

In the **Model Builder** window, under **Results** click **Data Sets**.

Surface 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Surface**.

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

3 From the **Data set** list, choose **Submodel/Solution 2 (3) (sol2)**.

4 Select Boundary 1 only.

5 Locate the **Parameterization** section. From the **x- and y-axes** list, choose **YZ-plane**.

Surface 2

1 In the **Model Builder** window, under **Results** right-click **Stress comparison** and choose **Surface**.

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

3 From the **Data set** list, choose **Surface 1**.

4 Locate the **Expression** section. In the **Expression** text field, type `solid2.mises`.

5 Locate the **Quality** section. From the **Resolution** list, choose **No refinement**.

6 From the **Smoothing** list, choose **None**.

7 Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Deformation 1

1 Right-click **Results>Stress comparison>Surface 2** and choose **Deformation**.

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

3 In the **x component** text field, type `11`.

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

5 Locate the **Scale** section. Select the **Scale factor** check box.

6 In the associated text field, type 1.

7 On the **Stress comparison** toolbar, click **Plot**.

8 Click the **Zoom Extents** button on the **Graphics** toolbar.

The solution is computed, follow the steps below if you want to save and reuse the model in the future.

FULL MODEL

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for the **Solid Mechanics 2 (solid 2)** interface.

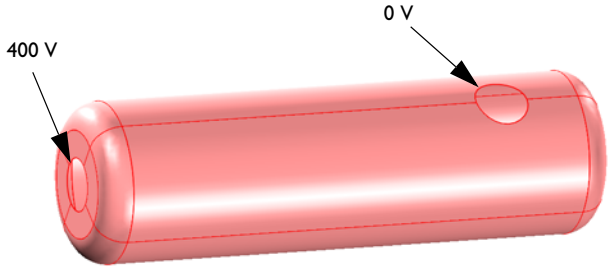
Shell Diffusion in a Tank

Introduction

A goal for many applications is to predict physics in thin structures, such as shells, without modeling the thickness of the structure. This is because large aspect ratios can cause meshing and geometry analysis problems. The model reported here demonstrates how to use the *tangential derivative variables* in COMSOL Multiphysics to solve partial differential equations in curved 3D shells and 2D boundaries without modeling their thickness.

Model Definition

The steel tank shown below has two pipe connections. One is grounded and the other connects to a dead current source. This model calculates the current density in the tank shell along with the potential distribution across the surface.



EQUATIONS

The fundamental equation to solve is the current conduction, or charge conservation, equation.

$$\nabla \cdot (-\sigma \nabla V) = 0 \tag{1}$$

Here, σ is the electrical conductivity (S/m) and V is the electric potential (V).

The material is a 1 mm thick steel sheet with a conductivity of $4.032 \cdot 10^6$ S/m. You are working with a surface in 3D so there is no thickness in the model. To account for the charge conservation in Equation 1 you must multiply the current flux expression with the shell thickness d:

$$\nabla \cdot (-\sigma d \nabla V) = 0 \tag{2}$$

Figure 1 shows the potential distribution across the surface.

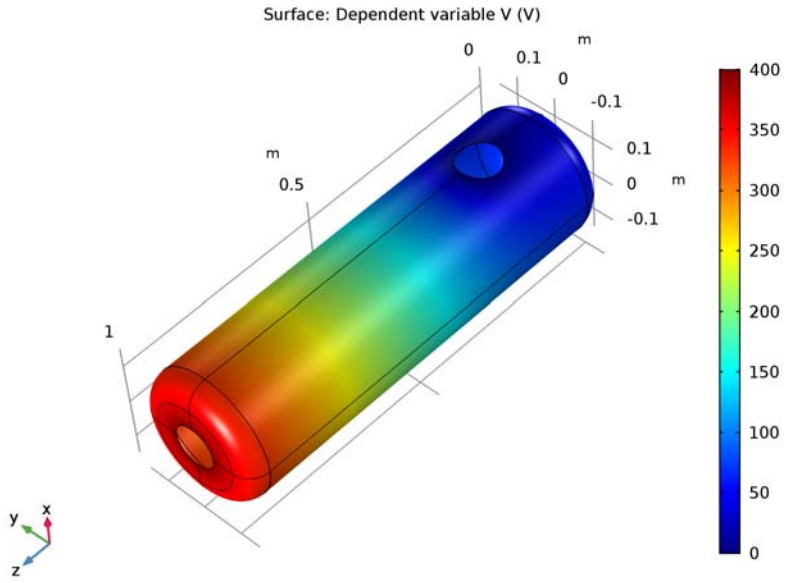


Figure 1: Electric potential distribution across the surface (V).

Figure 2 adds the current field as an arrow plot, showing clearly how the current collects toward the grounded connection.

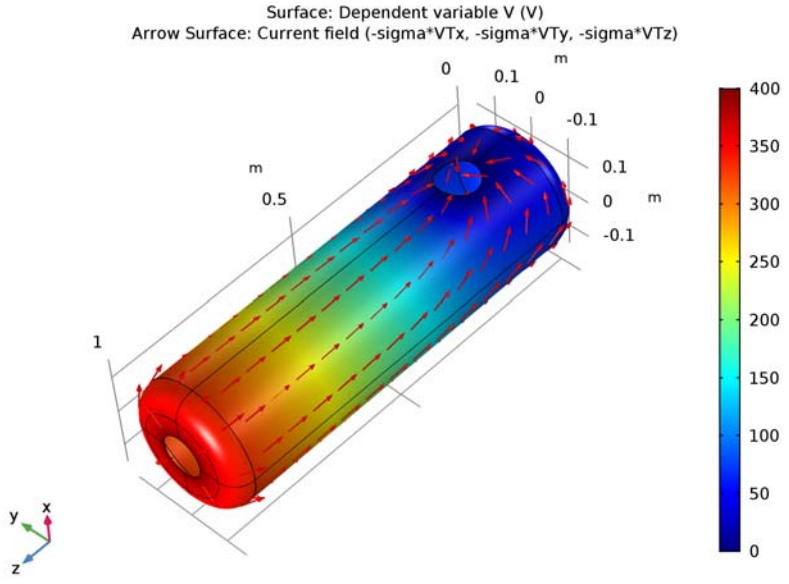


Figure 2: Arrow plot of the local current field.

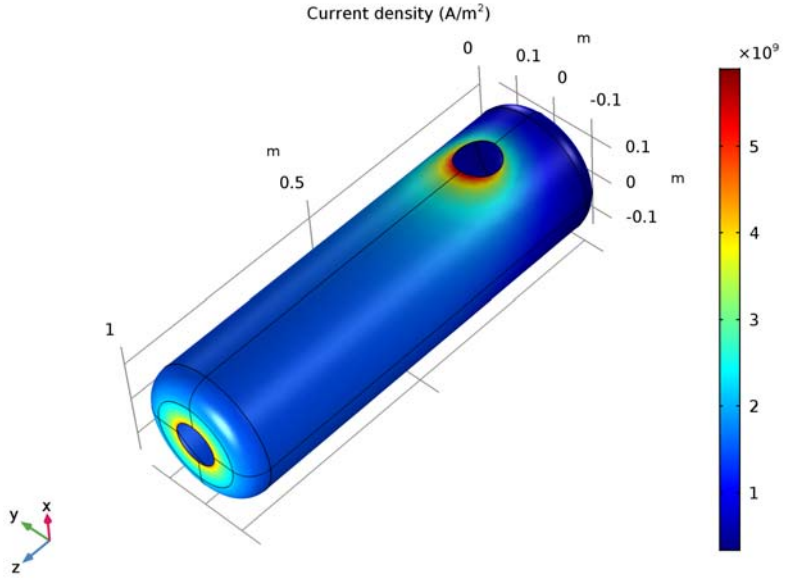


Figure 3: Local magnitude of the electric current density (A/m^2).

The plot of the magnitude of the local current density in Figure 3 is interesting because you can use it to calculate the resistive heating in the material as an extension to the model.

Notes About the COMSOL Implementation

Model Equation 2, the current conduction equation, using a Coefficient Form Boundary PDE interface, setting the diffusion coefficient $c = \sigma d$. To define the current field components use tangential derivative variables, which you access in COMSOL Multiphysics by adding a `T` suffix to the variable name before specifying the gradient component. So, for example, the tangential derivative $(\partial u/\partial x)_T$ is represented by the variable `uTx`.

Application Library path: `COMSOL_Multiphysics/Equation_Based/shell_diffusion`

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 **Mathematics>PDE Interfaces>Lower Dimensions>Coefficient Form Boundary PDE (cb)**.
3. Click **Add**.
4. In the **Dependent variables** table, enter the following settings:

V

5. Click **Study**.
6. In the **Select Study** tree, select **Preset Studies>Stationary**.
7. Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
sigma	4.032e6[S/m]	4.032E6 S/m	Conductivity
d	1[mm]	0.001 m	Shell thickness

GEOMETRY 1

Create the geometry. To simplify this step, insert a prepared geometry sequence.

1 On the **Geometry** toolbar, click **Insert Sequence**.

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

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

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

COEFFICIENT FORM BOUNDARY PDE (CB)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Coefficient Form Boundary PDE (cb)**.

2 In the **Settings** window for **Coefficient Form Boundary PDE**, locate the **Units** section.

3 Click **Select Dependent Variable Quantity**.

4 In the **Physical Quantity** dialog box, In the associated text field, type `id:electricpotential`.

5 Click **Filter**.

6 In the tree, select **Maxwell>Electric potential (V)**.

7 Click **OK**.

8 In the **Settings** window for **Coefficient Form Boundary PDE**, locate the **Units** section.

9 In the **Source term quantity** table, enter the following settings:

Source term quantity	Unit
Custom unit	A*m^-2

Coefficient Form PDE 1

1 In the **Model Builder** window, expand the **Coefficient Form Boundary PDE (cb)** node, then click **Coefficient Form PDE 1**.

2 In the **Settings** window for **Coefficient Form PDE**, locate the **Diffusion Coefficient** section.

3 In the *c* text field, type `sigma*d`.

4 Locate the **Source Term** section. In the *f* text field, type `0`.

These settings specify the charge conservation equation (Equation 2) for the shell surface.

Go on to set the values of the potential at the pipe connections by adding Dirichlet boundary conditions.

Dirichlet Boundary Condition 1

1 On the **Physics** toolbar, click **Edges** and choose **Dirichlet Boundary Condition**.

2 Select Edges 14, 15, 25, and 29 only.

3 In the **Settings** window for **Dirichlet Boundary Condition**, locate the **Dirichlet Boundary Condition** section.

4 In the *r* text field, type `400`.

Dirichlet Boundary Condition 2

1 On the **Physics** toolbar, click **Edges** and choose **Dirichlet Boundary Condition**.

2 Select Edges 40–43 only.

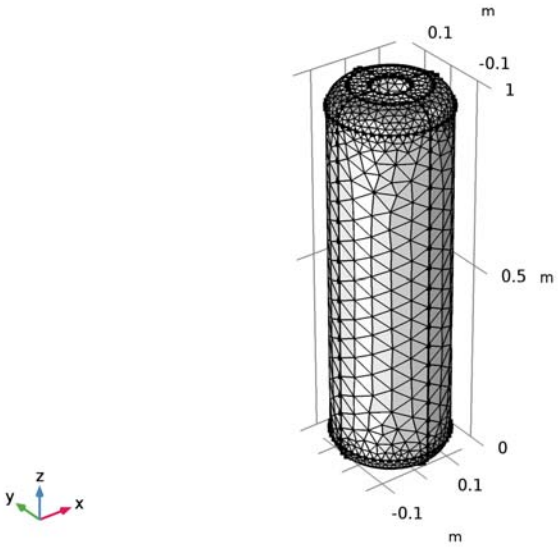
MESH 1

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

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

3 From the **Element size** list, choose **Finer**.

4 Click **Build All**.



STUDY 1

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

RESULTS

3D Plot Group 1

The default plot shows the potential distribution.

1 Click the **Zoom Extents** button on the **Graphics** toolbar.

Rotate the geometry so that you see both pipe connections. Compare the result with the plot in Figure 1.

Add an arrow surface plot of the current field as follows:

Arrow Surface 1

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

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

3 In the **X component** text field, type `-sigma*VTx`.

4 In the **Y component** text field, type `-sigma*VTy`.

5 In the **Z component** text field, type `-sigma*VTz`.

6 Select the **Description** check box.

7 In the associated text field, type `Current field (-sigma*VTx, -sigma*VTy, -sigma*VTz)`.

8 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Normalized**.

9 On the **3D Plot Group 1** toolbar, click **Plot**.

The plot in the **Graphics** window should now look like that in Figure 2.

To visualize the magnitude of the local current density, follow the steps given below.

3D Plot Group 2

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

2 In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Current density (A/m<sup>2</sup>)`.

Surface 1

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

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

3 In the **Expression** text field, type `sigma*sqrt(VTx^2+VTy^2+VTz^2)`.

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

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

Tapered Cantilever with Two Load Cases

Introduction

This example, taken from a NAFEMS benchmark collection (Ref. 1), demonstrates how to apply and evaluate different boundary conditions acting on a cantilever beam.

Model Definition

The cantilever beam has a thickness of 0.1 m, width of 4 m and is 4 m on its long edge and 2 m on its short edge. Two cases are considered. In the first one a gravity load, mg, acts in the negative y direction with an acceleration of 9.81 m/s^2. The left end boundary is fully fixed (no displacements). In the second case gravity load is not present and instead a uniformly distributed horizontal load, F, of 10 MN/m acts along the right end. At the left end there is no displacement in the x direction. Also at a midpoint location the left end is fixed in the y direction; see Figure 1.

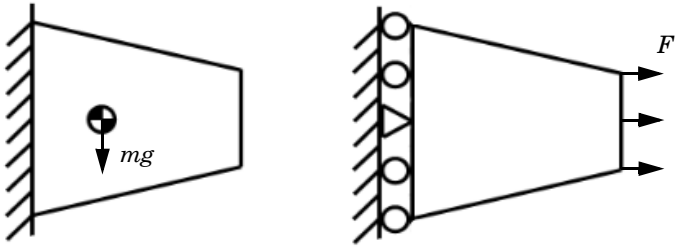


Figure 1: Schematic description of two load cases.

MATERIAL MODEL

The model uses the following material properties:

- The material is isotropic.
- The Young's modulus (elasticity modulus) is 210 GPa.
- The Poisson ratio is 0.3.
- The density is 7000 kg/m^3.

Results and Discussion

For the gravity case shear stress is evaluated. The benchmark target value of –0.200 MPa at the point (0, 2) is in good agreement with the model results. Using the default Normal

mesh size, the COMSOL Multiphysics solution gives a value of -0.199 MPa; see Figure 2.

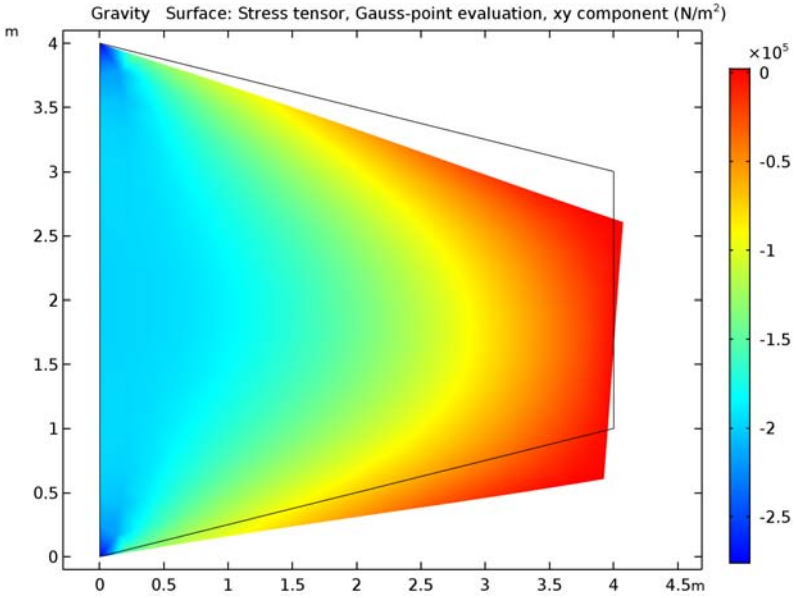


Figure 2: In plane shear stress due to gravity load.

For the load case, the horizontal (x direction) normal stress is evaluated. The benchmark target value of 61.3 MPa at the point $(0, 2)$ is in good agreement with the results. Using the default mesh size, the COMSOL Multiphysics solution gives a value of 61.4 MPa; see Figure 3.

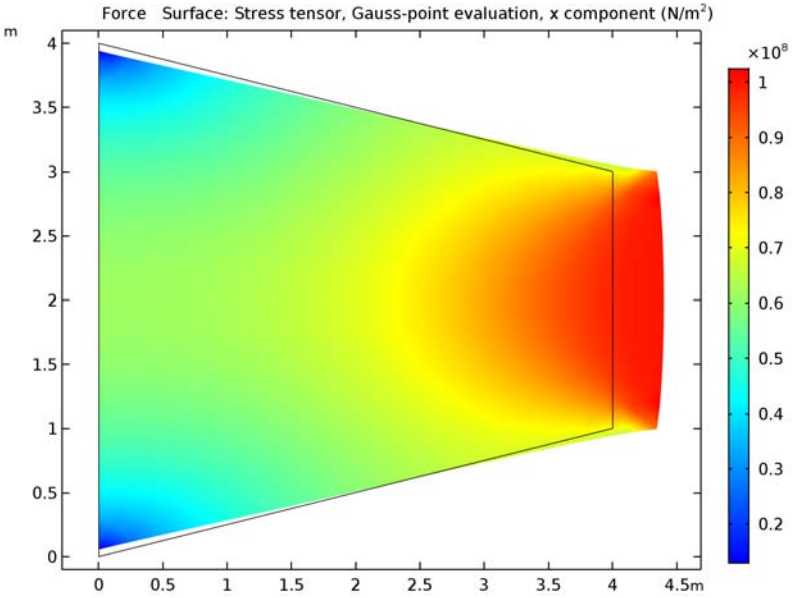


Figure 3: Counteracting stress due to applied edge load.

Notes About the COMSOL Implementation

Use predefined variables for the acceleration of gravity and density to enter the gravity load as a force/volume. COMSOL then computes the load using the thickness of the geometry.

Use the Solid Mechanics interface to perform a stress analysis. The finite element model uses the default second-order triangular Lagrange elements. To show convergence toward the benchmark value, create a finer mesh.

Use the Load Group and Constraint Group features to collect conditions that are enabled in different studies. Define a group in Global Definitions and assign its content directly from the load or constraint itself. You define load cases in the Study Extensions of a Study Type and enable active load and constraint groups; see Figure 4, where lg denotes a load

group and cg denotes a constraint group. Weight is used as a multiplication factor for the corresponding load group.

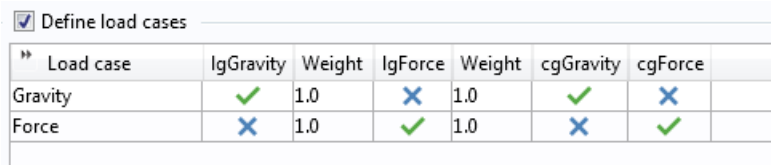


Figure 4: Definition of load cases.

Reference

1. D. Hitchings, A. Kamoulakos, and G.A.O. Davies, *Linear Statics Benchmarks Vol. 1*, NAFEMS, Glasgow, 1987.

Application Library path: `COMSOL_Multiphysics/Structural_Mechanics/tapered_cantilever`

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 **Structural Mechanics>Solid Mechanics (solid)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Stationary**.
6. Click **Done**.

GEOMETRY 1

Bézier Polygon 1 (b1)

1. On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

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

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **2**, set **x** to `4` and **y** to `1`.

5 Find the **Added segments** subsection. Click **Add Linear**.

6 Find the **Control points** subsection. In row **2**, set **y** to `3`.

7 Find the **Added segments** subsection. Click **Add Linear**.

8 Find the **Control points** subsection. In row **2**, set **x** to `0` and **y** to `4`.

9 Find the **Added segments** subsection. Click **Add Linear**.

10 Find the **Control points** subsection. Click **Close Curve**.

Point 1 (pt1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Point**.

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

3 In the **y** text field, type `2`.

Form Union (fin)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

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
Young's modulus	E	`210[GPa]`	Pa	Basic
Poisson's ratio	nu	`0.3`	1	Basic
Density	rho	`7000[kg/m^3]`	kg/m³	Basic

GLOBAL DEFINITIONS

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Load Group**.

2 In the **Settings** window for **Load Group**, type `Load Group Gravity` in the **Label** text field.

3 In the **Parameter name** text field, type `lgGravity`.

4 Right-click **Global Definitions** and choose **Load Group**.

5 In the **Settings** window for **Load Group**, type `Load Group Force` in the **Label** text field.

6 In the **Parameter name** text field, type `lgForce`.

7 Right-click **Global Definitions** and choose **Constraint Group**.

8 In the **Settings** window for **Constraint Group**, type `Constraint Group Gravity` in the **Label** text field.

9 In the **Parameter name** text field, type `cgGravity`.

10 Right-click **Global Definitions** and choose **Constraint Group**.

11 In the **Settings** window for **Constraint Group**, type `Constraint Group Force` in the **Label** text field.

12 In the **Parameter name** text field, type `cgForce`.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **2D Approximation** section.

3 From the list, choose **Plane stress**.

4 Locate the **Thickness** section. In the d text field, type `0.1`.

First, define all constraints for both load cases and then assign them to different load and constraint groups.

Fixed Constraint 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Select Boundaries 1 and 3 only.

3 On the **Physics** toolbar, click **Constraint Group** and choose **Constraint Group 1**.

Body Load 1

1 On the **Physics** toolbar, click **Domains** and choose **Body Load**.

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

3 In the **Settings** window for **Body Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_V$ vector as

`0`	`x`
`-g_const*solid.rho`	`y`

5 On the **Physics** toolbar, click **Load Group** and choose **Load Group Gravity**.

Roller 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Roller**.

2 Select Boundaries 1 and 3 only.

3 On the **Physics** toolbar, click **Constraint Group** and choose **Constraint Group 2**.

Boundary Load 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundary 5 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 From the **Load type** list, choose **Force per unit length**.

5 Specify the $\mathbf{F}_L$ vector as

10[MN/m]	x
0	y

6 On the **Physics** toolbar, click **Load Group** and choose **Load Group Force**.

Fixed Constraint 2

1 On the **Physics** toolbar, click **Points** and choose **Fixed Constraint**.

2 Select Point 2 only.

MESH 1

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

2 In the **Settings** window for **Mesh**, click **Build All**.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.

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

3 Locate the **Study Extensions** section. Select the **Define load cases** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Load case	lgGravity	Weight	lgForce	Weight	cgGravity	cgForce
Gravity	√	1.0		1.0	√	

6 Click **Add**.

7 In the table, enter the following settings:

Load case	lgGravity	Weight	lgForce	Weight	cgGravity	cgForce
Force		1.0	√	1.0		√

The study extension for the load cases should look like Figure 4.

8 On the **Home** toolbar, click **Compute**.

RESULTS

Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 In the **Settings** window for **2D Plot Group**, type `Normal stress` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Results>Normal stress** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics> Stress (Gauss points)>Stress tensor, Gauss-point evaluation (spatial frame)>solid.sGpx - Stress tensor, Gauss-point evaluation, x component**.

3 On the **Normal stress** toolbar, click **Plot**.

Normal stress 1

1 In the **Model Builder** window, under **Results** right-click **Normal stress** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Shear stress` in the **Label** text field.

3 Locate the **Data** section. From the **Load case** list, choose **Gravity**.

Surface 1

1 In the **Model Builder** window, expand the **Results>Shear stress** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `solid.sGpxy`.

4 On the **Shear stress** toolbar, click **Plot**.

Point Evaluation 1

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

2 In the **Settings** window for **Point Evaluation**, type `Point Evaluation - normal stress` in the **Label** text field.

3 Select Point 2 only.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Solid Mechanics>Stress (Gauss points)>Stress tensor, Gauss-point evaluation (spatial frame)>solid.sGpx - Stress tensor, Gauss-point evaluation, x component**.

5 Click **Evaluate**.

Point Evaluation 2

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

2 In the **Settings** window for **Point Evaluation**, type `Point Evaluation - shear stress` in the **Label** text field.

3 Select Point 2 only.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Solid Mechanics>Stress (Gauss points)>Stress tensor, Gauss-point evaluation (spatial frame)>solid.sGpxy - Stress tensor, Gauss-point evaluation, xy component**.

5 Click **Evaluate**.

Joule Heating of a Microactuator

Introduction

This simple tutorial model simulates the resistive heating — also known as Joule heating — of a two-hot-arm thermal actuator. The model couples the physics phenomena involved in one way only. However, as explained below, you can easily modify it to simulate a two-way coupling between the electric current and the heating of the actuator.

Model Definition

Figure 1 shows the actuator's parts and dimensions as well as its position on top of a substrate surface.

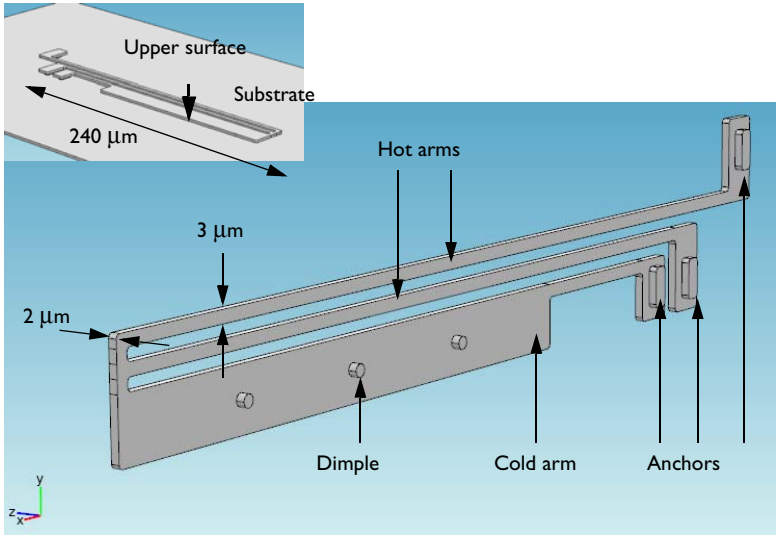


Figure 1: The thermal microactuator.

MATERIAL DATA

This model uses the material properties listed in Table 1 for the Joule Heating Model equations. The assumption of constant material properties means that the coupling between physics phenomena is one way only: the electric current through the actuator heats up the material, but the current itself is not affected by the temperature rise. By choosing the linearized resistivity option for the electrical conductivity (available in the

Joule Heating Model node's settings window) and providing the associated material properties you can turn this into a two-way coupling.

TABLE 1: MATERIAL DATA

PROPERTY	NAME	VALUE
Electrical conductivity	σ	$5 \cdot 10^4$ S/m
Relative permeability	ε_r	4.5
Thermal conductivity	k	40 W/(m·K)
Density	ρ	2300 kg/m^3
Heat capacity at constant pressure	C_p	600 J/(kg·K)

BOUNDARY CONDITIONS

An electric potential is applied between the bases of the hot arms' anchors. The cold arm anchor and all other surfaces are electrically insulated.

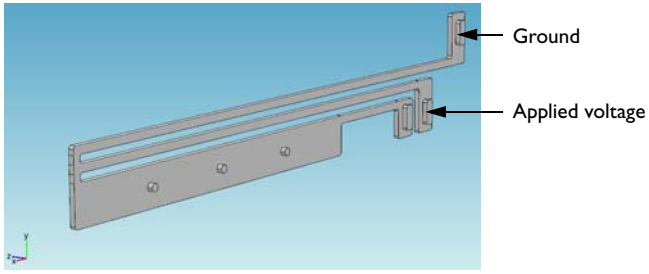


Figure 2: Electrical boundary conditions.

The temperature of the base of the three anchors and the three dimples is fixed to that of the substrate's constant temperature. Because the structure is sandwiched, all other boundaries interact thermally with the surroundings by conduction through thin layers of air.

The heat transfer coefficient is given by the thermal conductivity of air divided by the distance to the surrounding surfaces for the system. This exercise uses different heat transfer coefficients for the actuator's upper and other surfaces.

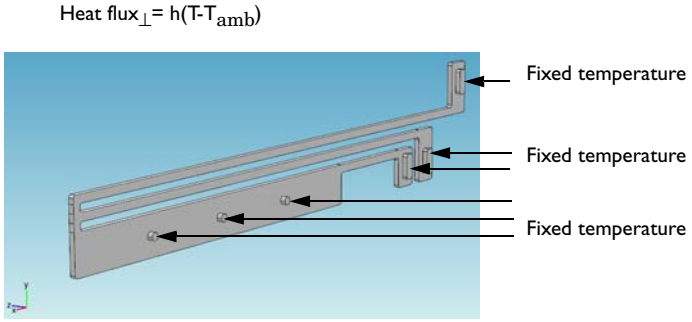


Figure 3: Heat-transfer boundary conditions.

Results

Figure 4 shows the temperature distribution on the actuator's surface. The line graph in Figure 5 provides more detailed information about the temperature along a single edge facing the substrate plane.

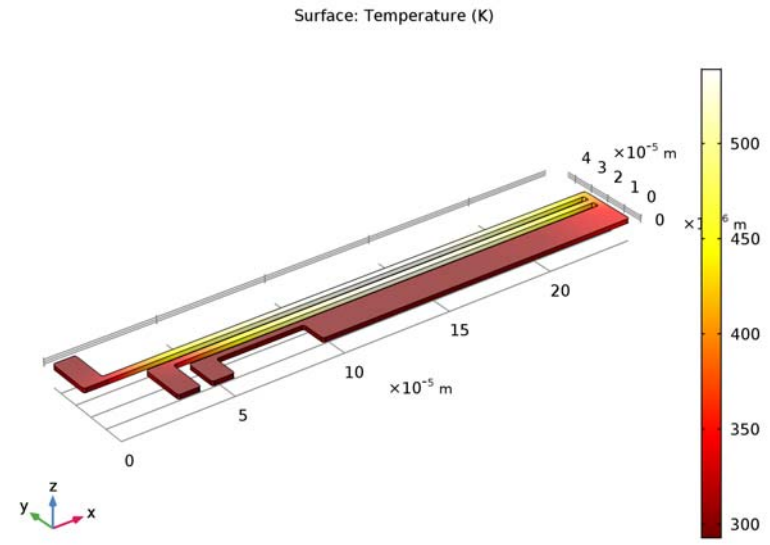


Figure 4: The temperature distribution on the actuator surface.

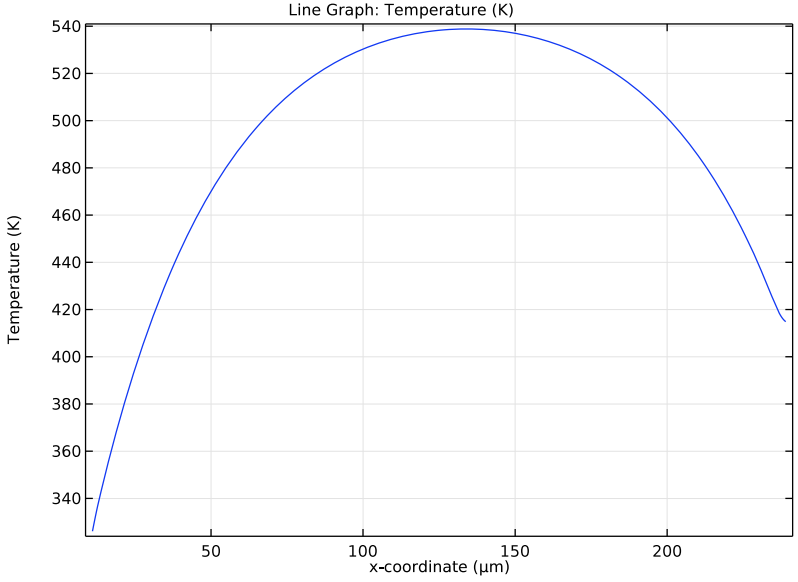


Figure 5: Temperature along the actuators longest edge facing the substrate.

Application Library path: `COMSOL_Multiphysics/Multiphysics/thermal_actuator_jh`

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 **Heat Transfer>Electromagnetic Heating>Joule Heating**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.
6. Click **Done**.

COMPONENT 1 (COMP1)

1. In the **Model Builder** window, right-click **Component 1 (comp1)** and choose **Rename**.
2. In the **Rename Component** dialog box, type `Thermal Actuator` in the **New label** text field.
3. Click **OK**.

GLOBAL DEFINITIONS

Parameters

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
htc_s	0.04[W/(m*K)]/2[um]	20000 W/(m²·K)	Heat transfer coefficient
htc_us	0.04[W/(m*K)]/100[um]	400 W/(m²·K)	Heat transfer coefficient, upper surface
DV	5[V]	5 V	Applied voltage

GEOMETRY 1

Import 1 (imp1)

1 On the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

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

5 Click **Build All Objects**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

DEFINITIONS

Explicit 1

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

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

3 In the **Rename Explicit** dialog box, type `Substrate Contact` in the **New label** text field.

4 Click **OK**.

5 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

7 Select Boundaries 10, 30, 50, 70, 76, and 82 only.

MATERIALS

Material 1 (mat1)

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

By default, the first material you define applies to all domains.

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
Electrical conductivity	sigma	5e4	S/m	Basic
Relative permittivity	epsilonr	4.5	1	Basic
Thermal conductivity	k	40	W/(m·K)	Basic
Density	rho	2.3e3	kg/m³	Basic
Heat capacity at constant pressure	Cp	600	J/(kg·K)	Basic

ELECTRIC CURRENTS (EC)

Ground 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.

2 Select Boundary 10 only.

Electric Potential 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

2 Select Boundary 30 only.

3 In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.

4 In the V_0 text field, type `DV`.

HEAT TRANSFER IN SOLIDS (HT)

In the **Model Builder** window, under **Thermal Actuator (comp1)** click **Heat Transfer in Solids (ht)**.

Heat Flux 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

This boundary condition applies to all boundaries except the top-surface boundary and those in contact with the substrate. A **Temperature** condition on the `substrate_contact` boundaries will override this **Heat Flux** condition so you do not explicitly need to exclude those boundaries. In contrast, because the **Heat Flux** boundary condition is additive, you must explicitly exclude the top-surface boundary from the selection. Implement this selection as follows:

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

3 From the **Selection** list, choose **All boundaries**.

4 In the **Graphics** window, click on the top surface and then right-click to remove it from the selection.

A convective heat flux is used to model the heat flux through a thin air layer. The heat transfer coefficient, `htc_s` is defined as the ratio of the air thermal conductivity to the gap thickness.

5 Locate the **Heat Flux** section. Click the **Convective heat flux** button.

6 In the h text field, type `htc_s`.

Heat Flux 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 Select Boundary 4 only.

A convective heat flux is used to model the heat flux through a thin air layer. The heat transfer coefficient, `htc_us` is defined as the ratio of the air thermal conductivity to the gap thickness.

3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

4 Click the **Convective heat flux** button.

5 In the h text field, type `htc_us`.

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

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

3 From the **Selection** list, choose **Substrate Contact**.

MESH 1

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

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

3 From the **Element size** list, choose **Fine**.

Free Triangular 1

1 Right-click **Thermal Actuator (comp1)>Mesh 1** and choose **More Operations> Free Triangular**.

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

3 From the **Selection** list, choose **Substrate Contact**.

4 Click **Paste Selection**.

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

6 Click **OK**.

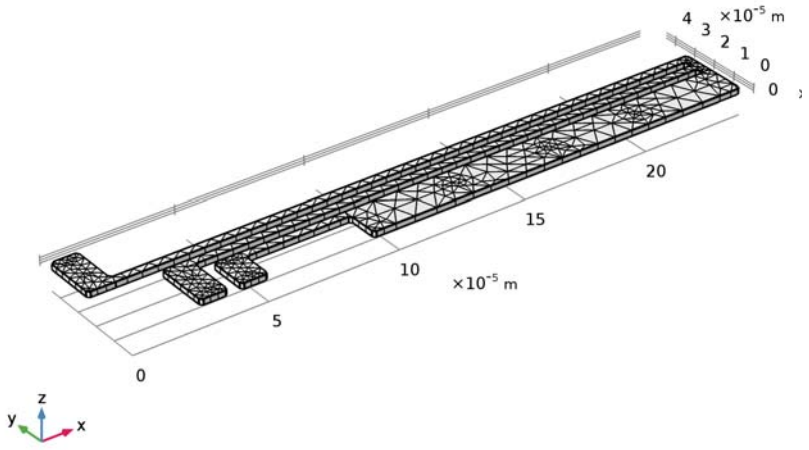
7 In the **Settings** window for **Free Triangular**, click **Build Selected**.

Swept 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

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

3 In the **Settings** window for **Mesh**, click **Build All**.



STUDY 1

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

RESULTS

Electric Potential (ec)

The first default plot group shows the electric potential distribution.

Temperature (ht)

The second default plot group shows the temperature distribution on the surface (see Figure 4).

1 Click the **Zoom Extents** button on the **Graphics** toolbar.

Reproduce the plot in Figure 5 by following these steps:

Line Graph 1

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

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

3 Select Edge 52 only.

4 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Thermal Actuator> Heat Transfer in Solids>Temperature>T - Temperature**.

5 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Thermal Actuator>Geometry>Coordinate>x - x-coordinate**.

6 Locate the **x-Axis Data** section. From the **Unit** list, choose **µm**.

7 On the **1D Plot Group 4** toolbar, click **Plot**.

Joule Heating of a Microactuator — Distributed Parameter Version

Introduction

The purpose of this example is to demonstrate how to access the cluster computing functionality in COMSOL from the COMSOL Desktop and use it to submit a batch job to a cluster through a job scheduler. The model takes advantage of the distributed parameter functionality in COMSOL. The model also demonstrates how you can measure the speedup of COMSOL on your cluster. The speedup is defined as the quotient between the total runtime using only one physical node and one core of the cluster and the runtime using several physical nodes and all cores of each physical node of the cluster. For detailed information about the model, see Joule Heating of a Microactuator.

Note: This application requires a Floating Network License.

Application Library path: `COMSOL_Multiphysics/Tutorials/thermal_actuator_jh_distributed`

Modeling Instructions

APPLICATION LIBRARIES

1 On the **Home** toolbar, click **Application Libraries**.

2 In the **Application Library** tree, select **COMSOL Multiphysics>Multiphysics> thermal_actuator_jh**.

3 Click **Open Application**.

STUDY 1

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Study Options** in the menu.

With this setting active, Cluster Computing is available from the Study node's context menu.

Parametric Sweep

1 On 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
DV	`range(5,0.5,15)`	

5 Click to expand the **Study Extensions** section. Select the **Distribute parametric sweep** check box.

Cluster Computing

1 In the **Model Builder** window, right-click **Study 1** and choose **Cluster Computing**.

2 In the **Settings** window for Cluster Computing, locate the **Batch Settings** section.

Set the number of nodes to the number of nodes you want to use. If you are testing Cluster Computing, set the number of nodes to 2 or less to make sure everything works.

Choose from one of the following settings the type of job scheduler to use:

General: Use the General scheduler if you intend to submit a job to a job scheduler that you have configured COMSOL to run on.

HPCS 2008: Use the HPCS 2008 scheduler if you intend to submit a job to a Windows HPC Server 2008 using the Windows HPC Cluster Manager.

WCCS 2003: Use the WCCS 2003 scheduler if you intend to submit a job to a Windows Compute Cluster Server 2003.

3 Locate the **Batch Settings** section. From the **Scheduler type** list, choose **Not distributed**.

Not Distributed: Use this setting when you have configured COMSOL to run on job scheduler but only intend to run on a single node of the cluster.

Make sure you configure the batch directories and COMSOL installation directories correctly. It is good practice to save these values as default once you have good settings.

4 Click the **Save as Default** button.

Batch 1

1 On the **Study** toolbar, click **Compute**.

In the General section, click Open to open the file containing the model generated by the batch job associated with this external process in a new COMSOL session.

In the Process Status section, the log shows the total solution time. COMSOL automatically takes advantage of all cores; to measure the speedup, set the number of cores to 1 and run a new job.

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

3 In the **Settings** window for Batch, locate the **General** section.

4 From the **Defined by study step** list, choose **User defined**.

5 Select the **Number of cores** check box.

6 Locate the **Files** section. In the **Filename** text field, enter a name of your choice.

Next, change the filename to create a new External Process node for the run:

7 In the **Filename** text field, enter a name of your choice.

Cluster Computing 1

If you are not using the cluster type Not distributed, make sure to set the Number of nodes to 1.

Batch 1

1 In the **Model Builder** window, under **Study 1>Job Configurations** right-click **Cluster Computing 1** and choose **Run**.

When the process has finished, compare the total time in the log for the new external process with the previous value. The speedup is equal to the previous value divided by the new value. The speedup depends on the mesh size. To improve the numbers, try refining the mesh.

Thermal Microactuator Simplified

Introduction

This example model consists of a two-hot-arm thermal actuator made of polysilicon. The actuator is activated through thermal expansion. The temperature increase required to deform the two hot arms, and thus displace the actuator, is obtained through Joule heating (resistive heating). The greater expansion of the hot arms, compared to the cold arm, causes a bending of the actuator.

The material properties of polysilicon are temperature dependent, which means that the involved physics phenomena are fully coupled. The electric current through the hot arms increases the temperature in the actuator, which in turn causes thermal expansion and changes the electrical conductivity of the material.

The actuator's operation thus involves three coupled physics phenomena: electric current conduction, heat conduction with heat generation, and structural stresses and strains due to thermal expansion.

In this example the thermal expansion is included manually using the Equation View. Furthermore, only linear strains are considered, which is a valid approximation provided deformations are small. Using the Structural Mechanics Module or the MEMS Module, you can directly include the thermal equation in the physics and also take into account possible large deformations; see the model Thermal Microactuator in the Structural Mechanics Module or MEMS Module Application Libraries.

Figure 1 shows the actuator's parts and dimensions as well as its position on top of a substrate surface.

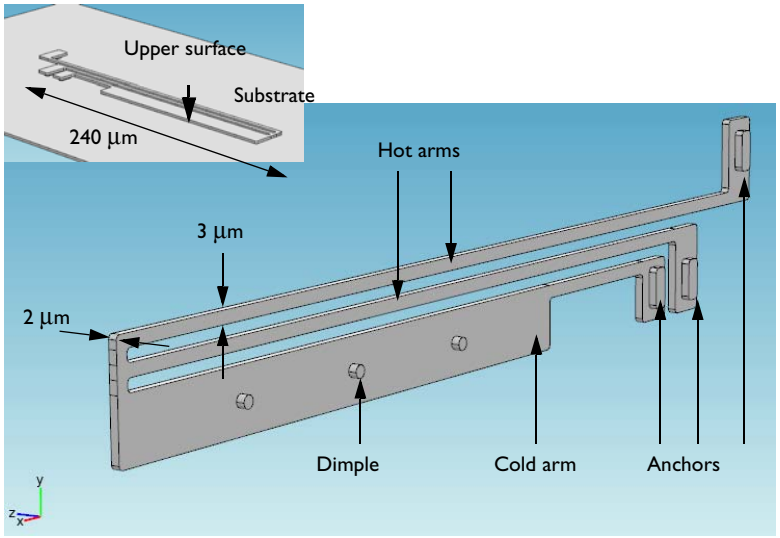


Figure 1: The thermal microactuator.

BOUNDARY CONDITIONS AND CONSTRAINTS

An electric potential is applied between the bases of the hot arms' anchors. The cold arm anchor and all other surfaces are electrically insulated.

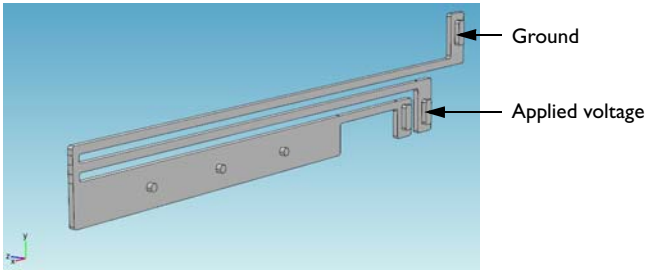


Figure 2: Electrical boundary conditions.

The temperature of the base of the three anchors and the three dimples is fixed to that of the substrate's constant temperature. Because the structure is sandwiched, all other

boundaries interact thermally with the surroundings by conduction through thin layers of air.

The heat transfer coefficient is given by the thermal conductivity of air divided by the distance to the surrounding surfaces for the system. This exercise uses different heat transfer coefficients for the actuator's upper and other surfaces.

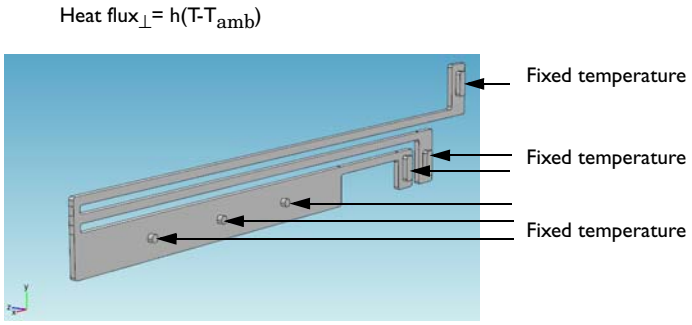


Figure 3: Heat-transfer boundary conditions.

All three arms are mechanically fixed at the base of the three anchors. The dimples can move freely in the plane of the substrate (the *xy*-plane in the figure) but do not move in the direction perpendicular to the substrate (the *z* direction).

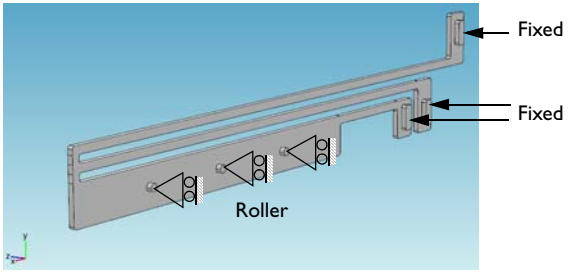


Figure 4: Structural boundary conditions and constraints.

GEOMETRY PROGRAMING OPERATION

The model geometry is implemented using the geometry programing feature operation such as the conditional if statement. The model geometry is implemented using two distinct design: a two-arms or a three-arms actuator design. It is possible to switch from one design to the other by means of a parameter.

Results

Figure 5 shows the surface temperature distribution for the actuator design with two arms. It also illustrates the displacement field through a deformation plot.

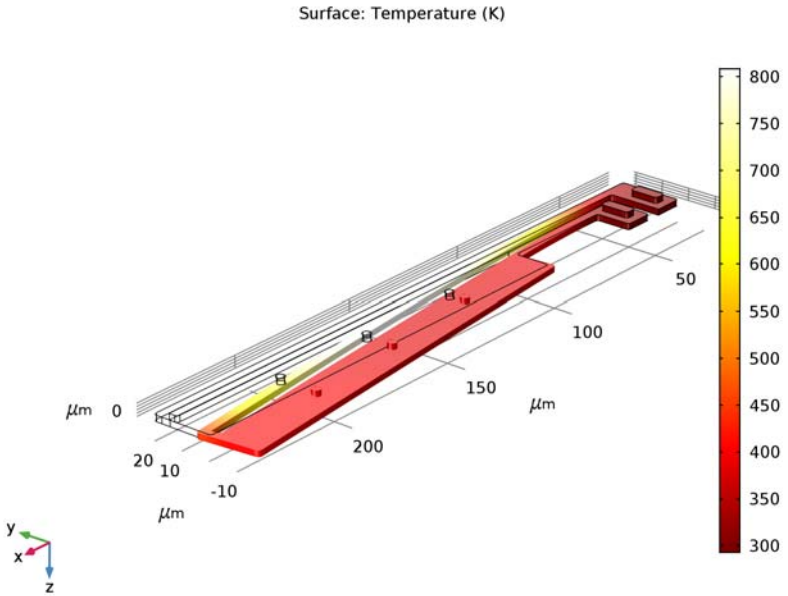


Figure 5: Temperature (surface) and displacement (deformation) in the two-arms actuator design.

For the two-arms design, the computed displacement at the tip of the actuator is about 3.3 μm.

Figure 6 shows the surface temperature distribution for the actuator design with 3 arms only.

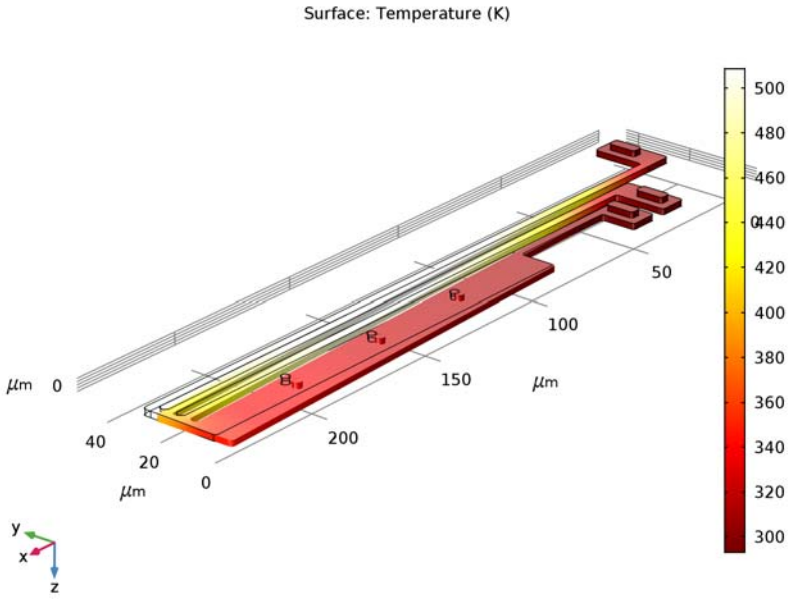


Figure 6: Temperature (surface) and displacement (deformation) in the three arms actuator design.

For the two-arms design, the computed displacement at the tip of the actuator is about 1.1 μm.

Notes About the COMSOL Implementation

Enable the Equation View to access the equation level. There one can modify the strain definition used by the linear elastic material model.

The thermal strain to be added is represented below:

$$\varepsilon_{th} = \alpha \cdot (T - T_0)$$

Where α is the material thermal expansion coefficient, T the strain temperature and T_0 the strain reference temperature.

Enable the equation view by clicking the **Show** button ( ) and select **Equation View**. You can then access the equation view by expanding each node of the model tree.

Application Library path: `COMSOL_Multiphysics/Multiphysics/thermal_actuator_simplified`

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 **Heat Transfer>Electromagnetic Heating>Joule Heating**.
3. Click **Add**.
4. In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
5. Click **Add**.
6. Click **Study**.
7. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.
8. Click **Done**.

COMPONENT 1 (COMP1)

1. In the **Model Builder** window, right-click **Component 1 (comp1)** and choose **Rename**.
2. In the **Rename Component** dialog box, type `Thermal Actuator` in the **New label** text field.
3. Click **OK**.

GLOBAL DEFINITIONS

Parameters

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
d	3[um]	3E-6 m	Height of the hot arm
dw	15[um]	1.5E-5 m	Height of the cold arm
gap	3[um]	3E-6 m	Gap between arms
wb	10[um]	1E-5 m	Width of the base
wv	25[um]	2.5E-5 m	Difference in length between hot arms
L	240[um]	2.4E-4 m	Actuator length
L1	L-wb	2.3E-4 m	Length of the longest hot arm
L2	L-wb-wv	2.05E-4 m	Length of the shortest hot arm
L3	L-2*wb-wv-L/48-L/6	1.5E-4 m	Length of the cold arm, thick part
L4	L/6	4E-5 m	Length of the cold arm, thin part
htc_s	0.04[W/(m*K)]/2[um]	20000 W/(m²·K)	Heat transfer coefficient
htc_us	0.04[W/(m*K)]/100[um]	400 W/(m²·K)	Heat transfer coefficient, upper surface
DV	5[V]	5 V	Applied voltage
alphaps	2.6e-6[1/K]	2.6E-6 1/K	Coefficient of thermal expansion
T0	293.15[K]	293.15 K	Strain reference temperature
noa	3	3	Number of arms

GEOMETRY 1

1 In the **Model Builder** window, under **Thermal Actuator (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **µm**.

If 1 (if1)

1 On the **Geometry** toolbar, click **Programming** and choose **If + End If**.

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

3 In the **Condition** text field, type `(noa==3)`.

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `L3`.

4 In the **Height** text field, type `dw`.

5 Locate the **Position** section. In the **xw** text field, type `L-L3`. Leave **yw** at the default value `0`.

Rectangles 2-9

Proceed to create eight additional rectangles with the following settings:

Name	Width	Height	xw	yw
Rectangle 2 (r2)	`L4`	`d`	`L-L3-L4`	`dw-d`
Rectangle 3 (r3)	`wb`	`dw`	`L-L3-L4-wb`	`0`
Rectangle 4 (r4)	`L2`	`d`	`L-L2`	`dw+gap`
Rectangle 5 (r5)	`wb`	`dw+gap+d`	`L-L2-wb`	`0`
Rectangle 6 (r6)	`L1`	`d`	`L-L1`	`dw+d+2*gap`
Rectangle 7 (r7)	`wb`	`dw+gap+d`	`0`	`dw+d+2*gap`
Rectangle 8 (r8)	`d`	`gap`	`L-d`	`dw+gap+d`
Rectangle 9 (r9)	`d`	`gap`	`L-d`	`dw`

Union 1 (uni1)

1 On the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Union**.

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

3 In the **Settings** window for **Union**, locate the **Union** section.

4 Clear the **Keep interior boundaries** check box.

Fillet 1 (fil1)

1 On the **Work Plane** toolbar, click **Fillet**.

2 On the object **uni1**, select Points 1, 2, 4–9, 11–14, 16, 17, 19–23, and 28 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `d/3`.

5 In the **Model Builder** window, click **Geometry 1**.

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (μm)
2

Work Plane 2 (wp2)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `wb-2*d`.

4 In the **Height** text field, type `2.5*(wb-2*d)`.

5 Locate the **Position** section. In the **xw** text field, type `d`.

6 In the **yw** text field, type `(dw+d+2*gap)+(dw+gap+d)-2.5*(wb-2*d)-d`.

Rectangle 2 (r2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `wb-2*d`.

4 In the **Height** text field, type `2.5*(wb-2*d)`.

5 Locate the **Position** section. In the **xw** text field, type `L-L2-wb+d`.

6 In the **yw** text field, type `d`.

Rectangle 3 (r3)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `wb-2*d`.

4 In the **Height** text field, type `2.5*(wb-2*d)`.

5 Locate the **Position** section. In the **xw** text field, type `L-L3-L4-wb+d`.

6 In the **yw** text field, type `d`.

Fillet 1 (fil1)

1 On the **Work Plane** toolbar, click **Fillet**.

2 Click the **Zoom Extents** button on the **Graphics** toolbar.

3 On each of the rectangles **r1**, **r2**, and **r3**, in turn, select all four corner points.

4 In the **Settings** window for **Fillet**, locate the **Radius** section.

5 In the **Radius** text field, type `d/3`.

Circle 1 (c1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `d/2`.

4 Locate the **Position** section. In the **xw** text field, type `L-L3/4`.

5 In the **yw** text field, type `dw/2`.

Circle 2 (c2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `d/2`.

4 Locate the **Position** section. In the **xw** text field, type `L-L3/2`.

5 In the **yw** text field, type `dw/2`.

Circle 3 (c3)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `d/2`.

4 Locate the **Position** section. In the **xw** text field, type `L-3*L3/4`.

5 In the **yw** text field, type `dw/2`.

6 In the **Model Builder** window, click **Geometry 1**.

Extrude 2 (ext2)

1 On the **Geometry** toolbar, click **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (μm)
2

4 Select the **Reverse direction** check box.

Union 1 (uni1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Click the **Select Box** button on the **Graphics** toolbar.

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

4 In the **Settings** window for **Union**, locate the **Union** section.

5 Clear the **Keep interior boundaries** check box.

6 Right-click **Union 1 (uni1)** and choose **Build Selected**.

Explicit Selection 1 (sel1)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

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

4 On the object **uni1**, select Boundary 10 only.

5 Locate the **Resulting Selection** section. Click **New**.

6 Clear the **Keep selection** check box.

7 In the **New Cumulative Selection** dialog box, type `Ground` in the **Name** text field.

8 Click **OK**.

Explicit Selection 2 (sel2)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

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

4 On the object **uni1**, select Boundary 29 only.

5 Locate the **Resulting Selection** section. Click **New**.

6 Clear the **Keep selection** check box.

7 In the **New Cumulative Selection** dialog box, type `Applied Voltage` in the **Name** text field.

8 Click **OK**.

Explicit Selection 3 (sel3)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

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

4 On the object **uni1**, select Boundary 48 only.

5 Locate the **Resulting Selection** section. Click **New**.

6 Clear the **Keep selection** check box.

7 In the **New Cumulative Selection** dialog box, type `Third` in the **Name** text field.

8 Click **OK**.

Explicit Selection 4 (sel4)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

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

4 On the object **uni1**, select Boundary 4 only.

5 Locate the **Resulting Selection** section. Click **New**.

6 Clear the **Keep selection** check box.

7 In the **New Cumulative Selection** dialog box, type `Upper Surface` in the **Name** text field.

8 Click **OK**.

Explicit Selection 5 (sel5)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

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

4 Click the **Select Box** button on the **Graphics** toolbar.

5 On the object **uni1**, select Boundaries 1–3, 5–9, 11–28, 30–47, and 49–92 only.

6 Locate the **Resulting Selection** section. Click **New**.

7 Clear the **Keep selection** check box.

8 In the **New Cumulative Selection** dialog box, type `Other Surface` in the **Name** text field.

9 Click **OK**.

Explicit Selection 6 (sel6)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

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

4 Click the **Zoom Box** button on the **Graphics** toolbar.

5 On the object **uni1**, select Point 154 only.

6 Locate the **Resulting Selection** section. Click **New**.

7 Clear the **Keep selection** check box.

8 In the **New Cumulative Selection** dialog box, type `Tip` in the **Name** text field.

9 Click **OK**.

Explicit Selection 7 (sel7)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

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

4 On the object **uni1**, select Boundaries 67, 72, and 77 only.

5 Locate the **Resulting Selection** section. Click **New**.

6 Clear the **Keep selection** check box.

7 In the **New Cumulative Selection** dialog box, type `Dimples` in the **Name** text field.

8 Click **OK**.

End If 1 (endif1)

In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **End If 1 (endif1)** and choose **Build Selected**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
noa	2	2	Number of arms

GEOMETRY 1

If 2 (if2)

1 On the **Geometry** toolbar, click **Programming** and choose **If + End If**.

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

3 In the **Condition** text field, type `(noa==2)`.

Work Plane 3 (wp3)

1 In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **Work Plane 1 (wp1)** and choose **Duplicate**.

2 Expand the **Work Plane 3 (wp3)** node.

Plane Geometry

Ctrl-click to select **Rectangle 6 (r6)**, **Rectangle 7(r7)**, and **Rectangle 8 (r8)** simultaneously, then right-click and choose **Disable**.

Extrude 1 (ext1)

In the **Model Builder** window, expand the **Thermal Actuator (comp1)>Geometry 1> Work Plane 3 (wp3)>Plane Geometry** node.

Extrude 3 (ext3)

1 Right-click **Thermal Actuator (comp1)>Geometry 1>Extrude 1 (ext1)** and choose **Duplicate**.

2 In the **Settings** window for **Extrude**, locate the **General** section.

3 From the **Work plane** list, choose **Work Plane 3 (wp3)**.

4 Find the **Input objects** subsection. Select the **Active** toggle button.

5 Select the object **wp3** only.

6 Right-click **Thermal Actuator (comp1)>Geometry 1>Extrude 3 (ext3)** and choose **Build Selected**.

Work Plane 4 (wp4)

1 In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **Work Plane 2 (wp2)** and choose **Duplicate**.

2 Expand the **Work Plane 4 (wp4)** node.

Rectangle 1 (r1)

1 In the **Model Builder** window, expand the **Thermal Actuator (comp1)>Geometry 1> Work Plane 4 (wp4)>Plane Geometry** node, then click **Rectangle 1 (r1)**.

2 Right-click **Thermal Actuator (comp1)>Geometry 1>Work Plane 4 (wp4)> Plane Geometry>Rectangle 1 (r1)** and choose **Disable**.

Extrude 4 (ext4)

1 In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **Extrude 2 (ext2)** and choose **Duplicate**.

2 In the **Settings** window for **Extrude**, locate the **General** section.

3 From the **Work plane** list, choose **Work Plane 4 (wp4)**.

4 Find the **Input objects** subsection. Select the **Active** toggle button.

5 Select the object **wp4** only.

6 Right-click **Thermal Actuator (comp1)>Geometry 1>Extrude 4 (ext4)** and choose **Build Selected**.

Union 2 (uni2)

1 In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **Union 1 (uni1)** and choose **Duplicate**.

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

3 Find the **Input objects** subsection. Select the **Active** toggle button.

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

Explicit Selection 8 (sel8)

1 In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **Explicit Selection 1 (sel1)** and choose **Duplicate**.

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

3 Find the **Entities to select** subsection. Select the **Active** toggle button.

4 On the object **uni2**, select Boundary 28 only.

5 Right-click **Thermal Actuator (comp1)>Geometry 1>Explicit Selection 8 (sel8)** and choose **Build Selected**.

Explicit Selection 9 (sel9)

1 In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **Explicit Selection 2 (sel2)** and choose **Duplicate**.

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

3 Find the **Entities to select** subsection. Select the **Active** toggle button.

4 On the object **uni2**, select Boundary 10 only.

5 Right-click **Thermal Actuator (comp1)>Geometry 1>Explicit Selection 9 (sel9)** and choose **Build Selected**.

Explicit Selection 10 (sel10)

1 In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **Explicit Selection 4 (sel4)** and choose **Duplicate**.

2 On the object **uni2**, select Boundary 4 only.

3 Right-click **Thermal Actuator (comp1)>Geometry 1>Explicit Selection 10 (sel10)** and choose **Build Selected**.

Explicit Selection 11 (sel11)

1 In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **Explicit Selection 5 (sel5)** and choose **Duplicate**.

2 Click the **Select Box** button on the **Graphics** toolbar.

3 On the object **uni2**, select Boundaries 1–3, 5–9, 11–27, and 29–66 only.

4 Right-click **Thermal Actuator (comp1)>Geometry 1>Explicit Selection 11 (sel11)** and choose **Build Selected**.

Explicit Selection 12 (sel12)

1 In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **Explicit Selection 6 (sel6)** and choose **Duplicate**.

2 On the object **uni2**, select Point 108 only.

3 Right-click **Thermal Actuator (comp1)>Geometry 1>Explicit Selection 12 (sel12)** and choose **Build Selected**.

Explicit Selection 13 (sel13)

1 In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **Explicit Selection 7 (sel7)** and choose **Duplicate**.

2 On the object **uni2**, select Boundaries 47, 52, and 57 only.

3 Right-click **Thermal Actuator (comp1)>Geometry 1>Explicit Selection 13 (sel13)** and choose **Build Selected**.

Form Union (fin)

In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

DEFINITIONS

Union 1

1 On the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, locate the **Geometric Entity Level** section.

3 From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Ground**, **Applied Voltage**, and **Third**.

6 Click **OK**.

7 Right-click **Union 1** and choose **Rename**.

8 In the **Rename Union** dialog box, type `Surface Contact` in the **New label** text field.

9 Click **OK**.

ADD MATERIAL

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

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

3 In the tree, select **Built-In>Polysilicon**.

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

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

ELECTRIC CURRENTS (EC)

Electric Potential 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

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

3 From the **Selection** list, choose **Applied Voltage**.

4 Locate the **Electric Potential** section. In the V_0 text field, type `DV`.

Ground 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.

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

3 From the **Selection** list, choose **Ground**.

HEAT TRANSFER IN SOLIDS (HT)

In the **Model Builder** window, under **Thermal Actuator (comp1)** click **Heat Transfer in Solids (ht)**.

Heat Flux 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

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

3 From the **Selection** list, choose **Other Surface**.

A convective heat flux is used to model the heat flux through a thin air layer. The heat transfer coefficient, `htc_s` is defined as the ratio of the air thermal conductivity to the gap thickness.

4 Locate the **Heat Flux** section. Click the **Convective heat flux** button.

5 In the h text field, type `htc_s`.

Heat Flux 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

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

3 From the **Selection** list, choose **Upper Surface**.

A convective heat flux is used to model the heat flux through a thin air layer. The heat transfer coefficient, `htc_us` is defined as the ratio of the air thermal conductivity to the gap thickness.

4 Locate the **Heat Flux** section. Click the **Convective heat flux** button.

5 In the h text field, type `htc_us`.

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

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

3 From the **Selection** list, choose **Surface Contact**.

SOLID MECHANICS (SOLID)

In the **Model Builder** window, under **Thermal Actuator (comp1)** click **Solid Mechanics (solid)**.

Fixed Constraint 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

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

3 From the **Selection** list, choose **Surface Contact**.

Roller 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Roller**.

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

3 From the **Selection** list, choose **Dimples**.

4 In the **Model Builder** window's toolbar, click the **Show** button and select **Equation View** in the menu.

Linear Elastic Material 1

1 Click the **Refresh Equations** button.

2 In the **Model Builder** window, expand the **Thermal Actuator (comp1)>Solid Mechanics (solid)>Linear Elastic Material 1** node, then click **Equation View**.

3 In the **Settings** window for **Equation View**, locate the **Variables** section.

4 In the table, enter the following settings:

Name	Expression	Unit	Description	Selection	Details
solid.eXX	`uX-alphaps*(T-T0)`	1	Strain tensor, XX component	Domain 1	+†operation
solid.eYY	`vY-alphaps*(T-T0)`	1	Strain tensor, YY component	Domain 1	+†operation
solid.eZZ	`wZ-alphaps*(T-T0)`	1	Strain tensor, ZZ component	Domain 1	+†operation

STUDY 1

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

RESULTS

Point Evaluation 1

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

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

3 From the **Selection** list, choose **Tip**.

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

Expression	Unit	Description
`solid.disp`	`µm`	`Total displacement`

5 Click **Evaluate**.

Volume Maximum 1

1 On the **Results** toolbar, click **More Derived Values** and choose **Maximum> Volume Maximum**.

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

3 From the **Selection** list, choose **All domains**.

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

Expression	Unit	Description
`T`	K	`Temperature`

5 Click **Evaluate**.

Surface

In the **Model Builder** window, expand the **Temperature (ht)** node.

Deformation 1

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

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** check box.

4 In the associated text field, type `5`.

5 On the **Temperature (ht)** toolbar, click **Plot**.

GLOBAL DEFINITIONS

Parameters

Continue by computing the solution for the three arms actuator geometry case.

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
noa	3	3	Number of arms

STUDY 1

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

RESULTS

Point Evaluation 1

1 In the **Model Builder** window, under **Results>Derived Values** click **Point Evaluation 1**.

2 In the **Settings** window for **Point Evaluation**, click **Evaluate**.

Volume Maximum 1

1 In the **Model Builder** window, under **Results>Derived Values** click **Volume Maximum 1**.

2 In the **Settings** window for **Volume Maximum**, click **Evaluate**.

Thin-Film Resistance

Introduction

When modeling transport by diffusion or conduction in thin layers, large differences in dimensions of the different domains are common. If the model has a sandwich structure, you can replace the thinnest layers with a thin-layer approximation, provided that the difference in thickness is large.

Model Definition

This study explains the principle of the thin-layer approximation in direct current conduction problems. A comparison of a structure with three domains to a simplified model that replaces the domain in the middle with a thin-layer approximation shows the benefit of this approach (see Figure 1).

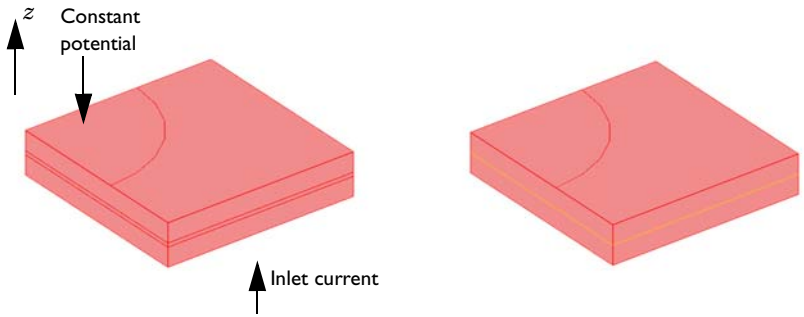


Figure 1: Exact domain description (left) and approximation (right). The current flows from the base plate to the circular plate on the upper surface of the device.

Equation 1 below describes the current balance in all three domains in the real sandwich structure:

$$\nabla \cdot (-\sigma \nabla V) = 0 \tag{1}$$

In this equation, σ represents the conductivity and V the electric potential. In this case, there is a substantial difference in conductivity between the thin and thicker layers of the structure. The boundary conditions include a current inlet in the base plate of the device and a constant potential at the upper circular boundary (see Figure 1). All other boundaries are insulated.

The simplified model is based on the assumption that the components of the current density vector in the x and y directions are small and that the dominating transport through the thin structure is obtained in the z direction. For the middle layer, this implies

that you can approximate Equation 1 by the one-dimensional equation

$$-\sigma\frac{d^2V}{dz^2} = 0 \tag{2}$$

It is possible to solve this equation analytically if the potential is given at the lower and upper surfaces of the middle layer:

$$V_{\delta = 0} = V_1 \tag{3}$$

$$V_{\delta = \delta_1} = V_2 \tag{4}$$

You can integrate Equation 2 analytically to give:

$$V = az + b$$

where a and b are integration constants. If you arbitrarily place $z = 0$ at the lower boundary of the middle layer, you get the constants a and b from the boundary conditions in Equation 3 and Equation 4:

$$V_1 = b$$

$$V_2 = a\delta + b$$

This gives:

$$b = V_1$$

$$a = \frac{V_2 - V_1}{\delta}$$

The resulting equation for the potential is thus

$$V = \left(\frac{V_2 - V_1}{\delta}\right)z + V_1 \tag{5}$$

The current density is defined as

$$J_z = -\sigma\frac{dV}{dz} \tag{6}$$

Combining Equation 5 and Equation 6 gives

$$J_z = -\sigma\left(\frac{V_2 - V_1}{\delta}\right) \tag{7}$$

In the thin-film approximation the potential is discontinuous at the film boundary. Use the Contact Impedance node on interior boundaries to model a thin layer of resistive material.

It is also possible to derive the expression for the current density in Equation 7 by approximating the gradient using the potential difference over the thin layer. This example includes the previous tedious derivation to show that this is exactly what you obtain from the solution of Equation 2.

The approximation presented in this example is not limited to direct current problems: You can also use it for modeling of diffusion, heat conduction, flow through porous media using Darcy's law, and other types of physics that the divergence of a gradient flux describes.

In general, the application of this simplification is appropriate in cases where the differences in thickness are so large that the mesh generator cannot even mesh the domain. In some cases, the mesh generator might be able to mesh the domain but then creates a very large number of elements.

Results and Discussion

Figure 2 shows a comparison between the exact solution of the problem using three conductive layers and the thin-film approximation. The comparison reveals an excellent agreement in the potential and current distribution despite that the middle film in this study is relatively thick. The approximation becomes even more accurate as the film thickness between the upper and lower domain decreases.

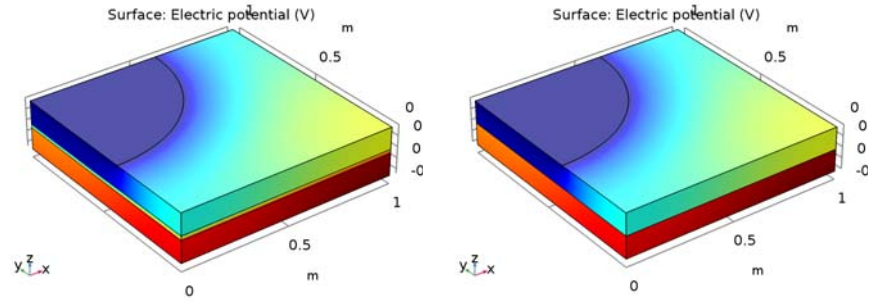


Figure 2: Potential distribution in the modeled device. The value of the potential loss over the device at a current of 0.3 A is almost identical in the two models: the full model (left) and thin-film approximation (right).

Figure 3 shows a cross-section plot of the potential through the structure's center for the full model and for the approximation. The plots show the excellent agreement obtained between the two models.

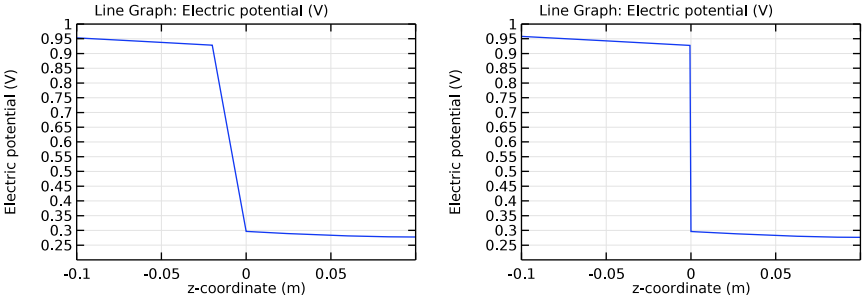


Figure 3: Potential distribution along the z direction in the middle of the device. Solution for the full model (left) and for the thin-film approximation (right).

Application Library path: `COMSOL_Multiphysics/Electromagnetics/thin_film_resistance`

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 **AC/DC>Electric Currents (ec)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Stationary**.
6. Click **Done**.

GEOMETRY 1

Work Plane 1 (wp1)

1. On the **Geometry** toolbar, click **Work Plane**.
2. In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.
3. In the **z-coordinate** text field, type `0.1`.
4. Locate the **Unite Objects** section. Clear the **Unite objects** check box.
5. Click **Show Work Plane**.

Circle 1 (c1)

1. On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.
2. In the **Settings** window for **Circle**, locate the **Size and Shape** section.
3. In the **Radius** text field, type `0.6`.
4. Locate the **Position** section. In the **yw** text field, type `1`.
5. On the **Work Plane** toolbar, click **Build All**.

Square 1 (sq1)

1. On the **Work Plane** toolbar, click **Primitives** and choose **Square**.
2. Click **Build All**.
3. Click the **Zoom Extents** button on the **Graphics** toolbar.

Intersection 1 (int1)

1. On the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Intersection**.
2. Click in the **Graphics** window and then press Ctrl+A to select both objects.
3. On the **Work Plane** toolbar, click **Build All**.

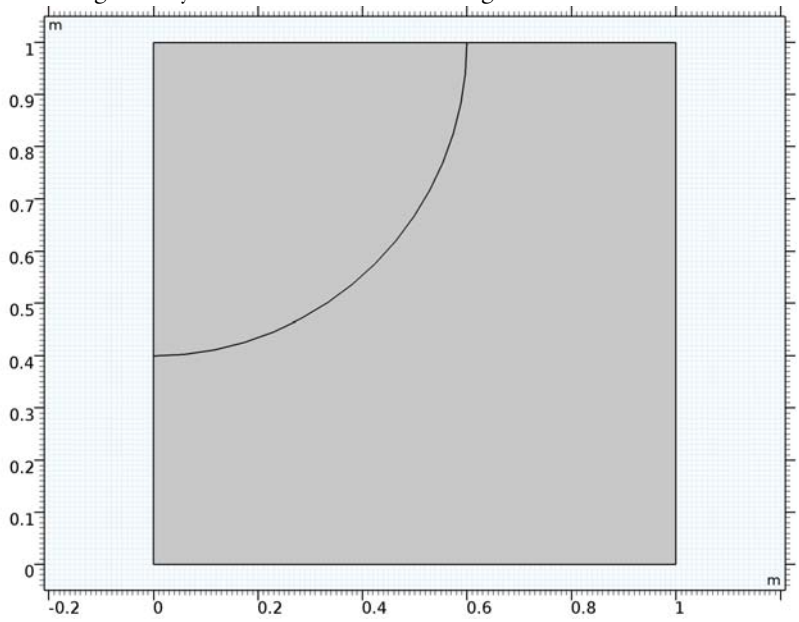
Square 2 (sq2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Square**.

2 Click **Build All**.

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

The 2D geometry should now look as in the figure below.



Work Plane 1 (wp1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 1 (wp1)**.

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

2 Select the object **wp1.sq2** only.

3 In the **Settings** window for **Extrude**, locate the **Distances** section.

4 In the table, enter the following settings:

Distances (m)
-0.1

Block 1 (blk1)

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

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

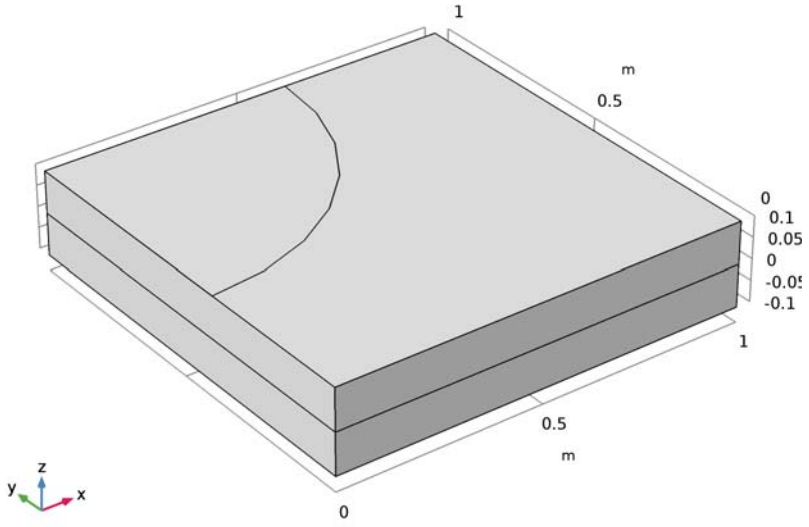
3 In the **Height** text field, type `0.1`.

4 Locate the **Position** section. In the **z** text field, type `-0.1`.

5 Click **Build All Objects**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

The final geometry should look as in the figure below.



ELECTRIC CURRENTS (EC)

Current Conservation 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Electric Currents (ec)** node, then click **Current Conservation 1**.

2 In the **Settings** window for **Current Conservation**, locate the **Conduction Current** section.

3 From the σ list, choose **User defined**. In the associated text field, type `1`.

Normal Current Density 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Normal Current Density**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Normal Current Density**, locate the **Normal Current Density** section.

4 In the J_n text field, type `0.3`.

Ground 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.

2 Select Boundary 8 only.

Contact Impedance 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Contact Impedance**.

2 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

3 Select Boundary 6 only.

4 Click the **Wireframe Rendering** button on the **Graphics** toolbar to restore the rendering setting.

5 In the **Settings** window for **Contact Impedance**, locate the **Contact Impedance** section.

6 In the d_s text field, type `0.02`.

7 From the σ list, choose **User defined**. Keep the default value.

8 From the ε_r list, choose **User defined**. Again, the default value applies.

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, because you will add the desired plots manually.

4 On the **Home** toolbar, click **Compute**.

RESULTS

The following steps show you how to reproduce the surface plot of the potential (right panel of Figure 2).

Surface 1

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

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

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

4 Clear the **Color legend** check box.

5 On the **3D Plot Group 1** toolbar, click **Plot**.

Follow the steps below to visualize the potential distribution along the z-direction in the middle of the device (the right panel of Figure 3).

Cut Line 3D 1

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

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

3 In row **Point 1**, set **X** to `0.5`, **y** to `0.5`, and **z** to `-0.1`.

4 In row **Point 2**, set **X** to `0.5`, **y** to `0.5`, and **z** to `0.1`.

1D Plot Group 2

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

2 In the **Settings** window for **1D Plot Group**, click to expand the **Axis** section.

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

4 In the **x minimum** text field, type `-0.1`.

5 In the **x maximum** text field, type `0.1`.

6 In the **y minimum** text field, type `0.2`.

7 In the **y maximum** text field, type `1`.

Line Graph 1

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

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

3 From the **Data set** list, choose **Cut Line 3D 1**.

4 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Component 1>Geometry>Coordinate>z - z-coordinate**.

5 On the **1D Plot Group 2** toolbar, click **Plot**.

Comparing the Thin-Film Approximation with the Full 3D Model

To reproduce the full 3D model that served as a reference model for comparison in this example, proceed with the following steps:

GEOMETRY 1

Move 1 (mov1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Move**.

2 Select the object **blk1** only.

3 In the **Settings** window for **Move**, locate the **Displacement** section.

4 In the **z** text field, type `-0.02`.

Block 2 (blk2)

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

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

3 In the **Height** text field, type `0.02`.

4 Locate the **Position** section. In the **z** text field, type `-0.02`.

ELECTRIC CURRENTS (EC)

Current Conservation 2

1 On the **Physics** toolbar, click **Domains** and choose **Current Conservation**.

2 Select Domain 2 only.

3 In the **Settings** window for **Current Conservation**, locate the **Conduction Current** section.

4 From the σ list, choose **User defined**. In the associated text field, type `0.01`.

Contact Impedance 1

In the **Model Builder** window, under **Component 1 (comp1)>Electric Currents (ec)** right-click **Contact Impedance 1** and choose **Disable**.

STUDY 1

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

RESULTS

3D Plot Group 1

This plot shows the potential distribution (the left panel of Figure 2).

1D Plot Group 2

This plot reproduces the left panel of Figure 3.

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

2 On the **1D Plot Group 2** toolbar, click **Plot**.

Now, follow these steps to reset the model to use the thin-film approximation.

GEOMETRY 1

Move 1 (mov1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Move 1 (mov1)** and choose **Disable**.

Block 2 (blk2)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Block 2 (blk2)** and choose **Disable**.

ELECTRIC CURRENTS (EC)

Contact Impedance 1

In the **Model Builder** window, under **Component 1 (comp1)>Electric Currents (ec)** right-click **Contact Impedance 1** and choose **Enable**.

Current Conservation 2

In the **Model Builder** window, under **Component 1 (comp1)>Electric Currents (ec)** right-click **Current Conservation 2** and choose **Disable**.

STUDY 1

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

Tuning Fork

Introduction

This example simulates a tuning fork for tuning musical instruments and, if correctly design, should sound the note of A, 440 Hz. It computes the fundamental eigenfrequency and eigenmode for the tuning fork. Although the example seems to be somewhat academic in nature, the eigenfrequencies and eigenmodes of microscopic tuning forks are also used in quartz watches and other electronic devices.

Model Definition

The model geometry is shown in Figure 1. The fundamental frequency of the fork is determined by the length of the prongs, the cross-section geometry of the prongs, and the material properties of the fork.

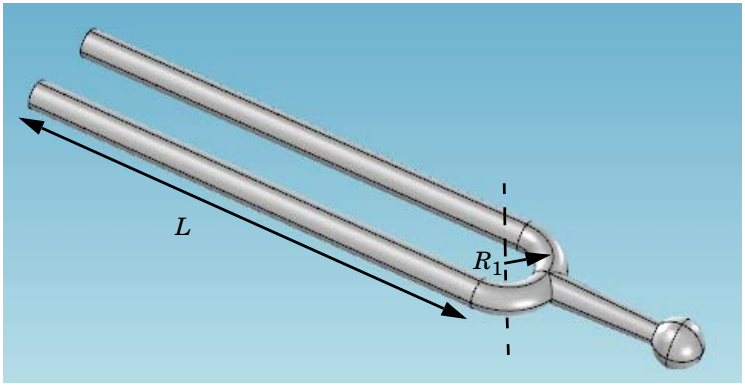


Figure 1: Tuning fork geometry.

The following formula gives a theoretical estimation for the fundamental frequency of a tuning fork with cylindrical cross section of the prong (Ref. 1):

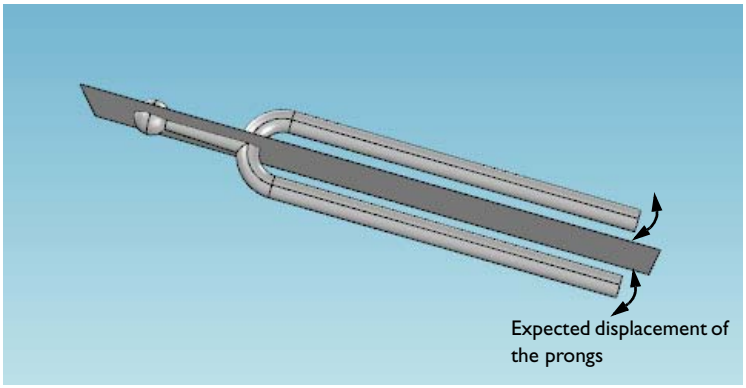
$$f = \frac{1.875^2 R_2}{4\pi L_p^2}\sqrt{\frac{E}{\rho}} \tag{1}$$

where R_2 is the radius of the cross section of the prongs, E denotes Young's modulus, and ρ is the density. The length of the prong can be estimated as

$$L_p = L + \frac{1}{2}\pi R_1 \tag{2}$$

where R_1 the radius of the base, and L is the length of the straight cylindrical part, see Figure 1.

In the fundamental eigenmode, the prongs move according to the figure below. Thus, the eigenmode is symmetric with a symmetry plane placed between the prongs.



The advantage with the shape of the fundamental eigenmode is that the relative displacements in the handle are very small, which makes it possible to hold the fork without damping the vibration. This also allows to make use of the theoretical estimation for the frequency Equation 1 which is based on the solution for a cantilever beam representing each prong.

The parameters used in the model are: R_1 = 7.5 mm and R_2 = 2.5 mm. The fork material is Steel AISI 4340, for which E = 205 GPa and ρ = 7850 kg/m^3.

For the frequency f = 440 Hz, Equation 1 and Equation 2 give the length of the prong cylindrical part as L = 7.8 cm. This presents an underestimation because the part of the prong near the base has larger bending stiffness compared to that for a straight cantilever beam.

To fine-tune the fork, you will use parametrized geometry and gradually increase the cylinder length starting from the above given estimation. To achieve this, you set up a parametric sweep with respect to parameter L.

Reference

1. *Tuning fork*, http://en.wikipedia.org/wiki/Tuning_fork

Application Library path: COMSOL_Multiphysics/Structural_Mechanics/tuning_fork

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 **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Eigenfrequency**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
L	7.8[cm]	0.078 m	Cylinder length
R1	7.5[mm]	0.0075 m	Base radius
R2	2.5[mm]	0.0025 m	Prong radius

GEOMETRY 1

You can build up the fork geometry efficiently using predefined geometry primitives.

Cone 1 (cone1)

1 On the **Geometry** toolbar, click **Cone**.

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

3 In the **Bottom radius** text field, type `R2`.

4 In the **Height** text field, type `2e-2`.

5 From the **Specify top size using** list, choose **Angle**.

6 In the **Semi-angle** text field, type `2`.

7 Locate the **Position** section. In the **x** text field, type `R1`.

8 In the **z** text field, type `-R1`.

9 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.

10 In the **z** text field, type `-1`.

Sphere 1 (sph1)

1 On the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for **Sphere**, locate the **Size** section.

3 In the **Radius** text field, type `4e-3`.

4 Locate the **Position** section. In the **x** text field, type `R1`.

5 In the **z** text field, type `-(R1+2.25e-2)`.

Torus 1 (tor1)

1 On the **Geometry** toolbar, click **Torus**.

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

3 In the **Major radius** text field, type `R1`.

4 In the **Minor radius** text field, type `R2`.

5 In the **Revolution angle** text field, type `180`.

6 Locate the **Position** section. In the **x** text field, type `R1`.

7 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.

8 In the **z** text field, type `0`.

9 In the **y** text field, type `1`.

10 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.

Union 1 (uni1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

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

3 Clear the **Keep interior boundaries** check box.

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

This completes the handle and base of the fork.

Add two cylinders to represent the prongs.

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

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

3 In the **Radius** text field, type `R2`.

4 In the **Height** text field, type `L`.

Cylinder 2 (cyl2)

1 On the **Geometry** toolbar, click **Cylinder**.

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

3 In the **Radius** text field, type `R2`.

4 In the **Height** text field, type `L`.

5 Locate the **Position** section. In the **x** text field, type `2*R1`.

Use virtual geometry operations to avoid short edges and narrow regions. This will improve the mesh generation.

Ignore Edges 1 (ige1)

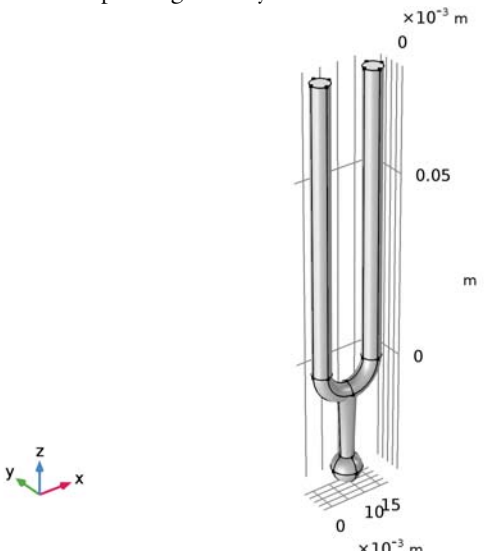
1 On the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Edges**.

2 On the object **fin**, select Edges 22, 23, 29, 32, 33, 39, 42, and 43 only.

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

4 Click the **Go to Default View** button on the **Graphics** toolbar.

The completed geometry should look as shown in the following figure:



ADD MATERIAL

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

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

3 In the tree, select **Built-In>Steel AISI 4340**.

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

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

MESH 1

1 In the **Settings** window for **Mesh**, locate the **Mesh Settings** section.

2 From the **Element size** list, choose **Fine**.

Free Triangular 1

1 Right-click **Component 1 (comp1)>Mesh 1** and choose **More Operations>Free Triangular**.

2 Select Boundaries 6 and 24 only.

3 In the **Settings** window for **Free Triangular**, click **Build Selected**.

Swept 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

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

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

4 Select Domains 1 and 3 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>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 `50`.

Swept 1

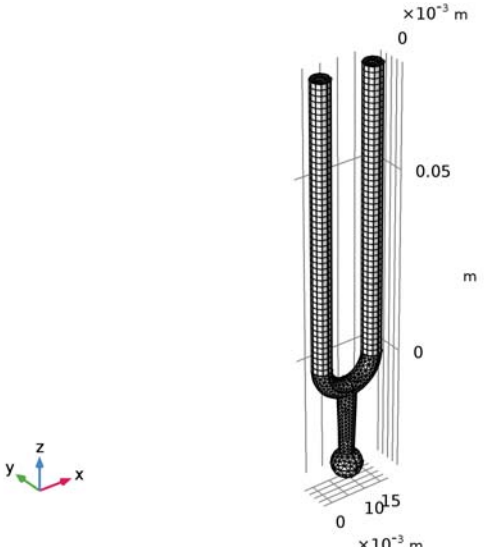
1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Swept 1**.

2 In the **Settings** window for **Swept**, click **Build Selected**.

3 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Tetrahedral**.

4 In the **Settings** window for **Mesh**, click **Build All**.

The meshed geometry should look like that in the figure below.



STUDY 1

Set up a parametric sweep with respect to the cylinder length L and search for an eigenfrequency in the vicinity of 440 Hz.

Parametric Sweep

1 On 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
L (Cylinder length)	`range(0.078,1e-4,0.0795)`

Step 1: Eigenfrequency

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

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

3 Select the **Desired number of eigenfrequencies** check box.

4 In the associated text field, type `1`.

5 In the **Search for eigenfrequencies around** text field, type `440`.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Eigenvalue Solver 1**.

3 In the **Settings** window for **Eigenvalue Solver**, locate the **General** section.

4 In the **Relative tolerance** text field, type `1e-3`.

5 On the **Study** toolbar, click **Compute**.

RESULTS

Mode Shape (solid)

To see all computed eigenfrequencies as a table, follow these steps:

Global Evaluation 1

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

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

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

4 From the **Table columns** list, choose **Inner solutions**.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Solid Mechanics>Global>solid.freq - Frequency**.

6 Click **New Table**.

TABLE

1 Go to the **Table** window.

You can see that the eigenfrequency closest to 440 Hz occurs for the cylinder length of 0.0791 m. Further fine-tuning can be performed if necessary.

RESULTS

To see the eigenmode that corresponds to this frequency, do the following:

Mode Shape (solid)

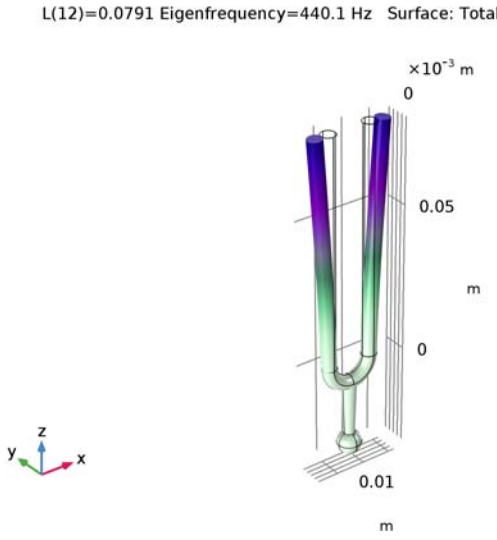
1 In the **Model Builder** window, under **Results** click **Mode Shape (solid)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Parameter value (L)** list, choose **0.0791**.

4 On the **Mode Shape (solid)** toolbar, click **Plot**.

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



In this figure, you can clearly see that mode is symmetric, and the displacements at the handle are very small compared to those of the prongs. This means that holding the tuning fork at the handle will dampen the vibrations negligibly.

Stresses and Strains in a Wrench

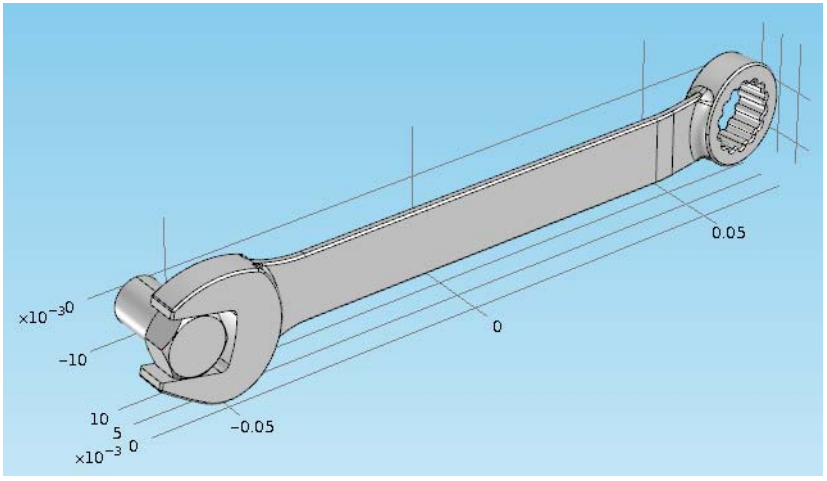
Introduction

This tutorial demonstrates how to set up a simple static structural analysis. The analysis is exemplified on a combination wrench during the application of torque on a bolt.

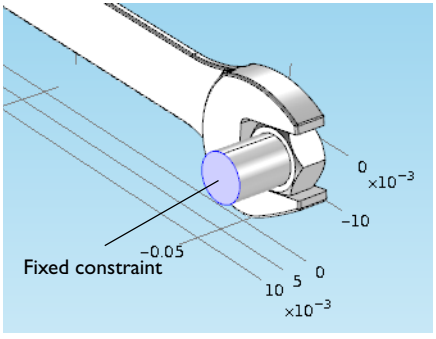
Despite its simplicity, and the fact that very few engineers would run a structural analysis before trying to turn a bolt, the example provides an excellent overview of structural analysis in COMSOL Multiphysics.

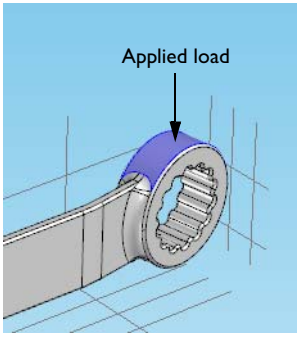
Model Definition

The model geometry is shown below.



The bolt's fixed constraint is at the cross section shown below. A load is applied at the box end of the combination wrench.





Here, assume that there is perfect contact between the wrench and the bolt. A possible extension is to apply a contact condition between the wrench and the bolt where the friction and the contact pressure determines the position of the contact surface.

Application Library path: `COMSOL_Multiphysics/Structural_Mechanics/wrench`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

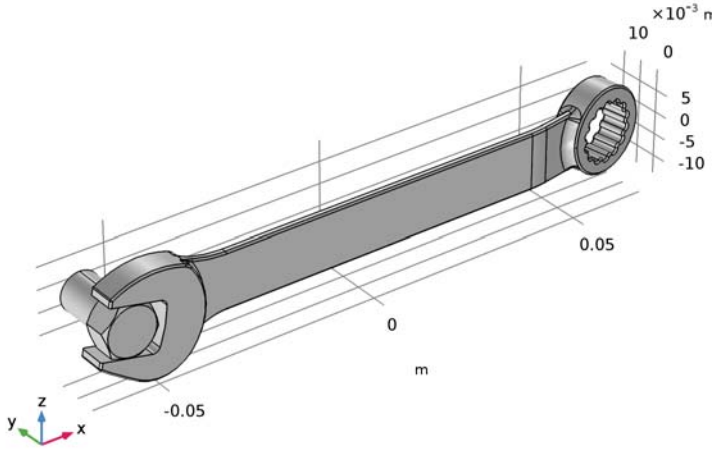
1. In the **Model Wizard** window, click **3D**.
2. In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Stationary**.
6. Click **Done**.

GEOMETRY 1

Import 1 (imp1)

1. On the **Home** toolbar, click **Import**.
2. In the **Settings** window for **Import**, locate the **Import** section.
3. Click **Browse**.
4. Browse to the model's Application Libraries folder and double-click the file `wrench.mphbin`.
5. Click **Build All Objects**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.



ADD MATERIAL

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

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

3 In the tree, select **Built-In>Structural steel**.

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

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

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
F	150[N]	150 N	Applied force

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

3 Select Boundary 35 only.

Boundary Load 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundary 111 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 From the **Load type** list, choose **Total force**.

5 Specify the $\mathbf{F}_{\text{tot}}$ vector as

0	x
0	y
-F	z

The minus sign means that the force is applied downwards.

MESH 1

Use finer mesh because the geometry contains small edges and faces.

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

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

3 From the **Element size** list, choose **Finer**.

4 Click **Build All**.

STUDY 1

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1** node.

4 Right-click **Suggested Iterative Solver (solid)** and choose **Enable**.

5 In the **Settings** window for **Iterative**, locate the **General** section.

6 From the **Preconditioning** list, choose **Right**.

7 On the **Study** toolbar, click **Compute**.

RESULTS

Stress (solid)

The default plot group shows the von Mises stress in a **Surface** plot with the displacement visualized using a **Deformation** subnode. Change to a more suitable unit as follows.

Surface 1

1 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Surface 1**.

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

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

4 On the **Stress (solid)** toolbar, click **Plot**.

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

Stress (solid) 1

1 In the **Model Builder** window, under **Results** right-click **Stress (solid)** and choose **Duplicate**.

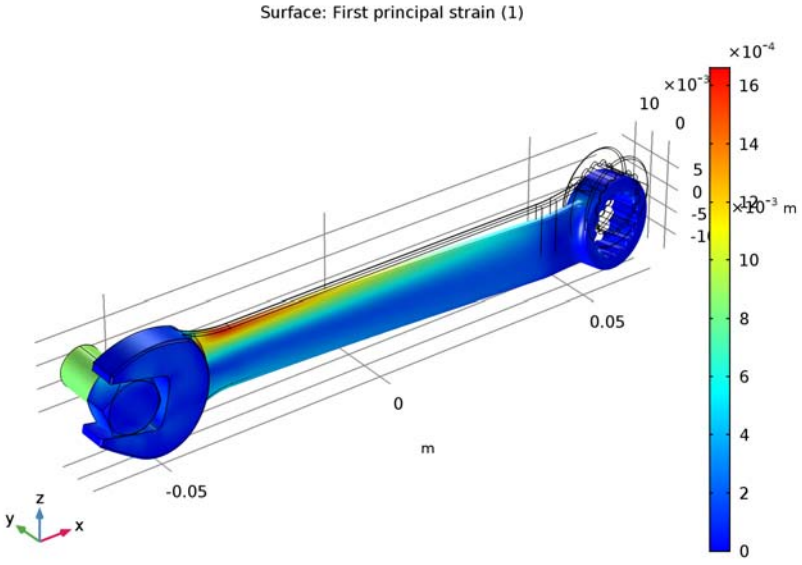
2 In the **Settings** window for **3D Plot Group**, type `First Principal Strain` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Results>First Principal Strain** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics>Strain>Principal strains>solid.ep1 - First principal strain**.

3 On the **First Principal Strain** toolbar, click **Plot**.



Notice that the maximum principal strain is lower than 2%, a result that satisfies the small strain assumption.