Structural Mechanics Module

Application Library Manual

5.2a

COMSOL

Structural Mechanics Module Application Library Manual

Version: COMSOL 5.2a

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: CM021102

Fluid-Structure Interaction in Aluminum Extrusion

Introduction

Out of all metals, the most frequently extruded is aluminum. Aluminum extrusion entails using a hydraulic ram to squeeze an aluminum bar through a die. This process will form the metal into a particular shape. Extruded aluminum is used in many manufacturing applications, such as building components for example. In massive forming processes like rolling or extrusion, metal alloys are deformed in a hot solid state with material flowing under ideally plastic conditions. Such processes can be simulated effectively using computational fluid dynamics, where the material is considered as a fluid with a very high viscosity that depends on velocity and temperature. Internal friction of the moving material acts as a heat source, so that the heat transfer equations are fully coupled with those ruling the fluid dynamics part. This approach is especially advantageous when large deformations are involved.

This model is adapted from a benchmark study in Ref. 1. The original benchmark solves a thermal-structural coupling, because it is common practice in the simulation of such processes to use specific finite element codes that have the capability to couple the structural equations with heat transfer. The alternative scheme discussed here couples non-Newtonian flow with heat transfer equations. In addition, because it is useful to know the stress in the die due to fluid pressure and thermal loads, the model adds a structural mechanics analysis.

The die design is courtesy of Compes S.p.A., while the die geometry, boundary conditions, and experimental data are taken from Ref. 1.

Note: This application requires the Heat Transfer Module and the Structural Mechanics Module. In addition, it uses the Material Library.

Model Definition

The model considers steady-state conditions, assuming a billet of infinite length flowing through the die. In the actual process, the billet is pushed by the ram through the die and its volume is continuously reducing.

Figure 1 shows the original complete geometry with four different profiles. To have a model with reasonable dimensions, consider only a quarter of the original geometry. The simplification involved in neglecting the differences between the four profiles does not affect the numerical scheme proposed. Figure 2 shows the resulting model geometry.

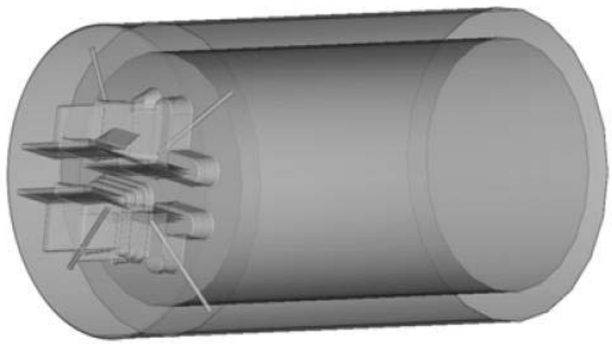


Figure 1: Original benchmark geometry.

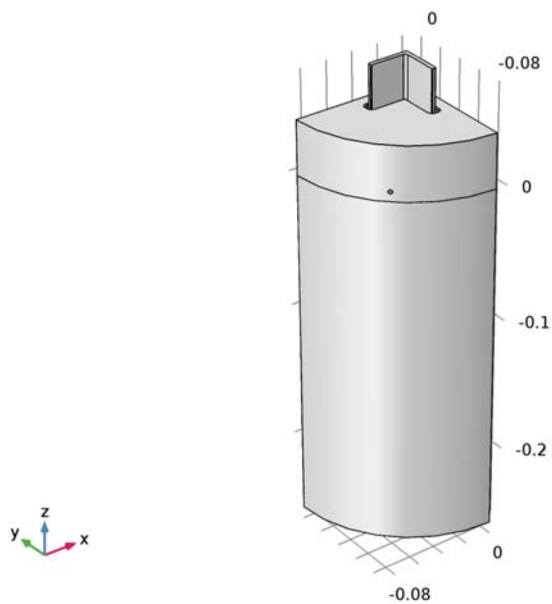


Figure 2: Quarter of the original geometry considered in the model.

MATERIAL PROPERTIES

The documentation for the benchmark model (Ref. 1) serves as the data source for properties of the two main materials: AISI steel for the die and the container (the ram is not considered here) and aluminum for the billet.

Structural Analysis

Because only the steel part is active in the structural analysis, consider a simple linear elastic behavior where the elastic properties are those of the material H11 mod (AISI 610) that can be found in the COMSOL Multiphysics Material Library.

Heat Transfer Analysis

The benchmark model uses the following properties for aluminum and steel:

ALUMINUM	VALUE	DESCRIPTION
k_{al}	210 N/(s·K)	Thermal conductivity
ρ_{al}	2700 kg/m^3	Density
$C_{p\mathrm{al}}$	2.94 N/(mm^2·K)/ρ_{al}	Heat capacity at constant pressure

STEEL	VALUE	DESCRIPTION
k_{fe}	24.33 N/(s·K)	Thermal conductivity
ρ_{fe}	7850 kg/m^3	Density
$C_{p\mathrm{fe}}$	4.63 N/(mm^2·K)/ρ_{fe}	Heat capacity at constant pressure

Non-Newtonian Flow

The properties of the aluminum were experimentally determined and then checked using literature data for the same alloy and surface state. However the benchmark proposes an experimental constitutive law, suited for the structural mechanics codes usually used to simulate such processes, in the form of the flow stress data. For this model this requires a recalculation of the constitutive law to derive a general expression for the viscosity. The equivalent von Mises stress, σ_{eqv}, can be defined in terms of the total contraction of the deviatoric stress tensor as

$$\sigma_{\mathrm{eqv}} = \sqrt{\frac{3}{2}\tau:\tau}$$

or, using $\tau = 2\eta\dot{\varepsilon}$ where $\dot{\varepsilon}$ is the strain rate and η is the viscosity, as

$$\sigma_{\mathrm{eqv}} = \sqrt{6\eta^2\dot{\varepsilon}:\dot{\varepsilon}} \qquad (1)$$

Introducing the equivalent strain rate

$$\dot{\phi}_{\mathrm{eqv}} \equiv \sqrt{\frac{2}{3}\dot{\varepsilon}:\dot{\varepsilon}}$$

Equation 1 can be expressed as

$$\sigma_{eqv} = 3\eta\dot{\phi}_{eqv}$$

The strain rate tensor is defined as (Ref. 2)

$$\dot{\varepsilon} = \frac{\nabla\mathbf{u} + (\nabla\mathbf{u})^T}{2} = \frac{1}{2}\dot{\gamma}$$

The shear rate $\dot{\gamma}$ is defined as

$$\dot{\gamma} = |\dot{\gamma}| = \sqrt{\frac{1}{2}\dot{\gamma}:\dot{\gamma}}$$

so that

$$\phi_{eqv} = \frac{1}{\sqrt{3}}\dot{\gamma}$$

The flow rule

$$\sigma_{eqv} = \kappa_f$$

states that plastic yielding occurs if the equivalent stress, σ_{eqv}, reaches the flow stress, κ_f. The viscosity is defined as (see Ref. 2 for further details)

$$\eta = \frac{\kappa_f}{3\dot{\phi}_{eqv}}$$

The organizers of the benchmark propose specific flow-stress data expressed in terms of a generalized Zener-Hollomon function

$$\eta = \frac{\operatorname{asinh}\left(\left(\frac{Z}{A}\right)^{\frac{1}{n}}\right)}{\sqrt{3}\alpha\dot{\gamma}}$$

where $A = 2.39 \cdot 10^8\ \mathrm{s}^{-1}$, $n = 2.976$, $\alpha = 0.052\ \mathrm{MPa}^{-1}$, and

$$Z = \frac{1}{\sqrt{3}}\dot{\gamma}e^{\left(\frac{Q}{RT}\right)}$$

with $Q = 153$ kJ/mol and $R = 8.314$ J/(K·mol).

SOURCES, INITIAL CONDITIONS, AND BOUNDARY CONDITIONS

Structural Analysis

Because the model geometry is a quarter of the actual geometry, use symmetric boundary conditions for the two orthogonal planes. On the external surfaces of the die, apply roller boundary conditions because in reality other dies, not considered here, are present to increase the system's stiffness.

The main loads are the thermal loads from the heat transfer analysis and pressures from the fluid dynamics analysis.

Heat Transfer Analysis

For the billet, use a volumetric heat source related to the viscous heating effect.

The external temperature of the ram and the die is held constant at 450 °C (723 K). The ambient temperature is 25 °C (298 K). For the heat exchange between aluminum and steel, use the heat transfer coefficient of 11 N/(s·mm·K). Also consider convective heat exchange with air outside the profiles with a fixed convective heat transfer coefficient of 15 $W/(m^2 \cdot K)$.

Apply initial temperatures as given in the following table:

PART	VALUE
Ram	380 °C (653 K)
Container	450 °C (723 K)
Billet	460 °C (733 K)
Die	404 °C (677 K)

Non-Newtonian Flow

At the inlet, the ram moves with a constant velocity of 0.5 mm/s. Impose this boundary condition by simply applying a constant inlet velocity. At the outlet, a normal stress condition with zero external pressure applies. On the surfaces placed on the two symmetry planes, use symmetric conditions. Finally, apply slip boundary conditions on the boundaries placed outside the profile.

Results and Discussion

The general response of the proposed numerical scheme, especially in the zone of the profile, is in good accordance with the experience of the designers. A comparison between the available experimental data and the numerical results of the simulation shows good agreement.

On the basis of the results from the simulation, the engineer can improve the preliminary die design by adjusting relevant physical parameters and operating conditions. For this purpose, the volume plot in Figure 3 showing the temperature field inside the profile gives important information. Furthermore, the combined streamline and slice plot in Figure 4 reveals any imbalances in the velocity field that could result in a crooked profile. A proper design should also ensure that different parts of the profile travel at the same speed. Figure 5 shows the von Mises equivalent stress in the steel part considering the thermal load and the pressure load due to the presence of the fluid.

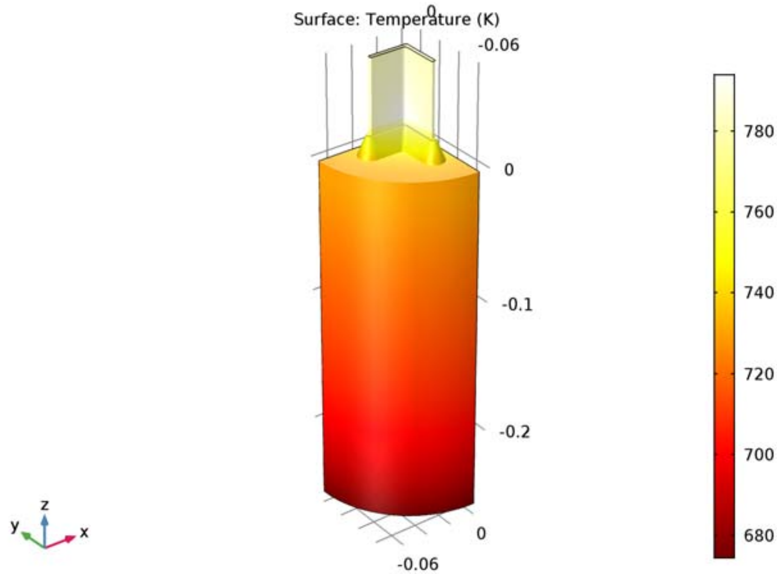


Figure 3: Temperature distribution in the billet.

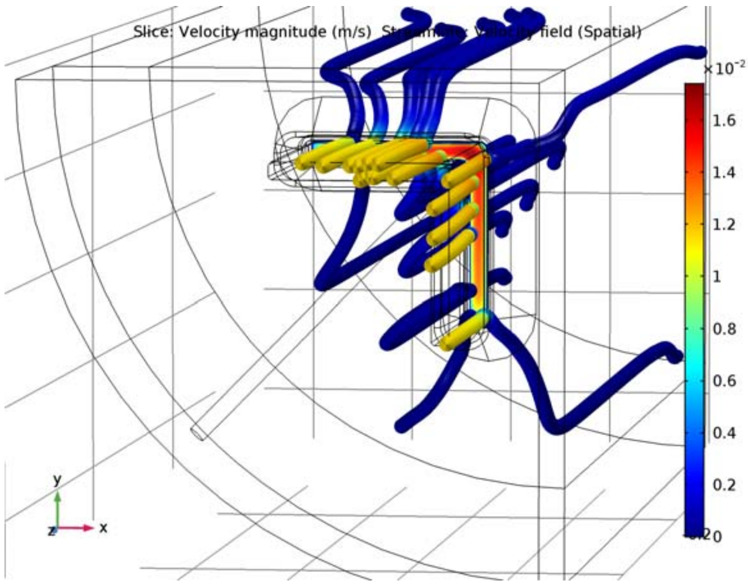


Figure 4: Velocity field and streamlines at the profile section.

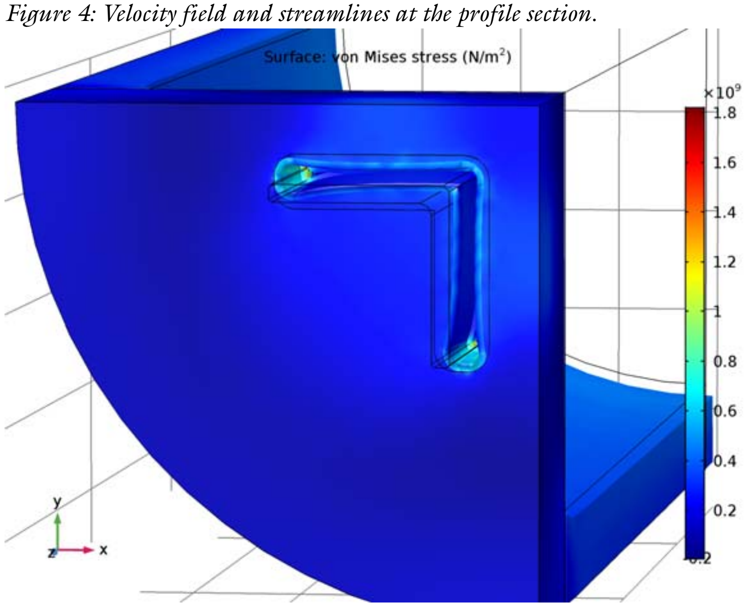


Figure 5: Equivalent von Mises stress distribution in the container.

References

1. M. Schikorra, L. Donati, L. Tomesani, and A.E. Tekkaya, "The Extrusion Benchmark 2007," *Proceedings of the Extrusion Workshop 2007 and 2nd Extrusion Benchmark Conference*, Bologna, Italy, http://diemtech.ing.unibo.it/extrusion07.

2. E.D. Schmitter, "Modelling massive forming processes with thermally coupled fluid dynamics," *Proceedings of the COMSOL Multiphysics User's Conference 2005 Frankfurt*, Frankfurt, Germany.

Application Library path: `Structural_Mechanics_Module/Fluid-Structure_Interaction/aluminum_extrusion_fsi`

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>Conjugate Heat Transfer>Laminar Flow**.
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**.

GEOMETRY 1

Import 1 (imp1)

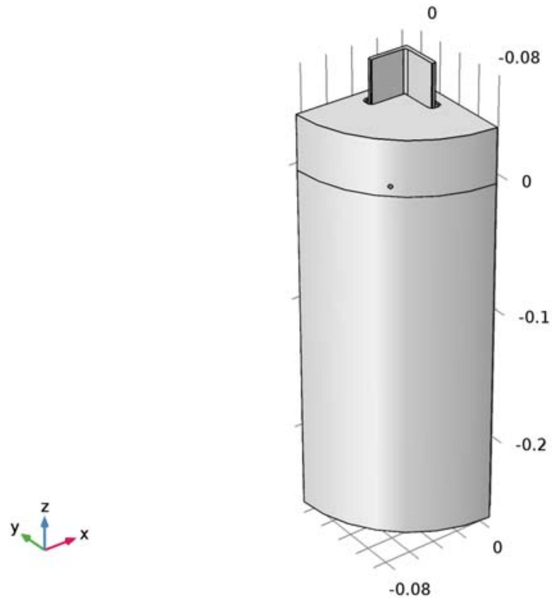
1 On the **Home** toolbar, click **Import**.
2 In the **Settings** window for Import, locate the **Import** section.
3 Click **Browse**.

4 Browse to the application's Application Libraries folder and double-click the file `aluminum_extrusion_fsi.mphbin`.

5 Click **Import**.

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

You should now see the following geometry.



GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, 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_alfe	1[mm]	0.001 m	Thickness of the high conductive layer
Heat_alfe	11[N/(s*mm*K)]	11000 W/(m²·K)	Aluminum-steel heat exchange coefficient
T_billet	460[degC]	733.15 K	Billet temperature

Name	Expression	Value	Description
T_container	450[degC]	723.15 K	Container temperature
T_ram	380[degC]	653.15 K	Ram temperature
T_pd1	404[degC]	677.15 K	Initial temperature around thermocouple at point PD1
V_ram	0.5[mm/s]	5E-4 m/s	Ram velocity
P_init	0[bar]	0 Pa	External reference pressure
T_air	25[degC]	298.15 K	Ambient temperature
Q_eta	153000[J/mol]	1.53E5 J/mol	Parameter Q for the generalized Zener-Hollomon function
n_eta	2.976	2.976	Parameter n for the generalized Zener-Hollomon function
A_eta	2.39e8[1/s]	2.39E8 1/s	Parameter A for the generalized Zener-Hollomon function
alpha_eta	0.0521[1/MPa]	5.21E-8 1/Pa	Parameter alpha for the generalized Zener-Hollomon function
H_conv	15	15	Convective heat exchange coefficient with air
F	sqrt(1/3)	0.57735	Factor for the conversion of the shear rate to COMSOL's definition

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
Z_eta	F*spf.sr*exp(Q_eta/(R_const*T))	1/s	Zener-Hollomon parameter
mu_al	asinh((Z_eta/A_eta)^(1/n_eta))/(3*alpha_eta*F*spf.sr+sqrt(eps))	Pa·s	Viscosity of aluminum

Create the selections to simplify the model specification.

Explicit 1

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

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

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

4 Select Boundaries 35–38, 42, 43, 48, 49, 51, 53, 68, 69, 76–79, 84, 85, 91, 93, 100, 101, 103, and 105 only.

For more convenience in selecting these boundaries, you can click the **Paste Selection** button and paste the above numbers.

Explicit 2

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

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

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

4 Select Boundaries 8, 11, 14, 15, 19, 20, 24, 29–34, 41, 47, 50, 56–58, 62, 67, 70, 71, 74, 75, 83, 86–90, 99, 102, 106, and 107 only.

Before creating the materials for the model, specify the fluid and solid domains. Using this information, the software can detect which material properties are needed.

HEAT TRANSFER (HT)

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer (ht)** click **Fluid 1**.

2 Select Domains 3 and 4 only.

3 In the **Settings** window for Fluid, locate the **Domain Selection** section.

4 Click **Create Selection**.

5 In the **Create Selection** dialog box, type `Billet` in the **Selection name** text field.

6 Click **OK**.

LAMINAR FLOW (SPF)

On the **Physics** toolbar, click **Heat Transfer (ht)** and choose **Laminar Flow (spf)**.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 In the **Settings** window for Laminar Flow, locate the **Physical Model** section.

3 From the **Compressibility** list, choose **Incompressible flow**.

4 Locate the **Domain Selection** section. From the **Selection** list, choose **Billet**.

SOLID MECHANICS (SOLID)

On the **Physics** toolbar, click **Laminar Flow (spf)** and choose **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 **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domains 1 and 2 only.

Now, define the material for each domain.

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 **Material Library>Tool Steels>H11 mod (AISI 610)>H11 mod (AISI 610) [solid]>H11 mod (AISI 610) [solid,triple tempered]**.

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

MATERIALS

H11 mod (AISI 610) [solid,triple tempered] (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **H11 mod (AISI 610) [solid,triple tempered] (mat1)**.

2 Select Domains 1 and 2 only.

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

4 In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Heat capacity at constant pressure	Cp	`4.63[N/(mm^2*K)]/rho(T[1/K])[kg/m^3]`	J/(kg·K)	Basic

Because the heat capacity only enters the transient heat transfer equation, this setting does not affect the steady-state simulation described here; it is provided for completeness in case you want to extend the model to perform transient simulations.

Material 2 (mat2)

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 `Billet` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Billet**.

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

Property	**Name**	**Value**	**Unit**	**Property group**
Thermal conductivity	k	`210`	W/(m·K)	Basic
Density	rho	`2700`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`2.94[N/(mm^2*K)]/rho`	J/(kg·K)	Basic
Ratio of specific heats	gamma	`1`	1	Basic
Dynamic viscosity	mu	`mu_al`	Pa·s	Basic

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

With the materials defined, you can set up the remaining physics of the model.

LAMINAR FLOW (SPF)

In the current model the viscosity in the fluid flow part is large, which implies that the model is diffusion dominated. Pseudo time stepping works poorly for this model because it is based on the scale of the convective flux.

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

2 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

3 In the **Settings** window for Laminar Flow, click to expand the **Advanced settings** section.

4 Locate the **Advanced Settings** section. Find the **Pseudo time stepping** subsection. Clear the **Use pseudo time stepping for stationary equation form** check box.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Laminar Flow (spf)** click **Initial Values 1**.

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

3 In the p text field, type `P_init`.

4 In the **Model Builder** window, click **Laminar Flow (spf)**.

Symmetry 1

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

2 Select Boundaries 9 and 112 only.

Inlet 1

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

2 Select Boundary 10 only.

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

4 Click the **Velocity field** button.

5 Specify the $\mathbf{u}_0$ vector as

0	x
0	y
V_ram	z

Wall 2

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

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

3 From the **Selection** list, choose **Outside**.

4 Locate the **Boundary Condition** section. From the **Boundary condition** list, choose **Slip**.

Outlet 1

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

2 Select Boundary 40 only.

3 In the **Settings** window for Outlet, locate the **Pressure Conditions** section.

4 In the p_0 text field, type `P_init`.

HEAT TRANSFER (HT)

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer (ht)** click **Initial Values 1**.
2 In the **Settings** window for Initial Values, type `T_container` in the T text field.
3 In the **Model Builder** window, click **Heat Transfer (ht)**.

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.
2 Select Boundaries 2, 5, and 7 only.
3 In the **Settings** window for Temperature, locate the **Temperature** section.
4 In the T_0 text field, type `T_container`.

Heat Flux 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
2 Select Boundary 10 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 `Heat_alfe`.
6 In the T_{ext} text field, type `T_ram`.

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 **Outside**.
4 Locate the **Heat Flux** section. Click the **Convective heat flux** button.
5 In the h text field, type `H_conv`.
6 In the T_{ext} text field, type `T_air`.

Outflow 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outflow**.
2 Select Boundary 40 only.

Thin Layer 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Thin Layer**.
2 In the **Settings** window for Thin Layer, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Interior**.

4 Locate the **Thin Layer** section. From the **Specify** list, choose **Thermal resistance**.

5 In the R_s text field, type `1/Heat_alfe`.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 Select Domains 1 and 2 only.

For faster convergence use linear elements. You can always refine the solution using the default quadratic elements.

3 In the **Model Builder** window's toolbar, click the **Show** button and select **Discretization** in the menu.

4 In the **Settings** window for Solid Mechanics, click to expand the **Discretization** section.

5 From the **Displacement field** list, choose **Linear**.

Roller 1

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

2 Select Boundaries 2, 5, and 7 only.

Symmetry 1

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

2 Select Boundaries 1, 4, 110, and 111 only.

Boundary Load 1

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

2 Select Boundaries 8, 11, 14, 15, 19, 20, 24, 29–34, 41, 47, 50, 56–58, 62, 67, 70, 71, 74, 75, 83, 86–90, 99, 102, 106, and 107 only.

3 In the **Settings** window for Boundary Load, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.

5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

0	t1
0	t2
-p	n

MULTIPHYSICS

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Non-Isothermal Flow 1 (nitf1)**.

2 In the **Settings** window for Non-Isothermal Flow, locate the **Flow Heating** section.

3 Select the **Include viscous dissipation** check box.

Thermal Expansion 1 (te1)

1 On the **Physics** toolbar, click **Multiphysics** and choose **Domain>Thermal Expansion**.

2 Select Domain 1 only.

3 In the **Settings** window for Thermal Expansion, locate the **Thermal Expansion Properties** section.

4 In the T_{ref} text field, type `T_container`.

Thermal Expansion 2 (te2)

1 On the **Physics** toolbar, click **Multiphysics** and choose **Domain>Thermal Expansion**.

2 Select Domains 2–4 only.

3 In the **Settings** window for Thermal Expansion, locate the **Thermal Expansion Properties** section.

4 In the T_{ref} text field, type `T_pd1`.

MESH 1

On the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

Free Triangular 1

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

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

3 Select Boundary 40 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1** and choose **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. Select the **Maximum element size** check box.

5 In the associated text field, type `0.0014`.

6 Select the **Curvature factor** check box.

7 In the associated text field, type `0.2`.

8 Click **Build Selected**.

9 On the **Mesh** toolbar, click **Swept**.

Swept 1

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

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

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

4 Select Domain 4 only.

5 On the **Mesh** toolbar, click **Distribution**.

Distribution 1

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

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 On the **Mesh** toolbar, click **Free Tetrahedral**.

Free Tetrahedral 1

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

Size 1

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

2 Click the **Custom** button.

3 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

4 In the associated text field, type `0.0085`.

Free Tetrahedral 1

Right-click **Free Tetrahedral 1** and choose **Size**.

Size 2

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

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

3 Select Boundaries 12 and 13 only.

4 Locate the **Element Size** section. Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `0.002`.

Free Tetrahedral 1

Right-click **Free Tetrahedral 1** and choose **Size**.

Size 3

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

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

3 Select Boundaries 24, 31, 32, 70, 88, and 106 only.

4 Locate the **Element Size** section. Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Minimum element size** check box.

6 In the associated text field, type `1e-5`.

Size

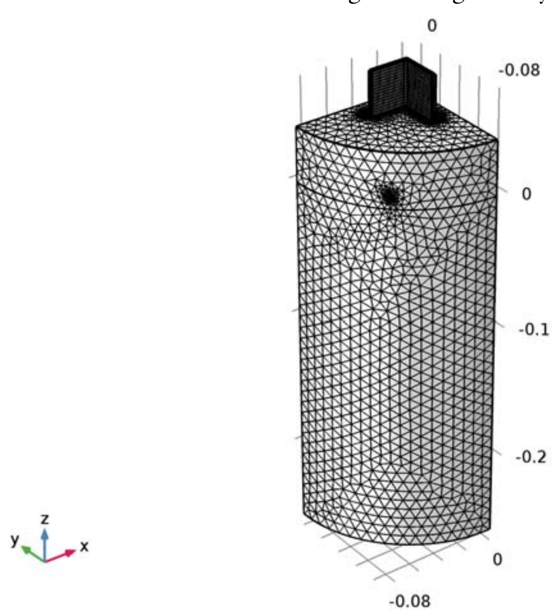
1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

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

3 From the **Predefined** list, choose **Finer**.

4 Click **Build All**.

You should now see the following meshed geometry.



STUDY 1

Step 1: Stationary

Use two stationary study steps. Solve first for the fluid dynamics and heat transfer to determine the thermal load and the pressure load and then for the structural mechanics.

1 In the **Model Builder** window, expand the **Study 1** node, then 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** interface.

Stationary 2

On the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

Step 2: Stationary 2

1 In the **Settings** window for Stationary, locate the **Physics and Variables Selection** section.

2 Select the **Modify physics tree and variables for study step** check box.

3 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Laminar Flow (spf)**.

4 Click **Disable in Solvers**.

5 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Heat Transfer (ht)**.

6 Click **Disable in Solvers**.

7 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Multiphysics> Non-Isothermal Flow 1 (nitf1)**.

8 Click **Disable in Solvers**.

For the structural analysis, use a memory efficient iterative solver to make it possible to solve the problem also on computers with limited memory.

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.

4 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 2** node.

5 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 2** and choose **Iterative**.

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

RESULTS

Temperature (ht)

The first default plot shows the temperature (Figure 3).

1 In the **Model Builder** window, under **Results** click **Temperature (ht)**.
2 In the **Settings** window for 3D Plot Group, locate the **Data** section.
3 From the **Data set** list, choose **Exterior Walls**.
4 On the **Temperature (ht)** toolbar, click **Plot**.

Data Sets

Modify the third default plot to see the velocity field and streamlines at the profile section (Figure 4).

Study 1/Solution Store 1 (sol2)

In the **Model Builder** window, expand the **Data Sets** node, then click **Study 1/Solution Store 1 (sol2)**.

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 **Domain**.
4 From the **Selection** list, choose **Billet**.

Velocity (spf)

1 In the **Model Builder** window, under **Results** click **Velocity (spf)**.
2 In the **Settings** window for 3D Plot Group, locate the **Data** section.
3 From the **Data set** list, choose **Study 1/Solution Store 1 (sol2)**.

Slice

1 In the **Model Builder** window, expand the **Velocity (spf)** node, then click **Slice**.
2 In the **Settings** window for Slice, locate the **Plane Data** section.
3 From the **Plane** list, choose **XY-planes**.
4 From the **Entry method** list, choose **Coordinates**.
5 In the **Z-coordinates** text field, type `0.0151`.
6 On the **Velocity (spf)** toolbar, click **Plot**.

Velocity (spf)

In the **Model Builder** window, under **Results** right-click **Velocity (spf)** and choose **Streamline**.

Streamline 1

1 In the **Settings** window for Streamline, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Laminar Flow>Velocity and pressure>u,v,w - Velocity field (Spatial)**.

2 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Start point controlled**.

3 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.

4 Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Slice**.

Color Expression 1

1 Right-click **Results>Velocity (spf)>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 **Model>Component 1>Laminar Flow>Velocity and pressure>spf.U - Velocity magnitude**.

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

To get a better view, rotate the geometry in the **Graphics** window and use the **Zoom Box** tool to obtain a close-up. You can preserve a view for a plot by creating a **View** feature node as follows:

4 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Results Options** in the menu.

Streamline 1

In the **Model Builder** window, expand the **Results>Velocity (spf)>Streamline 1** node.

View 3D 2

1 Right-click **Results>Views** and choose **View 3D**.

2 Use the **Graphics** toolbox to get a satisfying view.

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

4 Select the **Lock camera** check box.

Next, apply the view to the velocity plot.

Velocity (spf)

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

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

3 From the **View** list, choose **View 3D 2**.

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

Stress (solid)

The last plot shows the von Mises stress and deformation distribution in the container. To reproduce the Figure 5, apply the View 3D 2.

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.
2 In the **Settings** window for 3D Plot Group, locate the **Plot Settings** section.
3 From the **View** list, choose **View 3D 2**.
4 On the **Stress (solid)** toolbar, click **Plot**.

Velocity (spf)

1 In the **Settings** window for 3D Plot Group, locate the **Data** section.
2 From the **Data set** list, choose **Study 1/Solution 1 (sol1)**.
3 On the **Velocity (spf)** toolbar, click **Plot**.

Fluid-Structure Interaction in a Network of Blood Vessels

Introduction

This example studies a portion of the vascular system, in particular the upper part of the aorta (Figure 1). The aorta and its ramified blood vessels are embedded in biological tissue, specifically the cardiac muscle. The flowing blood applies pressure to the artery's internal surfaces and its branches, thereby deforming the tissue. The analysis consists of two distinct but coupled procedures: first, a fluid-dynamics analysis including a calculation of the velocity field and pressure distribution in the blood (variable in time and in space); second, a mechanical analysis of the deformation of the tissue and artery. Any change in the shape of the vessel walls does not influence the fluid domain, which implies that there is only a one-way fluid-structural coupling.

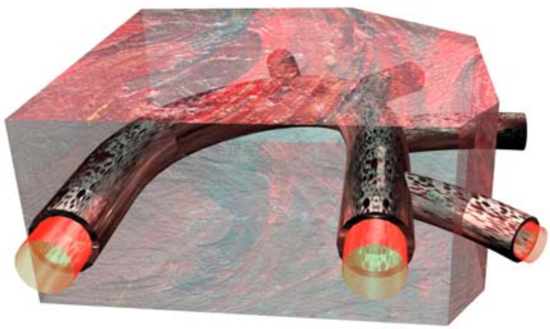


Figure 1: The model domain consists of part of the aorta, its branches, and the surrounding tissue.

Model Definition

Figure 2 shows two views of the model domain, one with and one without the cardiac muscle. The mechanical analysis must consider the cardiac muscle because it presents a stiffness that resists artery deformation due to the applied pressure.

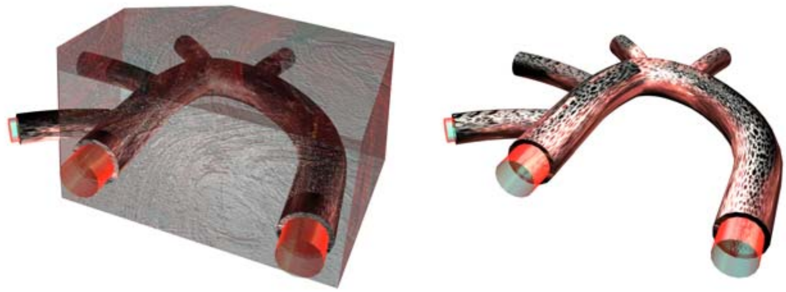


Figure 2: A view of the aorta and its ramification (branching vessels) with blood contained, shown both with (left) and without (right) the cardiac muscle.

The main characteristics of the analyses are:

- *Fluid dynamics analysis*

  Here the Navier-Stokes equations are solved in the blood domain. At each surface where the model brings a vessel to an abrupt end, it represents the load with a known pressure distribution.

- *Mechanical analysis*

  Only the domains related to the biological tissues are active in this analysis. The model represents the load with the total stress distribution it computes during the fluid-dynamics analysis.

ANALYSIS OF RUBBER-LIKE TISSUE AND ARTERY MATERIAL MODELS

Generally, the modeling of biological tissue is an advanced subject for several reasons:

- The material can undergo very large strains (finite deformations).
- The stress-strain relationship is generally nonlinear.
- Many hyperelastic materials are almost incompressible. You must then revise standard displacement-based finite element formulations in order to arrive at correct results (mixed formulations).

You must pay particular attention to the definition of stress and strain measures. In a geometrically nonlinear analysis the assumptions about infinitesimal displacements are no longer valid. It is necessary to consider geometrical nonlinearity in a model when:

- Significant rigid-body rotations occur (finite rotations).

- The strains are no longer small (larger than a few percent).
- The loading of the body depends on the deformation.

All of these issues are dealt with in the hyperelastic material model built-in the Nonlinear Structural Materials Module.

In this case, the displacements and strains are so small that it is sufficient to use a linear elastic material model. The material data is given for a neo-Hookean hyperelastic material, but in the small strain limit the interpretation of the material constants is the same for a linear elastic material.

MATERIALS

The following material properties are used:

- Blood
  - density = 1060 kg/m^3
  - dynamic viscosity = 0.005 Ns/m^2
- Artery
  - density = 960 kg/m^3
  - Neo-Hookean hyperelastic behavior: the coefficient μ equals $6.20 \cdot 10^6$ N/m^2, while the bulk modulus equals 20μ and corresponds to a value for Poisson's ratio, ν, of 0.45. An equivalent elastic modulus equals $1.0 \cdot 10^7$ N/m^2.
- Cardiac muscle
  - density = 1200 kg/m^3
  - Neo-Hookean hyperelastic behavior: the coefficient μ equals $7.20 \cdot 10^6$ N/m^2, while the bulk modulus equals 20μ and corresponds to a value for Poisson's ratio, ν, of 0.45. An equivalent elastic modulus equals $1.16 \cdot 10^6$ N/m^2.

FLUID DYNAMICS ANALYSIS

The fluid dynamics analysis considers the solution of the 3D Navier-Stokes equations. You can do so in both a stationary case or in the time domain. To establish the boundary conditions, six pressure conditions are applied with the configuration shown in Figure 3.

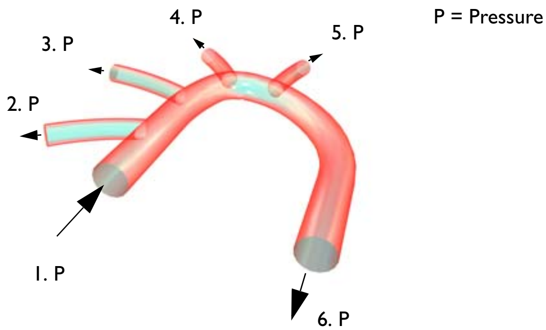


Figure 3: Boundary conditions for the fluid-flow analysis.

The pressure conditions are:

- Section 1: 126.09 mmHg
- Section 2: 125.91 mmHg
- Section 3: 125.415 mmHg
- Section 4: 125.415 mmHg
- Section 5: 125.415 mmHg
- Section 6: 125.1 mmHg

Those pressure values are the mean values over a heart beating cycle. During a cycle the pressure varies between a minimal and a maximal values which are calculated thanks to a relative amplitude α. For the time-dependent analysis, a simple trigonometric function is used for varying the pressure distribution over time:

$$f(t) = \begin{cases} (1-\alpha)\sin(\pi t) & 0 \le t \le 0{,}5\mathrm{s} \\ 1-\alpha\cos(2\pi(t-0.5)) & 0{,}5\mathrm{s} \le t \le 1{,}5\mathrm{s} \end{cases}$$

The first piece of function between 0 and 0.5 s has no physical significance, it is just a ramp that enable to calculate the initial state. The second piece of function makes the pressure vary between its minimal and maximal value during a 1 s cycle.

You implement this effect in COMSOL Multiphysics using Piecewise function.

The flow field at the time t=1 s is displayed in Figure 4 as a slice plot.

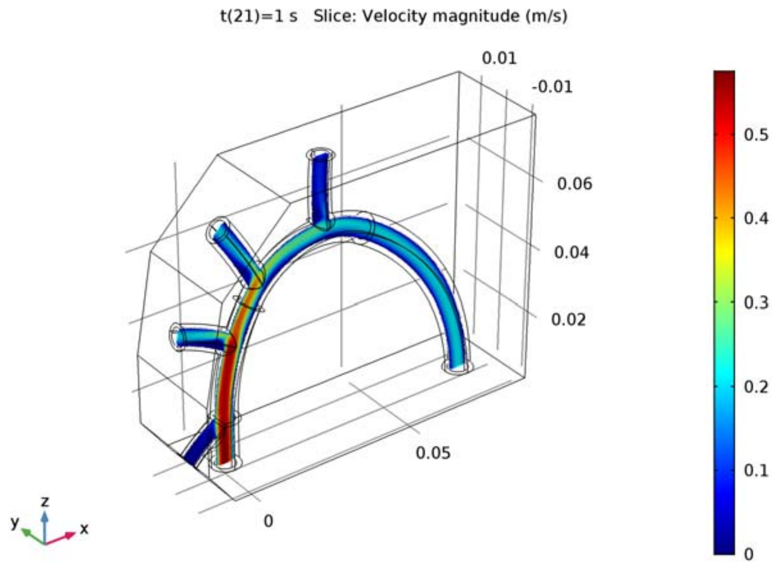


Figure 4: Velocity field in the aorta and its ramification (branching).

Figure 5 shows the total displacement at the peak load (after 1 s). The displacements are in the order of 4 μm, which suggests that the one-way multiphysics coupling is a reasonable approximation.

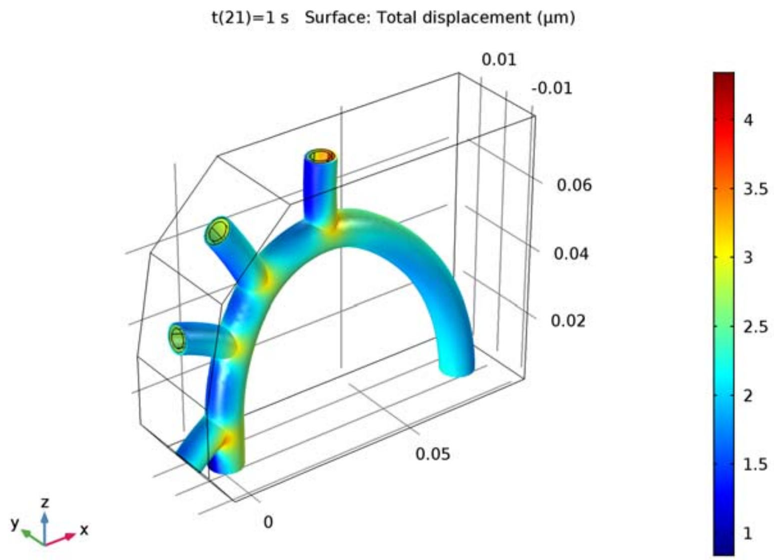


Figure 5: Displacements in the blood vessel.

Notes About the COMSOL Implementation

In this example, and many other cases, an analysis which is time dependent for one physics can be treated as quasi-static from the structural mechanics point of view. You can handle this by running the structural analysis as a parametric sweep over a number of static load cases, where the time is used as the parameter. This method is used here.

Application Library path: `Structural_Mechanics_Module/Bioengineering/blood_vessel`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Fluid Flow>Fluid-Structure Interaction, Fixed Geometry**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Time Dependent**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, 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
t	0[s]	0 s	Time continuation parameter
alpha	1/3	0.3333	Relative pressure amplitude during heart's beating

Piecewise 1 (pw1)

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

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

3 Locate the **Definition** section. In the **Argument** text field, type `t`.

4 Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
0	0.5	(1-alpha)*sin(pi*t)
0.5	1.5	1-alpha*cos(2*pi*(t-0.5))

5 Locate the **Units** section. In the **Arguments** text field, type `s`.

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

7 Click **Plot**.

GEOMETRY 1

The geometry for this model is available as an MPHBIN-file. Import this file as follows.

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 application's Application Libraries folder and double-click the file `blood_vessel.mphbin`.
5. Click **Import**.

   The length unit in the imported geometry is centimeters, while the default length unit in COMSOL Multiphysics is meters. Therefore, you need to rescale the geometry.

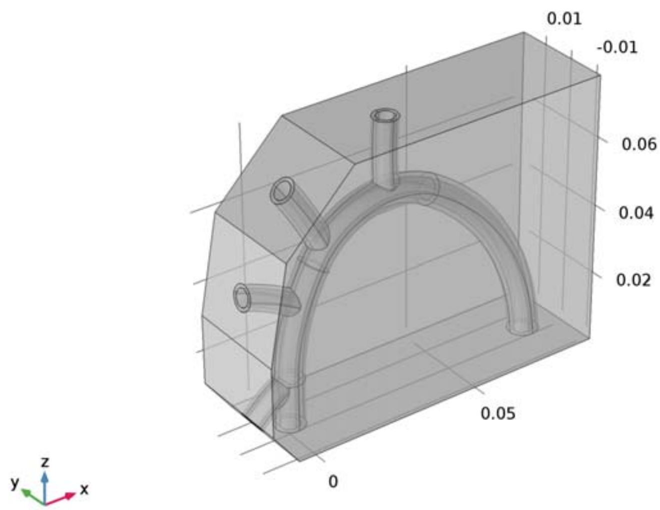
Scale 1 (sca1)

1. On the **Geometry** toolbar, click **Transforms** and choose **Scale**.
2. In the **Settings** window for Scale, locate the **Scale Factor** section.
3. In the **Factor** text field, type `0.01`.
4. Select the object **imp1** only.
5. Right-click **Scale 1 (sca1)** and choose **Build Selected**.
6. Click the **Go to Default 3D View** button on the **Graphics** toolbar.

Form Union (fin)

1. In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

2 Click the **Transparency** button on the **Graphics** toolbar to see the interior.



DEFINITIONS

Next, define a number of selections as sets of geometric entities for use in setting up the model.

Explicit 1

1. On the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for Explicit, type `Blood` in the **Label** text field.
3. Select Domain 3 only.

Explicit 2-10

1 Proceed to create nine explicit selections with the following settings:

Label	Geometric entity level	Selection
`Artery`	`Domain`	`2`
`Muscle`	`Domain`	`1`
`Inlet`	`Boundary`	`38`
`Outlet 1`	`Boundary`	`19`
`Outlet 2`	`Boundary`	`9`
`Outlet 3`	`Boundary`	`41`
`Outlet 4`	`Boundary`	`70`
`Outlet 5`	`Boundary`	`86`
`Roller boundaries`	`Boundary`	`1-6,12,26,27,30,33,64, 67,85,87`

The roller boundaries are the free boundaries of muscle and artery that are neither in contact with each other nor with blood.

Explicit 11

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

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

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

4 Select Boundaries 10, 20, 36, 42, and 68 only.

5 Select the **Group by continuous tangent** check box. The selection should now contains boundaries 10-11, 16-17, 20-21, 23-24, 36-37, 39-40, 42-43, 45-46, 50-53, 58-59, 61-62, 68-69, 75-76, 79-80, 82-83.

The loaded boundaries are the inner artery boundaries that are in contact with blood.

Explicit 12

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

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

3 Select Domain 2 only.

4 Locate the **Output Entities** section. From the **Output entities** list, choose **Adjacent boundaries**.

5 Select the **Interior boundaries** check box.

LAMINAR FLOW (SPF)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.
2. In the **Settings** window for Laminar Flow, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Blood**.

Inlet 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2. In the **Settings** window for Inlet, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Inlet**.
4. Locate the **Boundary Condition** section. From the list, choose **Pressure**.
5. Locate the **Pressure Conditions** section. In the p_0 text field, type `126.09[mmHg]*f(t)`.

Outlet 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.
2. In the **Settings** window for Outlet, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Outlet 1**.
4. Locate the **Pressure Conditions** section. In the p_0 text field, type `125.91[mmHg]*f(t)`.

Outlet 2-5

Proceed to add four outlet boundary nodes with the following settings:

Boundary Selection	**p0**
`Outlet 2`	`125.415[mmHg]*f(t)`
`Outlet 3`	`125.415[mmHg]*f(t)`
`Outlet 4`	`125.415[mmHg]*f(t)`
`Outlet 2`	`125.1[mmHg]*f(t)`

SOLID MECHANICS (SOLID)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.
2. Select Domains 1 and 2 only.

Linear Elastic Material 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.
2. In the **Settings** window for Linear Elastic Material, locate the **Linear Elastic Material** section.
3. From the **Specify** list, choose **Lamé parameters**.

4 In the **Model Builder** window, click **Solid Mechanics (solid)**.

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 **Roller boundaries**.

MULTIPHYSICS

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Fluid-Structure Interaction, Fixed Geometry 1 (fsifg1)**.

2 In the **Settings** window for Fluid-Structure Interaction, Fixed Geometry, locate the **Coupling Type** section.

3 From the list, choose **Fluid loading on structure** to ensure a one way coupling, from the fluid to the solid.

MATERIALS

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

Material 1 (mat1)

1 In the **Settings** window for Material, type `Blood` in the **Label** text field.

2 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Blood**.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Density	rho	`1060`	kg/m³	Basic
Dynamic viscosity	mu	`0.005`	Pa·s	Basic

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for Material, type `Artery` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Artery**.

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

Property	Name	Value	Unit	Property group
Lamé parameter λ	lambLame	`20*muLame-2*muLame/3`	N/m²	Lamé parameters

Property	Name	Value	Unit	Property group
Lamé parameter μ	muLame	`6.20e6`	N/m^2	Lamé parameters
Density	rho	`960`	kg/m^3	Basic

Material 3 (mat3)

1. Right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for Material, type `Muscle` in the **Label** text field.
3. Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Muscle**.
4. Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Lamé parameter λ	lambLame	`20*muLame-2*muLame/3`	N/m^2	Lamé parameters
Lamé parameter μ	muLame	`7.20e6`	N/m^2	Lamé parameters
Density	rho	`1200`	kg/m^3	Basic

MESH 1

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

Free Tetrahedral 1

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

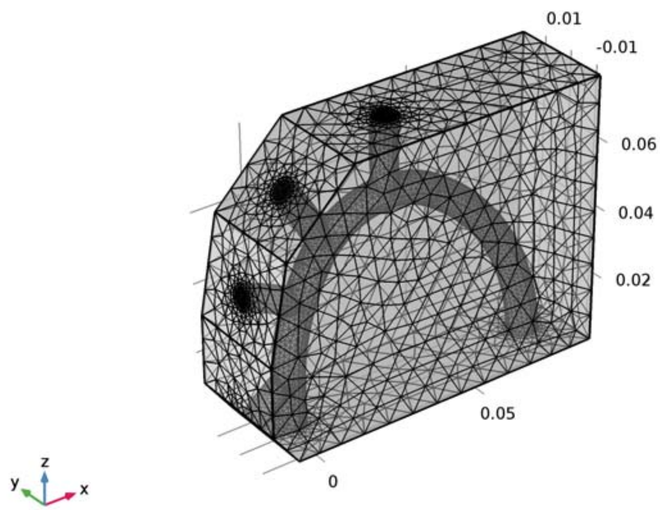
Size 1

1. In the **Settings** window for Size, locate the **Geometric Entity Selection** section.
2. From the **Geometric entity level** list, choose **Domain**.
3. From the **Selection** list, choose **Blood**.
4. Locate the **Element Size** section. Click the **Custom** button.
5. Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
6. In the associated text field, type `1e-3`.

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

The structural problem is quasi-static, so you can use the time just as a parameter for the parametric solver, together with a stationary solver. Thus the whole study can be divided into two steps. First run the transient study for the fluid-mechanics part of the problem and then use the stationary solver to solve the structural part using the solution from first transient study.

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(0,0.05,1.5)`.
4. Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for the **Solid Mechanics** interface.

Stationary

On the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

Step 2: Stationary

1 In the **Settings** window for Stationary, locate the **Physics and Variables Selection** section.

2 In the table, clear the **Solve for** check box for the **Laminar Flow** interface.

3 Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
`t`	`range(0,0.05,1.5)`	`s`

6 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

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

8 From the **Study** list, choose **Study 1, Time Dependent**.

9 From the **Selection** list, choose **All**.

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 **Steps taken by solver** list, choose **Intermediate**. This way the solver computes at least once between each output time step in order to reduce possible interpolation error in the fluid load evaluation.

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

RESULTS

Velocity (spf)

1 Click the **Transparency** button on the **Graphics** toolbar to restore the original transparency state.

By default, you get a slice plot of the velocity and a contour plot of the fluid pressure on the wall surface.The plot in Figure 4 corresponds to the first default plot.

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

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

4 From the **Time (s)** list, choose **1**.

Slice

1 In the **Model Builder** window, under **Results>Velocity (spf)** click **Slice**.

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

3 From the **Plane** list, choose **ZX-planes**.

4 In the **Planes** text field, type `1`.

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

6 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

Pressure (spf)

The default unit for pressure plot is Pascal. As the `mmHg` unit is not available in the selection list, type it directly in the text field.

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

2 In the **Settings** window for 3D Plot Group, click to expand the **Title** section.

Pressure

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

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

3 In the **Unit** field, type `mmHg`.

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

Data Sets

To reproduce the plot shown in Figure 5, begin by defining a selection for the solution data set to make interior boundaries visible in the plot.

Study 1/Solution 1 (3) (sol1)

On the **Results** toolbar, click **More Data Sets** and choose **Solution**.

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 **Artery walls**.

Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 In the **Settings** window for 3D Plot Group, type `Displacement (solid)` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1/Solution 1 (3) (sol1)**.

4 From the **Time (s)** list, choose **1**.

Surface 1

1 In the **Model Builder** window, expand the **Results>Displacement (solid)** 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>solid.disp - Total displacement**.

3 Locate the **Expression** section. From the **Unit** list, choose **µm**.

Deformation

1 In the **Model Builder** window, expand the **Surface 1** node, then click **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 `300`.

5 On the **Displacement (solid)** toolbar, click **Plot**.

6 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

Created in COMSOL Multiphysics 5.2a

Bracket Geometry

This is a template MPH-file containing the bracket geometry. For a description of this model, including detailed step-by-step instructions showing how to build it, see the section “The Fundamentals: A Static Linear Analysis” in the book *Introduction to the Structural Mechanics Module*.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_basic`

Created in COMSOL Multiphysics 5.2a

Bracket—Contact Analysis

Introduction

The various examples of a bracket form a suite of tutorials which summarizes the fundamentals when modeling structural mechanics problems in COMSOL Multiphysics and the Structural Mechanics Module.

This example illustrates how to solve a structural contact problem between two elastic bodies. You learn how to manually add a **Contact pair** node and define the boundaries to be in contact, then add the **Contact** boundary condition to enable the structural contact between the two parts of the assembly. It also shows how to compute the pre-tension in a bolt.

It is recommended that you review the *Introduction to the Structural Mechanics Module*, which includes background information and discusses the `bracket_basic.mph` model relevant to this example.

Model Definition

This tutorial is an extension to the model example described in the section "The Fundamentals: A Static Linear Analysis" in the *Introduction to the Structural Mechanics Module*. In the original model, a displacement constraint is used to represent the mounting bolts, while in the current model the bolts and the fixation plate are modeled (see Figure 1). The contact pressures between the bracket, the bolts and the plate are computed. In the first study, the bolts are assumed to be bonded to the bracket and the bolt, and the pre-tension in the bolts is computed. In a second study an external load is applied to the bracket arm and contact forces including friction is computed between all parts of the assembly.

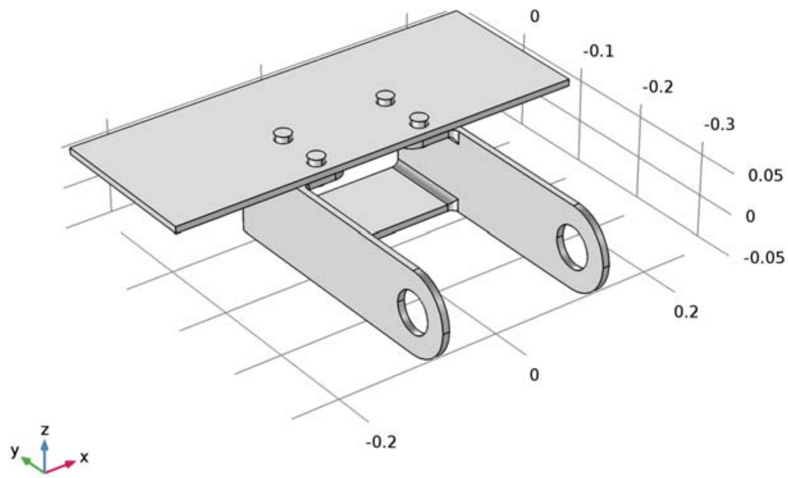


Figure 1: The geometry of the bracket, the bolts and the mounting plate.

Results and Discussion

Figure 2 shows the displacement with only the pre-tension in bolt. The bolts are assumed to be bonded with the plate and the bracket. The maximum displacement in the bracket is around the bolt region.

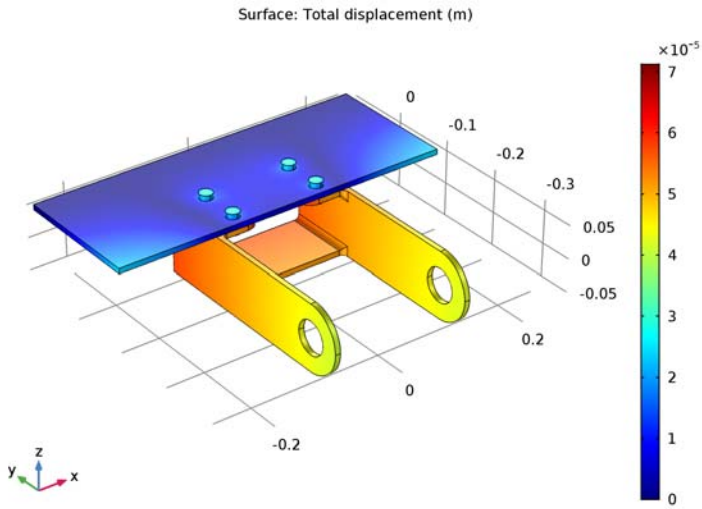


Figure 2: Total displacement under bolt pre-tension load case.

Figure 3 shows the contact pressure distribution between the bracket and the plate under bolt pre-tension load case. The contact pressure is computed using the penalty method.

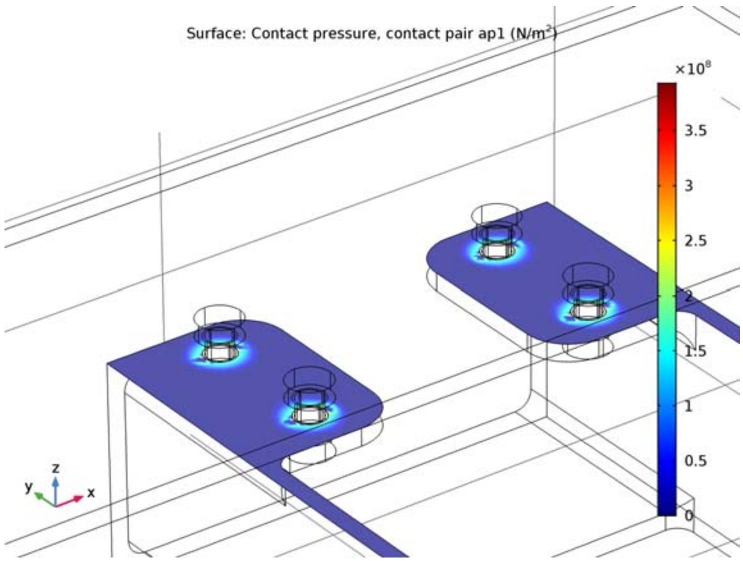


Figure 3: Contact pressure distribution between the bracket and the plate under bolt pre-tension load case.

Figure 4 shows the displacement when the contact pressure is computed between each part of the assembly under external load and bolt pre-tension.

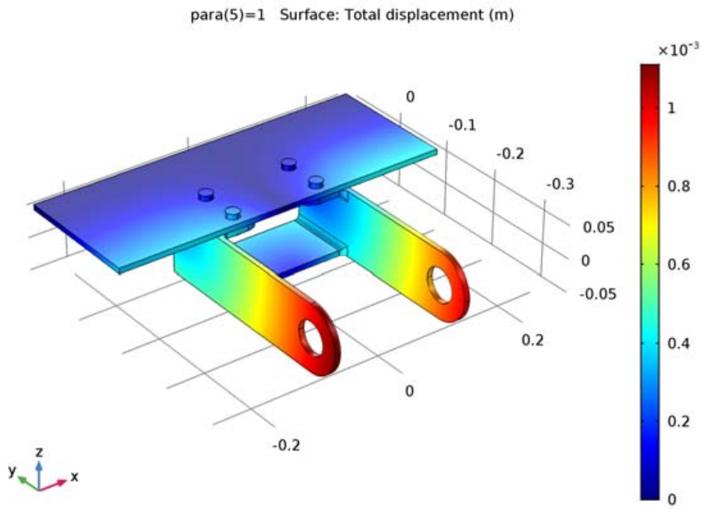


Figure 4: Total displacement of the assembly under external load and bolt pre-tension.

Figure 5 shows the contact pressure distribution between the bracket and the plate under external load and bolt pre-tension load case. The contact pressure is computed using the augmented lagrangian method.

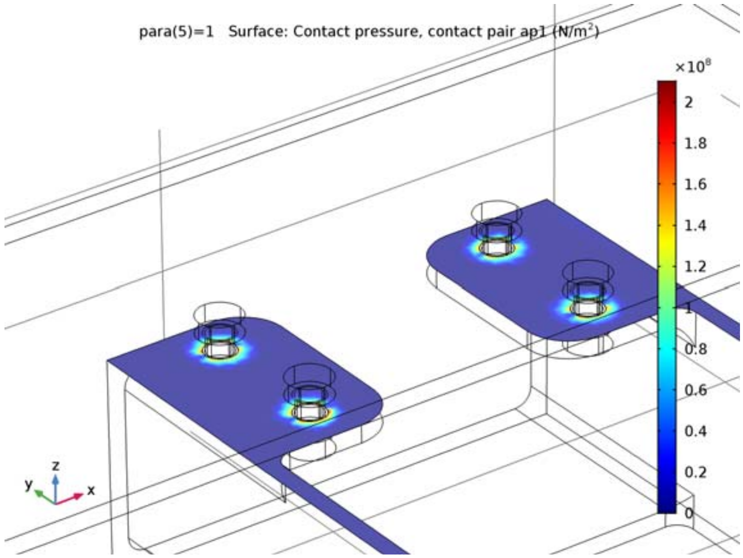


Figure 5: Contact pressure distribution between the bracket and the plate under external load and bolt pre-tension.

Figure 6 shows the contact pressure distribution under the bold heads with external load and bolt pre-tension load case. The contact pressure is computed using the augmented Lagrangian method.

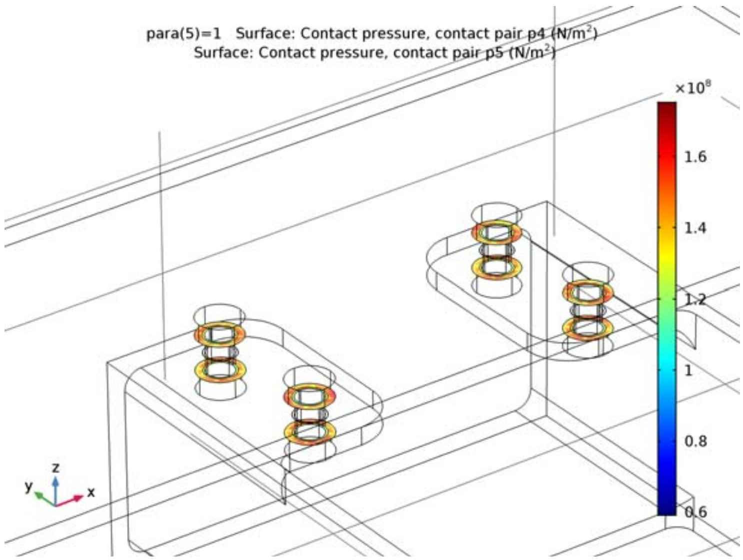


Figure 6: Contact pressure distribution under the bolt heads with external load and bolt pre-tension.

Figure 7 shows the friction force distribution between under the bolt heads with external load and bolt pre-tension load case.

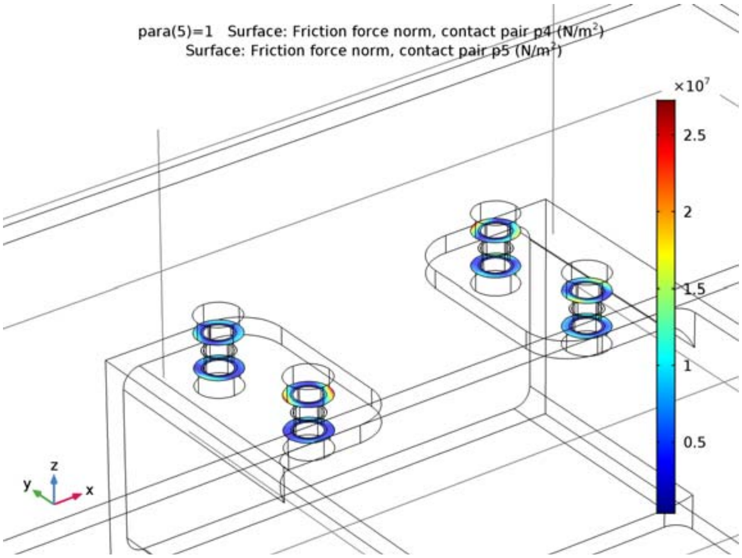


Figure 7: Contact pressure distribution under the bolt heads with external load and bolt pre-tension.

Notes About the COMSOL Implementation

In COMSOL Multiphysics, the contact pressure is evaluated as a function of the gap distance between the parts that are set to be in contact. This gap value is evaluated in the **Contact pair** node, where you define the source and destination boundaries.

When modeling contact, it is recommended to use a finer mesh on the destination contact boundary than on the source contact boundary.

In contact problems, it is common that some components are not sufficiently constrained in the initial configuration. There are then possible rigid body motions, and it is not possible to find a converged solution. Also, when modeling a frictionless contact, sliding may be unconstrained even after contact is established.

In this example, bonded contact with the bolts is assumed in the first study. Bonded contact can be implemented using identity pairs and a continuity boundary condition. This way the displacements in all directions are continuous between the pair boundaries.

In the second study, weak springs are used to suppress the initial singularity. This is a common approach. It is often possible to stabilize a problem with springs that are so weak that the forces they generate in the converged state are negligible. It is also possible to remove the springs gradually during the solution, or in a separate study step.

To compute the contact pressure you can either choose the penalty method or the augmented Lagrangian method. The penalty provides a faster and more stable solution, while the augmented Lagrangian method ensures minimal penetration between the parts in contact and accurate contact stresses.

Read more about how to set contact problem in the Contact Modeling in the *Structural Mechanics User's Guide*.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_contact`

Modeling Instructions

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

Browse to the application's Application Libraries folder and double-click the file `bracket_basic.mph`.

GEOMETRY 1

Import 2 (imp2)

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 application's Application Libraries folder and double-click the file `bracket_bolt_and_support.mphbin`.
5. Click **Import**.

Explicit Selection 1 (sel1)

1. On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.
2. In the **Settings** window for Explicit Selection, type `Bolts` in the **Label** text field.
3. Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Object**.
4. Select the object **imp2(2)** only.

Form Union (fin)

The **Form Union/Assembly** node determines how the parts of the assembly are considered in the analysis. By using the default setting, **Form a union**, the parts of the assembly are considered to be one unique object. The mounting bolts are automatically bonded to the bracket and the support plate. Select **Form an assembly** to consider each part of the assembly as a separate object. The mounting bolts are not connected to the bracket or the support plate. You need to include pairs to connect assembly parts with each other.

1. In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.
2. In the **Settings** window for Form Union/Assembly, locate the **Form Union/Assembly** section.
3. From the **Action** list, choose **Form an assembly**.
4. From the **Pair type** list, choose **Contact pair**.
5. From the **Repair tolerance** list, choose **Relative**.
6. In the **Relative repair tolerance** text field, type `1E-3`.
7. Right-click **Component 1 (comp1)>Geometry 1>Form Union (fin)** and choose **Build Selected**.
8. 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>Titanium beta-21S**.
4. Click **Add to Component** in the window toolbar.
5. On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Titanium beta-21S (mat2)

1. In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Titanium beta-21S (mat2)**.
2. In the **Settings** window for Material, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Bolts**.

SOLID MECHANICS (SOLID)

Bolt Pre-Tension 1

1 On the **Physics** toolbar, click **Global** and choose **Bolt Pre-Tension**.
2 In the **Settings** window for Bolt Pre-Tension, locate the **Bolt Pre-Tension** section.
3 From the **Pre-tension type** list, choose **Pre-tension stress**.
4 In the σ_p text field, type `400[MPa]`.

Bolt Selection 1

1 In the **Model Builder** window, expand the **Bolt Pre-Tension 1** node, then click **Bolt Selection 1**.
2 Select Boundary 87 only.

Bolt Pre-Tension 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Bolt Pre-Tension 1**.

Bolt Selection 2

1 On the **Physics** toolbar, click **Attributes** and choose **Bolt Selection**.
2 Select Boundary 92 only.
3 In the **Settings** window for Bolt Selection, locate the **Bolt Selection** section.
4 In the **Bolt label** text field, type `Bolt_2`.

Bolt Pre-Tension 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Bolt Pre-Tension 1**.

Bolt Selection 3

1 On the **Physics** toolbar, click **Attributes** and choose **Bolt Selection**.
2 Select Boundary 129 only.
3 In the **Settings** window for Bolt Selection, locate the **Bolt Selection** section.
4 In the **Bolt label** text field, type `Bolt_3`.

Bolt Pre-Tension 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Bolt Pre-Tension 1**.

Bolt Selection 4

1 On the **Physics** toolbar, click **Attributes** and choose **Bolt Selection**.

2 Select Boundary 134 only.

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

4 In the **Bolt label** text field, type `Bolt_4`.

Fixed Constraint 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Fixed Constraint 1**.

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

3 Click **Clear Selection**.

4 Select Boundary 5 only.

Continuity 1

On the **Physics** toolbar, in the Boundary section, click **Pairs** and choose **Continuity**.

DEFINITIONS

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node, then click **Contact Pair 2 (ap2)**.

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

3 From the **Pair type** list, choose **Identity pair**.

4 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Contact Pair 3 (ap3)**.

5 In the **Settings** window for Pair, locate the **Pair Type** section.

6 From the **Pair type** list, choose **Identity pair**.

SOLID MECHANICS (SOLID)

Continuity 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Continuity 1**.

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

3 In the **Pairs** list, choose **Identity Pair 2 (ap2)** and **Identity Pair 3 (ap3)**.

Contact 1

1 On the **Physics** toolbar, in the Boundary section, click **Pairs** and choose **Contact**.

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

3 In the **Pairs** list, select **Contact Pair 1 (ap1)**.

4 Locate the **Contact Pressure Method** section. From the list, choose **Penalty**.

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.

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-4`.

5 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** node, then click **Displacement field (Material) (comp1.u)**.

6 In the **Settings** window for Field, locate the **Scaling** section.

7 In the **Scale** text field, type `1e-4`.

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

RESULTS

Stress (solid)

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

Surface 1

1 In the **Model Builder** window, expand the **Results>Stress (solid)** node, then click **Surface 1**.

2 In the **Settings** window for Surface, type `Displacement, bolt bonded` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `solid.disp`.

4 On the **Stress (solid)** toolbar, click **Plot**.

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, type `Contact pressure, penalty` in the **Label** text field.

Surface 1

1 Right-click **Contact pressure, penalty** and choose **Surface**.

2 On the **Contact pressure, penalty** toolbar, click **Plot**.

3 In the **Model Builder** window, under **Results>Contact pressure, penalty** click **Surface 1**.

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

5 In the **Expression** text field, type `solid.Tn_ap1`.

6 On the **Contact pressure, penalty** toolbar, click **Plot**.

DEFINITIONS

Contact Pair 4 (p4)

1 On the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 Select Boundary 4 only.

3 In the **Settings** window for Pair, locate the **Destination Boundaries** section.

4 Select the **Active** toggle button.

5 Select Boundaries 73, 81, 115, and 123 only.

Contact Pair 5 (p5)

1 On the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 Select Boundaries 39 and 49 only.

3 In the **Settings** window for Pair, locate the **Destination Boundaries** section.

4 Select the **Active** toggle button.

5 Select Boundaries 70, 78, 112, and 120 only.

SOLID MECHANICS (SOLID)

Contact 2

1 On the **Physics** toolbar, in the Boundary section, click **Pairs** and choose **Contact**.

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

3 In the **Pairs** list, choose **Contact Pair 4 (p4)** and **Contact Pair 5 (p5)**.

4 Locate the **Penalty Factor** section. From the **Tuned for** list, choose **Speed**.

5 Locate the **Initial Values** section. In the T_n text field, type `10[MPa]`.

Friction 1

1 On the **Physics** toolbar, click **Attributes** and choose **Friction**.

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

3 In the μ_{stat} text field, type 0.2.

4 Locate the **Initial Values** section. From the **Previous contact state** list, choose **In contact**.

Contact 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Contact 1**.

2 In the **Settings** window for Contact, locate the **Contact Pressure Method** section.

3 From the list, choose **Augmented Lagrangian**.

4 Locate the **Penalty Factor** section. From the **Tuned for** list, choose **Speed**.

5 Locate the **Initial Values** section. In the T_n text field, type solid.Tn_ap1.

Friction 1

1 On the **Physics** toolbar, click **Attributes** and choose **Friction**.

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

3 In the μ_{stat} text field, type 0.1.

4 Locate the **Initial Values** section. From the **Previous contact state** list, choose **In contact**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, 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	2.5[MPa]	2.5E6 Pa	Peak load intensity
YC	-300[mm]	-0.3 m	Y coordinate of hole center
para	1	1	Control parameter

DEFINITIONS

Analytic 1 (an1)

1 On the **Home** toolbar, click **Functions** and choose **Local>Analytic**.

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

3 Locate the **Definition** section. In the **Expression** text field, type F*cos(atan2(py, abs(px))).

4 In the **Arguments** text field, type F, py, px.

5 Locate the **Units** section. In the **Arguments** text field, type `Pa, m, m`.

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

Step 1 (step1)

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

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

3 In the **Location** text field, type `0.25`.

4 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `0.5`.

Step 2 (step2)

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

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

3 In the **Location** text field, type `0.75`.

4 In the **From** text field, type `1`.

5 In the **To** text field, type `0`.

6 Locate the **Smoothing** section. In the **Size of transition zone** text field, type `0.5`.

SOLID MECHANICS (SOLID)

Boundary Load 1

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

2 Select Boundaries 26 and 65 only.

3 In the **Settings** window for Boundary Load, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.

5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

`0`	t1
`0`	t2
`load(-P0,Y-YC,Z)*step1(para)`	n

Spring Foundation 1

1 On the **Physics** toolbar, click **Domains** and choose **Spring Foundation**.

2 Select Domains 2–10 only.

3 In the **Settings** window for Spring Foundation, locate the **Spring** section.

4 In the $\mathbf{k}_V$ text field, type `1e12*step2(para)`.

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.

STUDY 2

Step 1: Stationary

Disable the continuity condition.

1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

2 In the **Settings** window for Stationary, locate the **Physics and Variables Selection** section.

3 Select the **Modify physics tree and variables for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Continuity 1**.

5 Click **Disable**.

6 Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

7 Click **Add**.

8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para	`range(0,0.25,1)`	

9 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**.

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

11 From the **Study** list, choose **Study 1, Stationary**.

12 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

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

14 From the **Study** list, choose **Study 1, 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 **Dependent Variables 1**.

3 In the **Settings** window for Dependent Variables, locate the **General** section.

4 From the **Defined by study step** list, choose **User defined**.

5 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** node, then click **Contact pressure (comp1.solid.Tn_ap1)**.

6 In the **Settings** window for Field, locate the **Scaling** section.

7 In the **Scale** text field, type `2e8`.

8 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** click **Contact pressure (comp1.solid.Tn_p4)**.

9 In the **Settings** window for Field, locate the **Scaling** section.

10 In the **Scale** text field, type `2e8`.

11 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** click **Contact pressure (comp1.solid.Tn_p5)**.

12 In the **Settings** window for Field, locate the **Scaling** section.

13 In the **Scale** text field, type `2e8`.

14 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** click **Friction force (Spatial) (comp1.solid.Tt_ap1)**.

15 In the **Settings** window for Field, locate the **Scaling** section.

16 In the **Scale** text field, type `2e7`.

17 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** click **Friction force (Spatial) (comp1.solid.Tt_p4)**.

18 In the **Settings** window for Field, locate the **Scaling** section.

19 In the **Scale** text field, type `2e7`.

20 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** click **Friction force (Spatial) (comp1.solid.Tt_p5)**.

21 In the **Settings** window for Field, locate the **Scaling** section.

22 In the **Scale** text field, type `2e7`.

23 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** click **Pre-deformation (comp1.solid.pblt1.sblt1.d_pre)**.

24 In the **Settings** window for State, locate the **General** section.

25 Clear the **Solve for this state** check box.

26 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** click **Pre-deformation (comp1.solid.pblt1.sblt2.d_pre)**.

27 In the **Settings** window for State, locate the **General** section.

28 Clear the **Solve for this state** check box.

29 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** click **Pre-deformation (comp1.solid.pblt1.sblt3.d_pre)**.

30 In the **Settings** window for State, locate the **General** section.

31 Clear the **Solve for this state** check box.

32 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** click **Pre-deformation (comp1.solid.pblt1.sblt4.d_pre)**.

33 In the **Settings** window for State, locate the **General** section.

34 Clear the **Solve for this state** check box.

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

RESULTS

Stress (solid) 1

1 In the **Model Builder** window, under **Results** click **Stress (solid) 1**.

2 In the **Settings** window for 3D Plot Group, type `Displacement` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Results>Displacement** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `solid.disp`.

4 On the **Displacement** toolbar, click **Plot**.

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

Contact pressure, penalty

In the **Model Builder** window, under **Results** right-click **Contact pressure, penalty** and choose **Duplicate**.

Contact pressure, penalty 1

1 In the **Settings** window for 3D Plot Group, type `Contact pressure, augmented Lagrange` in the **Label** text field.

2 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

3 On the **Contact pressure, augmented Lagrange** toolbar, click **Plot**.

Contact pressure, augmented Lagrange

In the **Model Builder** window, under **Results** right-click **Contact pressure, augmented Lagrange** and choose **Duplicate**.

Contact pressure, augmented Lagrange 1

In the **Settings** window for 3D Plot Group, type `Contact pressure, bolts` in the **Label** text field.

Surface 1

1. In the **Model Builder** window, expand the **Results>Contact pressure, bolts** node, then click **Surface 1**.
2. In the **Settings** window for Surface, locate the **Expression** section.
3. In the **Expression** text field, type `solid.Tn_p4`.

Surface 2

1. Right-click **Results>Contact pressure, bolts>Surface 1** and choose **Duplicate**.
2. In the **Settings** window for Surface, locate the **Expression** section.
3. In the **Expression** text field, type `solid.Tn_p5`.
4. Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.
5. On the **Contact pressure, bolts** toolbar, click **Plot**.

Contact pressure, bolts

In the **Model Builder** window, under **Results** right-click **Contact pressure, bolts** and choose **Duplicate**.

Contact pressure, bolts 1

In the **Settings** window for 3D Plot Group, type `Friction force norm` in the **Label** text field.

Surface 1

1. In the **Model Builder** window, expand the **Results>Friction force norm** node, then click **Surface 1**.
2. In the **Settings** window for Surface, locate the **Expression** section.
3. In the **Expression** text field, type `solid.cnt2.fric1.Ttnorm_p4`.

Surface 2

1. In the **Model Builder** window, under **Results>Friction force norm** click **Surface 2**.

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

3 In the **Expression** text field, type `solid.cnt2.fric1.Ttnorm_p5`.

4 On the **Friction force norm** toolbar, click **Plot**.

If you want to generate a model which is identical to the one in the model library, follow the instructions below. Otherwise, the modeling is complete.

Displacement

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

STUDY 1

Step 1: Stationary

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

2 In the **Settings** window for Stationary, locate the **Physics and Variables Selection** section.

3 Select the **Modify physics tree and variables for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Contact 1>Friction 1**.

5 Click **Disable**.

6 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Contact 2**.

7 Click **Disable**.

8 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Boundary Load 1**.

9 Click **Disable**.

10 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Spring Foundation 1**.

11 Click **Disable**.

Bracket—Eigenfrequency Analysis

Introduction

In this example you learn how to perform an eigenfrequency analysis for both an unloaded structure and a prestressed structure.

In the case when the structure is subjected to a constant external load, the stiffness generated by the stress may affect the natural frequencies of the structure. Tensile stresses tend to increase the natural frequencies, while compressive stresses tend to decrease them.

It is recommended you review the *Introduction to the Structural Mechanics Module*, which includes background information and discusses the `bracket_basic.mph` model relevant to this example.

Model Definition

This tutorial is an extension to the example described in the section "The Fundamentals: A Static Linear Analysis" in the *Introduction to the Structural Mechanics Module*.

The model geometry is represented in Figure 1.

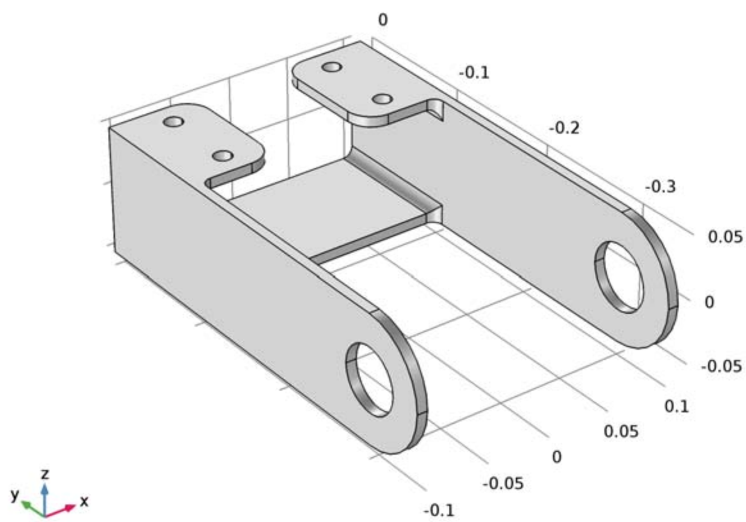


Figure 1: Geometry of the bracket.

The first case studies the natural frequency of the unloaded bracket, while in the second case the study considers how the natural frequencies are affected by an external load applied at the bracket holes. The left arm is under a pure tensile load while the right arm is under pure compressive load.

Results and Discussion

Figure 2 and Figure 3 show the first sixth eigenmode for both the unloaded and the prestressed case, respectively. The mode shape are listed in order from left to right and top to bottom. One can noticed the difference in the two first mode shape between the two load case.

The two first mode shapes correspond to the bending mode in x-direction in the bracket arm, for the unloaded case these are expected to about the same. For the prestressed load case however one expect a difference because of stress stiffening (left arm) and stress softening (right arm).

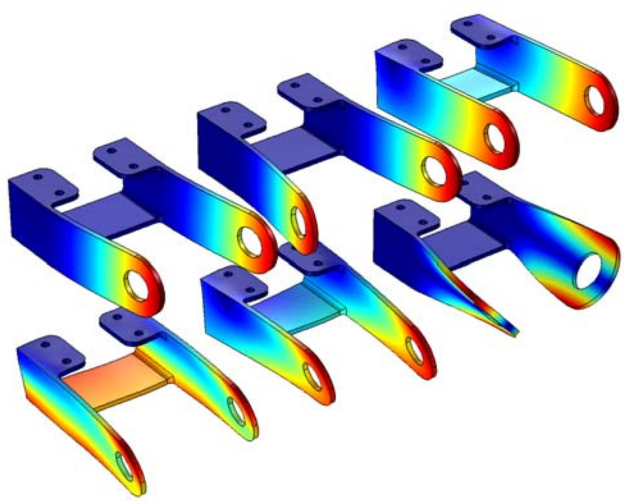


Figure 2: Six first eigenmode shapes for the unloaded case.

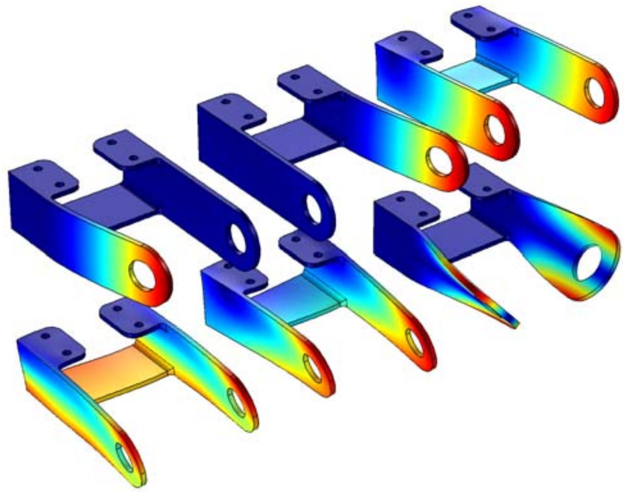


Figure 3: Six first eigenmode shapes for the prestressed case.

In Figure 4 below one can see the frequency shift in the two first eigenmode.

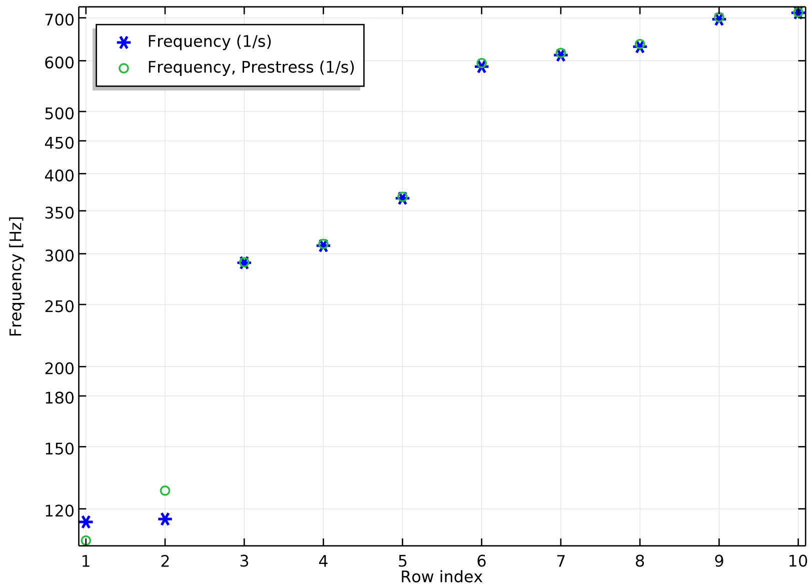


Figure 4: Six first eigenfrequencies for the unlaoaded case (star marker) and for the prestressed case (round marker).

For the unloaded case the two first eigenfrequency are about 115 Hz and correspond to the bending mode in the x-direction for both bracket arms. For the prestressed load case, the bending mode in both bracket arms are about 108 Hz for the right arm and 128 Hz for the left arm. Such a frequency shift are expected as a tensile load causes stress stiffening while a compressive load causes stress softening. The other mode shape are not significantly affected by the prestress load case.

Notes About the COMSOL Implementation

For a structural mechanics application in COMSOL Multiphysics, there are two predefined study types available for eigenfrequency analysis: **Eigenfrequency** and **Prestressed Analysis, Eigenfrequency**.

The eigenfrequency analysis compute the natural frequencies of the unloaded structure. The contribution of any load boundary condition is disregarded and the **Prescribed displacement** constraints are considered as having the value zero.

The prestressed eigenfrequency analysis, however first performs a stationary analysis to take into account the different loads and non-zero displacement constraints. The stress is then added automatically to the stiffness used in the eigenfrequency calculation.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_eigenfrequency`

Modeling Instructions

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

Browse to the application's Application Libraries folder and double-click the file `bracket_basic.mph`.

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>Eigenfrequency**.
4. Click **Add Study** in the window toolbar.
5. On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 1

Step 1: Eigenfrequency

In the Eigenfrequency study node you have the possibility to define the number of eigenfrequencies to compute, and the frequency around which you would like to search for the these frequencies. By default, the eigenvalue solver computes for the six lowest frequencies.

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 `10`.
5. On the **Home** toolbar, click **Compute**.

RESULTS

Mode Shape (solid)

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, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
freq	1/s	Frequency

4 Click **Evaluate**.

You can access the different eigenfrequency solutions in the 3D Plot Group Settings. Here you can see that, due to the symmetry, the eigenfrequency solver finds a frequency for each of the bracket arms, which is why every frequency is repeated in the solution list.

Note that the displacement values are normalized and have no physical significance. The normalization method can be changed in the **Eigenvalue Solver** node, available under the **Solver Configuration node**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, 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	30[MPa]	3E7 Pa	Peak load intensity
YC	-300[mm]	-0.3 m	Y coordinate of hole center

DEFINITIONS

Analytic 1 (an1)

1 On the **Home** toolbar, click **Functions** and choose **Local>Analytic**.

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

3 Locate the **Definition** section. In the **Expression** text field, type `F*cos(atan2(py, abs(px)))`.

4 In the **Arguments** text field, type `F, py, px`.

5 Locate the **Units** section. In the **Arguments** text field, type `Pa, m, m`.

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

SOLID MECHANICS (SOLID)

Boundary Load 1

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

Apply a boundary load to the bracket holes.

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

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

4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.

5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

`0`	t1
`0`	t2
`load(-P0,Z,Y-YC)*(Y>YC)`	n

Boundary Load 2

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

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

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

4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.

5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

`0`	t1
`0`	t2
`load(-P0,Z,Y-YC)*(Y<YC)`	n

ADD STUDY

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

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

The prestressed eigenfrequency analysis is available as a predefined study.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Prestressed Analysis, Eigenfrequency**.

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

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

STUDY 2

Step 1: Stationary

Note that the newly generated study combines one stationary analysis and one eigenfrequency analysis.

Step 2: Eigenfrequency

1 In the **Model Builder** window, under **Study 2** click **Step 2: 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 `10`.

5 In the **Search for eigenfrequencies around** text field, type `100`.

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

RESULTS

Mode Shape (solid) 1

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

In the settings for the second plot group you can see the list of the new eigenfrequencies.

Global Evaluation 1

1 In the **Model Builder** window, under **Results>Derived Values** click **Global Evaluation 1**.

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

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

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

Expression	Unit	Description
freq	1/s	Frequency, Prestress

5 Click **Evaluate**.

3D Plot Group 3

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

2 In the **Settings** window for 3D Plot Group, type `Prestress` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution Store 1 (sol3)**.

Surface 1

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

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

3 In the **Expression** text field, type `solid.mises`.

4 Right-click **Results>Prestress>Surface 1** and choose **Deformation**.

Deformation 1

On the **Prestress** 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 `Eigenfrequency` in the **Label** text field.

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

4 In the associated text field, type `Frequency [Hz]`.

5 Locate the **Grid** section. Select the **Manual spacing** check box.

6 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

Table Graph 1

1 On the **Eigenfrequency** toolbar, click **Table Graph**.

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

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

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

5 In the **Columns** list, choose **Frequency (1/s)** and **Frequency, Prestress (1/s)**.

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

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

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

9 From the **Positioning** list, choose **In data points**.

10 On the **Eigenfrequency** toolbar, click **Plot**.

11 Click the **y-Axis Log Scale** button on the **Graphics** toolbar.

Created in COMSOL Multiphysics 5.2a

Bracket—Frequency-Response Analysis

Introduction

The frequency response analysis solves for the linear steady-state response of a structure from harmonic loads. The problem is solved in the frequency domain and you can set a range of frequencies at which to compute the structural displacements.

In this example you learn how to perform a frequency response analysis of a structure under harmonic load but also how to perform a frequency response analysis of a prestressed structure.

It is recommended you review the *Introduction to the Structural Mechanics Module*, which includes background information and discusses the `bracket_basic.mph` model relevant to this example.

Model Definition

This model is an extension to the model example described in the section "The Fundamentals: A Static Linear Analysis" in the *Introduction to the Structural Mechanics Module*.

The model geometry is represented in Figure 1.

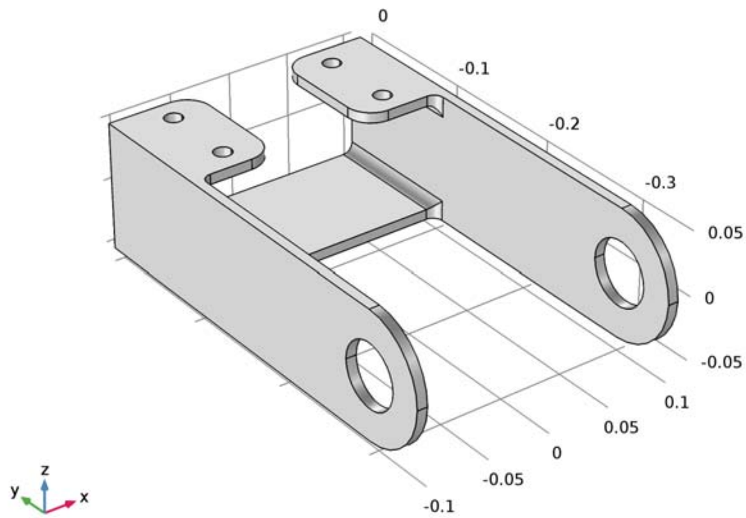


Figure 1: Bracket geometry

You study two load cases, the first one consists of a fully harmonic load case with an external load of 100 kPa applied to the bracket arms, the second load case consists of a combination of a static load and a harmonic perturbation.

An eigenfrequency analysis of this problem is performed in the tutorial Bracket—Eigenfrequency Analysis. It shows that the first resonance frequency is about 115 Hz. For the loaded case, the eigenfrequency solver shows that the first resonance frequency is about 108 Hz when the arm is under compressive load and about 128 Hz when the arm is under tensile load. In order to capture the resonance peak properly, you can refine the frequency step around these values.

Results and Discussion

Figure 2 shows the root mean square of the displacement at the tip of the left arm of the bracket for both the fully harmonic load case and the combined harmonic and static load cases.

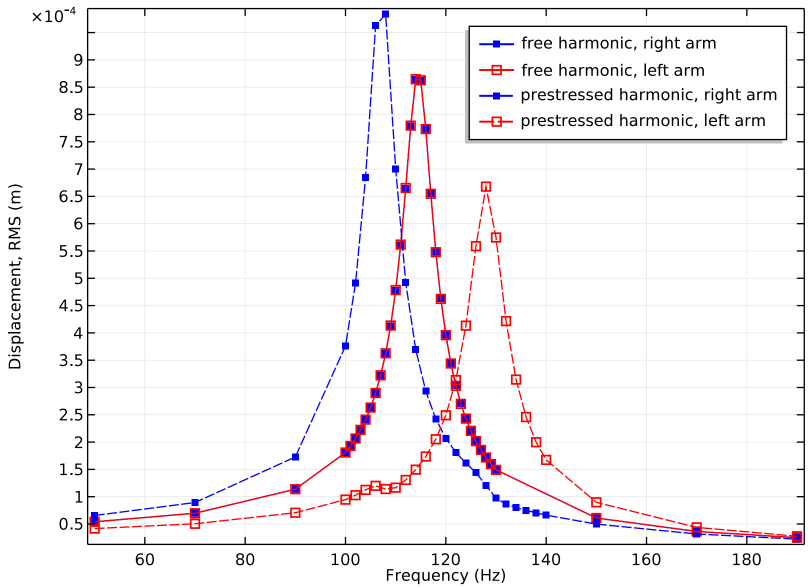


Figure 2: Root mean square of the displacement at the tip of the left (red) and right (blue) arms for both fully harmonic loaded case (solid) and a combined static and harmonic loaded case (dashed).

The curves show resonance peaks around 115 Hz for the unloaded structure in both bracket arms and the frequency shift for the loaded structure. These results are in agreement with the values predicted by solving with the eigenfrequency solver.

You can also verify that the deformation remains small even around the resonance frequency.

Figure 3 shows the phase of the x-displacement at the tip of the bracket right arm.

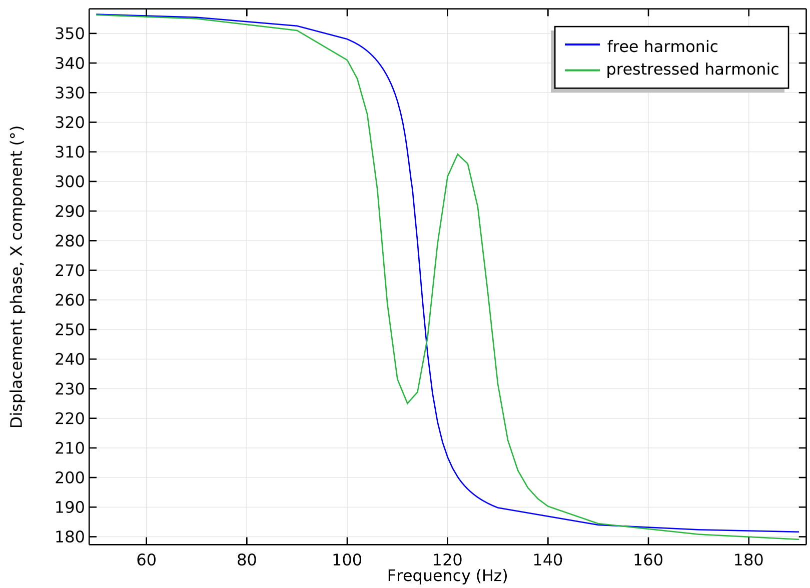


Figure 3: Phase of x-displacement at the tip of the bracket right arm.

Note the smooth transition around the resonance frequency which results from the damping using a 5% loss factor. The prestressed load case solution show a double phase change corresponding to the two resonance modes.

Notes About the COMSOL Implementation

For a structural mechanics application in COMSOL Multiphysics, there are three predefined study types available for frequency response analysis: **Frequency Domain**, **Frequency-Domain Modal** and **Prestressed Analysis, Frequency Domain**.

The frequency domain analysis computes for the structural response of an harmonic load. All loads and boundary conditions are assumed to be harmonic.

The frequency-domain modal analysis uses the modal solver to compute the frequency response. This analysis type significantly speed up the computation compare to the regular frequency domain analysis.

Use the prestressed frequency response analysis when a structure is subjected to both static and harmonic loads, and the stiffness induced by the static load case can affect the structural response to the harmonic load.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_frequency`

Modeling Instructions

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

Browse to the application's Application Library folder and double-click the file `bracket_basic.mph`.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

In the **Model Builder** window, expand the **Component 1 (comp1)>Solid Mechanics (solid)** node.

Linear Elastic Material 1

In the **Model Builder** window, expand the **Solid Mechanics (solid)** node, then click **Linear Elastic Material 1**.

Damping 1

1 On the **Physics** toolbar, click **Attributes** and choose **Damping**.

In the frequency domain you can use loss factor damping or Rayleigh damping. For this example use loss factor damping.

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

3 From the **Damping type** list, choose **Isotropic loss factor**.

MATERIALS

Structural steel (mat1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Materials** node, then click **Structural steel (mat1)**.

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

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Isotropic structural loss factor	eta_s	0.05	1	Basic

You can now apply an external harmonic load to the bracket arms.

SOLID MECHANICS (SOLID)

Boundary Load 1

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

2 Select Boundaries 4, 5, 42, and 43 only.

3 In the **Settings** window for Boundary Load, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

10[kPa]	x
0	y
0	z

In the frequency domain modal analysis, to define a harmonic load you need to mark the load as being a harmonic perturbation.

5 In the **Model Builder** window, right-click **Boundary Load 1** and choose **Harmonic Perturbation**.

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>Frequency-Domain Modal**.

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

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

STUDY 1

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 `2`.

Step 2: Frequency-Domain Modal

The frequency range will be 50 Hz-190 Hz with a refined frequency sweep step between 100 Hz and 130 Hz.

1 In the **Model Builder** window, under **Study 1** click **Step 2: Frequency-Domain Modal**.

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

3 In the **Frequencies** text field, type `50 70 90 range(100,1,130) 150 170 190`.

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

RESULTS

Stress (solid)

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

The default plot group shows the stress distribution on a deformed geometry for the final frequency. You can change the frequency for the plot evaluation in the **Parameter value** list in the settings for the plot group.

Plot the root mean square of the displacement at the tip of the left arm of the bracket.

1D Plot Group 2

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

2 In the **Settings** window for 1D Plot Group, type `Displacement, RMS` 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)`.

Point Graph 1

On the **Displacement, RMS** toolbar, click **Point Graph**.

Displacement, RMS

1 Select Point 1 only.

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

3 In the **Expression** text field, type `solid.disp_rms`.

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

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

6 From the **Positioning** list, choose **In data points**.

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

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

9 In the table, enter the following settings:

Legends
`full harmonic, right arm`

10 In the **Model Builder** window, click **Displacement, RMS**.

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

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

Point Graph 2

On the **Displacement, RMS** toolbar, click **Point Graph**.

Displacement, RMS

1 Select Point 75 only.

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

3 In the **Expression** text field, type `solid.disp_rms`.

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

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

6 From the **Positioning** list, choose **In data points**.

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

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

9 In the table, enter the following settings:

Legends
`full harmonic, left arm`

10 On the **Displacement, RMS** toolbar, click **Plot**.

Now plot the phase shift with respect to the applied load at a specified point location.

Cut Point 3D 1

On the **Results** toolbar, click **Cut Point 3D**.

Data Sets

1 In the **Settings** window for Cut Point 3D, locate the **Point Data** section.

2 In the **X** text field, type `0`.

3 In the **Y** text field, type `-50e-3`.

4 In the **Z** text field, type `-50e-3`.

5 Select the **Snap to closest boundary** check box.

1D Plot Group 3

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

2 In the **Settings** window for 1D Plot Group, type `Displacement phase, X component` in the **Label** text field.

3 Locate the **Title** section. 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 `Frequency (Hz)`.

Point Graph 1

On the **Displacement phase, X component** toolbar, click **Point Graph**.

Displacement phase, X component

1 In the **Settings** window for Point Graph, locate the **Data** section.

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

3 Locate the **y-axis data** section. Click **solid.uPhaseX - Displacement phase, X component** in the upper-right corner of the section. Locate the **y-Axis Data** section. From the **Unit** list, choose **°**.

4 Locate 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
`full harmonic`

7 On the **Displacement phase, X component** toolbar, click **Plot**.

In the solution data set feature, one can change the phase used to display the solution.

You will now consider a static load applied to the bracket and compute the prestressed frequency domain analysis.

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>Prestressed Analysis, Frequency Domain**.

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

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

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, 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	30[MPa]	3E7 Pa	Peak load intensity
YC	-300[mm]	-0.3 m	Y coordinate of hole center

DEFINITIONS

Analytic 1 (an1)

1 On the **Home** toolbar, click **Functions** and choose **Local>Analytic**.

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

3 Locate the **Definition** section. In the **Expression** text field, type `F*cos(atan2(py, abs(px)))`.

4 In the **Arguments** text field, type `F, py, px`.

5 Locate the **Units** section. In the **Arguments** text field, type `Pa, m, m`.

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

SOLID MECHANICS (SOLID)

Boundary Load 2

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

Apply a boundary load to the bracket holes.

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

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

4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.

5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

`0`	t1
`0`	t2
`load(-P0,Z,Y-YC)*(Y>YC)`	n

Boundary Load 3

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

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

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

4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.

5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

`0`	t1
`0`	t2
`load(-P0,Z,Y-YC)*(Y<YC)`	n

STUDY 2

Step 2: Frequency-Domain Perturbation

1 In the **Model Builder** window, under **Study 2** click **Step 2: Frequency-Domain Perturbation**.

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

3 In the **Frequencies** text field, type `50 70 90 range(100,2,140) 150 170 190`.

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

RESULTS

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

You have previously created a point graph plot for the unloaded case. Add a new point graph plot to the same figure but use the data set of the second load case.

Displacement, RMS

1 In the **Model Builder** window, under **Results>Displacement, RMS** select **Point Graph 1** and **Point Graph 2**, right click on any one of them and choose **Duplicate**.

2 In the **Model Builder** window, under **Results>Displacement, RMS** click **Point Graph 3**.

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

4 From the **Data set** list, choose **Study 2/Solution 3 (sol3)**.

5 Locate the **y-Axis Data** section. Clear the **Compute differential** check box.

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

Legends
`prestressed harmonic, right arm`

8 In the **Model Builder** window, under **Results>Displacement, RMS** click **Point Graph 4**.

9 In the **Settings** window for Point Graph, locate the **Data** section.

10 From the **Data set** list, choose **Study 2/Solution 3 (sol3)**.

11 Locate the **y-Axis Data** section. Clear the **Compute differential** check box.

12 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

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

Legends
`prestressed harmonic, left arm`

14 On the **Displacement, RMS** toolbar, click **Plot**.

Data Sets

1 In the **Model Builder** window, under **Results>Data Sets** right-click **Cut Point 3D 1** and choose **Duplicate**.

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

3 From the **Data set** list, choose **Study 2/Solution 3 (sol3)**.

Displacement phase, X component

1 In the **Model Builder** window, under **Results>Displacement phase, X component** right-click **Point Graph 1** and choose **Duplicate**.

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

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

4 On the **Displacement phase, X component** toolbar, click **Plot**.

5 Locate the **y-Axis Data** section. Clear the **Compute differential** check box.

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

Legends
`prestressed harmonic`

7 On the **Displacement phase, X component** toolbar, click **Plot**.

Stress (solid) 1

1 In the **Model Builder** window, under **Results** click **Stress (solid) 1**.

2 In the **Settings** window for 3D Plot Group, type `Stress (solid), prestressed` in the **Label** text field.

Created in COMSOL Multiphysics 5.2a

Bracket—Initial-Strain Analysis

Introduction

The various examples of a bracket form a suite of tutorials which summarizes the fundamentals when modeling structural mechanics problems in COMSOL Multiphysics and the Structural Mechanics Module.

In this example you learn how to introduce a prestrain to structure and investigate how it affects the assembly.

It is recommended you review the *Introduction to the Structural Mechanics Module*, which includes background information.

Model Definition

This tutorial is an extension to the example described in the section "The Fundamentals: A Static Linear Analysis" in the *Introduction to the Structural Mechanics Module*. The same model is also available as a stand alone model in the Application Libraries as *Bracket - Static Analysis*.

In the previous example, the pin was only considered as giving a load, whereas in this example the pin is actually modeled as shown in Figure 1.

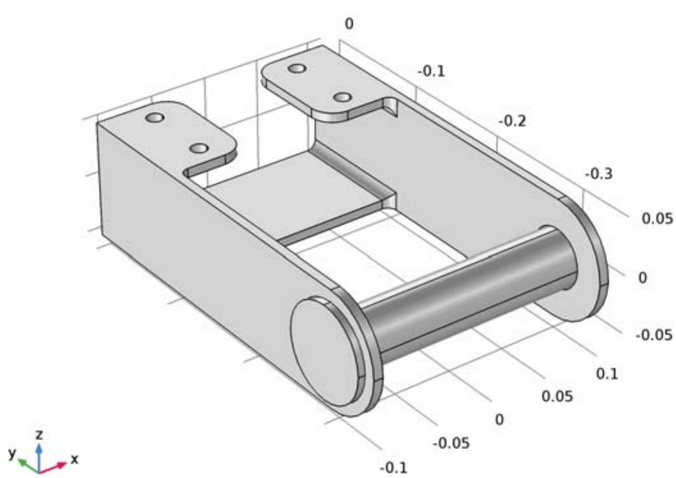


Figure 1: Bracket geometry.

An initial strain simulates that the pin is 1 mm too short in the axial direction. This could for example happen if there was a mismatch in dimensions due to manufacturing tolerances.

Results

Figure 2 shows how the pin compresses the bracket arms, and that the largest stresses are found in the region where the bracket arms are joined to the bolt supports.

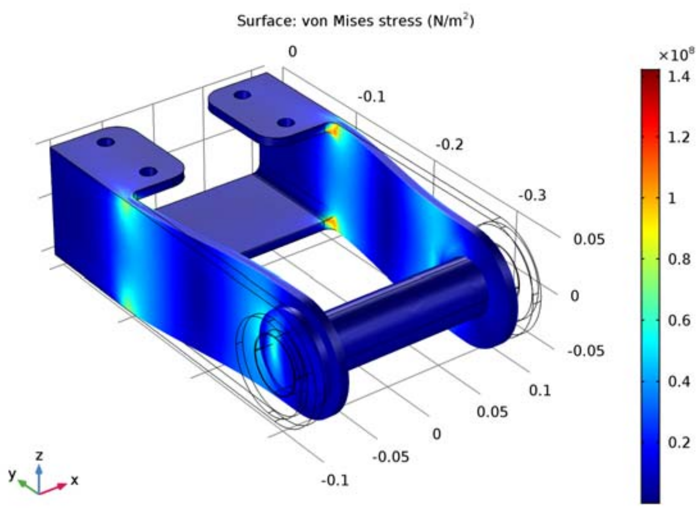


Figure 2: Von Mises stress distribution in the bracket.

Figure 3 shows the third principal strain to visualize the total strain in the structure. As the pin is stiff in comparison to the bracket, the total strain in the pin is almost the same as the initial strain given in the example.

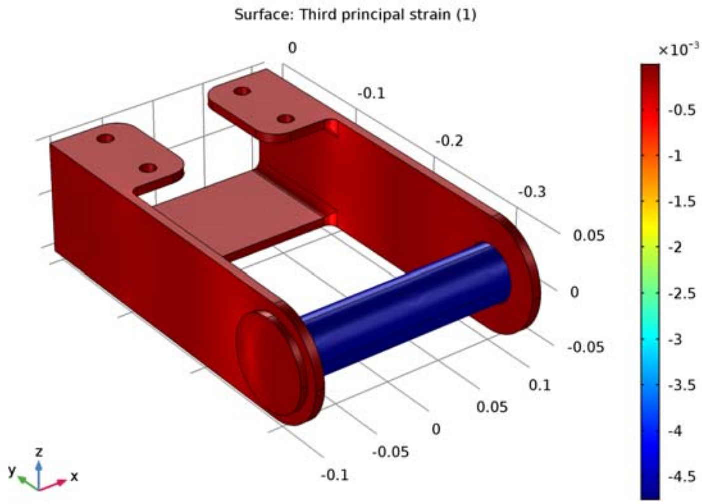


Figure 3: Strain distribution in the bracket.

Notes About the COMSOL Implementation

Initial stresses and strains can be specified in a subnode to a material model. Think of the strain or stress that you introduce as an inelastic contribution, which is not necessarily constant over the simulation. You can define a stress/strain distribution with constant values or as an expression which can, for example, be space- or time-dependent. The initial stresses and strains can also be results from another study, or even from another physics interface in the same study.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_initial_strain`

Modeling Instructions

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

Browse to the application's Application Libraries folder and double-click the file `bracket_basic.mph`.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

In the Parameters table, define a strain value that corresponds to a reduction of the pin length from 215 mm to 214 mm.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
`L0`	`215[mm]`	0.215 m	`Initial pin length`
`L`	`214[mm]`	0.214 m	`Current pin length`
`InitStrain`	`(L-L0)/L0`	-0.004651	`Pin strain`

GEOMETRY 1

Add the pin geometry to the bracket assembly by importing it into the existing geometry.

Import 2 (imp2)

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 application's Application Libraries folder and double-click the file `bracket_pin.mphbin`.

6 Click **Import**.

Form Union (fin)

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

2 In the **Settings** window for Form Union/Assembly, locate the **Form Union/Assembly** section.

3 From the **Repair tolerance** list, choose **Relative**.

4 In the **Relative repair tolerance** text field, type `1E-4`.

5 Right-click **Component 1 (comp1)>Geometry 1>Form Union (fin)** and choose **Build Selected**.

SOLID MECHANICS (SOLID)

Adding Initial Stress and Strain

Specify the initial strain under the Linear Elastic Material node.

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Solid Mechanics (solid)** node, then click **Linear Elastic Material 1**.

Initial Stress and Strain 1

1 On the **Physics** toolbar, click **Attributes** and choose **Initial Stress and Strain**.

2 In the **Settings** window for Initial Stress and Strain, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Manual**.

4 Select Domain 3 only.

The prestrain direction is the axial direction of the bolt, which coincides with the global x direction.

5 Locate the **Initial Stress and Strain** section. In the ε_0 table, enter the following settings:

InitStrain	0	0
0	0	0
0	0	0

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.

STUDY 1

Step 1: Stationary

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

RESULTS

Stress (solid)

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

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, type `Third principal strain` in the **Label** text field.

Surface 1

1 Right-click **Third principal strain** 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>Solid Mechanics>Strain>Principal strains>solid.ep3 - Third principal strain**.

3 On the **Third principal strain** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.2a

Bracket—Linear Buckling Analysis

Introduction

Buckling analysis is an important study type in structural analysis because it provides an estimate of the critical load that can cause sudden collapse of the structure. In this example you first learn how to perform a linear buckling analysis to compute for the critical buckling load, you also see in a second study how to compute the nonlinear deformation while increasing the applied load until the buckling is reached.

It is recommended you review the *Introduction to the Structural Mechanics Module*, which includes background information and discusses the `bracket_basic.mph` model relevant to this example.

Model Definition

This model is an extension to the example described in the section "The Fundamentals: A Static Linear Analysis" in the *Introduction to the Structural Mechanics Module*.

Due to nonlinearity in the numerical problem, it is recommended to refine the mesh in the area of interest. In this example, the geometry is subdivided in order to refine the mesh in the bracket right arm (see Figure 1) where the buckling load is applied.

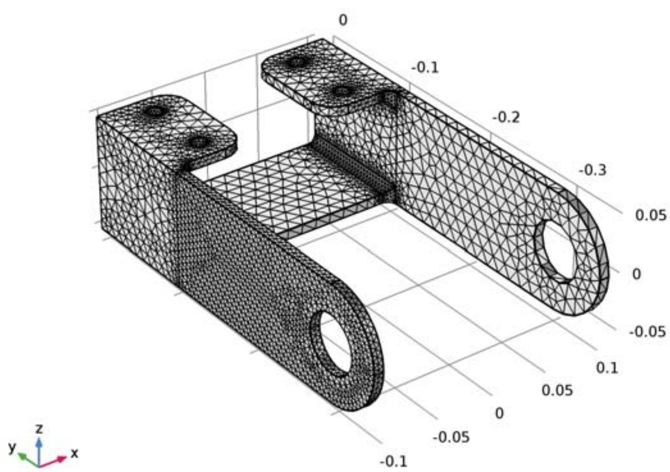


Figure 1: Refined mesh.

The analysis computes the critical compressive load with load vector resultant oriented in the positive y-direction. The loads are projected on the y-axis in order to avoid torsion modes due to the z-component of load.

Results and Discussion

Figure 2 shows the first buckling mode for the bracket geometry. The critical load factor is about 6e4, which correspond to the critical buckling load given in Newton.

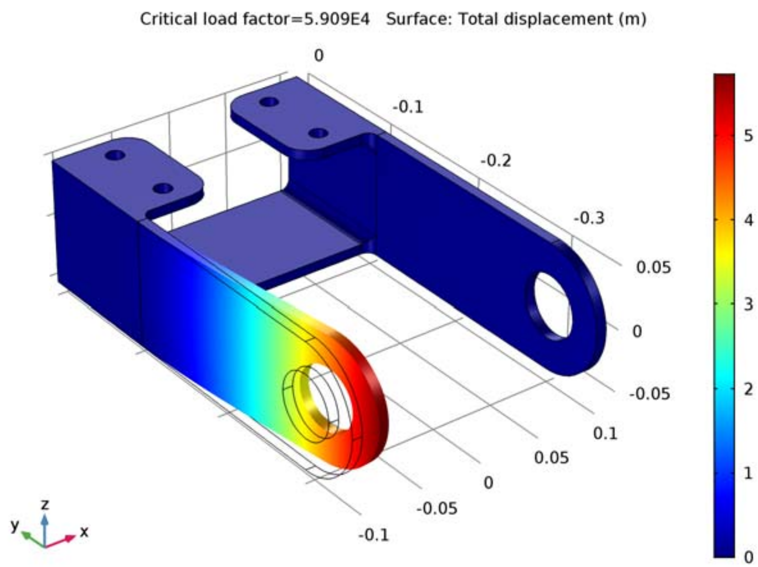


Figure 2: First buckling mode and critical load factor value.

Figure 3 shows the nonlinear displacement in the bracket right arm with respect to the increasing applied load (blue). You can see how the displacement deviates strongly from the linear response as the applied load gets closer to the critical buckling load computed with the linear buckling analysis. A deviation of 20% is obtained with an applied load force of 57 kN.

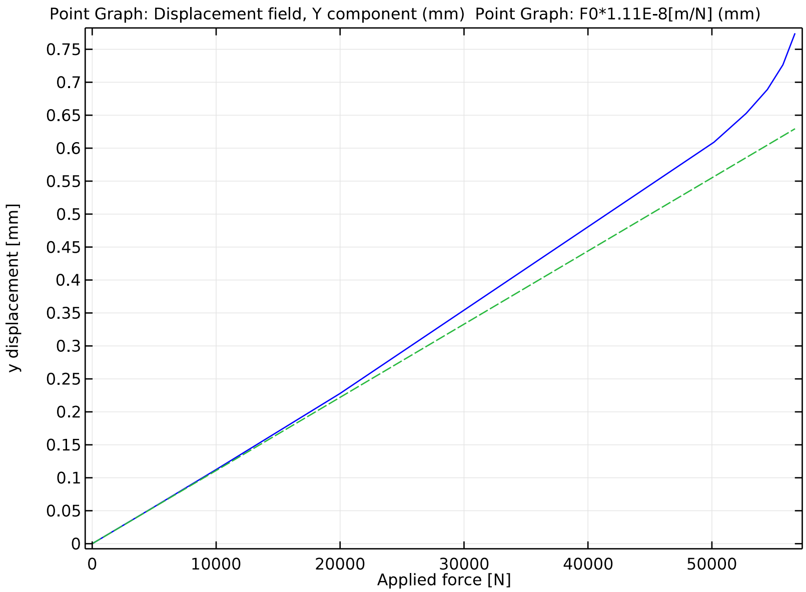


Figure 3: Bracket right arm y-displacement versus applied load.

Notes About the COMSOL Implementation

A linear buckling analysis consists of the following operations:

- Perform a static analysis using a unit load.
- Compute the eigenvalue problem including the stresses from the static load. The first eigenvalue corresponds to the value of the critical buckling load.

COMSOL Multiphysics runs automatically the sequence described above and the returns the value of the critical buckling load.

To perform a nonlinear buckling analysis, use the continuation solver to ramp up the load smoothly. You can use a stop condition to automatically stop the solver once a criteria is reached. In this example, the stop criterion is the deviation in the *y*-displacement of bracket right arm with respect to the linear displacement. The linear displacement response is predicted by using the solution computed under a unit load case for the linear buckling.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_linear_buckling`

Modeling Instructions

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

Browse to the application's Application Libraries folder and double-click the file `bracket_basic.mph`.

COMPONENT 1 (COMP1)

Prepare the geometry for a finer mesh on the arm which will buckle in this example.

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. From the **Plane** list, choose **zx-plane**.
4. From the **Offset type** list, choose **Through vertex**.
5. Find the **Offset vertex** subsection. Select the **Active** toggle button.
6. On the object **imp1**, select Point 17 only.

Partition Objects 1 (par1)

1. On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.
2. Select the object **imp1** only.
3. In the **Settings** window for Partition Objects, locate the **Partition Objects** section.
4. From the **Partition with** list, choose **Work plane**.

GLOBAL DEFINITIONS

Parameters

1. On the **Home** toolbar, 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**
F0	1[N]	1 N	Applied force
R	25[mm]	0.025 m	Hole radius
YC	-300[mm]	-0.3 m	Y coordinate of hole center
thick	8[mm]	0.008 m	Bracket arm thickness
P0	2/(thick*pi*R)*F0	3183 N/m²	Peak load intensity

DEFINITIONS

Analytic 1 (an1)

1. On the **Home** toolbar, click **Functions** and choose **Local>Analytic**.
2. In the **Settings** window for Analytic, type `load` in the **Function name** text field.
3. Locate the **Definition** section. In the **Expression** text field, type `F*cos(atan2(py, abs(px)))`.
4. In the **Arguments** text field, type `F, py, px`.
5. Locate the **Units** section. In the **Arguments** text field, type `Pa, m, m`.
6. In the **Function** text field, type `Pa`.

Boundary System 1 (sys1)

The load direction does not change with deformation.

1. In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Boundary System 1 (sys1)**.
2. In the **Settings** window for Boundary System, locate the **Settings** section.
3. From the **Frame** list, choose **Reference configuration**.

SOLID MECHANICS (SOLID)

Boundary Load 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
2. In the **Settings** window for Boundary Load, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Right hole**.
4. Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.

5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

0	t1
0	t2
`load(-P0,Z,Y-YC)*(Y>YC)`	n

MESH 1

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

Free Tetrahedral 1

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

Size 1

1. In the **Settings** window for Size, locate the **Geometric Entity Selection** section.
2. From the **Geometric entity level** list, choose **Domain**.
3. Select Domain 1 only.
4. Locate the **Element Size** section. Click the **Custom** button.
5. Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
6. In the associated text field, type `6e-3`.
7. In the **Model Builder** window, click **Mesh 1**.
8. In the **Settings** window for Mesh, click **Build All**.

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>Linear Buckling**.
4. Click **Add Study** in the window toolbar.
5. On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 1

Step 1: Stationary

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

RESULTS

Mode Shape (solid)

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

The default plot shows the mode shape of the first buckling mode.

Derived Values

Evaluate the y-displacement corresponding to the applied unit load.

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 **Study 1/Solution Store 1 (sol2)**.

4 Select Point 5 only.

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

Expression	Unit	Description
`v`	`m`	`Displacement field, Y component`

6 Click **Evaluate**.

You can now predict the linear relation between displacement and applied load. This can be used in the later analysis to predict when the nonlinear solution deviates from the linear one.

DEFINITIONS

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 **Point**.

4 Select Point 5 only.

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.

STUDY 2

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.
2 In the **Settings** window for Stationary, locate the **Study Settings** section.
3 Select the **Include geometric nonlinearity** check box.
4 Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
5 Click **Add**.
6 In the table, enter the following settings:

Parameter name	Parameter value list
F0	`1 1e3 5e3 1e4 2e4 5.924e4*log10(({range(2,1,10)}+1)/ 1.1)^(1/5)`

Solution 3 (sol3)

1 On the **Study** toolbar, click **Show Default Solver**.
2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node.
3 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 3 (sol3)>Stationary Solver 1** node.
4 Right-click **Parametric 1** and choose **Stop Condition**.
5 In the **Settings** window for Stop Condition, locate the **Stop Expressions** section.
6 Click **Add**.
7 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
`comp1.intop1(comp1.v)/ (F0*1.11E-8)/1.2`	true	√	`Stop expression 1`

8 Locate the **Output at Stop** section. From the **Add solution** list, choose **Step after stop**.
9 Clear the **Add warning** check box.
10 On the **Study** toolbar, click **Compute**.

RESULTS

Stress (solid)

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

Create Figure 3.

1D Plot Group 3

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 **Study 2/Solution 3 (sol3)**.

Point Graph 1

1. On the **1D Plot Group 3** toolbar, click **Point Graph**.
2. Select Point 5 only.
3. In the **Settings** window for Point Graph, locate the **y-Axis Data** section.
4. In the **Expression** text field, type `v`.
5. From the **Unit** list, choose **mm**.

1D Plot Group 3

1. In the **Model Builder** window, under **Results** click **1D Plot Group 3**.
2. In the **Settings** window for 1D Plot Group, type `Displacement` 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 `Applied force [N]`.
5. Select the **y-axis label** check box.
6. In the associated text field, type `y displacement [mm]`.

Point Graph 2

1. On the **Displacement** toolbar, click **Point Graph**.
2. Select Point 5 only.
3. In the **Settings** window for Point Graph, locate the **y-Axis Data** section.
4. In the **Expression** text field, type `F0*1.11E-8[m/N]`.
5. From the **Unit** list, choose **mm**.
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 **Displacement** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.2a

Bracket—Parametric Analysis

Introduction

The various examples of a bracket form a suite of tutorials which summarizes the fundamentals when modeling structural mechanics problems in COMSOL Multiphysics and the Structural Mechanics Module.

This example includes computing the solution to a case where the direction of the load is changed using a parametric sweep over a set of angles.

It is recommended you review the *Introduction to the Structural Mechanics Module*, which includes background information.

Model Definition

This model is an extension to the example described in the section “The Fundamentals: A Static Linear Analysis” in the *Introduction to the Structural Mechanics Module*. The same model is also available as a stand alone model in the Application Libraries as *Bracket - Static Analysis.*

The geometry is shown in Figure 1.

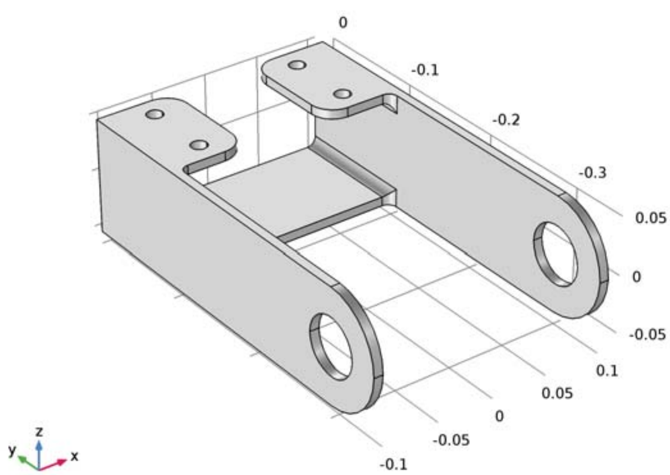


Figure 1: Bracket geometry.

In this analysis, the mounting bolts are assumed to be fixed and securely bonded to the bracket. To model the external load from the pin, you specify a surface load p with a trigonometric distribution on the inner surfaces of the two holes:

$$p = P_0 \cos(\alpha - \theta_0) \qquad -\frac{\pi}{2} < \alpha - \theta_0 < \frac{\pi}{2}$$

where P_0 is the peak load intensity. The main direction of the load is defined by θ_0, the angle from the y-axis. The load on the two holes act in opposite directions. The orientation of the load is controlled by a local coordinate system with axis directions generated using the sweep parameter `theta0`.

Results

Figure 2 shows the von Mises stress distribution corresponding to a bending load case, where the load acts in the positive z-direction in the left arm and in the negative z-direction in the right arm.

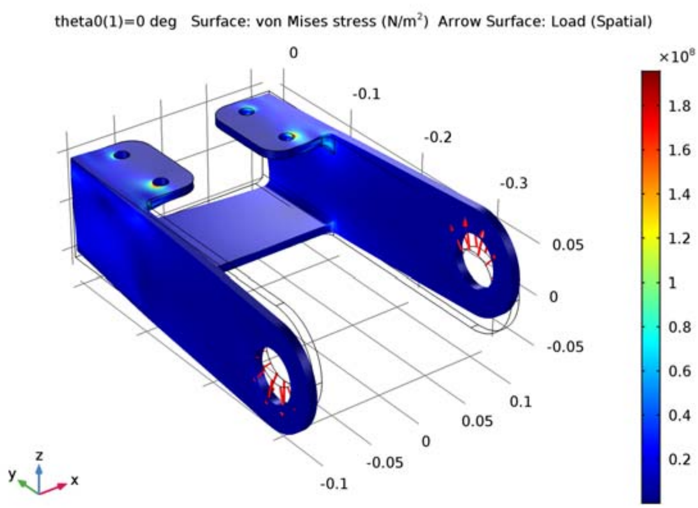


Figure 2: Von Mises stress under bending load case.

Figure 3 shows the von Mises stress distribution corresponding of a tensile load in the right arm and a compressive load in the left arm. The maximum von Mises stress value is lower in this case. A stress concentration can be seed also around the hole in the arm which is in tension.

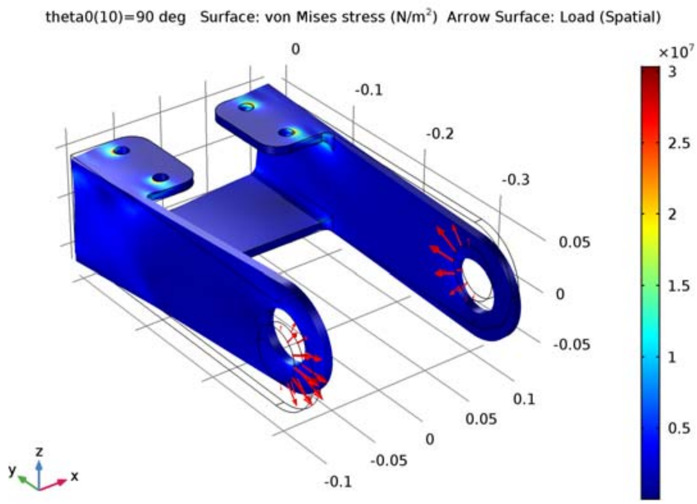


Figure 3: Von Mises stress under tensile and compressive load case.

Figure 4 shows the von Mises stress distribution corresponding of a load orientation of 130°.

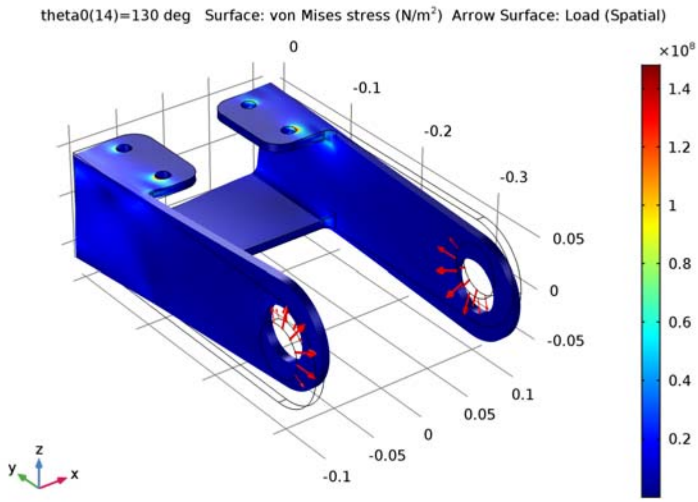


Figure 4: Von Mises stress for parameter theta0 = 130°.

Figure 5 shows that the x-component of the reaction force in all bolts for all load cases.

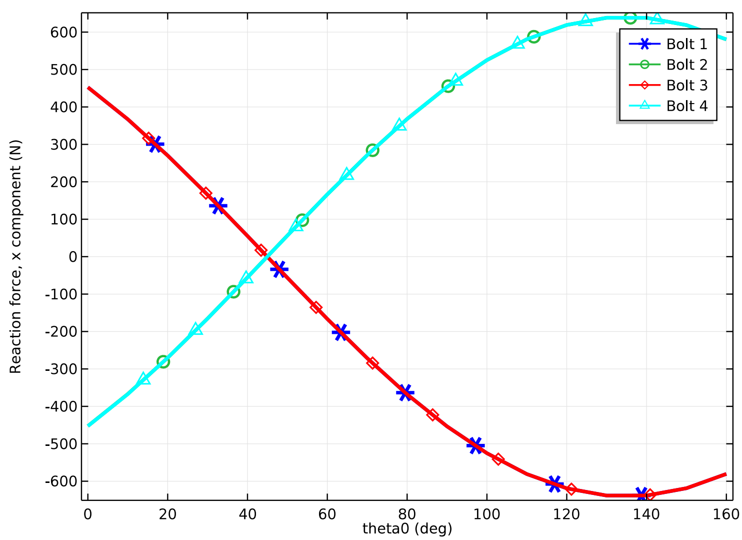


Figure 5: Reaction forces (x-component) as a function of angle.

[Figure 6](#) shows the y-component and [Figure 7](#) shows the z-component of the of the reaction force in all bolts for all load cases.

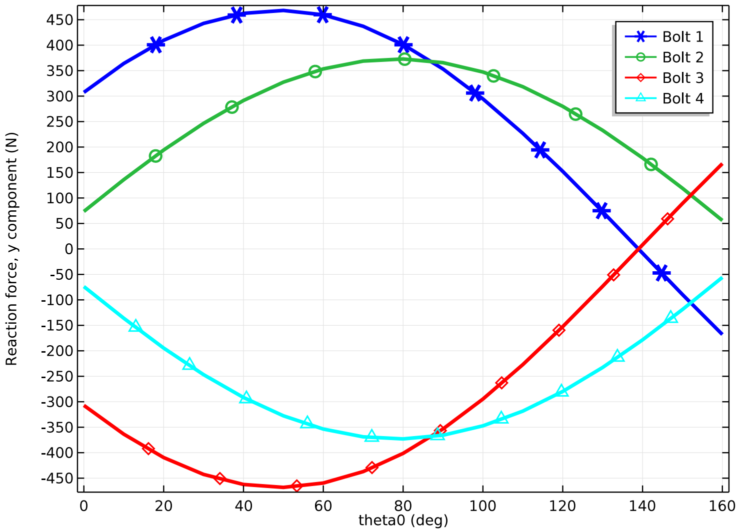


Figure 6: Reaction forces (y-component) as a function of angle.

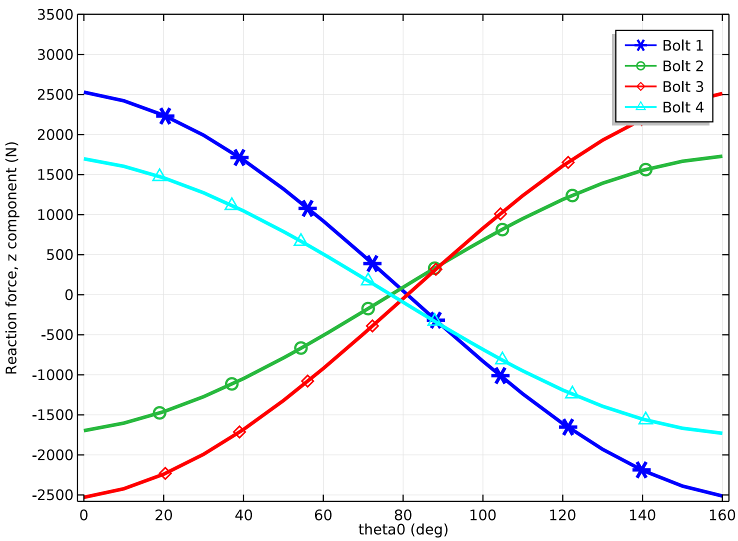


Figure 7: Reaction forces (z-component) as a function of load angle.

Notes About the COMSOL Implementation

COMSOL Multiphysics has two ways to perform parametric studies—using either a Parametric Sweep node or the Auxiliary sweep from the Stationary Solver node. In this example, either method can be used. and the continuation feature in the Solver node is used. An Auxiliary sweep is used here, but the continuation solver is not used. The continuation solver uses the solution from the previous parameter as an initial guess to calculate the current parameter value, and is the preferred option for nonlinear problems. Using the Parametric Sweep node is preferable for applications requiring, for example, geometric parametrization.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_parametric`

Parametric studies can be set up from scratch or, as in this example, added to an existing study.

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

Browse to the application's Application Libraries folder and double-click the file `bracket_static.mph`.

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

GLOBAL DEFINITIONS

In this model, the stress in the bracket is computed for different load orientations. First add a parameter to set the load direction angle.

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, enter the following settings:

Name	Expression	Value	Description
`theta0`	`0[deg]`	0 rad	`Load direction angle`

DEFINITIONS

Create a selection for the right hole carrying the load.

Explicit 5

1 On the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for Explicit, type `Right hole` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 4 only.
5 Select the **Group by continuous tangent** check box.

Create a selection for the left hole carrying the load.

Explicit 6

1 On the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for Explicit, type `Left hole` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 42 only.
5 Select the **Group by continuous tangent** check box.

You will now create a local coordinate system that will rotate with the load orientation.

Cylindrical System 2 (sys2)

1 On the **Definitions** toolbar, click **Coordinate Systems** and choose **Cylindrical System**.
2 In the **Settings** window for Cylindrical System, locate the **Settings** section.
3 Find the **Origin** subsection. In the table, enter the following settings:

x (m)	y (m)	z (m)
0	YC	0

4 Find the **Longitudinal axis** subsection. In the table, enter the following settings:

x	y	z
1	0	0

5 Find the **Direction of axis ϕ=0** subsection. In the table, enter the following settings:

x	y	z
0	sin(theta0)	cos(theta0)

Analytic 1 (load)

Change the expression of the load distribution.

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Analytic 1 (load)**.

2 In the **Settings** window for Analytic, locate the **Definition** section.

3 In the **Expression** text field, type `F*cos(p)`.

4 In the **Arguments** text field, type `F, p`.

5 Locate the **Units** section. In the **Arguments** text field, type `Pa, rad`.

SOLID MECHANICS (SOLID)

Change the boundary load to consider the parameterized direction.

Boundary Load 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Solid Mechanics (solid)** node, then click **Boundary Load 1**.

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

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

4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

load(-P0,sys2.phi)*(abs(sys2.phi)>pi/2)	r
0	phi
0	a

Boundary Load 2

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

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

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

4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

load(P0,sys2.phi)*(abs(sys2.phi)<pi/2)	r
0	phi
0	a

STUDY 1

Add an auxiliary sweep parameter, and compute the results.

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 **Auxiliary sweep** check box.
4. Click **Add**.
5. In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
theta0	range(0,10,160)	deg

6. From the **Run continuation for** list, choose **No parameter**.
7. On the **Home** toolbar, click **Compute**.

The default plot shows the solution for the last parameter value (160[deg]). You can easily change the parameter value to display the plot and then compare solutions for different load cases.

The following instructions reproduce Figure 2 to Figure 4.

RESULTS

Stress (solid)

1. In the **Model Builder** window, under **Results** click **Stress (solid)**.
2. In the **Settings** window for 3D Plot Group, locate the **Data** section.
3. From the **Parameter value (theta0 (deg))** list, choose **0**.
4. On the **Stress (solid)** toolbar, click **Plot**.
5. From the **Parameter value (theta0 (deg))** list, choose **90**.
6. On the **Stress (solid)** toolbar, click **Plot**.
7. From the **Parameter value (theta0 (deg))** list, choose **130**.
8. On the **Stress (solid)** toolbar, click **Plot**.
9. Click the **Zoom Extents** button on the **Graphics** toolbar.

Derived Values

You will now create a plot showing how the reaction forces vary with the load angle.

1. In the **Model Builder** window, under **Results** right-click **Derived Values** and choose **Clear All**.
2. On the **Results** toolbar, click **Evaluate All**.

1D Plot Group 4

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

2 In the **Settings** window for 1D Plot Group, type `Reaction force, x component` in the **Label** text field.

Table Graph 1

1 On the **Reaction force, x component** toolbar, click **Table Graph**.

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

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

4 In the **Columns** list, select **Reaction force, x component (N)**.

5 Locate the **Coloring and Style** section. In the **Width** text field, type `3`.

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

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

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

9 In the table, enter the following settings:

Legends
`Bolt 1`

Table Graph 2

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

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

3 From the **Table** list, choose **Reaction force, bolt 2**.

4 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

5 In the **Number** text field, type `7`.

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

Legends
`Bolt 2`

7 Repeat the steps above to add two more table graphs, with the following properties:

Table	Marker style	Number of markers	Legend
Reaction force, bolt 3	Diamond	9	Bolt 3
Reaction force, bolt 4	Triangle	10	Bolt 4

8 On the **Reaction force, x component** toolbar, click **Plot** to show Figure 5.

Reaction force, x component

In the **Model Builder** window, under **Results** right-click **Reaction force, x component** and choose **Duplicate**.

Reaction force, x component 1

In the **Settings** window for 1D Plot Group, type `Reaction force, y component` in the **Label** text field.

Table Graph 1

1 In the **Model Builder** window, expand the **Results>Reaction force, y component** node, then click **Table Graph 1**.
2 In the **Settings** window for Table Graph, locate the **Data** section.
3 In the **Columns** list, select **Reaction force, y component (N)**.
4 Make the same change in the next three table graphs.
5 On the **Reaction force, y component** toolbar, click **Plot** to show Figure 6.

Reaction force, y component

In the **Model Builder** window, under **Results** right-click **Reaction force, y component** and choose **Duplicate**.

Reaction force, y component 1

In the **Settings** window for 1D Plot Group, type `Reaction force, z component` in the **Label** text field.

Table Graph 1

1 In the **Model Builder** window, expand the **Results>Reaction force, z component** node, then click **Table Graph 1**.
2 In the **Settings** window for Table Graph, locate the **Data** section.
3 In the **Columns** list, select **Reaction force, z component (N)**.
4 Make the same change in the next three table graphs.
5 On the **Reaction force, z component** toolbar, click **Plot** to show Figure 7.

Created in COMSOL Multiphysics 5.2a

Bracket—Rigid Connector Analysis

Introduction

Rigid connectors provides an alternative way for modeling of geometrical parts that are of low interest in the analysis and have a negligible deformation. Such parts can be replaced with virtual rigid bodies with appropriate boundary condition. This saves computational time and memory.

In this example, you study the stress in a bracket connected to a pin where a load is applied. The pin is simulated as a rigid connector.

It is recommended you review the *Introduction to the Structural Mechanics Module*, which includes background information and discusses the `bracket_basic.mph` model relevant to this example.

Model Definition

This model is an extension to the example described in the section "The Fundamentals: A Static Linear Analysis" in the *Introduction to the Structural Mechanics Module*.

The model geometry is represented in Figure 1.

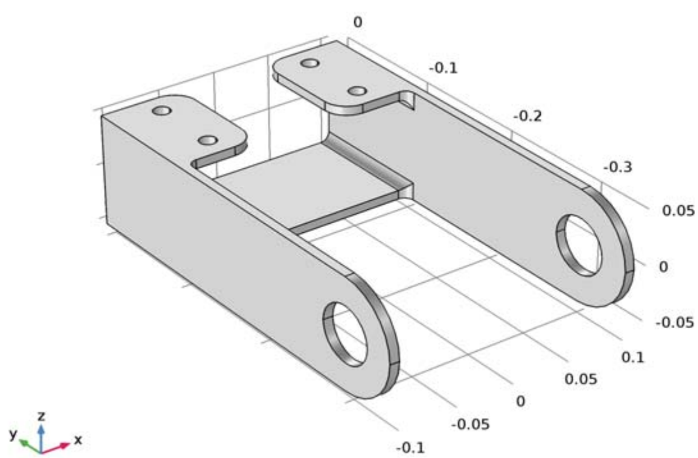


Figure 1: Bracket geometry.

A pin is connected between the bracket hole where a load is applied. The pin is assumed to be perfectly rigid and modeled with a rigid connector. Thus the pin is not represented in the model geometry. Figure 2 below shows the boundaries connected to the rigid connector and the position of its center of rotation.

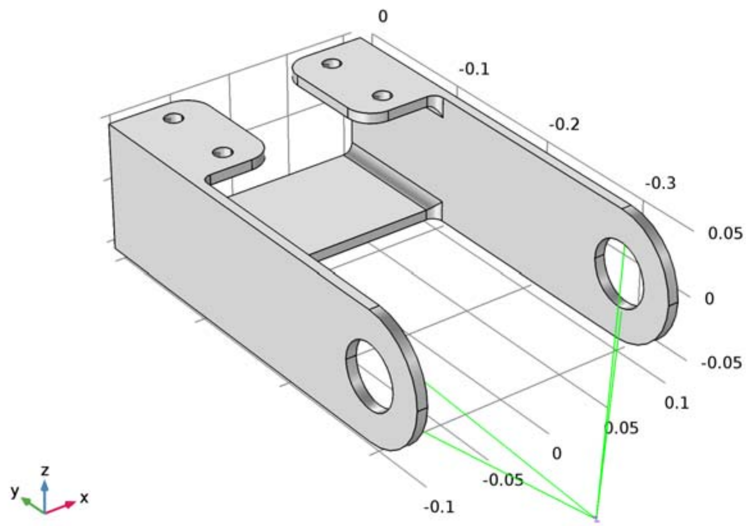


Figure 2: Center of rotation of the rigid connector applied to the bracket geometry.

Results and Discussion

Figure 3 shows the von Mises stress on a deformed geometry. You can see the effect that the pin's rotation and applied force has on the bracket arms.

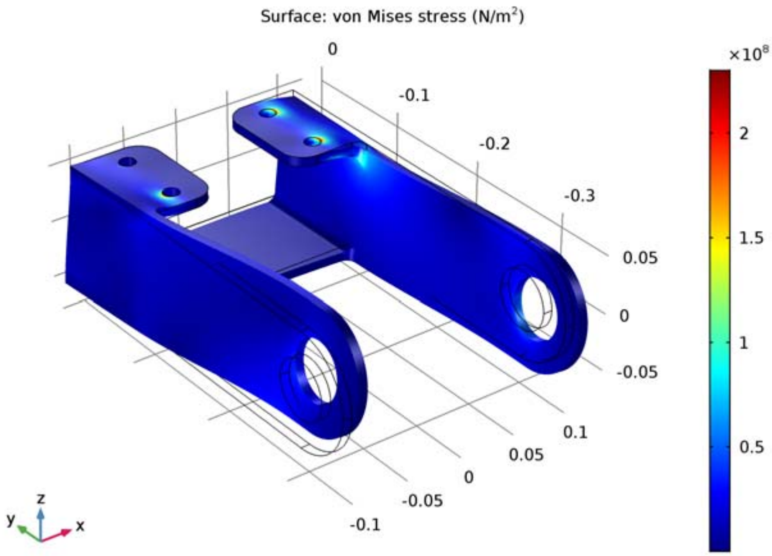


Figure 3: Von Mises stress distribution.

Notes About the COMSOL Implementation

The rigid connector has the translational and rotational degrees of freedom of a rigid body. Options for this feature include applying an external load or moment to the rigid body, or defining mass and moments of inertia in dynamic analyses. The rigid connector can also be used to apply loads and constraints at points so that these are distributed and thereby preventing singularities.

To visualize the position of the center of rotation you need to enable the physics symbols. This is done in the **File** menu, where you select **Preferences**. In the **Preferences** window, click **Graphics and Plot Windows**. In the **Visualization** section select **Show physics symbols**.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_rigid_connector`

Modeling Instructions

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

Browse to the application's Application Libraries folder and double-click the file `bracket_basic.mph`.

SOLID MECHANICS (SOLID)

Rigid Connector 1

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

Now add a rigid connector that connects the holes in the bracket arms to simulate the presence of the pin.

2 Select Boundaries 4, 5, 42, and 43 only.

By default the location of the center of rotation is computed automatically. You can also manually specify its location.

3 In the **Settings** window for Rigid Connector, locate the **Center of Rotation** section.

4 From the list, choose **User defined**.

5 Specify the $\mathbf{X}_c$ vector as

0	X
-0.40	Y
-0.10	Z

To visualize its position you need to enable the physics symbols. This is done in the **Options** menu, where you select **Preferences**. In the **Preferences** window, click **Graphics**. In the **Visualization** section select **Show physics symbols**.

The displacements of the rigid connector are constrained in the x and z directions at its center of rotation.

6 Locate the **Prescribed Displacement at Center of Rotation** section. Select the **Prescribed in x direction** check box.

7 Select the **Prescribed in z direction** check box.

8 Locate the **Prescribed Rotation** section. From the **By** list, choose **Prescribed rotation**.

Apply a prescribed rotation of the rigid body of 0.05 degrees around the y-axis.

9 Specify the Ω vector as

0	x
1	y
0	z

10 In the ϕ_0 text field, type `0.05[deg]`.

Applied Force 1

1 Right-click **Rigid Connector 1** and choose **Applied Force**.

Apply an external load of 10 kN at the center of rotation of the rigid body.

2 In the **Settings** window for Applied Force, locate the **Applied Force** section.

3 Specify the **F** vector as

0	x
-10[kN]	y
0	z

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.

STUDY 1

Step 1: Stationary

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

RESULTS

Stress (solid)

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

The default plot shows the von Mises stress on a deformed geometry.

Bracket—Shell Analysis

Introduction

The various examples of a bracket form a suite of tutorials which summarizes the fundamentals when modeling structural mechanics problems in COMSOL Multiphysics and the Structural Mechanics Module.

In this example you study the stress in a bracket subjected to external loads. The thin parts with constant thickness are modeled with the Shell interface and the transition regions where 3D effects are important are modeled using the Solid Mechanics interface. This example also shows how to connect shell elements with solid elements.

For thin geometries it can be more efficient to use shell elements instead of solid elements, thus saving computational time and memory. The Shell interface in the Structural Mechanics Module can be used to model structures approximated by thin and thick shells. There is also a similar Plate interface for 2D problems. The thickness of the shell or plate is taken into account in the equations instead of being explicitly modeled in the geometry.

It is recommended that you review the *Introduction to the Structural Mechanics Module*, which includes background information and discusses the `bracket_basic.mph` model relevant to this example.

Model Definition

The various models described in the section "The Fundamentals: A Static Linear Analysis" in the *Introduction to the Structural Mechanics Module*.

The parts of the geometry modeled with shells is shown in Figure 1 and the parts modeled with solids are shown in Figure 2. The shell surfaces are obtained from the 3D geometry using a partition operation.

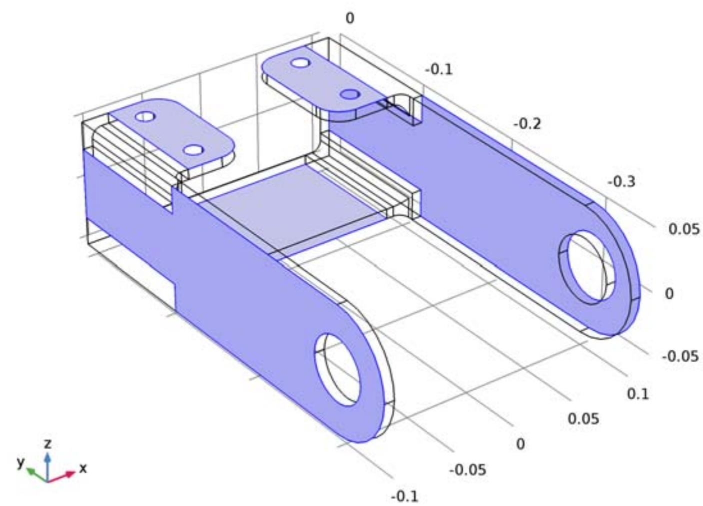


Figure 1: Shell domains in the bracket geometry.

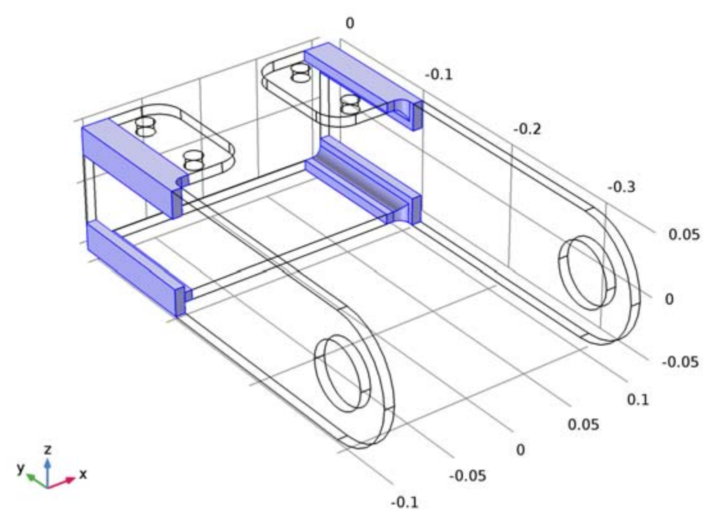


Figure 2: Solid domains in the bracket geometry.

The load is applied along the edge at the bracket holes.

Results and Discussion

The Shell interface generates a default plot which indicates the physical location of top and bottom surfaces, as well as the local coordinate system used for material input and stress output. Especially when working with offsets, this is an excellent tool for checking that the input data is correct. This plot is shown in Figure 3.

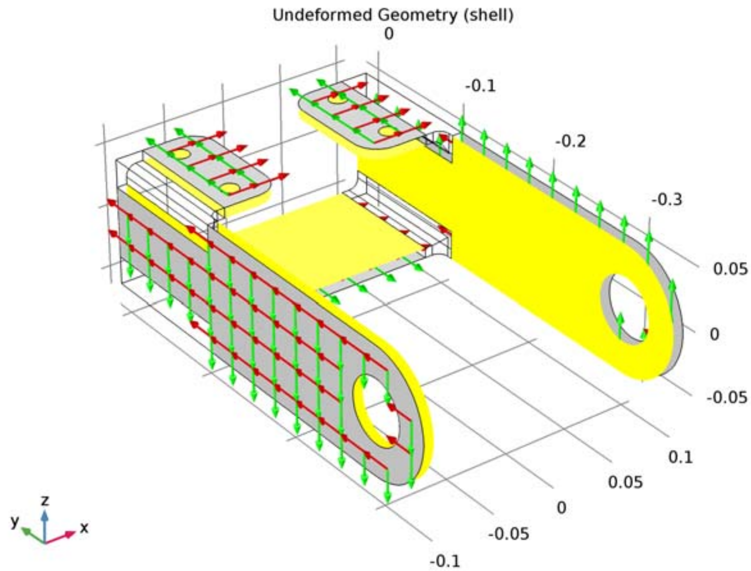


Figure 3: The shell geometry plot. Gray indicates top surface and yellow indicates bottom surface.

Figure 4 shows the first principal strain on a deformed geometry in the solid domain and at the top shell surfaces. The continuity over the transition between the shell and the solid is very good.

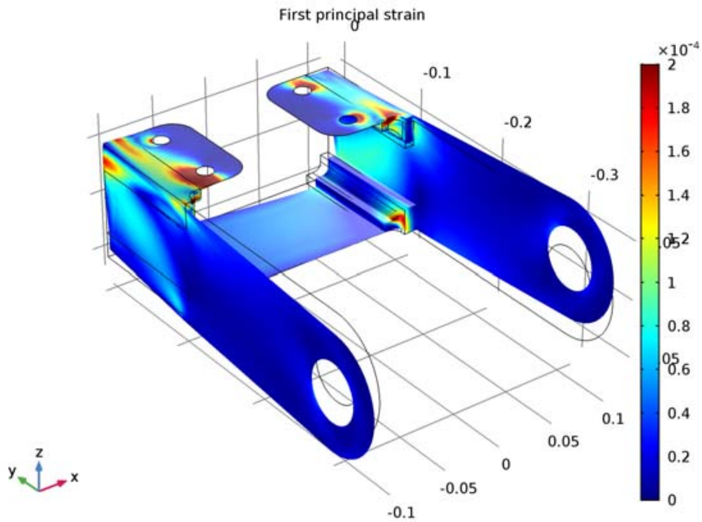


Figure 4: First principal strain distribution in the solid and at the top of the shell.

Figure 5 shows the principal stress at the bottom shell surfaces.

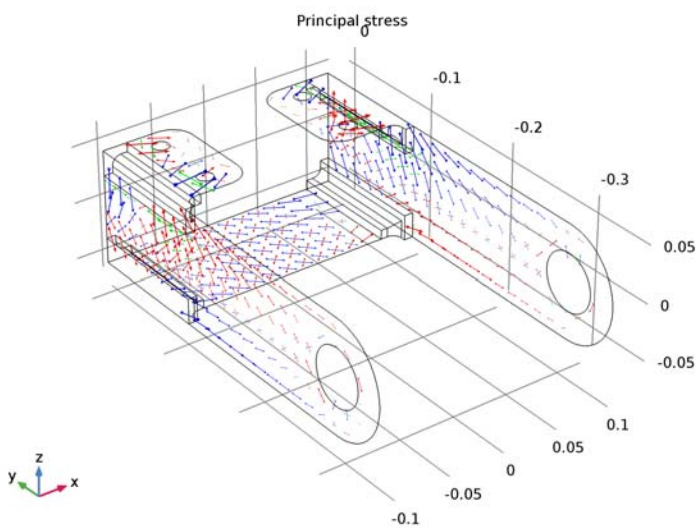


Figure 5: Principal stress at the bottom shell surface.

The table below lists the reaction force in the x, y and z directions in each bolts.

TABLE 1: REACTION FORCE IN BOLT

	REACTION FORCE, X-DIRECTION (KN)	REACTION FORCE, Y-DIRECTION (KN)	REACTION FORCE, Z-DIRECTION (KN)
Bolt 1	0.45	0.30	2.52
Bolt 2	-0.45	0.08	-1.69
Bolt 3	0.45	-0.30	-2.52
Bolt 4	-0.45	-0.08	1.69

You can compare the results obtained with the Shell interface and the initial model that used the Solid Mechanics interface (see the Results section in the tutorial *Bracket—Static Analysis*).

Notes About the COMSOL Implementation

You can specify an offset in the shell definition if the meshed surface is not the same as the midsurface of the real geometry.

The **Shell connection** and **Solid connection** nodes are used for connecting shell surfaces and edges to solid boundaries.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_shell`

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>Shell (shell)**.

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**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, 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	2.5[MPa]	2.5E6 Pa	Peak load intensity
YC	-300[mm]	-0.3 m	Y coordinate of hole center

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 application's Application Libraries folder and double-click the file `bracket.mphbin`.

6 Click **Import**.

Model the flat parts of the bracket with shell elements and the connections with solid elements.

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 From the **Plane type** list, choose **Face parallel**.

4 On the object **imp1**, select Boundary 8 only.

5 Click to expand the **Local coordinate system** section. Locate the **Local Coordinate System** section. From the **Origin** list, choose **Vertex projection**.

6 Find the **Vertex for origin** subsection. Select the **Active** toggle button.

7 On the object **imp1**, select Point 80 only.

8 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 `22[mm]`.

4 In the **Height** text field, type `22[mm]`.

Move 1 (mov1)

1 On the **Work Plane** toolbar, click **Transforms** and choose **Move**.

2 Select the object **r1** only.

3 In the **Settings** window for Move, locate the **Input** section.

4 Select the **Keep input objects** check box.

5 Locate the **Displacement** section. In the **yw** text field, type `0.1-22e-3`.

Move 2 (mov2)

1 On the **Work Plane** toolbar, click **Transforms** and choose **Move**.

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

3 In the **Settings** window for Move, locate the **Input** section.

4 Select the **Keep input objects** check box.

5 Locate the **Displacement** section. In the **xw** text field, type `0.215-0.022`.

Work Plane 1 (wp1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 1 (wp1)**.

Extrude 1 (ext1)

1 Right-click **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Extrude**.

3 In the **Settings** window for Extrude, locate the **Distances from Plane** section.

4 In the table, enter the following settings:

Distances (m)
0.112

5 Select the **Reverse direction** check box.

Partition Objects 1 (par1)

1 Right-click **Extrude 1 (ext1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

3 Select the object **imp1** only.

4 In the **Settings** window for Partition Objects, locate the **Partition Objects** section.

5 Find the **Tool objects** subsection. Select the **Active** toggle button.

6 Select the object **ext1** only.

7 Right-click **Partition Objects 1 (par1)** and choose **Build Selected**.

DEFINITIONS

Cylindrical System 2 (sys2)

1 On the **Definitions** toolbar, click **Coordinate Systems** and choose **Cylindrical System**.

2 In the **Settings** window for Cylindrical System, locate the **Settings** section.

3 Find the **Origin** subsection. In the table, enter the following settings:

x (m)	y (m)	z (m)
0	YC	0

4 Find the **Longitudinal axis** subsection. In the table, enter the following settings:

x	y	z
1	0	0

5 Find the **Direction of axis ϕ=0** subsection. In the table, enter the following settings:

x	y	z
0	0	1

Now define the load distribution to be applied on the bracket holes.

Analytic 1 (an1)

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

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

3 Locate the **Definition** section. In the **Expression** text field, type `F*abs(cos(p))`.

4 In the **Arguments** text field, type `F, p`.

5 Locate the **Units** section. In the **Arguments** text field, type `Pa, rad`.

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

Explicit 1

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

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

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

4 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

5 Select Boundaries 1, 32, 37, 50, and 86 only.

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

Explicit 2

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

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

3 Select Domains 2, 3, 7, and 8 only.

Explicit 3

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

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

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

4 Select Edges 84, 85, 89, 90, 120, 121, 125, and 126 only.

5 Select the **Group by continuous tangent** check box.

MATERIALS

Add structural steel material properties to both domains and 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>Structural steel**.

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

5 In the tree, select **Built-In>Structural steel**.

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

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

MATERIALS

Structural steel (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Structural steel (mat1)**.

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

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

4 From the **Selection** list, choose **Shell**.

Structural steel 1 (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Structural steel 1 (mat2)**.

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

3 From the **Selection** list, choose **Solid**.

SHELL (SHELL)

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

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

3 From the **Selection** list, choose **Shell**.

4 Locate the **Thickness** section. In the d text field, type `8[mm]`.

5 From the **Offset definition** list, choose **Physical offset**.

6 In the z_{offset} text field, type `-4[mm]`.

Fixed Constraint 1

1 On the **Physics** toolbar, click **Edges** and choose **Fixed Constraint**.

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

3 From the **Selection** list, choose **Bolt holes**.

Edge Load 1

1 On the **Physics** toolbar, click **Edges** and choose **Edge Load**.

2 Select Edges 4 and 188 only.

3 In the **Settings** window for Edge Load, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

5 Locate the **Force** section. From the **Load type** list, choose **Load defined as force per unit area**.

6 Specify the $\mathbf{F}_A$ vector as

`load(P0,sys2.phi)`	r
0	phi
0	a

You will now define shell edges connected to the solid boundaries.

Solid Connection 1

1. On the **Physics** toolbar, click **Edges** and choose **Solid Connection**.
2. Select Edges 12, 15, 17, 18, 66, 73, 130, 137, 191, and 193–195 only.
3. In the **Settings** window for Solid Connection, locate the **Solid Connection** section.
4. From the **Connection type** list, choose **Simplified**.

SOLID MECHANICS (SOLID)

Define the shell connection on the solid boundaries.

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.
2. In the **Settings** window for Solid Mechanics, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Solid**.

Shell Connection 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Shell Connection**.
2. Select Boundaries 9, 11, 13, 14, 30, 34, 58, 62, 80, 81, 83, and 84 only.
3. In the **Settings** window for Shell Connection, locate the **Shell Connection** section.
4. From the list, choose **Solid Connection 1 (shell)**.

MESH 1

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

Free Tetrahedral 1

1. In the **Settings** window for Free Tetrahedral, locate the **Domain Selection** section.
2. From the **Geometric entity level** list, choose **Domain**.
3. From the **Selection** list, choose **Solid**.

Free Triangular 1

1. In the **Model Builder** window, right-click **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 **Shell**.

4 Click **Build All**.

Size

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

2 From the **Predefined** list, choose **Finer**.

3 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 Right-click **Stationary Solver 1** and choose **Fully Coupled**.

4 In the **Model Builder** window, click **Solution 1 (sol1)**.

5 In the **Settings** window for Solution, click **Compute**.

RESULTS

Stress (shell)

1 In the **Model Builder** window, under **Results** click **Stress (shell)**.

2 On the **Stress (shell)** toolbar, click **Plot**.

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

The first default plot shows the maximum stress in the shell.

Undeformed Geometry (shell)

1 In the **Model Builder** window, under **Results** click **Undeformed Geometry (shell)**.

2 On the **Undeformed Geometry (shell)** toolbar, click **Plot**.

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

The second default plot shows the shell geometry, and the directions of the local coordinate axes.

3D Plot Group 4

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

2 In the **Settings** window for 3D Plot Group, type `Principal Stress Bottom` in the **Label** text field.

Principal Stress Surface 1

1 On the **Principal Stress Bottom** toolbar, click **More Plots** and choose **Principal Stress Surface**.

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

3 From the **Arrow length** list, choose **Logarithmic**.

You can also manually define where in the thickness direction of the shell you want to plot the results. Do this by changing the location in the Surface Settings window under the Parameters section. There the parameter `shell.z` defines the height of evaluation in shell: 1 for the top surface, 0 for the mid-plane, and -1 for the bottom surface.

4 Locate the **Principal Components** section. Find the **Parameters** subsection. In the table, enter the following settings:

Name	Value	Unit	Description
shell.z	-1	Height of evaluation in shell, [-1,1]	Local z-coordinate [-1,1] for thickness-dependent results

5 Locate the **Coloring and Style** section. In the **Number of arrows** text field, type `500`.

6 On the **Principal Stress Bottom** toolbar, click **Plot**.

To plot shell and solid results in the same figure as in Figure 3, follow the steps below.

3D Plot Group 5

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

2 In the **Settings** window for 3D Plot Group, type `Principal Strain` in the **Label** text field.

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

4 In the **Title** text area, type `First principal strain`.

Surface 1

1 Right-click **Principal Strain** and choose **Surface**.

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

3 In the **Expression** text field, type `shell.ep1`.

4 Click to expand the **Range** section. Select the **Manual color range** check box.

5 In the **Maximum** text field, type `2.0E-4`.

6 On the **Principal Strain** toolbar, click **Plot**.

7 Right-click **Results>Principal Strain>Surface 1** and choose **Deformation**.

Surface 1

1 In the **Model Builder** window, expand the **Results>Stress (solid)** node.

2 Right-click **Surface 1** and choose **Copy**.

Principal Strain

In the **Model Builder** window, under **Results** right-click **Principal Strain** and choose **Paste Surface**.

Surface 2

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

2 In the **Expression** text field, type `solid.ep1`.

3 Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

4 On the **Principal Strain** toolbar, click **Plot**.

Derived Values

The last step is to evaluate the reaction forces at the edges which represent the mounting bolts.

Line Integration 1

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

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

3 Select Edges 84 and 85 only.

4 In the **Settings** window for Line Integration, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Shell> Reactions>Reaction force (Spatial)>shell.RFx - Reaction force, x component**.

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

Expression	Unit	Description
shell.RFx	kN	Reaction force, x component

6 Click **Evaluate**.

7 In the **Settings** window for Line Integration, locate the **Expressions** section.

8 In the table, enter the following settings:

Expression	Unit	Description
shell.RFy	kN	Reaction force, y component

9 Click **Evaluate**.

10 In the **Settings** window for Line Integration, locate the **Expressions** section.

11 In the table, enter the following settings:

Expression	Unit	Description
`shell.RFz`	`kN`	`Reaction force, z component`

12 Click **Evaluate**.

Table 1

1 In the **Model Builder** window, expand the **Results>Tables** node, then click **Table 1**.

2 In the **Settings** window for Table, type `Reaction force bolt 1` in the **Label** text field.

Reaction Forces for the Remaining Bolts.

1 Compute the bolt forces for the remaining three bolts. Repeat the above instructions for **Line integration** and **Table** by adjusting for the data from the table below.

2 In the table, enter the following settings:

Line integration	Select edges	Default table name	New table label
Line integration 2	89, 90	Table 2	Reaction force bolt 2
Line integration 3	120, 121	Table 2	Reaction force bolt 3
Line integration 4	125, 126	Table 2	Reaction force bolt 4

Created in COMSOL Multiphysics 5.2a

Bracket—Spring Foundation Analysis

Introduction

A fixed, fully constrained, boundary condition contains the assumption that the analyzed structure is attached to an infinitely stiff support. While in many cases this is a good approximation, sometimes you may need to consider the stiffness of the supporting structure in your model. In COMSOL Multiphysics you can do this by using the Spring Foundation boundary condition.

In this example, you study the stress in a bracket subjected to external loads. The stiffness of the mounted bolts connecting support is modeled with spring foundations.

It is recommended you review the *Introduction to the Structural Mechanics Module*, which includes background information and discusses the `bracket_basic.mph` model relevant to this example.

Model Definition

This model is an extension to the model example described in the section "The Fundamentals: A Static Linear Analysis" in the *Introduction to the Structural Mechanics Module*.

The model geometry is represented in Figure 1.

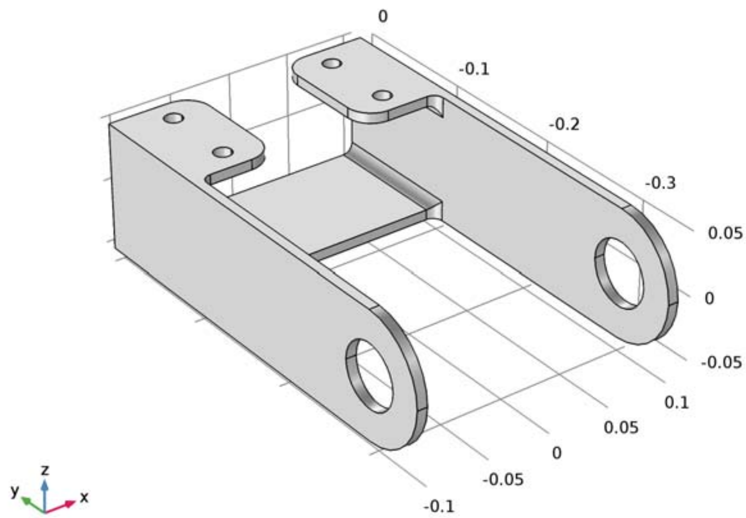


Figure 1: Bracket geometry.

The load is applied in the positive z-direction in the bracket left arm and in the negative z-direction in the bracket right arm.

The bolts are assumed to be elastic, defined with spring foundations.

Results and Discussion

Figure 2 shows the von Mises stress on a deformed geometry.

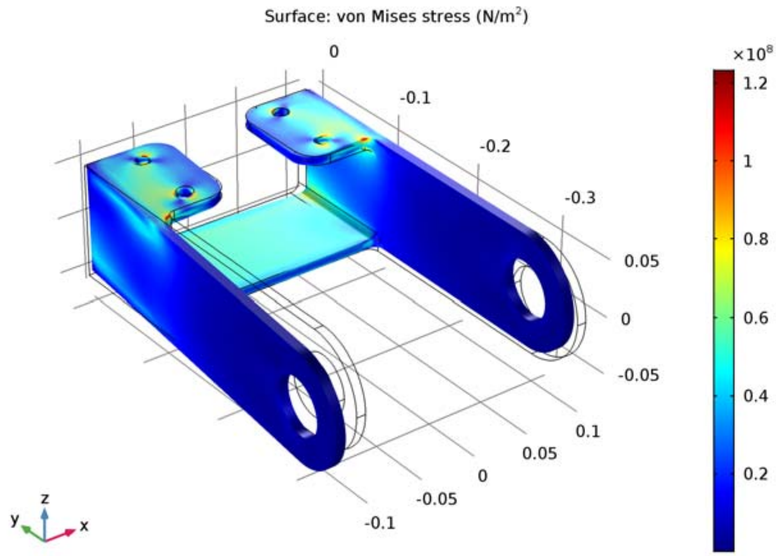


Figure 2: The Von Mises stress distribution.

The maximum stress in the bracket when connected using spring connection is about 124 MPa. This is slightly lower as compared to the case using rigidly mounted bolts (see the Results section in the tutorial *Bracket - Static analysis*) that give a maximum stress about 175 MPa. The difference in the stress is caused by the structural flexibility that a spring connection provides.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_spring`

Modeling Instructions

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

Browse to the application's Application Libraries folder and double-click the file `bracket_basic.mph`.

COMPONENT 1 (COMP1)

In the **Model Builder** window, expand the **Component 1 (comp1)** node.

GLOBAL DEFINITIONS

Add the two new parameters for the spring coefficients of the external structure to the table.

Parameters

1. On the **Home** toolbar, 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	2.5[MPa]	2.5E6 Pa	Peak load intensity
YC	-300[mm]	-0.3 m	Y coordinate of hole center
kxy	1e6[N/m]	1E6 N/m	Spring coefficient in x and y direction
kz	1e8[N/m]	1E8 N/m	Spring coefficient in z direction

DEFINITIONS

Analytic 1 (an1)

1. On the **Home** toolbar, click **Functions** and choose **Local>Analytic**.
2. In the **Settings** window for Analytic, type `load` in the **Function name** text field.
3. Locate the **Definition** section. In the **Expression** text field, type `F*cos(atan2(py, abs(px)))`.
4. In the **Arguments** text field, type `F, py, px`.
5. Locate the **Units** section. In the **Arguments** text field, type `Pa, m, m`.
6. In the **Function** text field, type `Pa`.

SOLID MECHANICS (SOLID)

Boundary Load 1

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

   Apply a boundary load to the bracket holes.
2. Select Boundaries 4 and 43 only.
3. In the **Settings** window for Boundary Load, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.

5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

0	t1
0	t2
load(-P0,Y-YC,Z)	n

Fixed Constraint 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** right-click **Fixed Constraint 1** and choose **Disable**.

Spring Foundation 1

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

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

3 From the **Selection** list, choose **Bolt holes**.

4 Locate the **Spring** section. From the **Spring type** list, choose **Total spring constant**.

5 From the list, choose **Diagonal**.

6 In the $\mathbf{k}_{tot}$ table, enter the following settings:

kxy	0	0
0	kxy	0
0	0	kz

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.

STUDY 1

Step 1: Stationary

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

RESULTS

Stress (solid)

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

   The default plot shows the von Mises effective stress distribution.

Created in COMSOL Multiphysics 5.2a

Bracket—Static Analysis

Introduction

The various examples of a bracket form a suite of tutorials which summarizes the fundamentals when modeling structural mechanics problems in COMSOL Multiphysics and the Structural Mechanics Module.

This is the most fundamental model in the suite, and includes the definition of material properties and boundary conditions. After the solution is computed, you learn how to analyze results and check the reaction forces.

Model Definition

The model used in this guide is a bracket made of steel. This type of bracket can be used to install an actuator that is mounted on a pin placed between the two holes in the bracket arms. The geometry is shown in Figure 1.

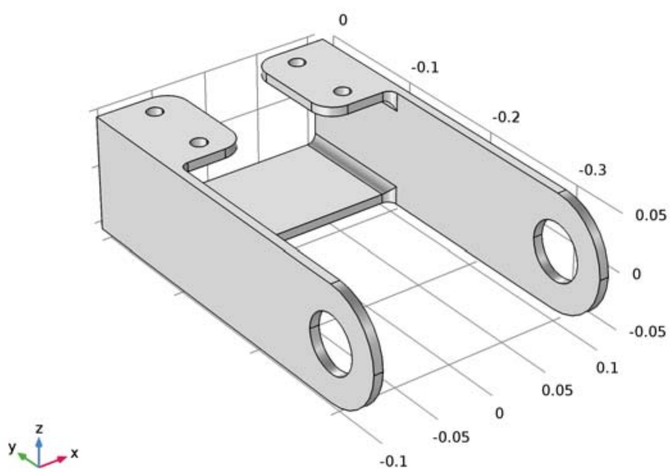


Figure 1: Bracket geometry.

In this analysis, the mounting bolts are assumed to be fixed and securely bonded to the bracket. One of the arms is loaded upwards, and the other downwards. The loads are applied on the normal of the inner surfaces of the holes, and their intensity is $P_0 \cos(\alpha)$, where α is the angle from the direction of the loads. Figure 2 below shows the loads applied to the bracket.

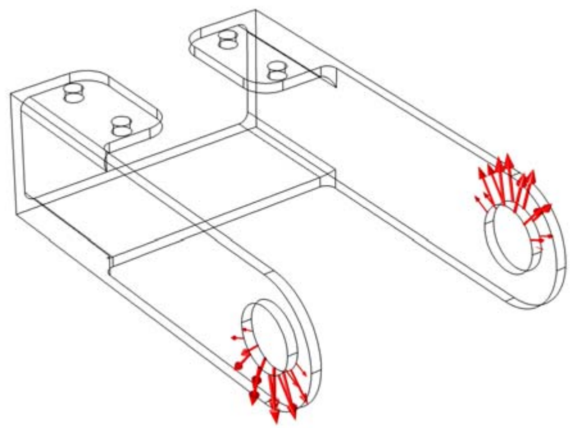


Figure 2: Load distribution in the bracket arms.

Results

Figure 3 shows the von Mises stress distribution together with an exaggerated (automatically scaled) picture of the deformation. The high stress values are located in the vicinity of the mounting bolts and at the transition between the plates. The maximum von Mises stress remains below the yield stress value for structural steel, which validates the choice of a linear elastic material.

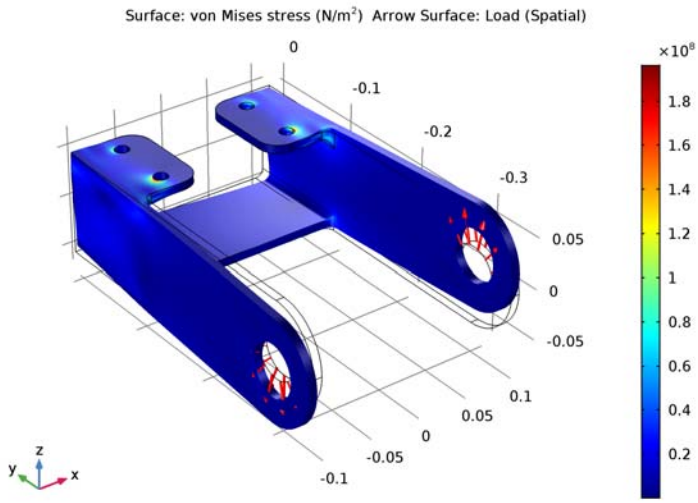


Figure 3: Von Mises stress distribution in the bracket under bending load in z-axis direction.

In Figure 4 you can see that the bracket base remains fixed while only the arms are deformed. The maximum total displacement is about 0.2 mm, which is in agreement with the assumption of small deformations.

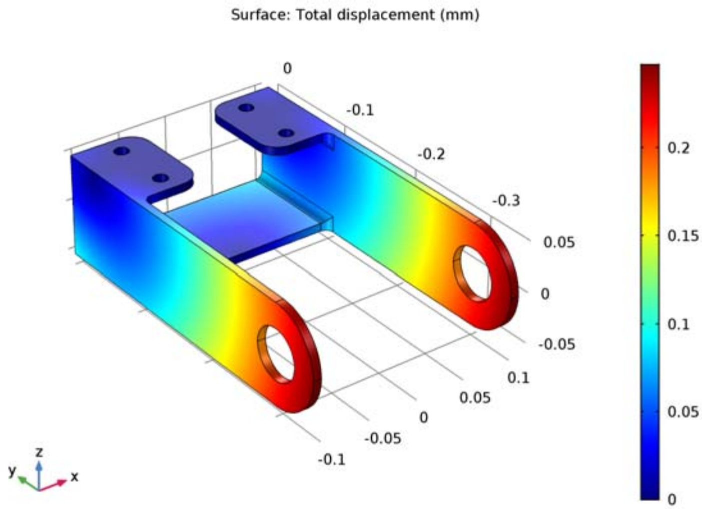


Figure 4: Bracket total displacement.

Figure 5 shows the principal stresses in the bracket. The largest principal stress is shown with red arrows, the intermediate principal stress with green arrows, and the smallest principal stress with blue arrows.

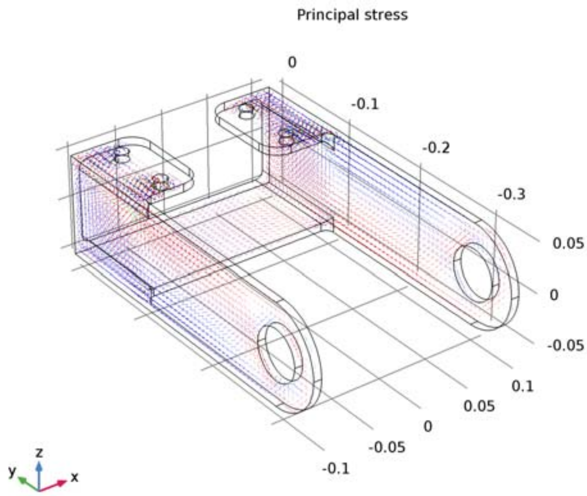


Figure 5: Principal stress in the bracket left arm.

In Table 1 you can see the reaction force in the x, y, and z directions in each bolt. In all directions they sum is zero, which is a good check, since in this model there are no resultant forces. The slight asymmetry can be attributed to that the mesh is not perfectly symmetric.

TABLE 1: REACTION FORCE IN BOLT

	REACTION FORCE, X-DIRECTION (N)	REACTION FORCE, Y-DIRECTION (N)	REACTION FORCE, Z-DIRECTION (N)
Bolt 1	451	300	2555
Bolt 2	-451	78	-1723
Bolt 3	453	-304	-2554
Bolt 4	-454	-75	1721

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_static`

Modeling Instructions

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

NEW

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

The first step to build a model is to open COMSOL and then specify the type of analysis you want to do - in this case, a stationary, solid mechanics analysis.

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

It is good modeling practice to gather the constants and parameters in one place so that you can change them easily. Using parameters will also improve the readability of your input data.

Parameters

1. On the **Home** toolbar, 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	2.5[MPa]	2.5E6 Pa	Peak load intensity
YC	-300[mm]	-0.3 m	Y coordinate of hole center

GEOMETRY 1

The next step is to create your geometry, which also can be imported from an external program. COMSOL Multiphysics supports a multitude of CAD programs and file formats. In this example, import a file in the COMSOL Multiphysics geometry file format (`.mphbin`).

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 application's Application Libraries folder and double-click the file `bracket.mphbin`.

6 Click **Import**.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

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

DEFINITIONS

Here you want to define an expression for the load applied to the load-carrying holes. Assume the load distribution to be defined with a trigonometric function.

Analytic 1 (an1)

1 On the **Home** toolbar, click **Functions** and choose **Local>Analytic**.

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

3 Locate the **Definition** section. In the **Expression** text field, type `F*cos(atan2(py, abs(px)))`.

4 In the **Arguments** text field, type `F, py, px`.

5 Locate the **Units** section. In the **Arguments** text field, type `Pa, m, m`.

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

Explicit 1

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

2 In the **Settings** window for Explicit, type `Bolt 1` in the **Label** text field.

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

4 Select Boundary 18 only.

5 Select the **Group by continuous tangent** check box.

6 Repeat the steps above to add three more explicit selections, with the following properties:

Default node label	New node label	Select this boundary
Explicit 2	Bolt 2	20
Explicit 3	Bolt 3	31
Explicit 4	Bolt 4	33

Union 1

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

2 In the **Settings** window for Union, type `Bolt holes` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. 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 **Bolt 1**, **Bolt 2**, **Bolt 3**, and **Bolt 4**.

6 Click **OK**.

MATERIALS

COMSOL Multiphysics is equipped with built-in material properties for a number of common materials. Here, choose structural steel. The material is automatically assigned to 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>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.

SOLID MECHANICS (SOLID)

By default, the Solid Mechanics interface assumes that the participating material models are linear elastic, which is appropriate for this example. All that is left to do is to set the constraints and loads.

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 **Bolt holes**.

Apply a boundary load to the bracket holes. The predefined boundary system is used for orienting the load in the normal direction.

Boundary Load 1

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

2 Select Boundaries 4 and 43 only.

3 In the **Settings** window for Boundary Load, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.

5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

0	t1
0	t2
load(-P0,Y-YC,Z)	n

MESH 1

Set the mesh size to be slightly finer than the default size.

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**.

The steps below show how to visualize the load distribution in the current geometry before computing the solution.

STUDY 1

Stress (solid)

On the **Study** toolbar, click **Get Initial Value**.

Note that the Study node automatically defines a solver sequence for the simulation based on the selected physics (Solid Mechanics) and study type (Stationary).

Since a mesh is required, and it has not been created yet, the Study node automatically generates this at the same time as the solver sequence. In practice, do not rely only on the standard mesh settings. For most real problems, suitable meshing parameters should be set up from the Mesh toolbar.

RESULTS

Stress (solid)

In the **Model Builder** window, under **Results** right-click **Stress (solid)** and choose **Arrow Surface**.

Arrow Surface 1

1 In the **Settings** window for Arrow Surface, locate the **Expression** section.
2 In the **X component** text field, type `solid.FperAreax`.
3 In the **Y component** text field, type `solid.FperAreay`.
4 In the **Z component** text field, type `solid.FperAreaz`.
5 Locate the **Coloring and Style** section. From the **Placement** list, choose **Mesh nodes**.
6 On the **Stress (solid)** toolbar, click **Plot**.

Now, solve the model.

STUDY 1

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

The default plot shows the von Mises stress distribution, together with an exaggerated (automatically scaled) picture of the deformation.

Add a plot group to display the displacement of the bracket.

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, type `Total displacement` in the **Label** text field.

Surface 1

1 Right-click **Total displacement** and choose **Surface**.
2 In the **Settings** window for Surface, locate the **Expression** section.
3 From the **Unit** list, choose **mm**.
4 On the **Total displacement** toolbar, click **Plot**.

Create another plot to display the principal stresses.

3D Plot Group 3

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

2 In the **Settings** window for 3D Plot Group, type `Principal stress` in the **Label** text field.

Principal Stress Volume 1

1 On the **Principal stress** toolbar, click **More Plots** and choose **Principal Stress Volume**.

2 In the **Settings** window for Principal Stress Volume, locate the **Positioning** section.

3 Find the **X grid points** subsection. In the **Points** text field, type `30`.

4 Find the **Y grid points** subsection. In the **Points** text field, type `60`.

5 Find the **Z grid points** subsection. In the **Points** text field, type `15`.

6 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.

7 On the **Principal stress** toolbar, click **Plot**.

A final check is to compute the total reaction force along the x, y, and z directions. Use a surface integration over the constrained boundaries.

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 **Bolt 1**.

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

Expression	Unit	Description
`solid.RFx`	`N`	`Reaction force, x component`
`solid.RFy`	`N`	`Reaction force, y component`
`solid.RFz`	`N`	`Reaction force, z component`

5 Click **Evaluate**.

Table 1

1 In the **Model Builder** window, expand the **Results>Tables** node, then click **Table 1**.

2 In the **Settings** window for Table, type `Reaction force, bolt 1` in the **Label** text field.

3 Repeat the previous steps three times to evaluate the reaction forces in all four bolts, using the information in the table below.

Node label	Choose selection	New table node name
Surface integration 2	Bolt 2	Reaction force, bolt 2

Node label	Choose selection	New table node name
Surface integration 3	Bolt 3	Reaction force, bolt 3
Surface integration 4	Bolt 4	Reaction force, bolt 4

Bracket—Thermal-Stress Analysis

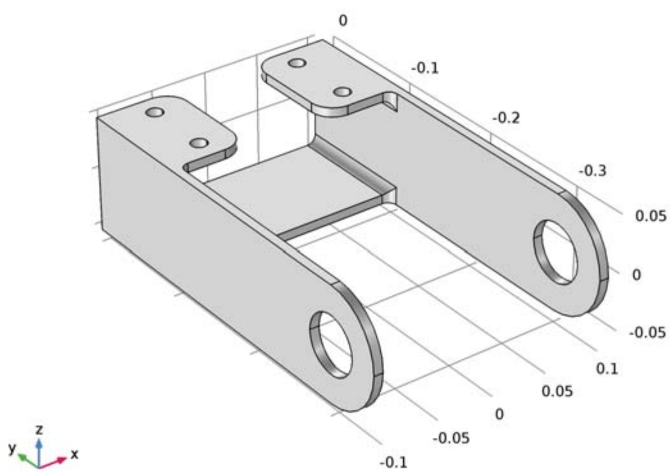
Introduction

The various examples of a bracket form a suite of tutorials which summarizes the fundamentals when modeling structural mechanics problems in COMSOL Multiphysics and the Structural Mechanics Module.

In this example you learn how to perform a thermal stress analysis.

Model Definition

The model used in this guide is an assembly of a bracket and its mounting bolts, which are all made of steel. This type of bracket can be used to install an actuator that is mounted on a pin placed between the two holes in the bracket arms. The geometry is shown in Figure 1.



In this example, a temperature distribution is computed in the bracket and the resulting thermal stresses are determined.

Results

Figure 1shows the temperature distribution in the bracket. The temperature is highest where the inward heat flux is prescribed, and decreases as heat is removed by convection from all other boundaries.

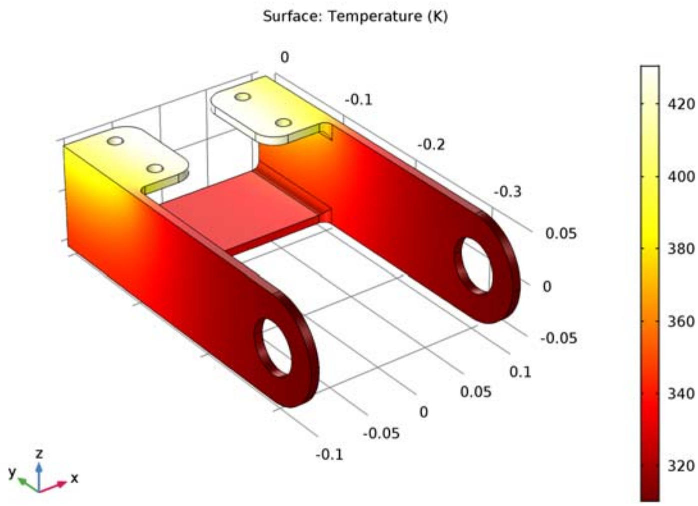


Figure 1: Temperature distribution in the bracket.

Figure 2 shows the von Mises stress distribution in the bracket. You can see how the bracket is deformed through thermal expansion. Due to the boundary conditions and the non-uniform temperature distribution, thermal stresses develop in the structure.

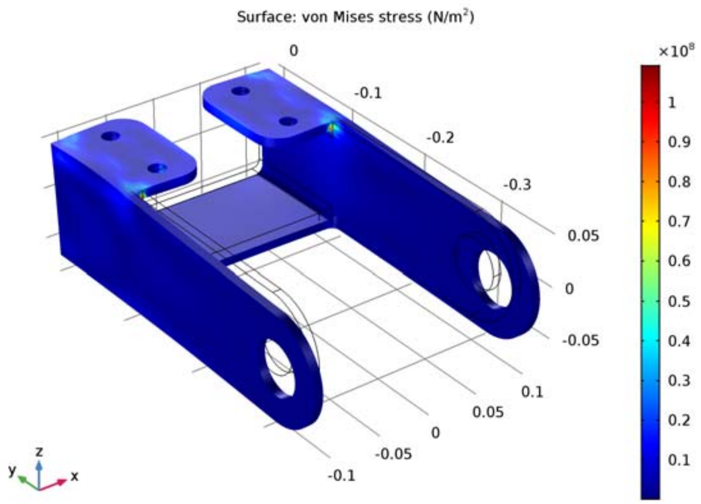


Figure 2: Von Mises stress distribution in the bracket.

Notes About the COMSOL Implementation

COMSOL Multiphysics contains physics interfaces for structural analysis as well as thermal analysis. You can set up the coupled analysis for thermal-structure interaction using three different methods:

1. Add a **Thermal Stress** interface as in this example. The coupling is predefined and appears in the **Thermal Expansion** and **Temperature Coupling** nodes under **Multiphysics**. This is the easiest approach.
2. Add separate **Solid Mechanics** and **Thermal Stress** interfaces. Then add **Thermal Expansion** and **Temperature Coupling** nodes under **Multiphysics**, and check the settings in them.
3. Add separate **Solid Mechanics** and **Thermal Stress** interfaces. Add a **Thermal Expansion** subnode under **Linear Elastic Material**, and do the appropriate settings there.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_thermal`

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>Thermal Stress**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.
6 Click **Done**.

GEOMETRY 1

The Thermal Stress interface is a multiphysics interface that combines a Solid Mechanics interface with a Heat Transfer in Solids interface. You can see the coupling between the physics interfaces under the **Multiphysics** node.

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 application's Application Libraries folder and double-click the file `bracket.mphbin`.
6 Click **Import**.
7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Form Union (fin)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

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.

SOLID MECHANICS (SOLID)

Now specify the boundary conditions for the Solid Mechanics interface.

Roller 1

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

2 Select Boundaries 17 and 27 only.

Spring Foundation 1

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

2 Select Boundaries 18–21 and 31–34 only.

3 In the **Settings** window for Spring Foundation, locate the **Spring** section.

4 From the list, choose **Diagonal**.

5 In the $\mathbf{k}_A$ table, enter the following settings:

1e7	0	0
0	1e7	0
0	0	0

HEAT TRANSFER IN SOLIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.

Heat Flux 1

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

2 From the Selection list, choose **All boundaries**. Then remove boundaries 17 and 27.

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 `10`.

Heat Flux 2

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

2 Select Boundaries 17 and 27 only.

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

4 In the q_0 text field, type `1e4`.

STUDY 1

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

RESULTS

Stress (solid)

Under the Results node, three plot groups are automatically added to show the default results for the structural and thermal analyses. The first plot group, `Stress (solid)`, shows the von Mises stresses on a scaled deformed geometry, as shown in Figure 2.

Temperature (ht)

The second plot group, `Temperature (ht)`, displays the temperature distribution as shown in Figure 1.

Bracket—Transient Analysis

Introduction

Transient analyses provide the time domain response of a structure subjected to time-dependent loads. A transient analysis can be important when the time scale of the load is such that inertial or damping effects might have a significant influence on the behavior of the structure.

In this example you learn how to add damping properties to the material, define external loads varying with time, set up time-stepping data for the study, and generate animations.

It is recommended you review the *Introduction to the Structural Mechanics Module*, which includes background information and discusses the `bracket_basic.mph` models relevant to this example.

Model Definition

This model is an extension to the example described in the section "The Fundamentals: A Static Linear Analysis" in the *Introduction to the Structural Mechanics Module*.

The model geometry is represented in Figure 1.

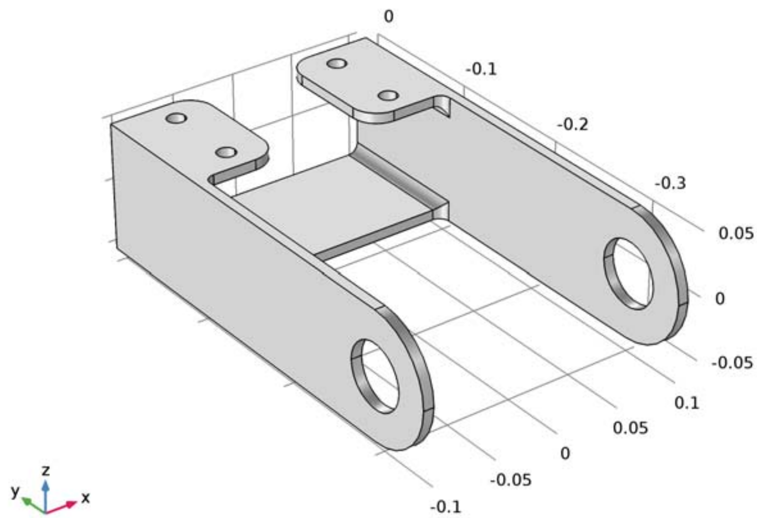


Figure 1: Bracket geometry.

A rigid body is assumed to be connected to the arms of the bracket on which the time-varying load is applied.

Results and Discussion

Figure 2 shows the rigid connector's displacements at the center of rotation versus the time.

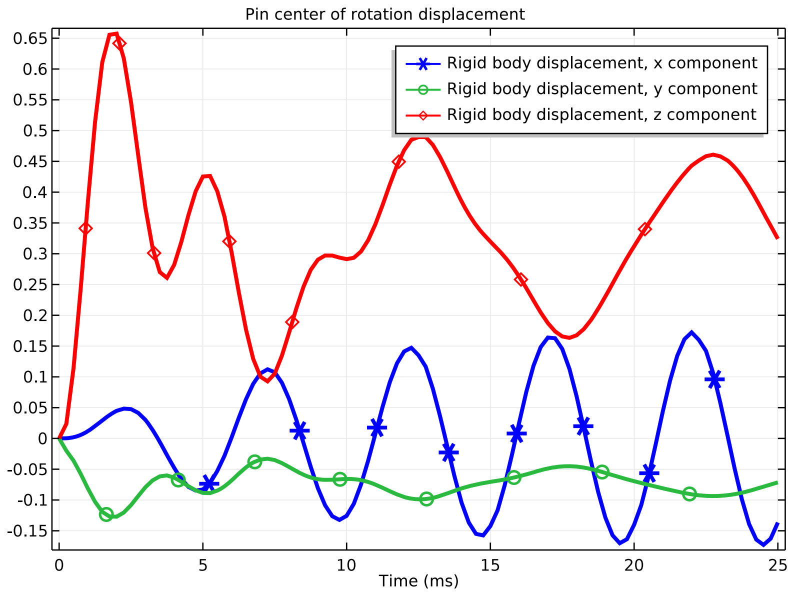


Figure 2: Displacement of the pin center of rotation vs time.

Notes About the COMSOL Implementation

To accurately model the physical problem, you need to apply damping in a dynamic analysis. In COMSOL you have the possibility to add damping of several type: isotropic loss factor, anisotropic loss factor or Rayleigh damping.

To implement time-dependent components you can enter expressions using the variable `t`, the time variable in COMSOL Multiphysics.

The scaling of the displacement variables can be changed to correspond to the expected deformations. Manual scaling is the default for the displacement variables. The default manual scaling used for the structural displacement is 1/100th of the geometry size, which in this example is about 200 mm. In this case the expected deformations are in the order of 0.1 mm. The manual scaling factor should thus be changed to 10^{-4}.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_transient`

Modeling Instructions

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

Browse to the application's Application Libraries folder and double-click the file `bracket_basic.mph`.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

In the **Model Builder** window, expand the **Component 1 (comp1)>Solid Mechanics (solid)** node, then click **Linear Elastic Material 1**.

Damping 1

1 On the **Physics** toolbar, click **Attributes** and choose **Damping**.

   In this example, you will use Rayleigh damping.

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

3 In the α_{dM} text field, type `50`.

4 In the β_{dK} text field, type `1e-4`.

   To represent the pin between the arms of the bracket you can use a rigid connector, to which a load can be applied.

Rigid Connector 1

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

2 Select Boundaries 4, 5, 42, and 43 only.

Applied Force 1

1 Right-click **Rigid Connector 1** and choose **Applied Force**.

2 In the **Settings** window for Applied Force, locate the **Applied Force** section.

3 Specify the **F** vector as

`100*sin(2*pi*t*200[1/s])`	x

`-11000`	y
`700*sin(2*pi*t*100[1/s])`	z

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.

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

STUDY 1

Step 1: Time Dependent

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

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

3 In the **Times** text field, type `range(0,0.25,25)`.

With this time stepping, the solver automatically stores the solution every 0.25 ms from 0 to 25 ms.

The time-dependent solver adapts its time stepping based on a tolerance criterion. This ensures that the solver takes small enough time steps if large variations occur between the specified output times.

4 Select the **Relative tolerance** check box.

5 In the associated text field, type `0.001`.

6 Click to expand the **Results while solving** section. Locate the **Results While Solving** section. Select the **Plot** check box.

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)>Dependent Variables 1** node, then click **Displacement field (Material) (comp1.u)**.

4 In the **Settings** window for Field, locate the **Scaling** section.

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

The computation can be speed up for a small model by disabling the convergence plot. To do this, go to the **Options** menu and select **Preferences**. Under **Results**, click to clear the **Generate convergence plots** check box.

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

RESULTS

Stress (solid)

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

The default plot shows the stress distribution at the final time. You can change the time for the plot display in the Time list of the plot group settings.

Plot the displacement of the center of rotation of the rigid body versus the time.

1D Plot Group 2

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

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 `Pin center of rotation displacement`.

5 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

Global 1

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

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

3 In the table, enter the following settings:

Expression	Unit	Description
solid.rig1.u	mm	Rigid body displacement, x component
solid.rig1.v	mm	Rigid body displacement, y component
solid.rig1.w	mm	Rigid body displacement, z component

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

5 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

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

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

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 **Legend** section.

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

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

To visualize the results in an animation, create a player.

Animation 1

1 On the **Results** toolbar, click **Animation** and choose **File**.

2 In the **Settings** window for Animation, locate the **Target** section.

3 From the **Target** list, choose **Player**.

4 Locate the **Frames** section. From the **Frame selection** list, choose **All**.

5 Click **Show Frame**.

COMSOL Multiphysics generates the movie and then plays it. To replay the movie, click the Play button on the Graphics toolbar.

If you want to export a movie in GIF, Flash, or AVI format, right-click **Export** and create an **Animation** feature.

Created in COMSOL Multiphysics 5.2a

Vibrations of a Disk Backed by an Air-Filled Cylinder

Introduction

The vibration modes of a thin or thick circular disc are well known, and it is possible to compute the corresponding eigenfrequencies with an arbitrary precision from a series solution. The same is true for the acoustic modes of an air-filled cylinder with perfectly rigid walls. A more interesting question to ask is: What happens if the cylinder is sealed in one end not by a rigid wall but by a thin disc? This is the question you address in this tutorial.

The application uses the Structural Mechanics Module's Shell interface and the Pressure Acoustics interface from COMSOL Multiphysics. If you have a license for the Acoustics Module, see Vibrations of a Disk Backed by an Air-Filled Cylinder in the Acoustics Module Application Libraries for a model version that uses the Acoustic-Shell Interaction, Frequency Domain multiphysics interface.

Model Definition

The geometry is a rigid steel cylinder with a height of 255 mm and a radius of 38 mm. One end is welded to a heavy slab, while the other is sealed with a steel disc only 0.38 mm thick. The disc is modeled using shell elements with the outer edge of the disc fixed. The acoustics in the cylinder is described in terms of the acoustic (differential) pressure. The eigenvalue equation for the pressure is

$$-\Delta p = \frac{\omega^2}{c^2}p$$

where c is the speed of sound and $\omega = 2\pi f$ defines the eigenfrequency, f.

A first step is to calculate the eigenfrequencies for the disc and the cylinder separately and compare them with theoretical values. This way you can verify the basic components of the model and assess the accuracy of the finite-element solution before modeling the coupled system. In a second step you simulate a decoupled problem, where the acoustic domain is completely surrounded by sound hard boundaries. In the third step you perform a coupled analysis where the boundary at the disc instead has the accelerations of the disc as boundary conditions. At the same time, the acoustic pressure supplies a load on the disc.

Results and Discussion

To be able to study the effects of the coupling, we first look at the solution of the uncoupled problem. Figure through Figure 4 show the two first uncoupled structural and acoustic modes.

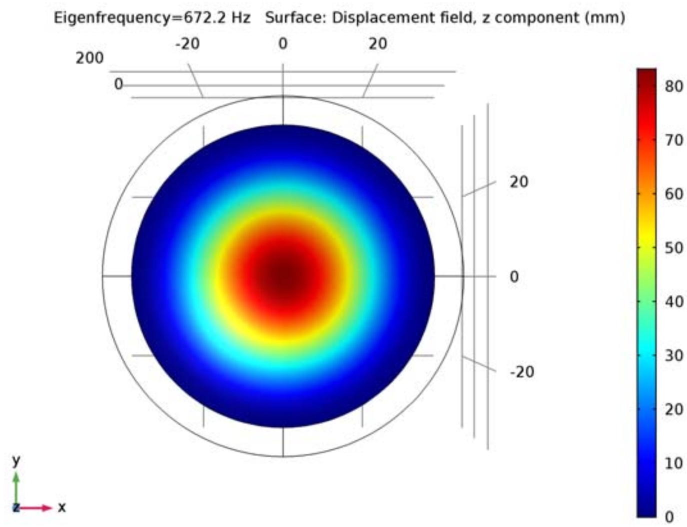


Figure 1: First uncoupled structural mode represented with vertical displacement.

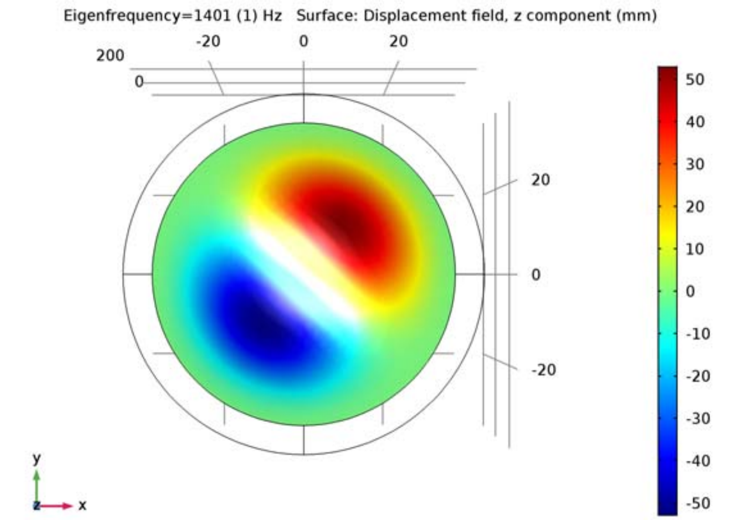


Figure 2: Second uncoupled structural mode represented with vertical displacement.

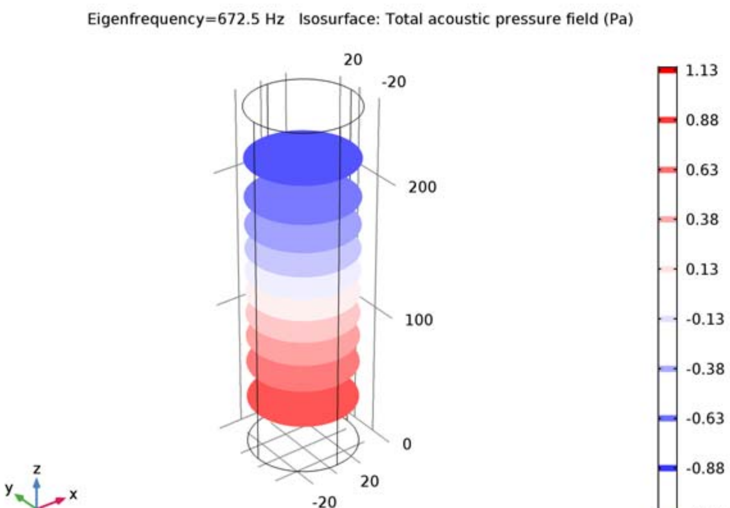


Figure 3: First uncoupled acoustic mode represented with pressure isosurfaces.

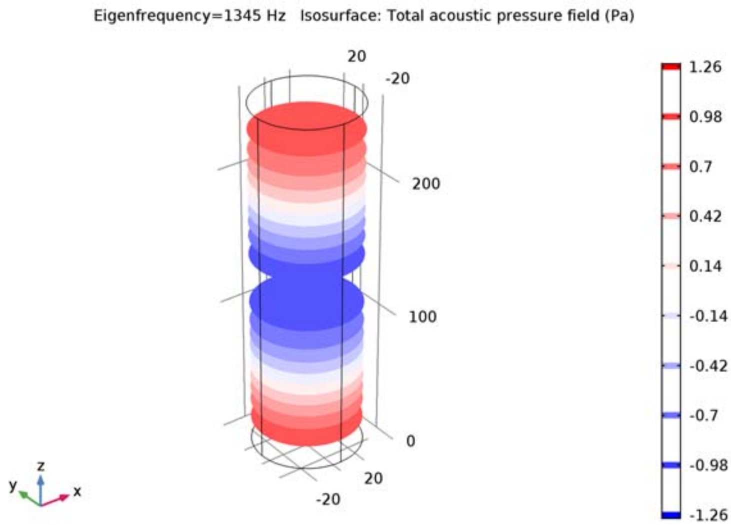


Figure 4: Second uncoupled acoustic mode represented with pressure isosurfaces.

In Ref. 1, D.G. Gorman and others have thoroughly investigated the model at hand, and they have developed a semi-analytical solution verified by experiments. Their results for the coupled problem are presented in Table 1, together with the computed results from the COMSOL Multiphysics analysis..

TABLE 1: RESULTS FROM SEMI-ANALYTICAL AND COMSOL MULTIPHYSICS ANALYSIS AND EXPERIMENTAL DATA

Dominated by	**Semi-analytical (Hz)**	**Computed (Hz)**	**Experimental (Hz)**
str/ac	636.9	637.2	630
str/ac	707.7	707.7	685
ac	1347	1347	1348
str	1394	1396/1397	1376
ac	2018	2018	2040
str	2289	2295/2307	2170
str/ac	2607	2628	2596
ac	2645	2646	–
str/ac	2697	2698	2689
ac	2730	2730	2756
ac	2968	2968	2971

As the table shows, the computed eigenfrequencies are in good agreement with both the theoretical predictions and the experimentally measured values. The table also states whether the mode is structurally dominated (str), acoustically dominated (ac), or tightly coupled (str/ac).The eigenfrequency precision is generally better for the acoustically dominated modes.

Most of the modes show rather weak coupling between the structural bending of the disc and the pressure field in the cylinder. It is, however, interesting to note that some of the uncoupled modes have been split into one covibrating and one contravibrating mode with distinct eigenfrequencies. This is, for example, the case for modes 1 and 2 in the FEM solution.

In Figure 5 the first coupled mode is shown in terms of disc displacements and air pressure. The coupling effect can be clearly displayed using a plot of pressure gradients, as in Figure 6.

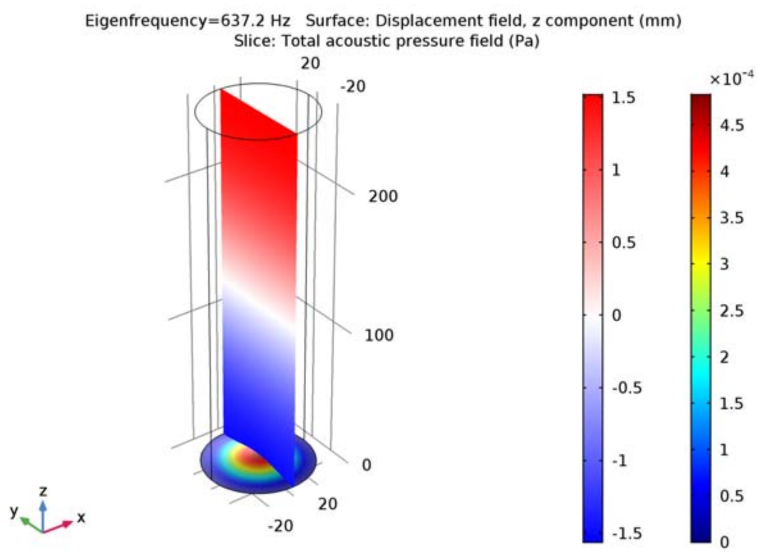


Figure 5: Disc deformation and pressure contours for the first coupled mode.

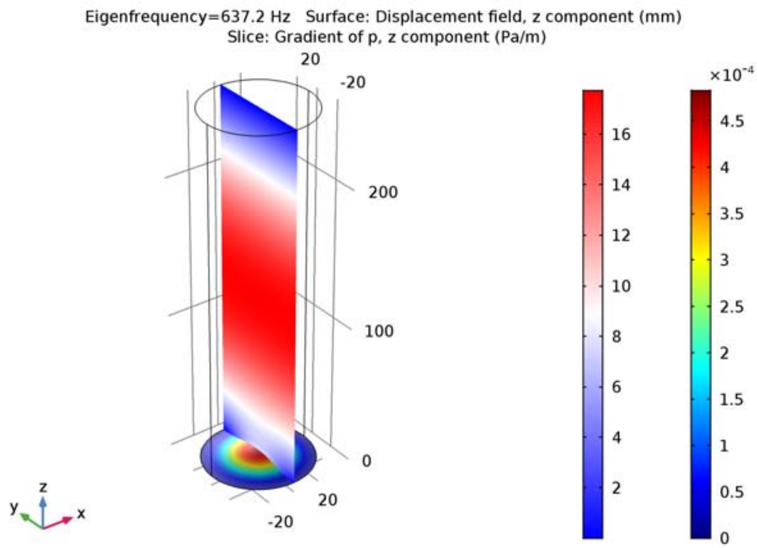


Figure 6: Disc deformation and pressure gradient contours for the first coupled mode.

Reference

1. D.G. Gorman, J.M. Reese, J. Horacek, and D. Dedouch, "Vibration Analysis of a Circular Disk Backed by a Cylindrical Cavity," *Proc. Instn. Mech. Engrs.*, Part C, vol. 215, no 11, pp 1303-1311, 2001.

Notes About the COMSOL Implementation

You specify the part of the physics for which to compute the uncoupled eigenvalues by selecting the variables for the eigenvalue solver.

When coupling the two types of physics, be careful when selecting the sign of the coupling terms, so that they act in the intended direction. You should specify the acceleration in the inward normal direction for the pressure acoustics domain, which in this case is the positive z-acceleration of the disc. The acceleration is denoted `wtt` as it is the second time-derivative of the variable `w`. The pressure on the shell can be given using global directions, so that a positive pressure acts as a face load in the negative z-direction. This is handled automatically by COMSOL in the setup in this example.

Application Library path: `Structural_Mechanics_Module/Acoustic-Structure_Interaction/coupled_vibrations_manual`

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>Shell (shell)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Eigenfrequency**.
8 Click **Done**.

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**.

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 `38`.
4 In the **Height** text field, type `255`.
5 Right-click **Cylinder 1 (cyl1)** and choose **Build Selected**.

Form Union (fin)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

SHELL (SHELL)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Shell (shell)**.
2. Select Boundary 3 only.
3. In the **Settings** window for Shell, locate the **Thickness** section.
4. In the d text field, type `0.38[mm]`.

Fixed Constraint 1

1. On the **Physics** toolbar, click **Edges** and choose **Fixed Constraint**.
2. Select Edges 2, 3, 7, and 10 only.

MATERIALS

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

Material 1 (mat1)

1. In the **Settings** window for Material, locate the **Material Contents** section.
2. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Density	rho	1.2	kg/m³	Basic
Speed of sound	c	343	m/s	Basic

Material 2 (mat2)

1. Right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for Material, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundary 3 only.
5. Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	2.1e11	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
Density	rho	7800	kg/m³	Basic

MESH 1

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

Free Quad 1

Select Boundary 3 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 Click the **Custom** button.
4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `10`.

Free Quad 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Quad 1**.
2 In the **Settings** window for Free Quad, click **Build Selected**.
3 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

Swept 1

In the **Settings** window for Swept, click **Build Selected**.

STUDY 1

In the first study, you solve the structural problem only.

1 In the **Model Builder** window, click **Study 1**.
2 In the **Settings** window for Study, type `Structural Analysis` in the **Label** text field.

STRUCTURAL ANALYSIS

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Structural Analysis** 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 `20`.
5 In the **Search for eigenfrequencies around** text field, type `500`.

   Exclude the Pressure Acoustics interface.

6 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for the **Pressure Acoustics, Frequency Domain (acpr)** interface.

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

RESULTS

Mode Shape (shell)

1 In the **Model Builder** window, under **Results** click **Mode Shape (shell)**.

2 In the **Settings** window for 3D Plot Group, type `Mode Shape, Structural Analysis` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Results>Mode Shape, Structural Analysis** 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>Shell>Displacement>Displacement field>w - Displacement field, z component**.

3 On the **Mode Shape, Structural Analysis** toolbar, click **Plot**.

4 Click the **Go to XY View** button on the **Graphics** toolbar.

5 Click the **Scene Light** button on the **Graphics** toolbar.

Mode Shape, Structural Analysis

1 In the **Model Builder** window, under **Results** click **Mode Shape, Structural Analysis**.

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

3 From the **Eigenfrequency (Hz)** list, choose **1401 (1)**.

4 On the **Mode Shape, Structural Analysis** toolbar, click **Plot**.

Undeformed Geometry (shell)

1 In the **Model Builder** window, under **Results** click **Undeformed Geometry (shell)**.

2 In the **Settings** window for 3D Plot Group, type `Undeformed Geometry, Structural Analysis` in the **Label** text field.

ROOT

Add the second study to solve the pure acoustics problem.

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>Eigenfrequency**.

For this study, exclude the Shell interface.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Shell (shell)** 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

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

2 In the **Settings** window for Study, type `Acoustics Analysis` in the **Label** text field.

ACOUSTICS ANALYSIS

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Acoustics Analysis** 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 `20`.

5 In the **Search for eigenfrequencies around** text field, type `500`.

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

RESULTS

Acoustic Pressure (acpr)

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

2 In the **Settings** window for 3D Plot Group, type `Acoustic Pressure, Acoustics Analysis` in the **Label** text field.

Sound Pressure Level (acpr)

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

2 In the **Settings** window for 3D Plot Group, type `Sound Pressure Level, Acoustics Analysis` in the **Label** text field.

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, type `Acoustic Pressure, Acoustics Analysis, Isosurfaces` in the **Label** text field.

3 Locate the **Data** section. From the **Eigenfrequency (Hz)** list, choose **672.5**.

Isosurface 1

1 In the **Model Builder** window, expand the **Results>Acoustic Pressure, Acoustics Analysis, Isosurfaces** node, then click **Isosurface 1**.
2 In the **Settings** window for Isosurface, locate the **Coloring and Style** section.
3 From the **Color table** list, choose **WaveLight**.
4 Select the **Symmetrize color range** check box.
5 On the **Acoustic Pressure, Acoustics Analysis, Isosurfaces** toolbar, click **Plot**.
6 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

Acoustic Pressure, Acoustics Analysis, Isosurfaces

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure, Acoustics Analysis, Isosurfaces**.
2 In the **Settings** window for 3D Plot Group, locate the **Data** section.
3 From the **Eigenfrequency (Hz)** list, choose **1345**.
4 On the **Acoustic Pressure, Acoustics Analysis, Isosurfaces** toolbar, click **Plot**.

   Add the boundary conditions that couple the Pressure Acoustics and Shell interfaces.

SHELL (SHELL)

Face Load 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Face Load**.
2 Select Boundary 3 only.

   The acoustic pressure exerts a normal load on the plate.

3 In the **Settings** window for Face Load, locate the **Force** section.
4 From the $\mathbf{F}_A$ list, choose **Acoustic load per unit area (acpr/fpam1)**.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Frequency Domain (acpr)**.

Normal Acceleration 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Normal Acceleration**.
2 Select Boundary 3 only.

   The inward normal acceleration at the plate equals the second time derivative of the vertical displacement.

3 In the **Settings** window for Normal Acceleration, locate the **Normal Acceleration** section.

4 From the **Type** list, choose **Acceleration**.

5 From the $\mathbf{a}_0$ list, choose **Acceleration (shell/emm1)**.

Add the third study for the coupled problem.

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>Eigenfrequency**.

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

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

STUDY 3

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

2 In the **Settings** window for Study, type `Coupled Analysis` in the **Label** text field.

COUPLED ANALYSIS

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Coupled Analysis** 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 `20`.

5 In the **Search for eigenfrequencies around** text field, type `500`.

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

RESULTS

Mode Shape (shell)

1 In the **Model Builder** window, click **Mode Shape (shell)**.

2 In the **Settings** window for 3D Plot Group, type `Mode Shape, Coupled Analysis` in the **Label** text field.

3 Locate the **Data** section. From the **Eigenfrequency (Hz)** list, choose **637.2**.

Surface 1

1 In the **Model Builder** window, expand the **Results>Mode Shape, Coupled Analysis** 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 **w - Displacement field, z component**.

Mode Shape, Coupled Analysis

In the **Model Builder** window, under **Results** right-click **Mode Shape, Coupled Analysis** and choose **Slice**.

Slice 1

1 In the **Settings** window for Slice, locate the **Plane Data** section.

2 In the **Planes** text field, type `1`.

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

4 On the **Mode Shape, Coupled Analysis** toolbar, click **Plot**.

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

Plot the pressure gradient to display the connection to the disk shape.

6 Locate the **Expression** section. In the **Expression** text field, type `pz`.

7 In the **Unit** field, type `Pa/m`.

8 On the **Mode Shape, Coupled Analysis** toolbar, click **Plot**.

Undeformed Geometry (shell)

1 In the **Model Builder** window, under **Results** click **Undeformed Geometry (shell)**.

2 In the **Settings** window for 3D Plot Group, type `Undeformed Geometry, Coupled Analysis` in the **Label** text field.

Acoustic Pressure (acpr)

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

2 In the **Settings** window for 3D Plot Group, type `Acoustic Pressure, Coupled Analysis` in the **Label** text field.

Sound Pressure Level (acpr)

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

2 In the **Settings** window for 3D Plot Group, type `Sound Pressure Level, Coupled Analysis` in the **Label** text field.

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, type `Acoustic Pressure, Coupled Analysis, Isosurfaces` in the **Label** text field.

Created in COMSOL Multiphysics 5.2a

Static and Eigenfrequency Analyses of an Elbow Bracket

Introduction

The component shown in Figure 1 is part of a support mechanism and is subjected to both mechanical loads and thermal loads. This tutorial model takes you through the steps to carry out a detailed analysis of the part using the Structural Mechanics Module.

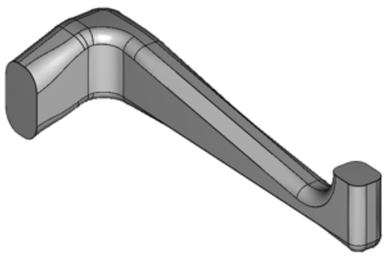


Figure 1: Geometry of the elbow bracket.

In the various parts of the example you are introduced to using the available basic analysis types, together with numerous postprocessing possibilities. These analysis types are:

- Static analysis
- Eigenfrequency analysis
- Damped eigenfrequency analysis

In an extended version of this model, also the following study types are covered:

- Transient analysis
- Modal based transient analysis
- Frequency response analysis
- Modal based frequency response analysis
- Parametric analysis
- Linear buckling analysis

This tutorial model comes in two versions:

- A short version, `elbow_bracket_brief`, treating the three first analysis types in the above list.
- A complete version, `elbow_bracket`, treating all nine analysis types.

Each of the listed analysis types corresponds to a *study* type; the available studies are described in the section Available Study Types in the *Structural Mechanics Module User's*

Guide. The chapter Structural Mechanics Modeling in the same manual provides further assistance.

Model Definition

The geometry for this part, see Figure 1, has been created with a CAD software, and it is available for you to import into COMSOL Multiphysics.

Material

Structural steel, as taken from the material library, with Young's modulus of 200 GPa, Poisson's ratio of 0.33, and coefficient of thermal expansion $12.3 \cdot 10^{-6}\ \mathrm{K}^{-1}$.

Damping

The Structural Mechanics Module supports Rayleigh damping and loss factor damping. You can also use no damping, which is the default option.

In some of the studies Rayleigh damping is used. Then you specify damping parameters that are proportional to the mass (α_{dM}) and stiffness (β_{dK}) in the following way:

$$C = \alpha_{dM} M + \beta_{dK} K$$

where C is the damping matrix, M is the mass matrix, and K is the stiffness matrix. The damping is specified locally in each domain; this means that you can specify different damping parameters in different parts of the model.

To find the values for the Rayleigh damping, you can use the relations between the critical damping ratio and the Rayleigh damping parameters. It is often easier to interpret the critical damping ratios, which are given by

$$\xi_i = \frac{\left(\frac{\alpha_{dM}}{\omega_i} + \beta_{dK} \cdot \omega_i\right)}{2}$$

where ξ_i is the critical damping ratio at a specific angular frequency ω_i. Knowing two pairs of corresponding ξ_i and ω_i results in a system of equations

$$\begin{bmatrix} \frac{1}{(2 \cdot \omega_1)} & \frac{\omega_1}{2} \\ \frac{1}{(2 \cdot \omega_2)} & \frac{\omega_2}{2} \end{bmatrix} \begin{bmatrix} \alpha_{dM} \\ \beta_{dK} \end{bmatrix} = \begin{bmatrix} \xi_1 \\ \xi_2 \end{bmatrix}$$

with the damping parameters as the unknown variables.

Assume that the structure has a constant damping ratio of 0.1. Select two frequencies near the excitation frequency, 400 Hz and 600 Hz. Solving the system of equations above gives the result is $\alpha_{dM} = 300$ and $\beta_{dK} = 3.2 \cdot 10^{-5}$.

For more information see the section about modeling damping and losses in the *Structural Mechanics Module User's Guide*.

If modal-based dynamic response studies are performed it is usually easier to give the critical damping ratios directly. This also gives more detailed control over the damping properties over a large frequency range.

Loads and Constraints

The displacement are fixed in all directions on the face shown in Figure 2. The load is described under each study, but in all cases it is distributed over the face shown in this figure.

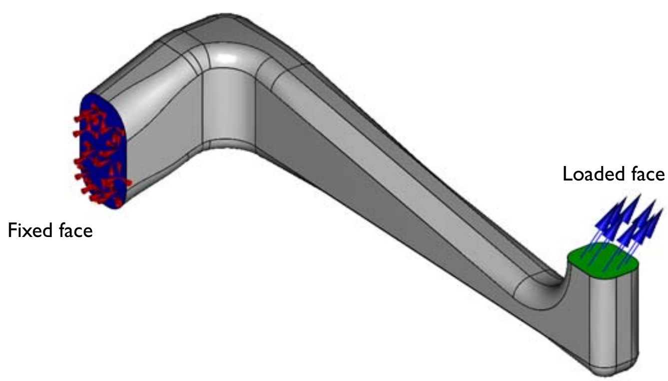


Figure 2: Constraint and loading of the bracket.

The Application Libraries note immediately below appears in the discussion of every model. The path indicates the location of the example file in the Application Libraries root directory. The most convenient way to open it is from the **Application Libraries** window in the COMSOL Desktop, which you can open from the **File** menu.

Application Library path: `Structural_Mechanics_Module/Tutorials/elbow_bracket_brief`

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**.

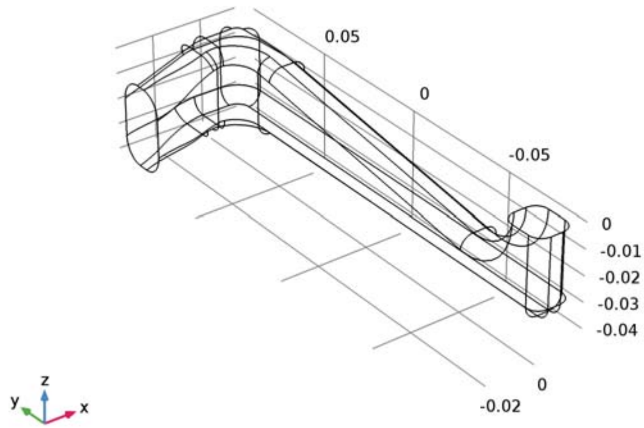
GEOMETRY 1

Import 1 (imp1)

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

7 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

The view in the Graphics window should look like that in the image below.



8 Click the **Wireframe Rendering** button on the **Graphics** toolbar to return to the default surface rendering.

Suppress some edges during meshing, in order to avoid generation of unnecessary small elements.

Ignore Edges 1 (ige1)

1 On the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Edges**.

2 On the object **fin**, select Edges 17, 21, 23, 27, 38, 40, 42, and 44 only.

MESH 1

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

Size

The default free mesher has nine predefined combinations of mesh parameter settings. They range from **Extremely fine** to **Extremely coarse**, with **Normal** as the default setting. Unless any other mesh parameters are set, this is the setting that is used if you use **Build All** or **Build Selected** to generate the mesh.

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

2 From the **Predefined** list, choose **Coarse**.

As a stress concentration can be expected in the corner of the bracket, put a finer mesh there.

Free Tetrahedral 1

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

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 Locate the **Element Size** section. From the **Predefined** list, choose **Extra fine**.
4 Select Boundaries 13 and 14 only.
5 In the **Model Builder** window, click **Mesh 1**.
6 In the **Settings** window for Mesh, click **Build All**.

MATERIALS

Next, specify the material properties. You can do this either by explicitly typing them in or by selecting a library material in the Material Browser. For this model, use a library material.

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.

Static Analysis

A static analysis has no explicit or implicit time dependencies. This situation corresponds to the steady state with constant (in time) boundary conditions and material properties.

The purpose of such analysis can be to find the maximum stress level and compare it with the material's yield strength, as well as to check that the deformation of the component is within the limits of the design criteria.

Results and Discussion

The analysis shows that the von Mises effective stress has a maximum value of about 190 MPa, which, compared with the material's yield strength of 350 MPa, results in a utilization factor of 54%.

The analysis also gives the maximum static displacements as 1.14 mm

Three different representations of the stress state are shown in Figure 3 through Figure 5.

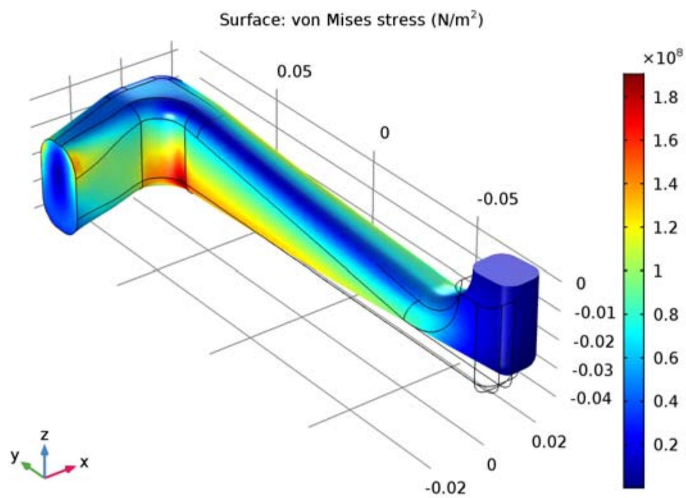


Figure 3: Effective stresses on the boundary of the domain.

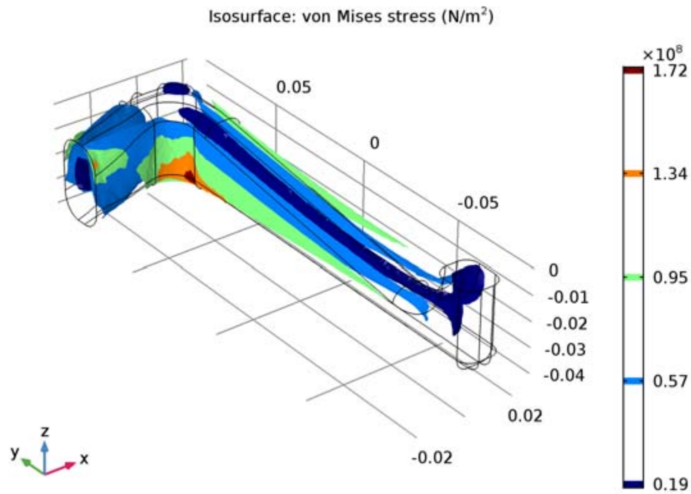


Figure 4: Isosurface plot of the effective stress.

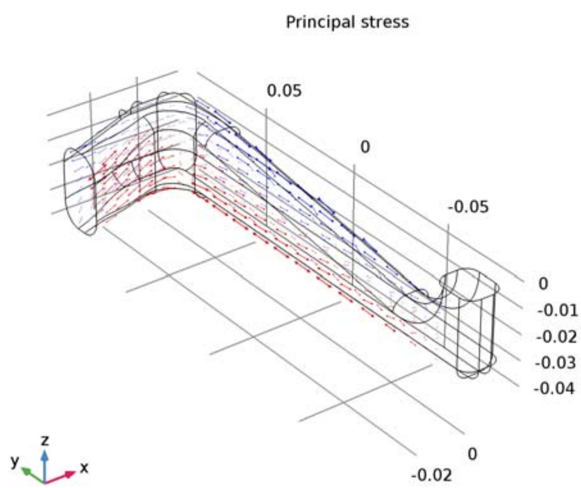


Figure 5: Arrow plot of the principal stresses.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

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

Boundary Load 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
2. Select Boundary 21 only.
3. In the **Settings** window for Boundary Load, locate the **Force** section.
4. Specify the $\mathbf{F}_A$ vector as

`3[MPa]`	x
`0`	y
`3[MPa]`	z

STUDY 1

In this model, where there are many different studies, it is a good idea to assign manual names to some nodes in the model tree.

1. In the **Model Builder** window, click **Study 1**.
2. In the **Settings** window for Study, type `Study 1 (Static)` in the **Label** text field.

   The default settings in the generated solver are OK for this model, so it can be run directly.
3. On the **Home** toolbar, click **Compute**.

Before moving on to analyzing the solution, rename the solver.

STUDY 1 (STATIC)

In the **Model Builder** window, expand the **Study 1 (Static)** node.

Solution 1 (sol1)

1. In the **Model Builder** window, expand the **Study 1 (Static)>Solver Configurations** node, then click **Solution 1 (sol1)**.
2. In the **Settings** window for Solution, type `Solution, Static` in the **Label** text field.

RESULTS

Similarly, rename the solution data set.

Data Sets

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

2 In the **Settings** window for Solution, type `Static Solution` in the **Label** text field.

In the Results branch, you can create various plot types, evaluate expressions, or animate the results. The result features can visualize any expression containing, for example, the solution variables, their derivatives, and the space coordinates. Many frequently used expressions are predefined as postprocessing variables, and they are directly available in the Expression section menus for the various plot types.

When the solver finishes, a default plot appears. It shows a surface plot of the von Mises stress with the deformed shape of the component. For future reference, you can rename it.

Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 In the **Settings** window for 3D Plot Group, type `Static Stress Contour` in the **Label** text field.

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

To evaluate the maximum displacement, use a `maximum` coupling operator.

DEFINITIONS

Maximum 1 (maxop1)

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

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

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

4 From the **Selection** list, choose **All boundaries**.

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
`U_max`	`maxop1(solid.disp)`	m	`Maximum deflection`

STUDY 1 (STATIC)

Solution, Static (sol1)

1 In the **Model Builder** window, under **Study 1 (Static)>Solver Configurations** right-click **Solution, Static (sol1)** and choose **Solution>Update**.

This step is necessary in order to access variables that were created after the solution was performed.

RESULTS

Global Evaluation 1

On the **Results** toolbar, click **Global Evaluation**.

Derived Values

1 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>U_max - Maximum deflection**.

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

Expression	Unit	Description
`U_max`	`mm`	`Maximum deflection`

3 Click **Evaluate**.

The result, approximately 1.1 mm appears in the **Table** window.

Next, add a second plot group and create an isosurface plot. The resulting plot should resemble that in Figure 4.

3D Plot Group 2

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

2 In the **Settings** window for 3D Plot Group, type `Static Stress Isosurface` in the **Label** text field.

Static Stress Isosurface

1 Right-click **Static Stress Isosurface** 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>Solid Mechanics>Stress>solid.mises - von Mises stress**.

3 Right-click **Results>Static Stress Isosurface>Isosurface 1** and choose **Deformation**.

4 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

With the following steps you can reproduce the principal stress arrow plot shown in Figure 5:

3D Plot Group 3

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

2 In the **Settings** window for 3D Plot Group, type `Static Principal Stress Arrow Plot` in the **Label** text field.

Principal Stress Volume 1

On the **Static Principal Stress Arrow Plot** toolbar, click **More Plots** and choose **Principal Stress Volume**.

Static Principal Stress Arrow Plot

1 In the **Settings** window for Principal Stress Volume, locate the **Positioning** section.

2 Find the **X grid points** subsection. In the **Points** text field, type `10`.

3 Find the **Y grid points** subsection. In the **Points** text field, type `15`.

4 Find the **Z grid points** subsection. In the **Points** text field, type `10`.

5 On the **Static Principal Stress Arrow Plot** toolbar, click **Plot**.

Eigenfrequency Analysis

An eigenfrequency analysis finds the eigenfrequencies and modes of deformation of a component. The eigenfrequencies f in the structural mechanics field are related to the eigenvalues λ returned by the solvers through

$$f = \frac{-\lambda}{2\pi i}$$

In COMSOL Multiphysics you can choose between working with eigenfrequencies and working with eigenvalues according to your preferences. Eigenfrequencies is the default option for all physics interfaces in the Structural Mechanics Module.

If no damping is included in the material, the undamped natural frequencies are computed.

The purpose of the following eigenfrequency analysis is to find the six lowest eigenfrequencies and corresponding mode shapes.

Results and Discussion

The first six eigenfrequencies, rounded to three digits, are:

EIGENFREQUENCY	FREQUENCY
f_1	417 Hz
f_2	571 Hz
f_3	1927 Hz
f_4	2455 Hz
f_5	3111 Hz
f_6	3933 Hz

The mode shapes corresponding to the two lowest eigenfrequencies are shown in Figure 6. The deformed plot indicates an oscillation in the xy-plane for the lowest eigenfrequency, while the second lowest eigenmode shows an oscillation in the yz-plane.

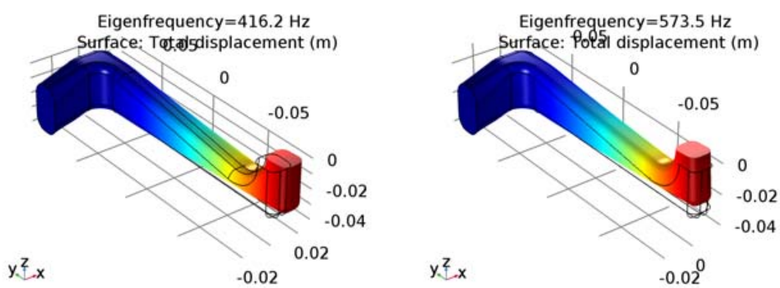


Figure 6: Eigenmodes of the two lowest eigenfrequencies.

Notes About the COMSOL Implementation

Any loads present on the model, such as the load from the static load case above, are ignored in the default eigenfrequency analysis. It is also possible to include effects from prestress. You can find an example of such an analysis in the example Vibrating String.

Add a new study to your model.

Add a new study to your model.

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>Eigenfrequency**.

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

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

STUDY 2

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

2 In the **Settings** window for Study, type `Study 2 (Eigenfrequency)` in the **Label** text field.

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

STUDY 2 (EIGENFREQUENCY)

Solution 2 (sol2)

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

2 In the **Settings** window for Solution, type `Solution, Eigenfrequency` in the **Label** text field.

RESULTS

Data Sets

1 In the **Model Builder** window, under **Results>Data Sets** click **Study 2 (Eigenfrequency)/ Solution, Eigenfrequency (sol2)**.

2 In the **Settings** window for Solution, type `Eigenfrequency Solution` in the **Label** text field.

Mode Shape (solid)

As a default, the first eigenmode is shown. Follow these steps to reproduce the plot in the left panel of Figure 6.

1 In the **Model Builder** window, expand the **Mode Shape (solid)** node, then click **Surface 1**.

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

3 Clear the **Color legend** check box.

The displacement values do not have any real significance for an eigenmode plot such as this one.

Take a look at the second mode as well.

4 In the **Model Builder** window, click **Mode Shape (solid)**.

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

6 From the **Eigenfrequency (Hz)** list, choose **573.5**.

7 On the **Mode Shape (solid)** toolbar, click **Plot**.

Compare the resulting plot to that to the right in Figure 6.

You can give the plot a more descriptive name:

8 In the **Label** text field, type `Undamped Mode Shapes`.

9 On the **Undamped Mode Shapes** toolbar, click **Animation** and choose **Player**.

Export

This creates an animation showing how the elbow bracket would deform if subjected to a harmonic load with a frequency near the selected eigenfrequency, in this case 571 Hz. To play the movie again, click the **Play** button on the Graphics toolbar.

The default animation sequence type when you add a player this way is **Dynamic data extension**. If you set the **Sequence type** to **Stored solutions** and then click the **Generate Frame** button, you get an animation where each frame corresponds to an eigenmode in the **Eigenfrequency** list. By using the **Frame number** slider in the **Frames** section you can then easily browse the eigenmodes.

Rename the player:

1 In the **Model Builder** window, under **Results>Export** click **Animation 1**.

2 In the **Settings** window for Animation, type `Mode Shape Animation` in the **Label** text field.

Damped Eigenfrequency Analysis

If the material has damping, the eigenvalue solver automatically switches to computation of the damped eigenfrequencies. The damped eigenfrequencies and eigenmodes are complex. The real part of the eigenfrequency corresponds to the frequency and the imaginary part is the damping.

Results and Discussion

The first six eigenfrequencies (rounded to three digits) are given below, and can be compared with the results from the undamped model.:

EIGENFREQUENCY	FREQUENCY	UNDAMPED FREQUENCY
f_1	415+41.3i Hz	417 Hz
f_2	568+56.6i Hz	571 Hz
f_3	1885+397i Hz	1927 Hz
f_4	2372+630i Hz	2454 Hz
f_5	2947+997i Hz	3111 Hz
f_6	3602+1579i Hz	3933 Hz

The relative damping of a certain mode is the ratio between the imaginary and the real part. It can be seen that the relative damping increases rapidly as the natural frequency increases. This is an effect of the Rayleigh damping model.

Notes About the COMSOL Implementation

As the eigenvalues exist as complex conjugate pairs, the first six damped eigenfrequencies correspond to the first three undamped eigenfrequencies. For this reason, twelve computed frequencies are requested.

Modeling Instructions

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>Eigenfrequency**.
4. Click **Add Study** in the window toolbar.
5. On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

SOLID MECHANICS (SOLID)

Add damping and specify the mass and stiffness parameters.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

Damping 1

1 On the **Physics** toolbar, click **Attributes** and choose **Damping**.
2 In the **Settings** window for Damping, locate the **Damping Settings** section.
3 In the α_{dM} text field, type `300`.
4 In the β_{dK} text field, type `3.2e-5`.

STUDY 3

1 In the **Model Builder** window, click **Study 3**.
2 In the **Settings** window for Study, type `Study 3 (Damped Eigenfrequency)` in the **Label** text field.
3 On the **Home** toolbar, click **Compute**.

STUDY 3 (DAMPED EIGENFREQUENCY)

Solution 3 (sol3)

1 In the **Model Builder** window, expand the **Study 3 (Damped Eigenfrequency)>Solver Configurations** node, then click **Solution 3 (sol3)**.
2 In the **Settings** window for Solution, type `Solution, Damped Eigenfrequency` in the **Label** text field.

RESULTS

Data Sets

1 In the **Model Builder** window, under **Results>Data Sets** click **Study 3 (Damped Eigenfrequency)/Solution, Damped Eigenfrequency (sol3)**.
2 In the **Settings** window for Solution, type `Damped Eigenfrequency Solution` in the **Label** text field.

Mode Shape (solid)

1 In the **Model Builder** window, under **Results** click **Mode Shape (solid)**.
2 In the **Settings** window for 3D Plot Group, type `Damped Mode Shapes` in the **Label** text field.

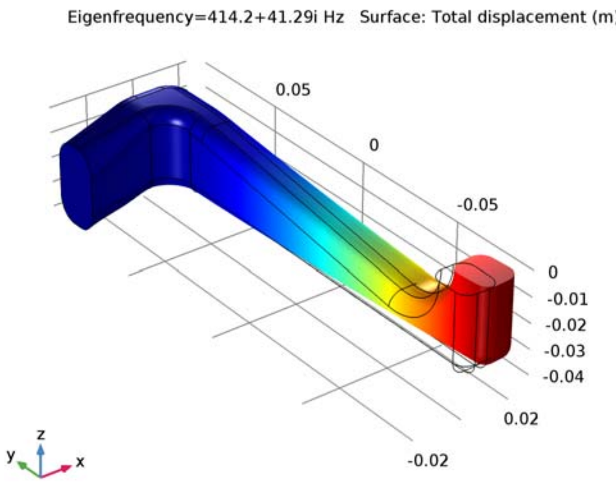
   The mode shape identical to the one obtained when solving the undamped problem. Only the frequency has changed.

Damped Mode Shapes

1 In the **Model Builder** window, expand the **Results>Damped Mode Shapes** node, then click **Surface 1**.

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

3 Clear the **Color legend** check box.



The second study should still produce undamped eigenfrequencies when it is run next time, so you must make sure that the newly added Damping node is ignored.

STUDY 2 (EIGENFREQUENCY)

Step 1: Eigenfrequency

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

2 In the **Settings** window for Eigenfrequency, locate the **Physics and Variables Selection** section.

3 Select the **Modify physics tree and variables for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Linear Elastic Material 1>Damping 1**.

5 Click **Disable**.

Heating Circuit

Introduction

Small heating circuits find use in many applications. For example, in manufacturing processes they heat up reactive fluids. Figure 1 illustrates a typical heating device for this application. The device consists of an electrically resistive layer deposited on a glass plate. The layer causes Joule heating when a voltage is applied to the circuit. The layer's properties determine the amount of heat produced.

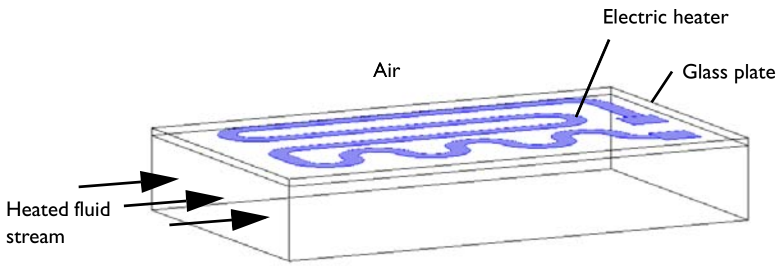


Figure 1: Geometry of a heating device.

In this particular application, you must observe three important design considerations:

- Non-invasive heating
- Minimal deflection of the heating device
- Avoidance of overheating the process fluid

The heater must also work without failure. You achieve the first and second requirements by inserting a glass plate between the heating circuit and the fluid; it acts as a conducting separator. Glass is an ideal material for both these purposes because it is non-reactive and has a low coefficient of thermal expansion.

You must also avoid overheating due to the risk of self-ignition of the reactive fluid stream. Ignition is also the main reason for separating the electrical circuit from direct contact with the fluid. The heating device is tailored for each application, making virtual prototyping very important for manufacturers.

For heating circuits in general, detachment of the resistive layer often determines the failure rate. This is caused by excessive thermally induced interfacial stresses. Once the layer has detached, it gets locally overheated, which accelerates the detachment. Finally, in the worst case, the circuit might overheat and burn. From this perspective, it is also important to study the interfacial tension due to the different thermal-expansion coefficients of the resistive layer and the substrate as well as the differences in temperature.

The geometric shape of the layer is a key parameter to design circuits for proper functioning. You can investigate all of the above-mentioned aspects by modeling the circuit.

This multiphysics example simulates the electrical heat generation, the heat transfer, and the mechanical stresses and deformations of a heating circuit device. The model uses the Heat Transfer in Solids interface of the Heat Transfer Module in combination with the Electric Currents, Shell interface from the AC/DC Module and the Solid Mechanics and Shell interfaces from the Structural Mechanics Module.

Note: This application requires the AC/DC Module, the Heat Transfer Module, and the Structural Mechanics Module.

Model Definition

Figure 2 shows a drawing of the modeled heating circuit.

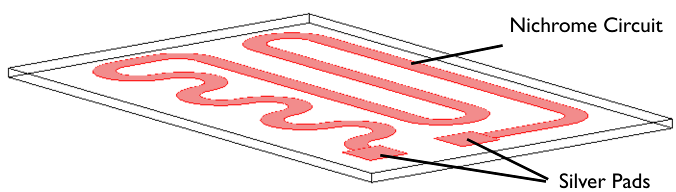


Figure 2: Drawing of the heating circuit deposited on a glass plate.

The device consists of a serpentine-shaped Nichrome resistive layer, 10 μm thick and 5 mm wide, deposited on a glass plate. At each end, it has a silver contact pad measuring 10 mm-by-10 mm-by-10 μm. When the circuit is in use, the deposited side of the glass plate is in contact with surrounding air, and the back side is in contact with the heated fluid. Assume that the edges and sides of the glass plate are thermally insulated.

Table 1 gives the resistor's dimensions.

TABLE 1: DIMENSIONS

OBJECT	LENGTH	WIDTH	THICKNESS
Glass Plate	130 mm	80 mm	2 mm
Pads and Circuit	-	-	10 μm

During operation the resistive layer produces heat. Model the electrically generated heat using the Electric Currents, Shell interface from the AC/DC Module. An electric potential of 12 V is applied to the pads. In the model, you achieve this effect by setting the potential at one edge of the first pad to 12 V and that of one edge of the other pad to 0 V.

To model the heat transfer in the thin conducting layer, use the Thin Layer feature from the Heat Transfer in Solids interface. The heat rate per unit area (measured in W/m^2) produced inside the thin layer is given by

$$q_{\mathrm{prod}} = dQ_{\mathrm{DC}} \tag{1}$$

where $Q_{\mathrm{DC}} = \mathbf{J} \cdot \mathbf{E} = \sigma|\nabla_{\mathbf{t}} V|^2$ (W/m^3) is the power density. The generated heat appears as an inward heat flux at the surface of the glass plate.

At steady state, the resistive layer dissipates the heat it generates in two ways: on its up side to the surrounding air (at 293 K), and on its down side to the glass plate. The glass plate is similarly cooled in two ways: on its circuit side by air, and on its back side by a process fluid (353 K). You model the heat fluxes to the surroundings using heat transfer coefficients, h. For the heat transfer to air, $h = 5\ W/(m^2{\cdot}K)$, representing natural convection. On the glass plate's back side, $h = 20\ W/(m^2{\cdot}K)$, representing convective heat transfer to the fluid. The sides of the glass plate are insulated.

The model simulates thermal expansion using static structural-mechanics analyses. It uses the Solid Mechanics interface for the glass plate, and the Shell interface for the circuit layer. The equations of these two interfaces are described in the *Structural Mechanics Module User's Guide*. The stresses are set to zero at 293 K. You determine the boundary conditions for the Solid Mechanics interface by fixing one corner with respect to x-, y-, and z-displacements and rotation.

Table 2 summarizes the material properties used in the model.

TABLE 2: MATERIAL PROPERTIES

MATERIAL	E [GPa]	ν	α [1/K]	k [W/(m·K)]	ρ [kg/m^3]	C_p [J/(kg·K)]
Silver	83	0.37	1.89e-5	420	10500	230
Nichrome	213	0.33	1e-5	15	9000	20
Glass	73.1	0.17	5.5e-7	1.38	2203	703

Results and Discussion

Figure 3 shows the heat that the resistive layer generates.

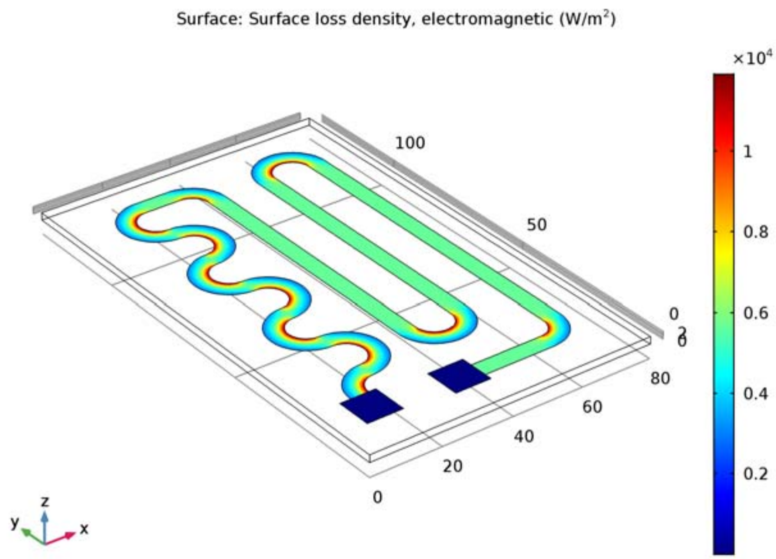


Figure 3: Stationary heat generation in the resistive layer when 12 V is applied.

The highest heating power occurs at the inner corners of the curves due to the higher current density at these spots. The total generated heat, as calculated by integration, is approximately 13.8 W.

Figure 4 shows the temperature of the resistive layer and the glass plate at steady state.

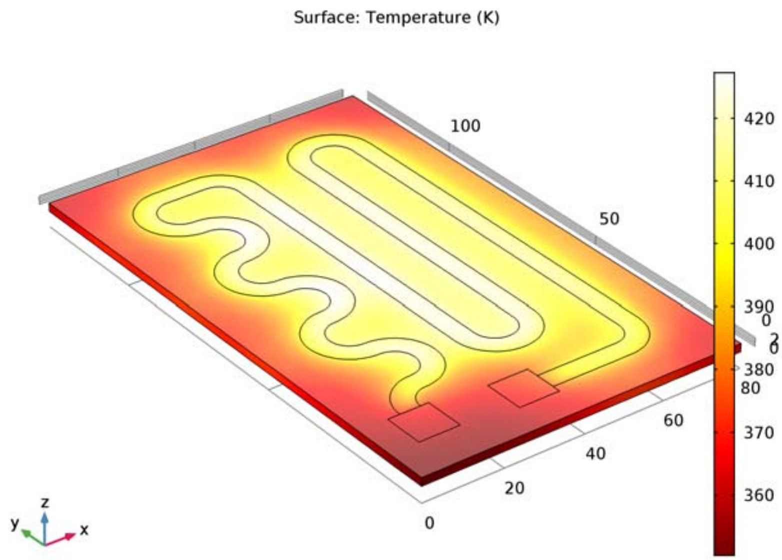


Figure 4: Temperature distribution in the heating device at steady state.

The highest temperature is approximately 428 K, and it appears in the central section of the circuit layer. It is interesting to see that the differences in temperature between the fluid side and the circuit side of the glass plate are quite small because the plate is very thin. Using boundary integration, the integral heat flux on the fluid side evaluates to approximately 8.5 W. This means that the device transfers the majority of the heat it generates—8.5 W out of 13.8 W—to the fluid, which is good from a design perspective, although the thermal resistance of the glass plate results in some losses.

The temperature rise also induces thermal stresses due the materials' different coefficients of thermal expansion. As a result, mechanical stresses and deformations arise in the layer and in the glass plate. Figure 5 shows the effective stress distribution in the device and the resulting deformations. During operation, the glass plate bends towards the air side.

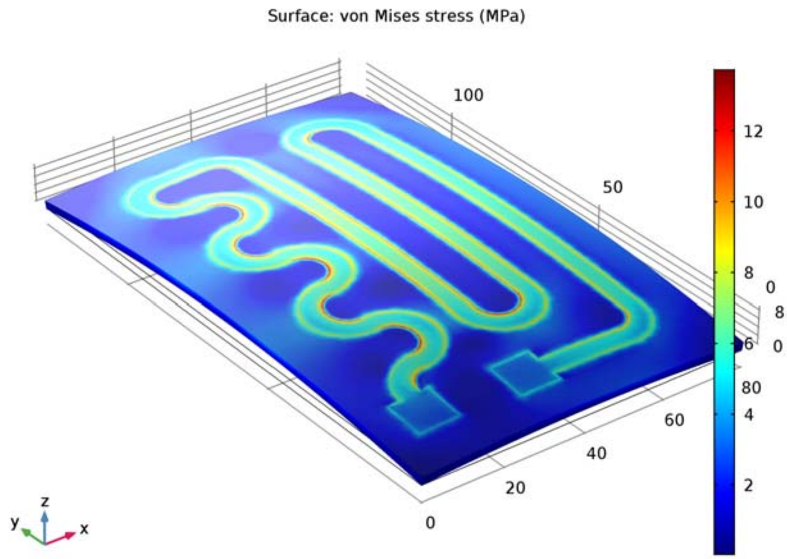


Figure 5: The thermally induced von Mises effective stress plotted with the deformation.

The highest effective stress, approximately 13 MPa, occurs at the inner corners of the curves of the Nichrome circuit. The yield stress for high quality glass is roughly 250 MPa, and for Nichrome it is 360 MPa. This means that the individual objects remain structurally intact for the simulated heating power loads.

You must also consider stresses in the interface between the resistive layer and the glass plate. Assume that the yield stress of the surface adhesion in the interface is in the region of 50 MPa—a value significantly lower than the yield stresses of the other materials in the device. If the effective stress increases above this value, the resistive layer locally detaches from the glass. Once it has detached, heat transfer is locally impeded, which can lead to overheating of the resistive layer and eventually cause the device to fail.

Figure 6 displays the effective forces acting on the adhesive layer during heater operation. As the figure shows, the device experiences a maximum interfacial stress that is an order of magnitude smaller than the yield stress. This means that the device are OK in terms of adhesive stress.

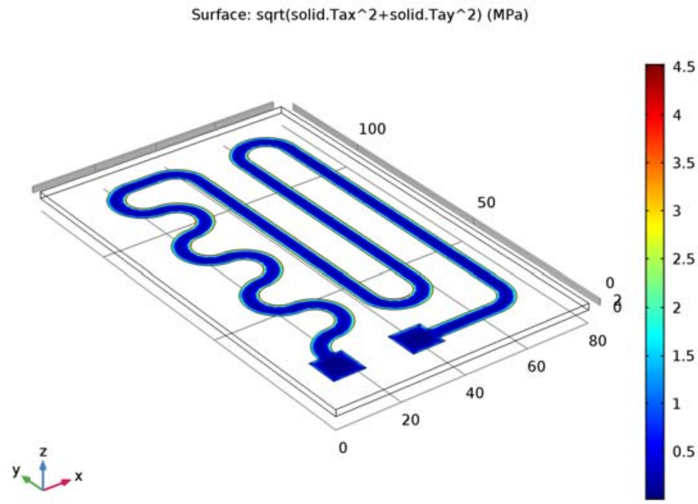


Figure 6: The effective forces in the interface between the resistive layer and the glass plate.

Finally study the device's deflections, shown in Figure 7.

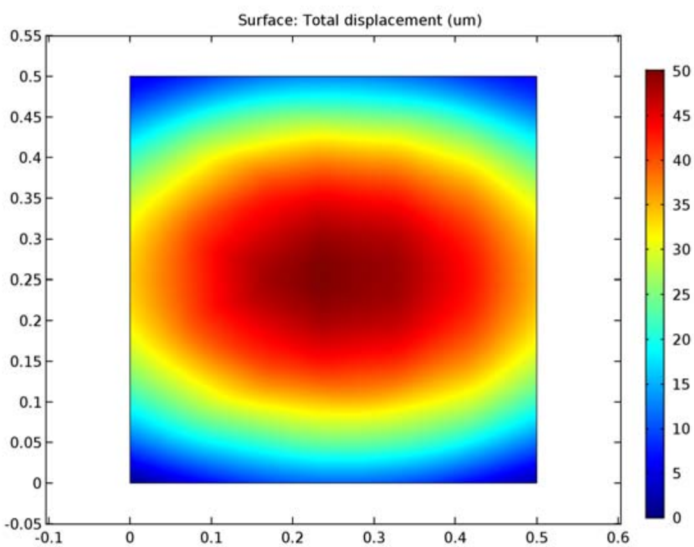


Figure 7: Total displacement on the fluid side of the glass plate.

The maximum displacement, located at the center of the plate, is approximately 50 µm. For high-precision applications, such as semiconductor processing, this might be a significant value that limits the device's operating temperature.

Application Library path: `Structural_Mechanics_Module/Thermal-Structure_Interaction/heating_circuit`

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>Thermal Stress**.

3 Click **Add**.

4 In the **Select Physics** tree, select **AC/DC>Electric Currents, Shell (ecs)**.

5 Click **Add**.

6 In the **Select Physics** tree, select **Structural Mechanics>Membrane (mbrn)**.

7 Click **Add**.

8 Click **Study**.

9 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.

10 Click **Done**.

GEOMETRY 1

The **Thermal Stress** interface includes **Heat Transfer in Solids** and **Solid Mechanics**. In the volume, these two interfaces solve for temperature and displacement, respectively. In the shell representing the circuit, the temperature, the electrical potential and displacement are solved by **Heat Transfer In Solids**, **Electric Currents, Shell**, and **Membrane** interfaces, respectively.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, 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	12[V]	12 V	Input voltage
d_layer	10[um]	1E-5 m	Layer thickness
sigma_silver	6.3e7[S/m]	6.3E7 S/m	Electric conductivity of silver
sigma_nichrome	9.3e5[S/m]	9.3E5 S/m	Electric conductivity of Nichrome
T_air	20[degC]	293.2 K	Air temperature
h_air	5[W/(m^2*K)]	5 W/(m²·K)	Heat transfer film coefficient, air
T_fluid	353[K]	353 K	Fluid temperature
h_fluid	20[W/(m^2*K)]	20 W/(m²·K)	Heat transfer film coefficient, fluid

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**.

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 80.

4 In the **Depth** text field, type 130.

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

Work Plane 1 (wp1)

1 Right-click **Block 1 (blk1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Work Plane**.

3 In the **Settings** window for Work Plane, locate the **Plane Definition** section.

4 In the **z-coordinate** text field, type `2`.

5 Click **Show Work Plane**.

Plane Geometry

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

Square 1 (sq1)

1 On the **Work Plane** 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 `10`.

4 Locate the **Position** section. In the **xw** text field, type `7`.

5 In the **yw** text field, type `10`.

6 Right-click **Square 1 (sq1)** and choose **Build Selected**.

Square 2 (sq2)

1 Right-click **Square 1 (sq1)** and choose **Duplicate**.

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

3 In the **xw** text field, type `30`.

4 In the **yw** text field, type `8`.

Polygon 1 (pol1)

1 Right-click **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)>Plane Geometry>Square 2 (sq2)** and choose **Build Selected**.

2 On the **Work Plane** toolbar, click **Primitives** and choose **Polygon**.

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

4 From the **Data source** list, choose **File**.

5 Click **Browse**.

6 Browse to the application's Application Libraries folder and double-click the file `heating_circuit_polygon.txt`.

Fillet 1 (fil1)

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

2 On the **Work Plane** toolbar, click **Fillet**.

3 On the object **pol1**, select Points 2–8, 23–29, 34, 36, 37, 41, and 42 only.

4 In the **Settings** window for Fillet, locate the **Radius** section.

5 In the **Radius** text field, type `10`.

Fillet 2 (fil2)

1 Right-click **Fillet 1 (fil1)** and choose **Build Selected**.

2 On the **Work Plane** toolbar, click **Fillet**.

3 On the object **fil1**, select Points 6–12, 26–31, 37, 40, 43, 46, 49, and 50 only.

4 In the **Settings** window for Fillet, locate the **Radius** section.

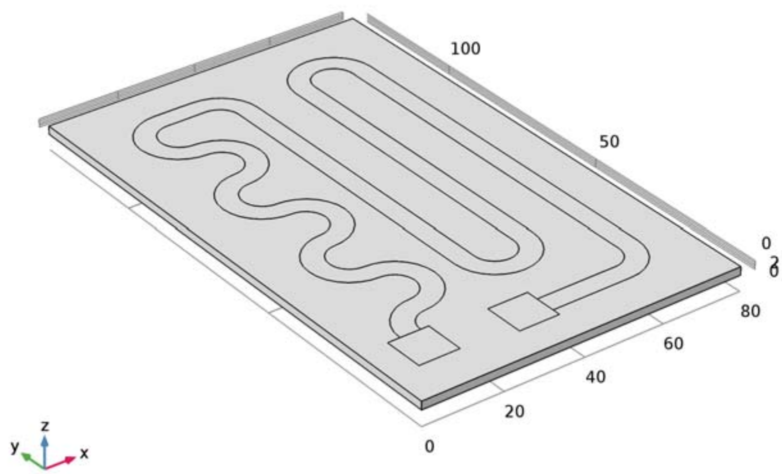
5 In the **Radius** text field, type `5`.

6 On the **Work Plane** toolbar, click **Build All**.

Form Union (fin)

1 On the **Home** toolbar, click **Build All**.

The geometry should look like the figure below.



DEFINITIONS

Add a selection that you can use later when applying boundary conditions and shell physics settings.

Explicit 1

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

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

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

4 Select Boundaries 6–8 only.

Before creating the materials for use in this model, it is a good idea to specify which boundaries are to be modeled as conducting shells. Using this information, COMSOL Multiphysics can detect which material properties are needed.

HEAT TRANSFER IN SOLIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.

Thin Layer 1

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

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

3 From the **Selection** list, choose **Circuit**.

4 Locate the **Thin Layer** section. From the **Layer type** list, choose **Thermally thin approximation**.

5 In the d_s text field, type `d_layer`.

ELECTRIC CURRENTS, SHELL (ECS)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electric Currents, Shell (ecs)**.

2 In the **Settings** window for Electric Currents, Shell, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Circuit**.

4 Locate the **Shell Thickness** section. In the d_s text field, type `d_layer`.

MEMBRANE (MBRN)

On the **Physics** toolbar, click **Electric Currents, Shell (ecs)** and choose **Membrane (mbrn)**.

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

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

3 From the **Selection** list, choose **Circuit**.

4 Locate the **Thickness** section. In the d text field, type `d_layer`.

5 Click to expand the **Dependent variables** section. Locate the **Dependent Variables** section. In the **Displacement field** text field, type `u`.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Membrane (mbrn)** click **Linear Elastic Material 1**.

Thermal Expansion 1

1 On the **Physics** toolbar, click **Attributes** and choose **Thermal Expansion**.

2 In the **Settings** window for Thermal Expansion, locate the **Model Inputs** section.

3 From the T list, choose **Temperature (ht)**.

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>Silica glass**.

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

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

MATERIALS

Silica glass (mat1)

Now set up the materials.

Material 2 (mat2)

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 `Silver` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Circuit**.

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

Property	**Name**	**Value**	**Unit**	**Property group**
Thermal conductivity	k	420	W/(m·K)	Basic
Density	rho	10500	kg/m³	Basic
Heat capacity at constant pressure	Cp	230	J/(kg·K)	Basic
Electrical conductivity	sigma	`sigma_silver`	S/m	Basic

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	1	1	Basic
Young's modulus	E	83e9	Pa	Basic
Poisson's ratio	nu	0.37	1	Basic
Coefficient of thermal expansion	alpha	18.9e-6	1/K	Basic

Material 3 (mat3)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for Material, type `Nichrome` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 7 only.

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

Property	Name	Value	Unit	Property group
Thermal conductivity	k	15	W/(m·K)	Basic
Density	rho	9000	kg/m³	Basic
Heat capacity at constant pressure	Cp	20	J/(kg·K)	Basic
Electrical conductivity	sigma	sigma_nichrome	S/m	Basic
Relative permittivity	epsilonr	1	1	Basic
Young's modulus	E	213e9	Pa	Basic
Poisson's ratio	nu	0.33	1	Basic
Coefficient of thermal expansion	alpha	10e-6	1/K	Basic

With the materials defined, set up the remaining physics of the model. In the next section, the resistive loss within the circuit is defined as a heat source for the thermal stress physics. The resistive loss is calculated automatically within the **Electric Currents, Shell** physics interface. Add the coupling feature **Boundary Electromagnetic Heat Source** to take the resistive loss into account.

MULTIPHYSICS

Boundary Electromagnetic Heat Source 1 (bemh1)

1 On the **Physics** toolbar, click **Multiphysics** and choose **Boundary>Boundary Electromagnetic Heat Source**.
2 In the **Settings** window for Boundary Electromagnetic Heat Source, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Circuit**.

HEAT TRANSFER IN SOLIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.

Heat Flux 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
2 Select Boundaries 4 and 6–8 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 `h_air`.
6 In the T_{ext} text field, type `T_air`.

Heat Flux 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
2 Select Boundary 3 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 `h_fluid`.
6 In the T_{ext} text field, type `T_fluid`.

Next, add constraints to restrain the glass plate movements.

SOLID MECHANICS (SOLID)

In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

Fixed Constraint 1

1 On the **Physics** toolbar, click **Points** and choose **Fixed Constraint**.
2 Select Point 1 only.

Prescribed Displacement 1

1. On the **Physics** toolbar, click **Points** and choose **Prescribed Displacement**.
2. Select Point 63 only.
3. In the **Settings** window for Prescribed Displacement, locate the **Prescribed Displacement** section.
4. Select the **Prescribed in y direction** check box.
5. Select the **Prescribed in z direction** check box.

Prescribed Displacement 2

1. On the **Physics** toolbar, click **Points** and choose **Prescribed Displacement**.
2. Select Point 3 only.
3. In the **Settings** window for Prescribed Displacement, locate the **Prescribed Displacement** section.
4. Select the **Prescribed in z direction** check box.

Finally, add a voltage and ground.

ELECTRIC CURRENTS, SHELL (ECS)

In the **Model Builder** window, under **Component 1 (comp1)** click **Electric Currents, Shell (ecs)**.

Electric Potential 1

1. On the **Physics** toolbar, click **Edges** and choose **Electric Potential**.
2. Select Edge 10 only.
3. In the **Settings** window for Electric Potential, locate the **Electric Potential** section.
4. In the V_0 text field, type `V_in`.

Ground 1

1. On the **Physics** toolbar, click **Edges** and choose **Ground**.
2. Select Edge 43 only.

MESH 1

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

Free Triangular 1

Select Boundaries 4 and 6–8 only.

Size 1

1. Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1** and choose **Size**.

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

3 From the **Selection** list, choose **Circuit**.

4 Locate the **Element Size** section. Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `2`.

7 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

Swept 1

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

Distribution 1

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

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

3 Click **Build All**.

STUDY 1

In order to improve the solver's performance, set the segregated solver to calculate temperature, voltage and displacement separately. The best order is `V`, `T`, `u`.

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>Segregated 1** node, then click **Segregated Step 2**.

4 In the **Settings** window for Segregated Step, type `Electric potential V` in the **Label** text field.

5 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1>Segregated 1>Electric potential V** and choose **Move Up**.

6 Locate the **General** section. In the **Variables** list, select **Displacement field (Material) (comp1.u)**.

7 Under **Variables**, click **Delete**.

8 In the **Variables** list, select **Normal strain (comp1.mbrn.unn)**.

9 Under **Variables**, click **Delete**.

10 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1>Segregated 1** click **Segregated Step 3**.

11 In the **Settings** window for Segregated Step, type `Displacement u` in the **Label** text field.

12 Locate the **General** section. Under **Variables**, click **Add**.

13 In the **Add** dialog box, select **Normal strain (comp1.mbrn.unn)** in the **Variables** list.

14 Click **OK**.

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

RESULTS

Stress (solid)

The default plots show the von Mises stress including the deformation (Figure 5) and the temperature (Figure 4) on the surface of the full 3D geometry, and the electric potential and the von Mises stress on the circuit layer.

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**.

Surface

1 In the **Model Builder** window, expand the **Results>Stress (mbrn)** node, then click **Surface**.

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

3 From the **Unit** list, choose **MPa**.

4 On the **Stress (mbrn)** toolbar, click **Plot**.

Study 1/Solution 1 (sol1)

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

2 Right-click **Study 1/Solution 1 (sol1)** and choose **Duplicate**.

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 **Circuit**.

To generate Figure 3 follow the steps below.

3D Plot Group 6

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

2 In the **Settings** window for 3D Plot Group, type `Surface Losses` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1/Solution 1 (2) (sol1)**.

Surface 1

1 Right-click **Surface Losses** 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>Electric Currents, Shell> Heating and losses>ecs.Qsh - Surface loss density, electromagnetic**.

3 On the **Surface Losses** toolbar, click **Plot**.

4 Click the **Scene Light** button on the **Graphics** toolbar.

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

The following steps generate a plot of the norm of the surface traction vector in the surface plane (see Figure 6):

3D Plot Group 7

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

2 In the **Settings** window for 3D Plot Group, type `Interface Stress` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1/Solution 1 (2) (sol1)**.

Surface 1

1 Right-click **Interface Stress** and choose **Surface**.

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

3 In the **Expression** text field, type `sqrt(solid.Tax^2+solid.Tay^2)`.

4 From the **Unit** list, choose **MPa**.

5 On the **Interface Stress** toolbar, click **Plot**.

Finally, to obtain Figure 7, proceed as follows:

Surface 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Surface**.

2 Select Boundary 3 only.

2D Plot Group 8

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

2 In the **Settings** window for 2D Plot Group, type `Displacement, Bottom Boundary` in the **Label** text field.

Surface 1

1 Right-click **Displacement, Bottom Boundary** 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>Solid Mechanics>Displacement>solid.disp - Total displacement**.

3 Locate the **Expression** section. In the **Unit** field, type `um`.

4 On the **Displacement, Bottom Boundary** toolbar, click **Plot**.

Derived Values

To calculate the values for the total generated heat and the integrated heat flux on the fluid side, perform a boundary integration:

Surface Integration 1

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

2 Select Boundary 3 only.

3 In the **Settings** window for Surface Integration, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Heat Transfer in Solids>Boundary fluxes>ht.q0 - Inward heat flux**.

4 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The result should be close to 8.5 W.

RESULTS

Surface Integration 2

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 **Circuit**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Electric Currents, Shell>Heating and losses>ecs.Qsh - Surface loss density, electromagnetic**.

5 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The result should be close to 13.8 W.

Created in COMSOL Multiphysics 5.2a

Assembly with a Hinge

Introduction

In mechanical assemblies, parts are sometimes connected so that they are free to move relative to each other in one or more degrees of freedom. Examples of such connections are ball joints, hinges, and different types of bearings. If the details of the connection are not the subjects of the analysis, it is often possible to model the connection using the Rigid Connector feature in COMSOL Multiphysics.

The current example illustrates how to model a barrel hinge connecting two solid objects in an assembly.

Model Definition

Figure 1 shows the model geometry.

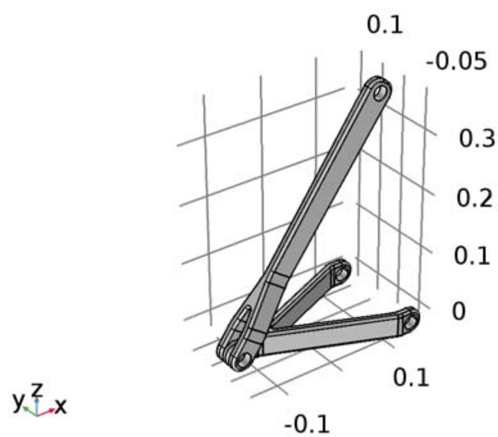


Figure 1: Model geometry.

The two parts of the assembly are connected through a barrel hinge that allows relative rotation only along the axis of the pin hole. All other degrees of freedom are common between the two parts.

The two holes of the forked bottom part are bolted, and can be considered as fully constrained.

The pin hole of the top part is constrained in the x direction so that it can slide in the y-z plane.

A force of 1 kN is applied in z direction at a 10 cm distance in the negative y direction from the center of the upper pin hole. The offset of the load thus introduces both tension and bending of the member.

Results and Discussion

The default plot shown in Figure 2 shows the von Mises stress in the model. You can see the bending of the top part due to the offset of the load. You can also see the stress that is transmitted around the hinge.

In Figure 3 the color gradient in the x displacement indicates that the lower part of the assembly undergoes bending around y axis. With a constraint rotation in y direction bending from one part would progress to the second part. The upper part however shows a fairly constant displacement through the height, which indicates that is not being bent in y direction and thus it has a free rotation in y direction in the connection point.

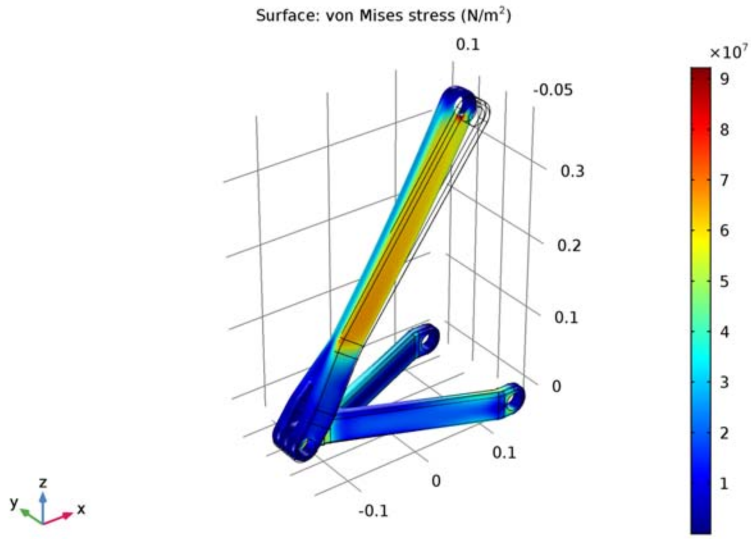


Figure 2: von Mises stress distribution in the hinge assembly.

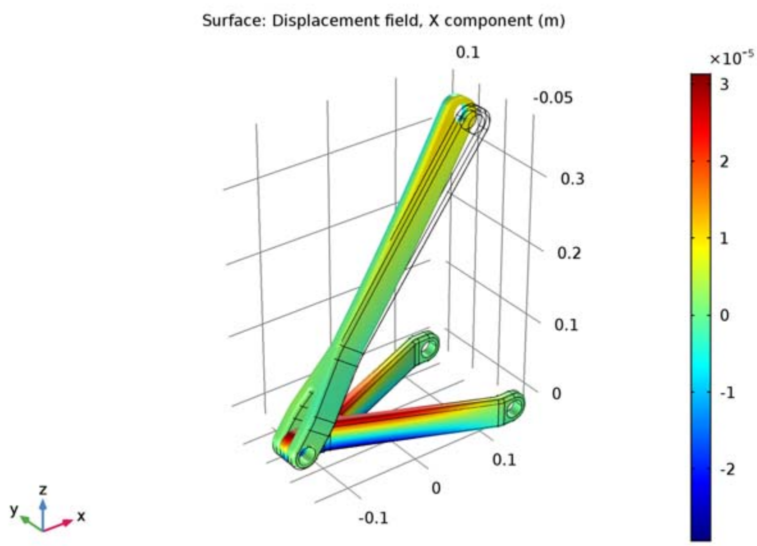


Figure 3: x-displacement.

Notes About the COMSOL Implementation

The approach when modeling mechanism is to attach rigid connectors to both parts, make sure that they have a common center of rotation, and then couple relevant degrees of freedom between them in order to obtain the desired function.

When you model a hinge, all translations and two rotations should be equal in the two parts. As in this case the displacement and the rotation in the hinge remain small, the procedure simply consists of linking the displacement in all directions as well as the rotation around the x and z directions. You connect the displacements of the rigid connectors directly using the prescribed displacement setting available in the rigid connector feature. In order to constrain the rotation directions independently, you need however to add two global constraints, one for each rotational degree of freedom. The rigid connector uses a quaternion representation of the rotation. For details, see the description in the *Structural Mechanics User's Guide*. In this specific example both the b and the d rotation variables are coupled between the two rigid connectors, because they directly correspond to the intended rotation angles for small rotations.

Another rigid connector is also used for applying the force to the upper pin hole.

Application Library path: `Structural_Mechanics_Module/`
`Connectors_and_Mechanisms/hinge_assembly`

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 On the **Home** toolbar, 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	1e3[N]	1000 N	Applied load

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 application's Application Libraries folder and double-click the file `hinge_assembly.mphbin`.

5 Click **Import**.

Form Union (fin)

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

2 In the **Settings** window for Form Union/Assembly, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Clear the **Create pairs** check box.

5 Right-click **Component 1 (comp1)>Geometry 1>Form Union (fin)** and choose **Build Selected**.

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.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

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

2 Select Boundaries 133–136 only.

Rigid Connector 1

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

2 Select Boundaries 67 and 68 only.

3 In the **Settings** window for Rigid Connector, locate the **Prescribed Displacement at Center of Rotation** section.

4 Select the **Prescribed in x direction** check box.

Applied Force 1

1 Right-click **Rigid Connector 1** and choose **Applied Force**.

2 In the **Settings** window for Applied Force, locate the **Location** section.

3 From the list, choose **User defined (absolute)**.

The center of rotation for a rigid connector is available in the variables `xcX_rig1`, `xcY_rig1` and `xcZ_rig1`. The default position is the center of gravity of the attached boundaries, which in this case will be the center of the hole.

4 Specify the $\mathbf{X}_p$ vector as

solid.xcX_rig1	X
solid.xcY_rig1-0.1	Y
solid.xcZ_rig1	Z

5 Specify the $\mathbf{F}$ vector as

0	x
0	y
F	z

Rigid Connector 2

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

2 Select Boundaries 75 and 76 only.

Rigid Connector 3

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

2 Select Boundaries 16–19 only.

Set the translations of this rigid connector equal to the translations of the second rigid connector.

3 In the **Settings** window for Rigid Connector, locate the **Prescribed Displacement at Center of Rotation** section.

4 Select the **Prescribed in x direction** check box.

5 In the u_{0x} text field, type `solid.rig2.u`.

6 Select the **Prescribed in y direction** check box.

7 In the u_{0y} text field, type `solid.rig2.v`.

8 Select the **Prescribed in z direction** check box.

9 In the u_{0z} text field, type `solid.rig2.w`.

10 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

Connect the rotations around the x and z directions, using the quaternion degrees of freedom 'b' and 'd'.

Global Constraint 1

1 On the **Physics** toolbar, click **Global** and choose **Global Constraint**.

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

3 In the **Constraint expression** text field, type `comp1.solid.rig2.b-comp1.solid.rig3.b`.

Global Constraint 2

1 On the **Physics** toolbar, click **Global** and choose **Global Constraint**.

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

3 In the **Constraint expression** text field, type `comp1.solid.rig2.d-comp1.solid.rig3.d`.

MESH 1

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

Free Tetrahedral 1

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

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 Select Boundaries 41, 42, and 53 only.

4 Locate the **Element Size** section. Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `0.002`.

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

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

RESULTS

Stress (solid)

The default plot shows the von Mises stress distribution on the surface of the assembly. Compare with Figure 2.

Finally, reproduce the x-displacement plot shown in Figure 3 with the following steps:

1 In the **Model Builder** window, under **Results** right-click **Stress (solid)** and choose **Duplicate**.

Surface 1

1 In the **Model Builder** window, expand the **Stress (solid) 1** 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)>u - Displacement field, X component**.

3 On the **Stress (solid) 1** toolbar, click **Plot**.

Stress (solid) 1

1 In the **Model Builder** window, under **Results** click **Stress (solid) 1**.

2 In the **Settings** window for 3D Plot Group, type `x-Displacement` in the **Label** text field.

Vibrations of an Impeller

Introduction

This tutorial model demonstrates the use of dynamic cyclic symmetry with postprocessing on the full geometry. A 3D impeller with eight identical blades can be divided into eight sectors of symmetry. The model computes the fundamental frequencies for the full impeller geometry and compares them to the values computed for a single sector with the cyclic symmetry boundary conditions applied on two sector boundaries. It also demonstrates how to set up a frequency response analysis for one sector of symmetry, and how to postprocess the results into the full geometry by using the sector data sets. The results for one sector are in very good agreement with the computations on the full geometry, while both the computational time and memory requirements are significantly reduced.

Model Definition

Figure 1 shows the impeller geometry. The problem is solved using the Cartesian coordinate system in 3D.

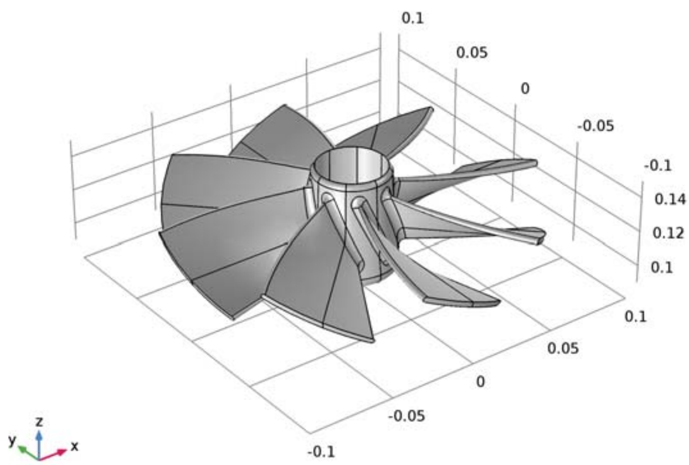


Figure 1: Impeller geometry.

The geometry can be divided into eight identical parts, each represented by a sector with an angle $\theta = \pi/4$ with respect to rotation around the z-axis; see Figure 2.

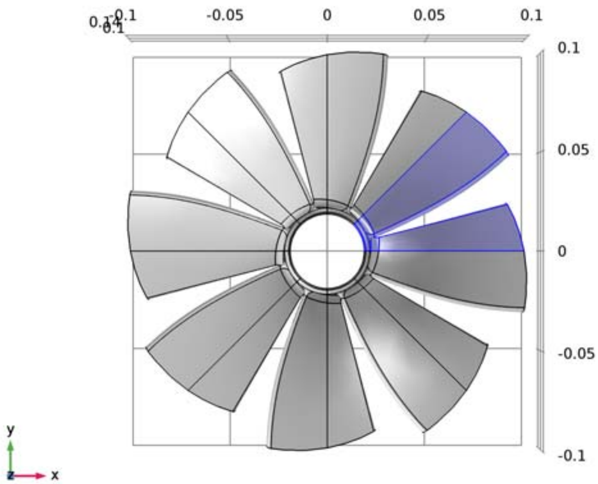


Figure 2: Sector of periodicity.

The impeller is made of aluminum, and is supposed to be mounted on a shaft. The mounting boundary is modeled via a fixed constraint, and all possible effects of the shaft rotation are neglected.

The analysis is based on the Floquet theory which can be applied to the problem of small-amplitude vibrations of spatially periodic structures, Ref. 1. This includes the case of cyclic symmetry studied in this example.

For an eigenfrequency study, one can show that all the eigenmodes of the full problem can be found by performing the analysis on one sector of symmetry only and imposing the cyclic symmetry of the eigenmodes with an angle of periodicity $\varphi = m\theta$, where the cyclic symmetry mode number m can vary from 0 to $N/2$, with N being the total number of sectors so that $\theta = 2\pi/N$.

Results and Discussion

In the first part of the analysis, you perform an eigenfrequency analysis of a single sector of periodicity, and then of the full geometry. A sweep over all required values of the cyclic symmetry parameter recovers all the eigenfrequencies of the full model with decent accuracy. See the Modeling Instructions section for in-detail comparison of the results and discussion of the performance gains.

In the second part, you perform a frequency-response analysis. Again, first of the sector of periodicity, and then of the full impeller geometry. The excitation is a pressure load applied to all free boundaries of the impeller. You enter it as a normal component of the boundary load using the expression

$$F_n = -p_0 \exp[-jm\,\mathrm{atan}(Y/X)]$$

using the magnitude of $p_0 = 10^4\ \mathrm{Pa}$ and cyclic symmetry parameter $m = 3$. The excitation frequency is 200 Hz. Figure 3 and Figure 4 show very good agreement between the results computed on the full and reduced geometry.

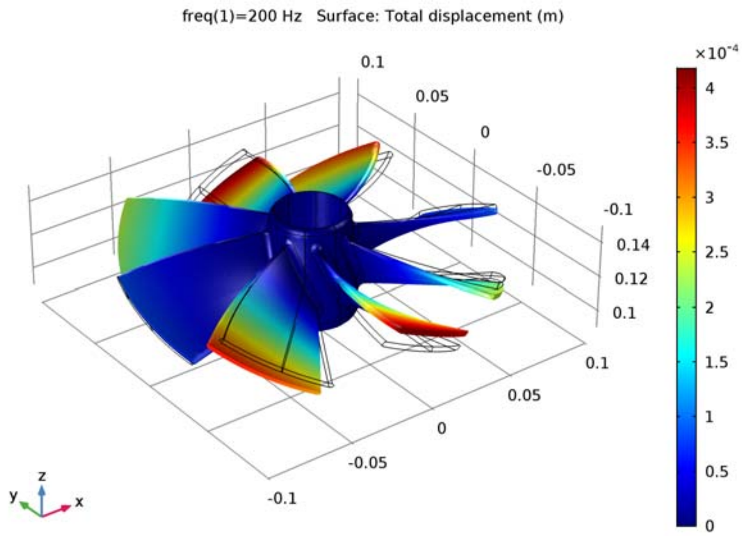


Figure 3: Frequency response computed on the sector of periodicity only, and then visualized over the full geometry.

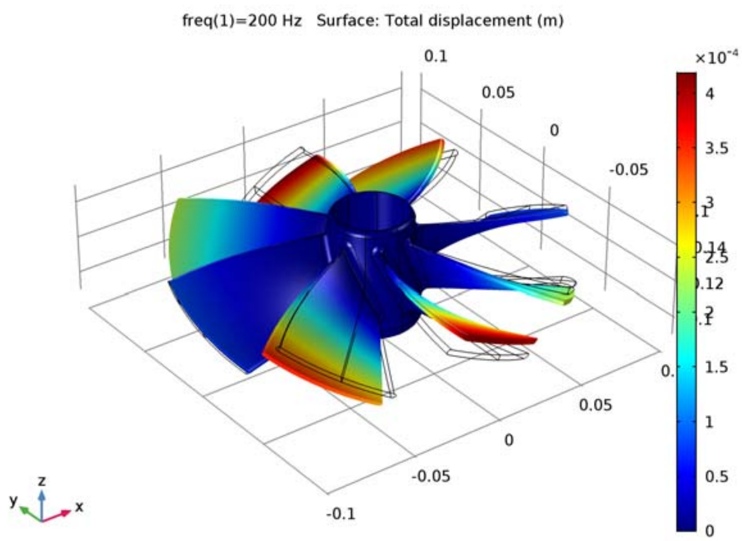


Figure 4: Frequency response computed for the full geometry.

MESHING

You use an unstructured mesh with the same size of the mesh elements for both calculations on one sector of symmetry and on the full geometry, see Figure 5. This helps to compare the results for this tutorial model. In practice, the mesh used for computations on the sector could be much finer, so that the results obtained via such geometry reduction would provide significantly better resolution of the results under the same memory requirements as for the full geometry (with a coarser mesh).

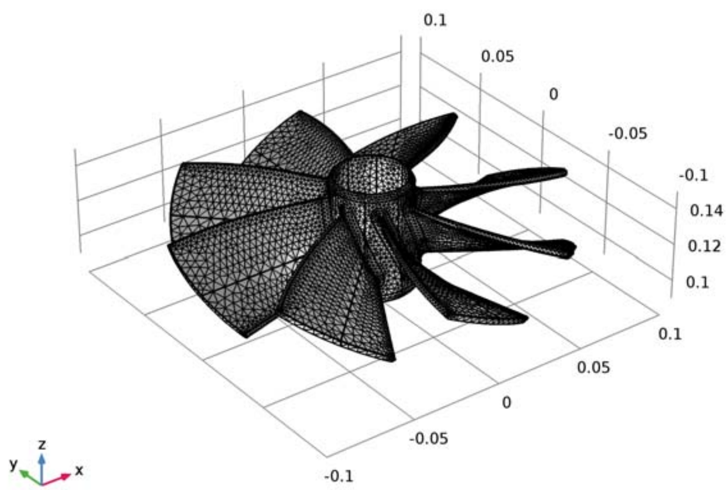


Figure 5: Meshed geometry.

CYCLIC SYMMETRY CONDITIONS AND POSTPROCESSING

To set up the cyclic symmetry conditions, you use the predefined functionality available in COMSOL Multiphysics within the Solid Mechanics interface under the Periodic Condition boundary feature. This imposes the proper boundary coupling condition on the sector boundaries.

You visualize the results computed for one sector over the full geometry by making use of the predefined type of derived dataset called Sector 3D, which is available under the Results in COMSOL Multiphysics.

Reference

1. B. Lalanne and M. Touratier, "Aeroelastic Vibrations and Stability in Cyclic Symmetric Domains," *Int. J. Rotating Machinery*, vol. 6, no. 6, pp 445–452, 2000.

Application Library path: `Structural_Mechanics_Module/Dynamics_and_Vibration/impeller`

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**.

GEOMETRY 1

Import the pre-built geometry for the impeller from a file.

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 application's Application Libraries folder and double-click the file `impeller.mphbin`.
5. Click **Import**.
6. Click the **Go to Default 3D View** button on the **Graphics** toolbar.
7. On the **Home** toolbar, click **Build All**.

8 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

9 Click the **Go to XY View** button on the **Graphics** toolbar.

The complete geometry should look similar to that shown in Figure 1 and Figure 2.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, 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
N	8	8	Number of sectors
theta	2*pi/N	0.7854	Unit sector angle
m	3	3	Azimuthal mode number
p0	1e4[Pa]	1E4 Pa	Load magnitude

COMPONENT 1 (COMP1)

Add a second Solid Mechanics interface to use for the computations on the reduced geometry only.

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

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

SOLID MECHANICS 2 (SOLID2)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics 2 (solid2)**.

2 Select Domain 8 only.

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>Aluminum**.

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

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

SOLID MECHANICS (SOLID)

In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

Fixed Constraint 1

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

2 Select Boundaries 54, 55, 67, 68, 85, 87, 109, and 113 only.

SOLID MECHANICS 2 (SOLID2)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics 2 (solid2)**.

2 On the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

3 Select Boundary 113 only.

For a reduced geometry, you set up the Cyclic symmetry condition on the sector boundaries.

Periodic Condition 1

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

2 Select Boundaries 112 and 134 only.

3 In the **Settings** window for Periodic Condition, locate the **Periodicity Settings** section.

4 From the **Type of periodicity** list, choose **Cyclic symmetry**.

5 In the m text field, type `m`.

Destination Selection 1

1 Right-click **Periodic Condition 1** and choose **Destination Selection**.

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

3 Click **Clear Selection**.

4 Select Boundary 112 only.

Follow these steps to create a free unstructured mesh that will be identical in all eight sectors.

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**.

Free Triangular 1

1 Right-click **Component 1 (comp1)>Mesh 1** and choose **More Operations>Free Triangular**.

2 Select Boundary 134 only.

Copy Face 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Copy Face**.

2 Select Boundary 134 only.

3 In the **Settings** window for Copy Face, locate the **Destination Boundaries** section.

4 Select the **Active** toggle button.

5 Select Boundary 112 only.

Free Tetrahedral 1

1 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 Select Domain 8 only.

5 Click **Build Selected**.

Copy Domain 1

1 Right-click **Mesh 1** and choose **More Operations>Copy Domain**.

2 Select Domain 8 only.

3 In the **Settings** window for Copy Domain, locate the **Destination Domains** section.

4 Select the **Active** toggle button.

5 Select Domains 1–5 only.

6 Click **Build Selected**.

Copy Domain 2

1 Right-click **Mesh 1** and choose **More Operations>Copy Domain**.

2 Select Domain 1 only.

3 In the **Settings** window for Copy Domain, locate the **Destination Domains** section.

4 Select the **Active** toggle button.

5 Select Domains 6 and 7 only.

6 In the **Model Builder** window, click **Mesh 1**.

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

8 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

The resulting mesh should look similar to that shown in Figure 5.

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 **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for the **Solid Mechanics 2** interface.

4 Locate the **Study Settings** section. Select the **Desired number of eigenfrequencies** check box.

5 In the associated text field, type `32`.

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)>Dependent Variables 1** node, then click **Displacement field (Material) (comp1.u2)**.

4 In the **Settings** window for Field, locate the **General** section.

5 Clear the **Store in output** check box.

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

ADD STUDY

1 On the **Study** 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>Eigenfrequency**.

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 **Study** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 2** 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 `4`.

To capture all possible eigenfrequencies, set up a sweep over the cyclic symmetry mode number m in the range from 0 to $N/2$, where N is the total number of sectors

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
m		

5 Click **Range**.

6 In the **Range** dialog box, type `0` in the **Start** text field.

7 In the **Stop** text field, type `N/2`.

8 In the **Step** text field, type `1`.

9 Click **Add**.

Solution 2 (sol2)

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** node, then click **Displacement field (Material) (comp1.u)**.

4 In the **Settings** window for Field, locate the **General** section.

5 Clear the **Store in output** check box.

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

RESULTS

Mode Shape (solid2)

Note a nearly eight times reduction in the number of degrees of freedom, and thus of the memory required to compute the reduced model.

However, the computational time is approximately the same because you need to perform a sweep over all values of the periodicity parameter.

Add a new dataset to visualize over the full geometry the eigenmode shape for the reduced model.

Sector 3D 1

1. On the **Results** toolbar, click **More Data Sets** and choose **Sector 3D**.
2. In the **Settings** window for Sector 3D, locate the **Data** section.
3. From the **Data set** list, choose **Study 2/Parametric Solutions 1 (sol3)**.
4. Locate the **Symmetry** section. In the **Number of sectors** text field, type `8`.
5. Click to expand the **Advanced** section. In the **Azimuthal mode number** text field, type `4`.

Mode Shape (solid2)

1. In the **Model Builder** window, under **Results** click **Mode Shape (solid2)**.
2. In the **Settings** window for 3D Plot Group, locate the **Data** section.
3. From the **Data set** list, choose **Sector 3D 1**.
4. On the **Mode Shape (solid2)** toolbar, click **Plot**.

Derived Values

Collect all the computed eigenfrequencies into tables.

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
`freq`	`1/s`	`Frequency`

4. Click **New Table**.

TABLE

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

   Note that the eigenfrequencies for the full geometry present groups of values very close to each other, eight frequencies in each group. This shows that vibrations of each of the eight blades of the impeller are only weakly coupled to the remaining structure, which is because the central part has significantly larger effective bending stiffness compared to that of each blade. Hence, the eigenfrequencies in each group are close to the natural frequencies of a single blade (if computed assuming a fully fixed footing).

RESULTS

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 2/Parametric Solutions 1 (sol3)**.
4 From the **Table columns** list, choose **Inner solutions**.
5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
freq	1/s	Frequency

6 Click **New Table**.

TABLE

1 Go to the **Table** window.

Compare the values of the egenfrequencies computed by using the periodicity conditions to those found for the full geometry.

Next, add a load representing a periodic pressure perturbation in the stream, and thus on all the external boundaries of the impeller.

SOLID MECHANICS (SOLID)

In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

Boundary Load 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
2 In the **Settings** window for Boundary Load, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **All boundaries**.
4 Select Boundaries 1–3, 5–18, 20, 21, 23–53, 56–66, 69–75, 77–84, 88–107, 110, 111, 114–133, and 135–152 only.

You can do this by selecting all boundaries first, and then removing from the selection all the constraint boundaries and all the internal boundaries of the periodicity sectors.

5 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.

6 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

0	t1
0	t2
`-p0*exp(-j*m*atan2(Y,X))`	n

SOLID MECHANICS 2 (SOLID2)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics 2 (solid2)**.

2 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

3 Select Boundaries 114, 115, 119, 123, 125, 130, 131, 133, 136, 137, 142–146, 150, and 151 only.

4 In the **Settings** window for Boundary Load, locate the **Coordinate System Selection** section.

5 From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.

6 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

0	t1
0	t2
`-p0*exp(-j*m*atan2(Y,X))`	n

ROOT

Set up and perform the frequency-response analysis, first for the full model, and then for a sector of periodicity.

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>Frequency Domain**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Solid Mechanics 2 (solid2)** interface.

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

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

STUDY 3

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 3** 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 `200`.

   Switch off the generation of the default plot as that would be a plot of the von Mises stress, while you will be comparing the full and reduced structure responses in terms of displacements.

4 In the **Model Builder** window, click **Study 3**.
5 In the **Settings** window for Study, locate the **Study Settings** section.
6 Clear the **Generate default plots** check box.

Solution 9 (sol9)

1 On the **Study** toolbar, click **Show Default Solver**.
2 In the **Model Builder** window, expand the **Solution 9 (sol9)** node.
3 In the **Model Builder** window, expand the **Study 3>Solver Configurations>Solution 9 (sol9)>Dependent Variables 1** node, then click **Displacement field (Material) (comp1.u2)**.
4 In the **Settings** window for Field, locate the **General** section.
5 Clear the **Store in output** check box.
6 On the **Study** toolbar, click **Compute**.

RESULTS

3D Plot Group 3

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2 In the **Settings** window for 3D Plot Group, type `Displacement (solid)` in the **Label** text field.
3 Locate the **Data** section. From the **Data set** list, choose **Study 3/Solution 9 (sol9)**.
4 Right-click **Displacement (solid)** and choose **Surface**.

Surface 1

In the **Model Builder** window, under **Results>Displacement (solid)** right-click **Surface 1** and choose **Deformation**.

Deformation 1

1 In the **Settings** window for Deformation, locate the **Scale** section.

2 Select the **Scale factor** check box.

3 In the associated text field, type 25.

4 On the **Displacement (solid)** toolbar, click **Plot**.

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>Frequency Domain**.

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 4

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 4** 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 200.

4 In the **Model Builder** window, click **Study 4**.

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

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

Solution 10 (sol10)

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

2 In the **Model Builder** window, expand the **Solution 10 (sol10)** node.

3 In the **Model Builder** window, expand the **Study 4>Solver Configurations>Solution 10 (sol10)>Dependent Variables 1** node, then click **Displacement field (Material) (comp1.u)**.

4 In the **Settings** window for Field, locate the **General** section.

5 Clear the **Store in output** check box.

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

For a frequency-response analysis, use of the reduced geometry gives significant gains in both the memory required and computational time needed.

RESULTS

Set up a displacement plot for the reduced geometry and compare it to that for the full geometry.

Sector 3D 2

1 On the **Results** toolbar, click **More Data Sets** and choose **Sector 3D**.

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

3 From the **Data set** list, choose **Study 4/Solution 10 (sol10)**.

4 Locate the **Symmetry** section. In the **Number of sectors** text field, type `8`.

5 Locate the **Advanced** section. In the **Azimuthal mode number** text field, type `3`.

3D Plot Group 4

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

2 In the **Settings** window for 3D Plot Group, type `Displacement (solid2)` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Sector 3D 2**.

Surface 1

1 Right-click **Displacement (solid2)** and choose **Surface**.

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

3 In the **Expression** text field, type `solid2.disp`.

Deformation 1

1 Right-click **Results>Displacement (solid2)>Surface 1** 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 `25`.

5 Locate the **Expression** section. In the **X component** text field, type `u2`.

6 In the **Y component** text field, type `v2`.

7 In the **Z component** text field, type `w2`.

8 On the **Displacement (solid2)** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.2a

Thermal Stresses in a Layered Plate

Introduction

This example contains an analysis of the thermal stress in a layered plate. The plate consists of three layers: the coating, the substrate, and the carrier. The coating is deposited on the substrate at a temperature of 800 °C. At this temperature both the coating and the substrate are stress-free. The temperature of the plate is then lowered to 150 °C, which induces thermal stresses in the coating/substrate assembly. At this temperature the coating/substrate assembly is epoxied to a carrier plate so that the coating/substrate has initial stresses when it is bonded to the carrier. Finally, the temperature is lowered to 20 °C.

Model Definition

The plate is restrained from moving in the z direction. This makes it possible to use the plane strain approximation in the 2D Solid Mechanics interface with. The assumption is then that the z-component of the strain is zero.

This model contains only thermal loads, which are introduced into the constitutive equations according to the following equations:

$$\sigma = D\varepsilon_{\mathrm{el}} + \sigma_0 = D(\varepsilon - \varepsilon_{\mathrm{th}} - \varepsilon_0) + \sigma_0$$

and

$$\varepsilon_{\mathrm{th}} = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{xz} \end{bmatrix}_{\mathrm{th}} = \alpha_{\mathrm{vec}}(T - T_{\mathrm{ref}})$$

where σ is the stress vector, D is the elasticity matrix, ε_x, ε_y, ε_z, γ_{xy}, γ_{yz}, γ_{xz} are the strain components, α_{vec} is the coefficient of thermal expansion, T is the actual temperature, and T_{ref} is the reference temperature.

The geometry of the plate is shown in Figure 1. The top layer in the geometry is the coating, the middle layer is the substrate, and the bottom layer is the carrier.

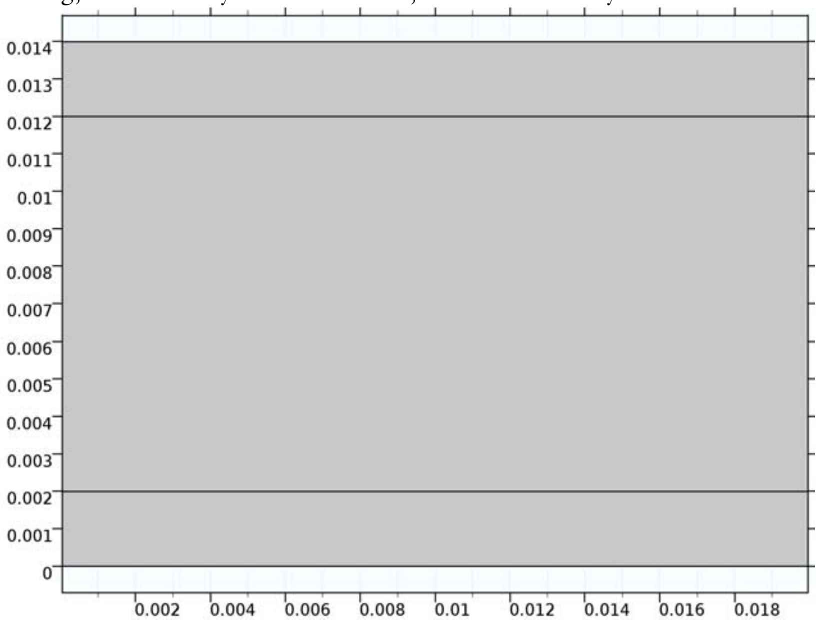


Figure 1: The plate geometry.

The analysis uses two steps:

STEP 1

In the first step you lower the temperature from 800 °C to 150 °C, which affects the coating layer and the substrate layer. The carrier layer is not active in this step.

In both steps the upper-left corner of the coating is fixed, and the upper-right corner of the coating is constrained in the *y* direction. This prevents rigid-body movements but does not affect the stress distribution.

STEP 2

In this step all three layers are active and you drop the temperature from 150 °C to room temperature, 20 °C. This step includes the initial stresses from Step 1.

Results and Discussion

Figure 2 depicts the normal stress in the *x* direction from the first analysis step. The substrate material has a higher coefficient of thermal expansion than the coating material. This means that the substrate shrinks more than the coating, causing tensile stresses in the substrate area next to the coating and compressive stresses in the coating.

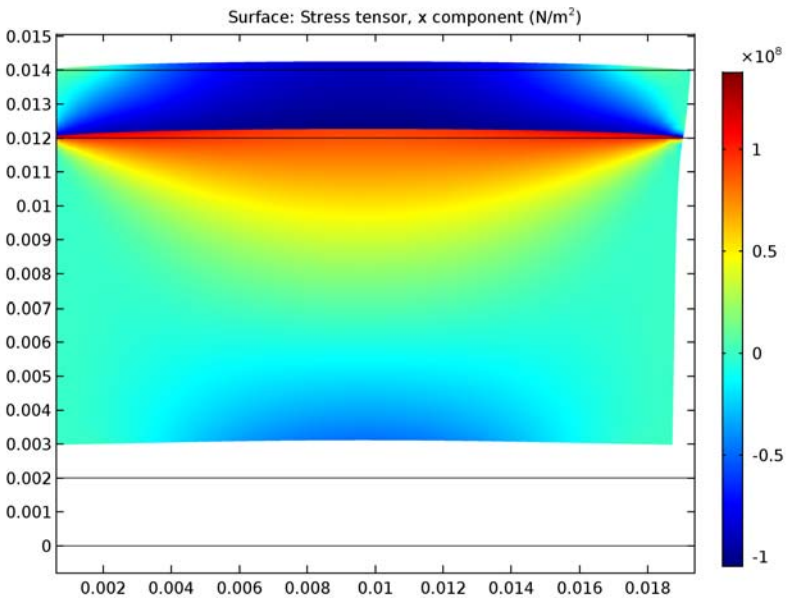


Figure 2: Normal stress in the x direction for the first analysis step.

Figure 3 shows the residual thermal *x*-stress in the final step where the temperature is lowered to 20 °C. The tensile stress levels have increased somewhat in the substrate area next to the coating, as have the compressive stress in the coating compared to the first process step. The main stress contribution is clearly the added initial stress from the first process step.

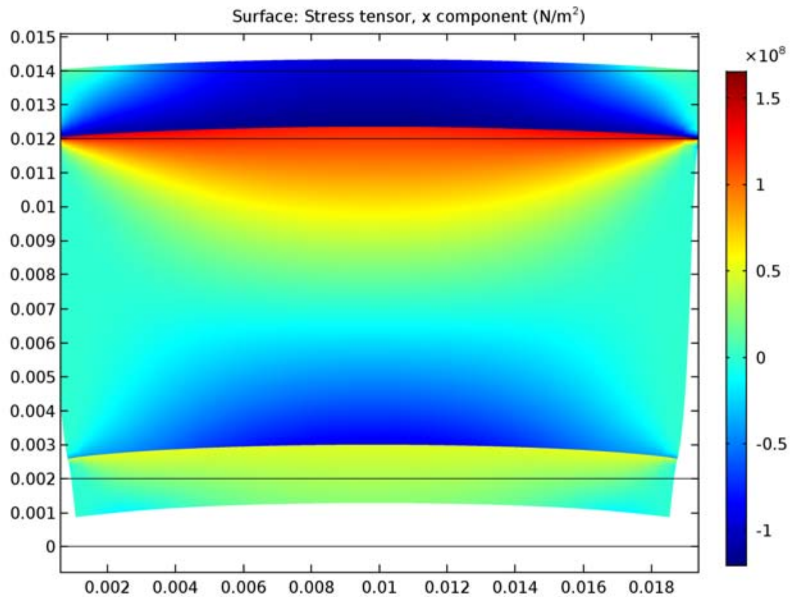


Figure 3: Residual thermal stress at room temperature.

Application Library path: `Structural_Mechanics_Module/Thermal-Structure_Interaction/layered_plate`

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)**.

Add two different solid interfaces, one for the structure before adding the carrier, and one for the complete structure.

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**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, 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
`Ttop`	`800[degC]`	1073 K	`Coating deposition temperature`
`Tbot`	`150[degC]`	423.2 K	`Temperature when the coating/ substrate is epoxied to the carrier`
`Troom`	`20[degC]`	293.2 K	`Room temperature`

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 **Height** text field, type `0.002`.

4 In the **Width** text field, type `0.02`.

5 Click **Build All Objects**.

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 `0.02`.

4 In the **Height** text field, type `0.01`.

5 Locate the **Position** section. In the **y** text field, type `0.002`.

6 Click **Build All Objects**.

Rectangle 3 (r3)

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 `0.02`.

4 In the **Height** text field, type `0.002`.

5 Locate the **Position** section. In the **y** text field, type `0.012`.

6 Click **Build All Objects**.

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

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, type `Two Layers` in the **Label** text field.

3 Select Domains 2 and 3 only.

TWO LAYERS (SOLID)

On the **Physics** toolbar, click **Solid Mechanics (solid)** and choose **Two Layers (solid)**.

Linear Elastic Material 1

In the **Model Builder** window, expand the **Component 1 (comp1)>Two Layers (solid)** node, then click **Linear Elastic Material 1**.

Thermal Expansion 1

1 On the **Physics** toolbar, click **Attributes** and choose **Thermal Expansion**.

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

3 In the T_{ref} text field, type `Ttop`.

4 Locate the **Model Inputs** section. In the T text field, type `Tbot`.

Fixed Constraint 1

1 On the **Physics** toolbar, click **Points** and choose **Fixed Constraint**.

2 Select Point 4 only.

Prescribed Displacement 1

1 On the **Physics** toolbar, click **Points** and choose **Prescribed Displacement**.

2 Select Point 8 only.

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

4 Select the **Prescribed in y direction** check box.

SOLID MECHANICS 2 (SOLID2)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics 2 (solid2)**.

2 In the **Settings** window for Solid Mechanics, type `Three Layers` in the **Label** text field.

THREE LAYERS (SOLID2)

On the **Physics** toolbar, click **Solid Mechanics 2 (solid2)** and choose **Three Layers (solid2)**.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Three Layers (solid2)** click **Linear Elastic Material 1**.

Thermal Expansion 1

1 On the **Physics** toolbar, click **Attributes** and choose **Thermal Expansion**.

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

3 In the T_{ref} text field, type `Tbot`.

4 Locate the **Model Inputs** section. In the T text field, type `Troom`.

Fixed Constraint 1

1 On the **Physics** toolbar, click **Points** and choose **Fixed Constraint**.

2 Select Point 4 only.

Prescribed Displacement 1

1 On the **Physics** toolbar, click **Points** and choose **Prescribed Displacement**.

2 Select Point 8 only.

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

4 Select the **Prescribed in y direction** check box.

Linear Elastic Material 1

Use the stresses from the two layer model as initial stresses for the three layer model.

1 In the **Model Builder** window, under **Component 1 (comp1)>Three Layers (solid2)** click **Linear Elastic Material 1**.

Initial Stress and Strain 1

1. On the **Physics** toolbar, click **Attributes** and choose **Initial Stress and Strain**.
2. Select Domains 2 and 3 only.
3. In the **Settings** window for Initial Stress and Strain, locate the **Initial Stress and Strain** section.
4. In the S_0 table, enter the following settings:

`solid.sx`	`solid.sxy`	`0`
solid.sxy	`solid.sy`	`0`
0	0	`solid.sz`

MATERIALS

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

Material 1 (mat1)

1. In the **Settings** window for Material, type `Carrier` in the **Label** text field.
2. Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Manual**.
3. Click **Clear Selection**.
4. Select Domain 1 only.
5. Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Young's modulus	E	`2.15e11`	Pa	Basic
Poisson's ratio	nu	`0.3`	1	Basic
Density	rho	`1000`	kg/m³	Basic
Coefficient of thermal expansion	alpha	`6e-6`	1/K	Basic

Material 2 (mat2)

1. Right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for Material, type `Substrate` in the **Label** text field.
3. Select Domain 2 only.

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

Property	**Name**	**Value**	**Unit**	**Property group**
Young's modulus	E	`1.3e11`	Pa	Basic
Poisson's ratio	nu	`0.28`	1	Basic
Density	rho	`1000`	kg/m³	Basic
Coefficient of thermal expansion	alpha	`3e-6`	1/K	Basic

Material 3 (mat3)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for Material, type `Coating` in the **Label** text field.

3 Select Domain 3 only.

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

Property	**Name**	**Value**	**Unit**	**Property group**
Young's modulus	E	`7e10`	Pa	Basic
Poisson's ratio	nu	`0.17`	1	Basic
Density	rho	`1000`	kg/m³	Basic
Coefficient of thermal expansion	alpha	`5e-7`	1/K	Basic

MESH 1

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

Size

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

2 From the **Predefined** list, choose **Extra fine**.

3 Click **Build All**.

STUDY 1

Add a static solution for the case with three layers.

Stationary 2

On the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

Step 2: Stationary 2

Use only one Solid Mechanics interface per solution by deactivating the other one.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** 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 **Three Layers**.

Step 2: Stationary 2

1 In the **Model Builder** window, under **Study 1** click **Step 2: Stationary 2**.

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 **Two Layers**.

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

RESULTS

Surface 1

1 In the **Model Builder** window, expand the **Stress (solid)** 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>Two Layers>Stress>Stress tensor (Spatial)>solid.sx - Stress tensor, x component**.

3 On the **Stress (solid)** toolbar, click **Plot**.

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

Surface 1

1 In the **Model Builder** window, expand the **Results>Stress (solid2)** 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>Three Layers>Stress>Stress tensor (Spatial)>solid2.sx - Stress tensor, x component**.

3 On the **Stress (solid2)** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.2a

Nonlinear Magnetostrictive Transducer

Introduction

Magnetostriction describes the change in dimensions of a material due to a change in its magnetization. This phenomenon is a manifestation of magnetoelastic coupling, which is exhibited by all magnetic materials to some extent. The effects related to magnetoelastic coupling are described by various names. The *Joule effect* describes the change in length due to a change in the magnetization state of the material. This *magnetostrictive effect* is used in transducers for applications in sonars, acoustic devices, active vibration control, position control, and fuel injection systems.

Magnetostriction has a quantum-mechanical origin. The magneto-mechanical coupling takes place at the atomic level due to spin-orbit coupling. From a system level, the material can be assumed to consist of a number of tiny ellipsoidal magnets which rotate due to the torque produced by the externally applied magnetic field. The rotation of these elemental magnets produces a dimensional change leading to free strain in the material. The strain (or magnetostriction) has a nonlinear dependence on the magnetic field and the mechanical stress in the material.

This tutorial demonstrates how to model the nonlinear response of a magnetostrictive material.

Model Definition

A typical magnetostrictive transducer shown in Figure 1 has a steel housing enclosing a drive coil. A magnetostrictive material is placed in the core that works as an actuator when a magnetic field is applied by passing a current through the drive coil.

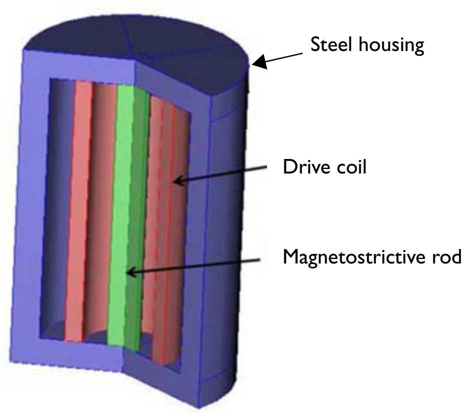


Figure 1: Sectional view of a cylindrical transducer.

Due to the symmetric nature of the geometry, the problem is solved as a 2D axisymmetric model, which leads to reduced computation time. The corresponding 2D axisymmetric geometry is shown in Figure 2.

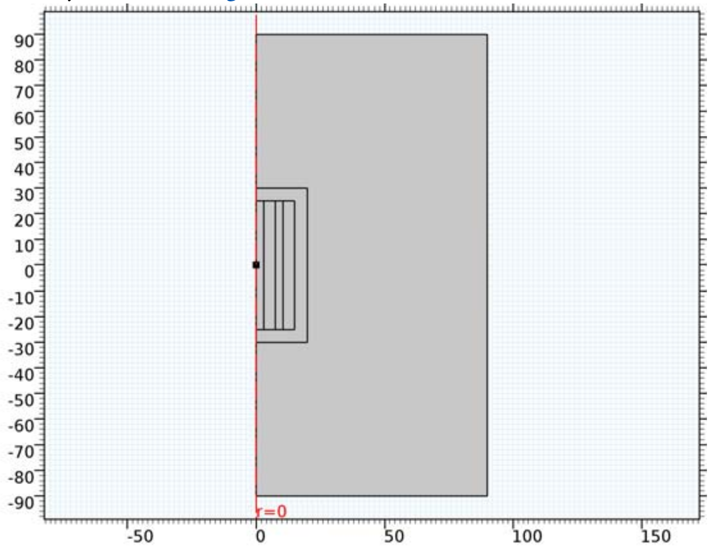


Figure 2: 2D axisymmetric view of a magnetostrictive transducer surrounded by an air domain. The geometric dimensions are in millimeters.

It is assumed that the current in the coil is DC, and hence it can be solved as a stationary problem. The first study performed considers a constant current density of 10^6 A/m^2 in the coil. A second study is set up where the current density in the drive coil is varied from 0 to 10^7 A/m^2 using the parametric sweep feature in COMSOL. The solution from this parametric sweep is then used to generate the characteristic nonlinear magnetostriction (λ) vs. magnetic field ($\boldsymbol{H}$) curve. The ramping of current density using the parametric sweep option is performed under the assumption that the current in the coil changes quasi-statically without producing any inductive effect.

NOTES ON THE MAGNETIC AND MAGNETOELASTIC PROBLEMS

An air domain is created around the transducer to realistically model the magnetic flux path. The boundaries of this air domain are magnetically insulated which ensures that flux does not diverge out of the modeling domain. An alternate technique of implementing this air domain in COMSOL involves the use of Infinite Elements. For more information on Infinite Elements, please refer to the AC/DC Module User's Guide.

The drive coil is modeled as a homogenized current carrying domain. Individual wires and their electrical conductivity are not resolved. It is assumed that the externally applied current density in the coil is known *a priori*. In a 2D axisymmetric model, the external current density is the total current through the coil divided by the longitudinal cross-section area (coil length times coil thickness). The coil can also be modeled alternately using the Multiturn Coil Domain feature available in the AC/DC Module. Please refer to the *AC/DC Module User's Guide* for more details on using this alternative technique.

Traditionally, the magnetic flux density (also called the **B**-field) is obtained as a function of the applied magnetic field (the **H**-field). Such relationship is usually called a BH curve. However, in COMSOL you need to specify the other way round while working with the built-in **Magnetic Fields** interface. The reason why you need an HB curve rather than a BH curve is that the formulation of the magnetic problem in COMSOL Multiphysics solves for the magnetic vector potential **A** whose curl yields the vector **B**-field. The **H**-field is then obtained as a function of the **B**-field. In order to avoid circular dependency, it is necessary to specify the **H**-field in terms of the **B**-field.

The steel housing used in this example is designed to create a closed magnetic flux path thereby minimizing flux leakage. The nonlinear magnetic behavior of the steel housing is modeled by using an HB curve to specify the magnetic constitutive relation in the steel housing. The nonlinear HB curve is obtained by choosing the material Soft iron without losses from AC/DC materials in COMSOL's Material Library. Incorporation of a nonlinear HB curve helps in modeling magnetic saturation effects at sufficiently high magnetic field. Furthermore, you can examine the results of the model to find out specific locations in a material where magnetic saturation has taken place whereas other regions of that material have remained unsaturated.

The stress in the magnetostrictive material is modeled as

$$S = \mathrm{C_H}[\varepsilon_{\mathrm{el}} - \varepsilon_{\mathrm{me}}(\mathbf{M})]$$

The material is assumed to be isotropic, so that the stiffness matrix $\mathrm{C_H}$ can be represented in terms of two parameters, the Young's modulus and Poisson's ration.

The magnetostrictive strain is modeled as the following quadratic isotropic function of the magnetization field **M**:

$$\varepsilon_{\mathrm{me}} = \frac{3}{2}\frac{\lambda_s}{M_s^2}\mathrm{dev}(\mathbf{M} \otimes \mathbf{M})$$

where the tensor product of two vectors is defined as

$$(\mathbf{M} \otimes \mathbf{M})_{ij} = M_i M_j$$

and λ_s is the saturation magnetostriction, which is the maximum magnetostrictive strain reached at the saturation magnetization M_s.

Note that the magnetostrictive strain is represented by a deviatoric tensor. This is because the deformation can be related to the magnetic domain rotation associated with the magnetization of the material; such process should not change the material volume.

Nonlinear magnetization in the magnetostrictive material is found from the following nonlinear relation:

$$\mathbf{M} = M_s L(|\mathbf{H}_{\mathrm{eff}}|)\frac{\mathbf{H}_{\mathrm{eff}}}{|\mathbf{H}_{\mathrm{eff}}|}$$

where L is the Langevin function

$$L = \coth\left(\frac{3\chi_m|\mathbf{H}_{\mathrm{eff}}|}{M_s}\right) - \frac{M_s}{3\chi_m|\mathbf{H}_{\mathrm{eff}}|}$$

with χ_m being the magnetic susceptibility in the initial linear region, and the effective magnetic field in the material is given by

$$\mathbf{H}_{\mathrm{eff}} = \mathbf{H} + \frac{3\lambda_s}{\mu_0 M_s^2} S_{\mathrm{ed}}\mathbf{M}$$

The second term in the above relation represents the mechanical stress contribution to the effective field, and thus to the material magnetization, which is called the *Villari effect*. The deviatoric elastic stress tensor is related to the elastic strain $\varepsilon_{\mathrm{el}}$ in the material as

$$S_{\mathrm{ed}} = \mathrm{dev}(\mathrm{C_H}\varepsilon_{\mathrm{el}})$$

In addition, the magnetization and magnetic field are related to each other and to the **B**-field by

$$\mathbf{B} = \mu_0(\mathbf{H} + \mathbf{M})$$

The material properties used to describe the magnetostrictive material are shown in .

TABLE 1: MATERIAL PROPERTIES OF THE MAGNETOSTRICTIVE MATERIAL

MATERIAL PROPERTY	VALUE	DESCRIPTION
E	60 x 10^9 Pa	Young's modulus
ν	0.45	Poisson's ratio
ρ	7870 kg/m^3	Density

TABLE 1: MATERIAL PROPERTIES OF THE MAGNETOSTRICTIVE MATERIAL

MATERIAL PROPERTY	VALUE	DESCRIPTION
σ	5.96 x 10^6 S/m	Electric conductivity
ε_r	1	Relative permittivity
λ_s	2 x 10^{-4}	Saturation magnetostriction
M_s	1.5 x 10^6 A/m	Saturation magnetization

COUPLING THE MAGNETIC AND STRUCTURAL PROBLEMS

The implementation is straightforwards as you make use of a predefined multiphysics coupling interface available in COMSOL and called Magnetostriction.

Selecting such interface in the model wizard will add **Structural Mechanics** and **Magnetic Fields** interfaces together with the corresponding multiphysics coupling feature **Magnetostriction**.

Most of the settings you need to access in order to configure the coupling can be found under the **Magnetostrictive Material** feature added under the **Structural Mechanics** interface.

Results and Discussion

The results obtained from the first study, where a constant external current density of 10^6 A/m^2 is applied to the coil. Figure 3 shows the von Mises stress in the magnetostrictive material as a surface plot. This plot indicates that the stress due to magnetostriction is uniformly zero everywhere except the region near the bottom surface of the rod due to the fixed constraint boundary condition that was applied to this end of the rod. This is because the free strain due to magnetostriction should not produce any stress unless the material is mechanically constrained. Figure 4 shows that the corresponding strain field caused by the magnetostriction is also fairly uniform in the material except at the fixed end of the rod.

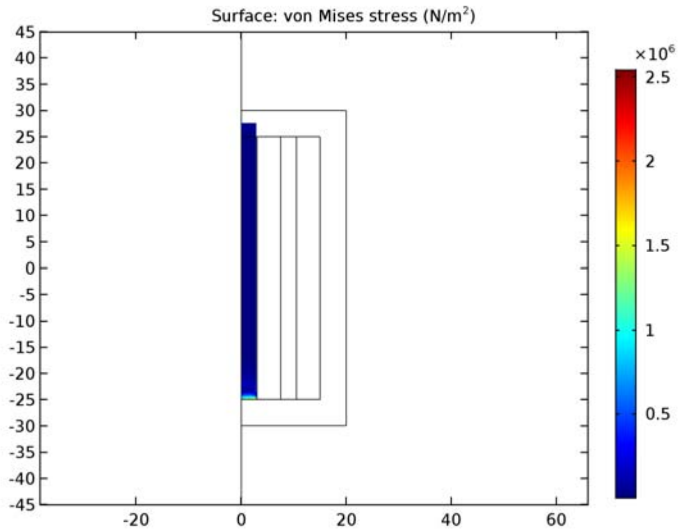


Figure 3: Surface plot of the von Mises stress and a scaled deformation plot of the displacement.

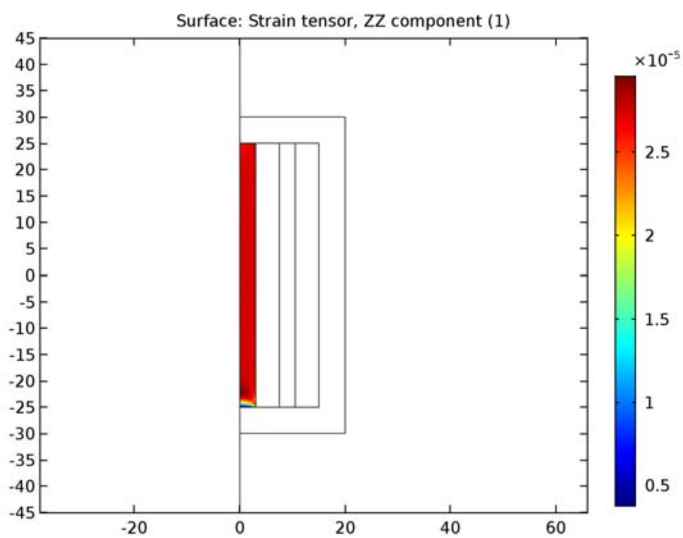


Figure 4: Surface plot of the axial strain component.

Figure 5 shows the magnetic the flux concentration in the magnetostrictive core due to the closed magnetic path provided by the steel housing. The magnetic flux density in the rod is mostly uniform. Fringe effects can be seen at both ends of the rod where majority of the magnetic flux is forced to curl into the steel housing.

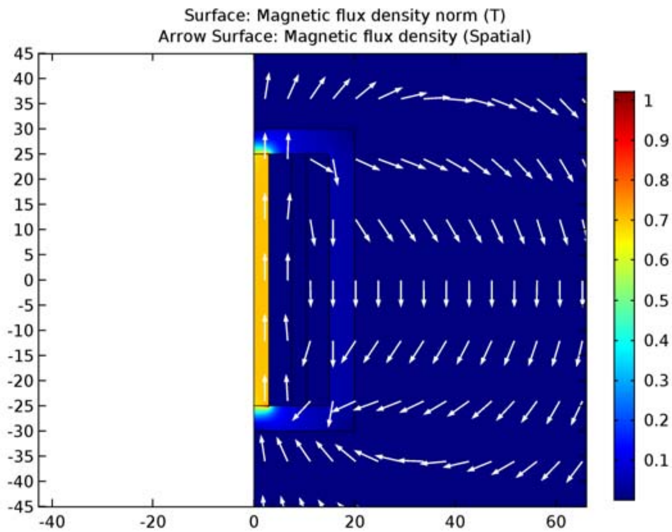


Figure 5: Surface plot of the norm of the magnetic flux density and a normalized arrow plot of its r and z-components showing the closed flux path in the model.

Figure 6 shows an interesting postprocessing feature in COMSOL. The solution obtained from the 2D axisymmetric model has been revolved by 225 degrees for 3D visualization of the solution. On solving a 2D axisymmetry model, COMSOL automatically creates a 3D solution data set by revolving the solution which is then plotted as a 3D plot.

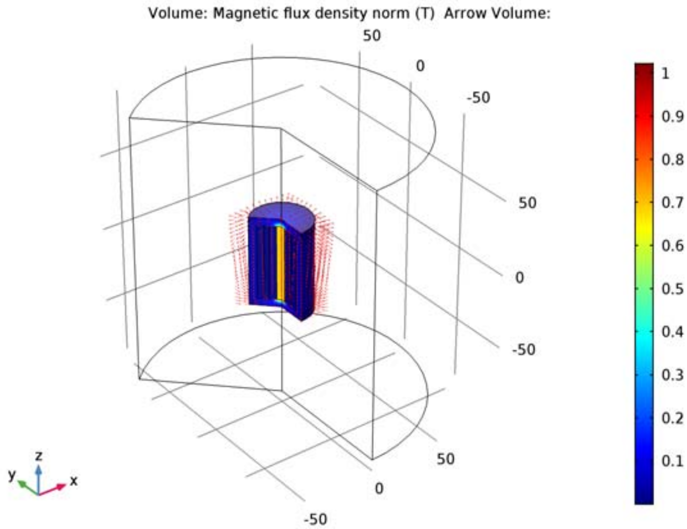


Figure 6: A 225 degree sectional view in 3D of the norm of the magnetic flux density in the magnetostrictive rod, steel housing and in the region within the housing. The solution in the outer air domain has been suppressed to get a better view. The normalized arrow plot shows the direction of the magnetic flux density.

Figure 7 shows the magnetostriction curve of the material obtained from the parametric study that simulated a quasi-static ramping up of the current density in the coil. The corresponding BH curve is shown in Figure 8. Because the magnetic field is oriented mostly along the axial direction, only the Z-components of the corresponding vectors are plotted. Note the significantly nonlinear behavior in the region where the magnetic field H_z varies between 5 to 20 kA/m.

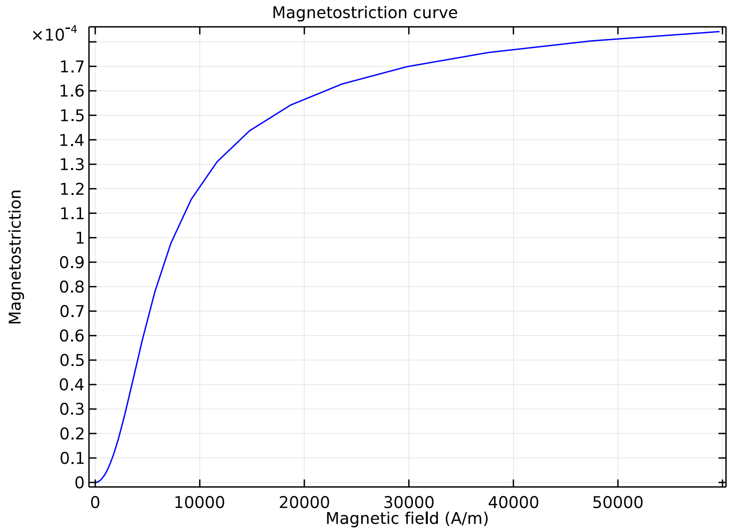


Figure 7: Magnetostriction versus magnetic field (at a point on the magnetostrictive material.

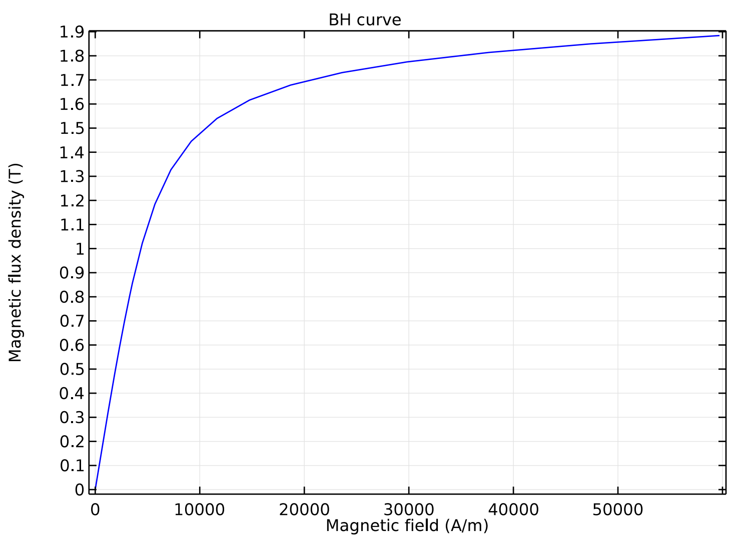


Figure 8: Magnetic flux density versus magnetic field at a point on the magnetostrictive material.

Reference

1. S. Chikazumi, *Physics of Ferromagnetism*, Oxford University Press, New York, 1997.

Application Library path: `Structural_Mechanics_Module/Magnetostrictive_Devices/nonlinear_magnetostriction`

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 **Structural Mechanics>Magnetostriction**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.
6 Click **Done**.

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 `3`.
4 In the **Height** text field, type `50`.
5 Locate the **Position** section. In the **z** text field, type `-25`.

Copy 1 (copy1)

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

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

3 Select the object **r1** only.

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

5 In the **r** text field, type `7.5`.

Rectangle 2 (r2)

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

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

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

4 In the **Width** text field, type `20`.

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

6 Locate the **Position** section. In the **z** text field, type `-30`.

Copy 2 (copy2)

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

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

3 Select the object **r2** only.

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

5 In the **z** text field, type `55`.

Rectangle 3 (r3)

1 Right-click **Copy 2 (copy2)** and choose **Build Selected**.

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

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

4 In the **Width** text field, type `5`.

5 In the **Height** text field, type `50`.

6 Locate the **Position** section. In the **r** text field, type `15`.

7 In the **z** text field, type `-25`.

Rectangle 4 (r4)

1 Right-click **Rectangle 3 (r3)** and choose **Build Selected**.

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

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

4 In the **Width** text field, type `90`.

5 In the **Height** text field, type `180`.

6 Locate the **Position** section. In the **z** text field, type `-90`.

7 Right-click **Rectangle 4 (r4)** and choose **Build Selected**.

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

Union 1 (uni1)

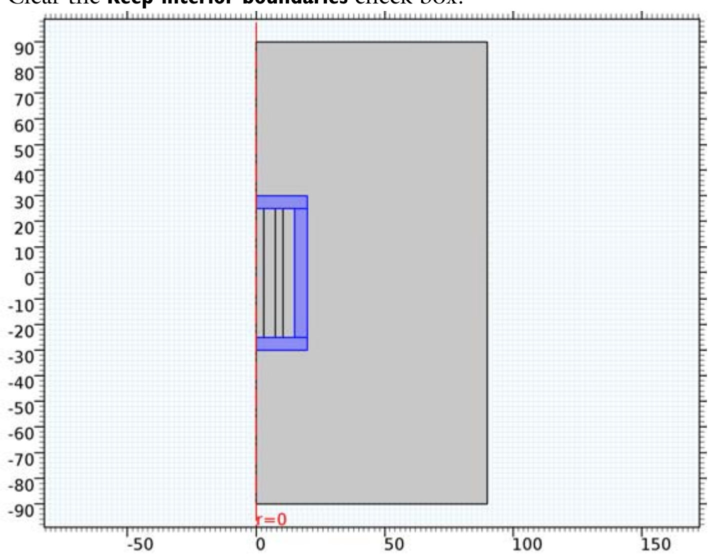
1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

See the figure below for the objects that need to be selected in the next step.

2 Select the objects **r3**, **r2**, and **copy2** only.

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

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



Point 1 (pt1)

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

2 Click **Build All Objects**.

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

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, 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
J0	1e6[A/m^2]	1E6 A/m²	Current density

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 **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 3 only.

The solid mechanics equations will be solved only in the magnetostrictive material.

MAGNETIC FIELDS (MF)

On the **Physics** toolbar, click **Solid Mechanics (solid)** and choose **Magnetic Fields (mf)**.

In the **Model Builder** window, under **Component 1 (comp1)** click **Magnetic Fields (mf)**.

Ampère's Law 2

1 On the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 Select Domain 2 only.

3 In the **Settings** window for Ampère's Law, locate the **Material Type** section.

4 From the **Material type** list, choose **Solid**.

5 Locate the **Magnetic Field** section. From the **Constitutive relation** list, choose **HB curve**.

SOLID MECHANICS (SOLID)

Magnetostrictive Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Magnetostrictive Material 1**.

2 In the **Settings** window for Magnetostrictive Material, click to collapse the **Model inputs** section.

3 Locate the **Magnetoelastic Properties** section. From the **Magnetostriction model** list, choose **Nonlinear isotropic**.

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

Air (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat1)**.

2 Select Domains 1 and 4–6 only.

ADD MATERIAL

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

2 In the tree, select **AC/DC>Soft Iron (without losses)**.

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

MATERIALS

Soft Iron (without losses) (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Soft Iron (without losses) (mat2)**.

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

3 Click **Paste Selection**.

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

5 Click **OK**.

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

Material 3 (mat3)

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 `Magnetostrictive` in the **Label** text field.

3 Select Domain 3 only.

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

Property	Name	Value	Unit	Property group
Density	rho	7870	kg/m³	Basic
Saturation magnetization	Ms	1.5e6	A/m	Magnetostrictive

Property	Name	Value	Unit	Property group
Initial magnetic susceptibility	chi	200	1	Magnetostrictive
Saturation magnetostriction	lambdas	2e-4	1	Magnetostrictive
Young's modulus	E	60e9	Pa	Basic
Poisson's ratio	nu	0.45	1	Basic
Electrical conductivity	sigma	5.96e6	S/m	Basic
Relative permittivity	epsilonr	1	1	Basic

MAGNETIC FIELDS (MF)

In the **Model Builder** window, under **Component 1 (comp1)** click **Magnetic Fields (mf)**.

External Current Density 1

1 On the **Physics** toolbar, click **Domains** and choose **External Current Density**.

2 Select Domain 5 only.

3 In the **Settings** window for External Current Density, locate the **External Current Density** section.

4 Specify the $\mathbf{J}_e$ vector as

0	r
J0	phi
0	z

Ampère's Law, Magnetostrictive 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Magnetic Fields (mf)** click **Ampère's Law, Magnetostrictive 1**.

2 In the **Settings** window for Ampère's Law, Magnetostrictive, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Manual**.

4 Click **Clear Selection**.

5 Select Domain 3 only.

SOLID MECHANICS (SOLID)

On the **Physics** toolbar, click **Magnetic Fields (mf)** and choose **Solid Mechanics (solid)**.

In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

Fixed Constraint 1

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

2 Select Boundary 6 only.

This boundary condition simulates that the lower surface of the magnetostrictive rod is fixed to the base of the transducer housing.

MESH 1

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

Free Quad 1

1 In the **Settings** window for Free Quad, locate the **Domain Selection** section.

2 From the **Geometric entity level** list, choose **Domain**.

3 Select Domains 2 and 3 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Quad 1** and choose **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. Select the **Maximum element size** check box.

5 In the associated text field, type `0.75`.

6 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Triangular**.

Free Triangular 1

In the **Settings** window for Free Triangular, click **Build All**.

STUDY 1

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

DEFINITIONS

View 1

In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

Axis

1 In the **Model Builder** window, expand the **View 1** node, then click **Axis**.

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

3 In the **r maximum** text field, type `60`.

4 In the **r minimum** text field, type `-52`.

5 In the **z minimum** text field, type `-45`.

6 In the **z maximum** text field, type `45`.

7 Click **Update**.

RESULTS

Stress (solid)

The first default plot shows the von Mises stress in the magnetostrictive core along with a scaled deformation plot, which should be similar to that shown in Figure 3.

Follow the steps outlined below to create Figure 4.

2D Plot Group 5

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

2 In the **Settings** window for 2D Plot Group, type `Strain (solid)` in the **Label** text field.

Surface 1

1 Right-click **Strain (solid)** 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>Solid Mechanics>Strain>Strain tensor (Material)>solid.eZZ - Strain tensor, ZZ component**.

3 On the **Strain (solid)** toolbar, click **Plot**.

Compare the resulting plot with that in Figure 4.

Follow the steps outlined below to create Figure 5.

Magnetic Flux Density Norm (mf)

In the **Model Builder** window, under **Results** right-click **Magnetic Flux Density Norm (mf)** and choose **Arrow Surface**.

Arrow Surface 1

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

2 In the **R component** text field, type `mf.Br`.

3 In the **Z component** text field, type `mf.Bz`.

4 Locate the **Arrow Positioning** section. Find the **R grid points** subsection. In the **Points** text field, type `20`.

5 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Normalized**.

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

7 On the **Magnetic Flux Density Norm (mf)** toolbar, click **Plot**.

Compare the resulting plot with that in Figure 5.

Follow the steps outlined below to create Figure 6.

Surface 1

1 In the **Model Builder** window, expand the **Magnetic Flux Density Norm, Revolved Geometry (mf)** node.

2 Right-click **Surface 1** and choose **Delete**.

Click **Yes** to confirm.

Magnetic Flux Density Norm, Revolved Geometry (mf)

In the **Model Builder** window, under **Results** right-click **Magnetic Flux Density Norm, Revolved Geometry (mf)** and choose **Volume**.

Volume 1

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

2 In the **Settings** window for Volume, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Magnetic Fields>Magnetic>mf.normB - Magnetic flux density norm**.

Filter 1

1 Right-click **Volume 1** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `dom!=1`.

This excludes the outer air domain from the plot.

Magnetic Flux Density Norm, Revolved Geometry (mf)

In the **Model Builder** window, under **Results** right-click **Magnetic Flux Density Norm, Revolved Geometry (mf)** and choose **Arrow Volume**.

Arrow Volume 1

1 In the **Settings** window for Arrow Volume, locate the **Expression** section.

2 In the **X component** text field, type `mf.Br*cos(rev2phi)- mf.Bphi*sin(rev2phi)`.

3 In the **Y component** text field, type `mf.Br*sin(rev2phi)+ mf.Bphi*cos(rev2phi)`.

4 In the **Z component** text field, type `mf.Bz`.

5 Locate the **Arrow Positioning** section. Find the **X grid points** subsection. From the **Entry method** list, choose **Coordinates**.

6 In the **Coordinates** text field, type `range(-20,4,20)`.

7 Find the **Y grid points** subsection. From the **Entry method** list, choose **Coordinates**.

8 In the **Coordinates** text field, type `range(-20,4,20)`.

9 Find the **Z grid points** subsection. From the **Entry method** list, choose **Coordinates**.

10 In the **Coordinates** text field, type `range(-30,2.5,30)`.

11 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Normalized**.

12 Select the **Scale factor** check box.

13 In the associated text field, type `5`.

14 On the **Magnetic Flux Density Norm, Revolved Geometry (mf)** toolbar, click **Plot**.

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

16 Click the **Zoom In** button on the **Graphics** toolbar.

17 Click the **Zoom In** button on the **Graphics** toolbar.

Next, perform a auxiliary continuation sweep on the external current density and plot the solution to view the saturation effect in the magnetostrictive core.

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.

STUDY 2

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

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

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

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2** 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 **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	**Parameter value list**	**Parameter unit**
J0		

6 Click **Range**.

7 In the **Range** dialog box, type `0` in the **Start** text field.

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

9 In the **Stop** text field, type `7.3`.

10 From the **Function to apply to all values** list, choose **exp10**.

11 Click **Add**.

Running the entire parametric study based on the settings above will take few minutes. The exact solution time will vary depending on the specification of the computer being used.

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

RESULTS

Follow the instructions below to create Figure 7 and Figure 8.

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 `Magnetostriction` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

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

5 In the **Title** text area, type `Magnetostriction curve`.

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

7 In the associated text field, type `Magnetic field (A/m)`.

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

9 In the associated text field, type `Magnetostriction`.

Point Graph 1

1 On the **Magnetostriction** toolbar, click **Point Graph**.

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

3 Click **Paste Selection**.

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

5 Click **OK**.

6 In the **Settings** window for Point Graph, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Strain>Strain tensor (Material)>solid.eZZ - Strain tensor, ZZ component**.

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

8 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Component 1>Magnetic Fields>Magnetic>Magnetic field (Material)>mf.HZ - Magnetic field, Z component**.

9 On the **Magnetostriction** toolbar, click **Plot**.

Magnetostriction

In the **Model Builder** window, under **Results** right-click **Magnetostriction** and choose **Duplicate**.

Magnetostriction 1

1 In the **Settings** window for 1D Plot Group, type `BH curve` in the **Label** text field.

2 Locate the **Title** section. In the **Title** text area, type `BH curve`.

3 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Magnetic flux density (T)`.

Point Graph 1

1 In the **Model Builder** window, expand the **Results>BH curve** node, then click **Point Graph 1**.

2 In the **Settings** window for Point Graph, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Magnetic Fields>Magnetic>Magnetic flux density (Material)>mf.BZ - Magnetic flux density, Z component**.

3 On the **BH curve** toolbar, click **Plot**.

Peristaltic Pump

Introduction

In a peristaltic pump, rotating rollers squeeze a flexible tube. As the pushed-down rollers move along the tube, fluids in the tube follow the motion. The main advantage of the peristaltic pump is that no seals, valves, or other internal parts ever touch the fluid. Due to their cleanliness, peristaltic pumps have found many applications in the pharmaceutical, chemical, and food industries. Besides this, the action of a peristaltic pump is very gentle, which is important if the fluid can be easily damaged. Peristaltic pumps are therefore used in medical applications, one of which is to move the blood through the body during open heart surgery. Other types of pumps would risk destroying the blood cells.

In this COMSOL Multiphysics example, a peristaltic pump is analyzed by combining structural mechanics (to model the squeezing of the tube) and fluid dynamics (to compute the fluid's motion). Thus, it is an example of a fluid-structure interaction (FSI) problem.

Model Definition

The analysis is set up in 2D axial symmetry (Figure 1). A nylon tube 0.1 m long has an inner radius of 1 cm and an outer radius of 1.5 cm; it contains fluid with the density $\rho = 1 \cdot 10^3$ kg/m^3 and viscosity $\mu = 5 \cdot 10^{-3}$ Pa·s. A time- and position-dependent force density is applied to the outer wall of the tube, in the radial direction. This force density could have been taken from real data from a peristaltic pump operation. For the sake of simplicity, this example models it with a Gaussian distribution along the length of the tube. The Gaussian distribution has a width of 1 cm and is moving with the constant velocity 0.03 m/s in the positive z direction. To represent the engagement of the roll, the force density, multiplied by a smoothed Heaviside function, kicks in at t = 0.1 s and takes the force to its full development at t = 0.5 s. Likewise, the disengagement of the roll starts at t = 1.0 s and ends at t = 1.4 s. The example models the tube's deformation during a full cycle of 1.5 s.

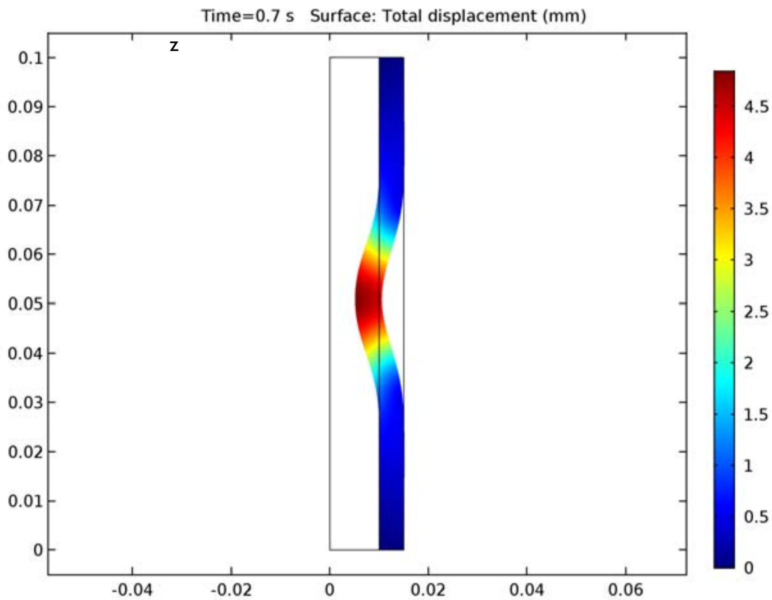


Figure 1: The geometry of the peristaltic pump as it is deforming under the pressure of the roll. The tube is rotationally symmetric with respect to the z-axis. The color shows the deformation of the tube material.

DOMAIN EQUATIONS

The structural mechanics computations use the assumption that the material is linear elastic, and they take geometric nonlinearities into account.

The fluid flow is described by the incompressible Navier-Stokes equations:

$$\rho\frac{\partial \mathbf{u}}{\partial t} + \rho\mathbf{u}\cdot\nabla\mathbf{u} = -\nabla p + \nabla\cdot\mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T)$$

$$\nabla\cdot\mathbf{u} = 0$$

where ρ denotes the density (SI unit: kg/m^3), $\mathbf{u}$ the velocity (SI unit: m/s), μ the viscosity (SI unit: Pa·s), and p the pressure (SI unit: Pa). The equations are set up and solved inside the tube.

The Navier-Stokes equations are solved on a freely moving deformed mesh, which constitutes the fluid domain. The deformation of this mesh relative to the initial shape of the domain is computed using Winslow smoothing. Inside the solid wall of the tube, the moving mesh follows the structural deformation. For more information, please refer to the chapter The Fluid-Structure Interaction Interface in the *Structural Mechanics Module User's Guide*.

BOUNDARY CONDITIONS

For the structural mechanics computations, the time- and coordinate-dependent load is prescribed as the boundary condition at the tube's outer surface. This is the load that drives the pump operation. The top and bottom ends of the tube are constrained along both coordinate axes.

For the fluid simulation, the boundary condition at the inlet and the outlet assumes that the total stress is zero, that is:

$$\mathbf{n}\cdot[-pI+\mu(\nabla\mathbf{u}+(\nabla\mathbf{u})^T)] = \mathbf{0}$$

The mesh is fixed to zero r displacement at the symmetry axis and zero z displacement at the top and the bottom of the tube.

At the fluid-solid boundary, the structural velocity is transmitted to the fluid. As a feedback, the stresses in the fluid flow act as a loading on the inner boundary of the solid wall of the tube.

COMPUTATION OF VOLUMETRIC FLOW RATES AND TOTAL VOLUME OF PUMPED FLUID

The model's dependent variables are the displacements of the tube wall together with the fluid velocity $\mathbf{u} = (u, v)$ and pressure p.

To get the volumetric flow rate of the fluid $\dot{V}$ in m^3/s and the total volume of pumped fluid, you need to perform some additional calculations. To obtain the volumetric flow rate at any instant t, compute a boundary integral over the pipe's inlet and outlet boundary:

$$\dot{V}\mathrm{in} = -\int_{s_{\mathrm{in}}} 2\pi r(\mathbf{n}\cdot\mathbf{u})ds$$

$$\dot{V}\mathrm{out} = \int_{s_{\mathrm{out}}} 2\pi r(\mathbf{n}\cdot\mathbf{u})ds$$

where $\mathbf{n}$ is the outward-pointing unit normal of the boundary, $\mathbf{u}$ is the velocity vector, and s is the boundary length parameter, along which you integrate. In this particular model, the inlet and outlet boundaries are horizontal so $\mathbf{n}\cdot\mathbf{u} = n_x u + n_y v$ simplifies to v or $-v$ depending on the direction of the flow.

It is of interest to track how much fluid is conveyed through the outlet during a peristaltic cycle, This can be calculated as the following time integral:

$$V_{\text{pump}}(t) = \int_0^t \dot{V}_{\text{out}}\, dt'$$

To compute this integral, specify the corresponding ODE in COMSOL Multiphysics

$$\frac{dV_{\text{pump}}}{dt} = \dot{V}_{\text{out}}$$

with proper initial conditions; the software then will integrate this equation.

Results

Figure 2 shows several snapshots from the peristaltic pump in action.

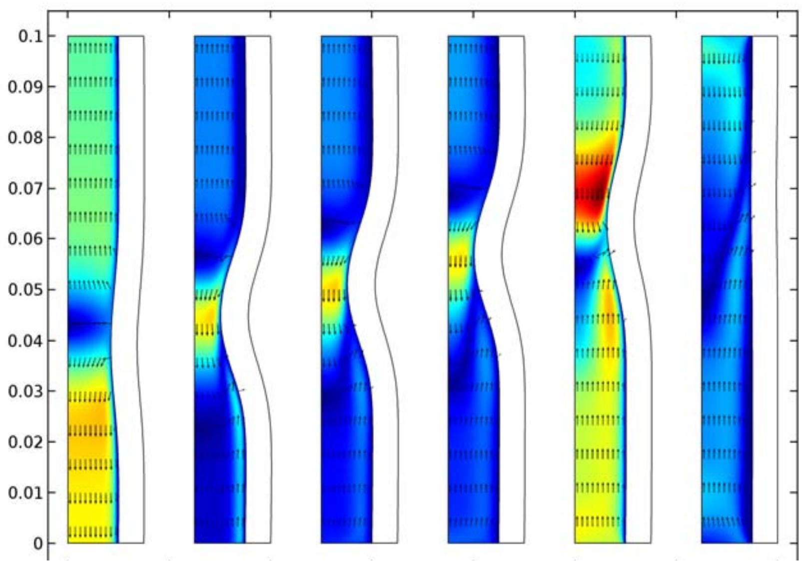


Figure 2: Snapshots of the velocity field and the shape of the inside of the tube at t = 0.3 s, t = 0.5 s, t = 0.7 s, t = 0.9 s, t = 1.1 s and t = 1.3 s. The colors represent the magnitude of the velocity, and the arrows its direction.

Figure 3 shows the inner volume of the tube as a function of time. At $t = 0.3$ s, the roll has begun its engagement phase, and it is increasing its pressure on the tube. As less and less space is left for the fluid, it is streaming out of the tube, through both the inlet and the outlet. At $t = 0.5$ s, the roll has been fully engaged for a while. As it is moving upward along the tube, so does the fluid, both at the inlet and at the outlet. This is where most of the net flow in the direction from the inlet to the outlet is created. Finally, at t = 1.3 s, the engagement process is reversed, and the roll is disengaging. As a result, the fluid is streaming into the tube from both ends.

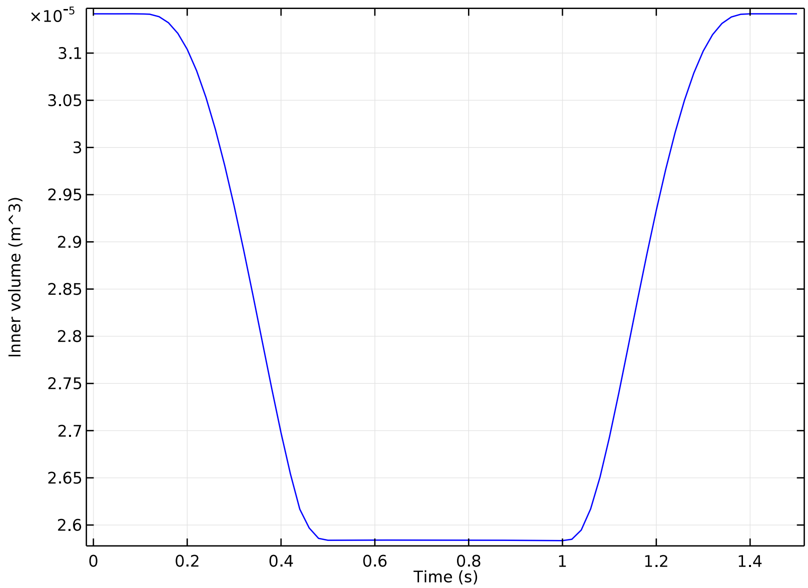


Figure 3: The inner volume (m^3) of the tube as a function of time (s).

Figure 4 shows the inlet and outlet flows, and it confirms the overall behavior indicated in the velocity snapshots. Note that a real peristaltic pump usually removes or minimizes the peaks associated with volume changes with the help of a second roll that engages at the same time as the first roll disengages. This way, there are hardly any volume changes, and the fluid flows forward all the time. Also note from Figure 4 that by taking the difference of the curves, $\dot{V}_{\mathrm{in}} - \dot{V}_{\mathrm{out}}$ and integrating over time, you generate Figure 3.

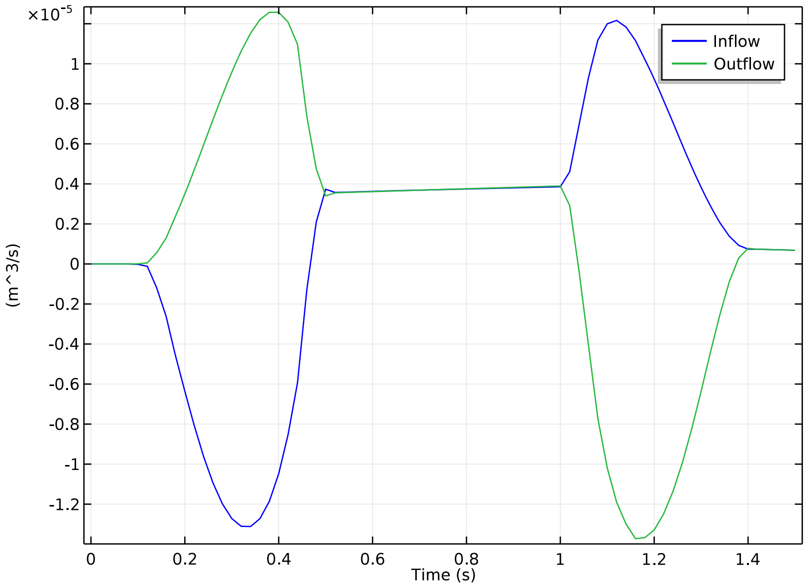


Figure 4: Inlet and outlet flow in m^3/s as functions of time. Positive values indicate that the fluid is flowing in through the inlet and out through the outlet.

[Figure 5](#) sums up the process, plotting the accumulated net flow versus time. It is worth noting that although the accumulated flow during the first 0.5 s of the cycle is zero or negative, it is well above zero after the full cycle.

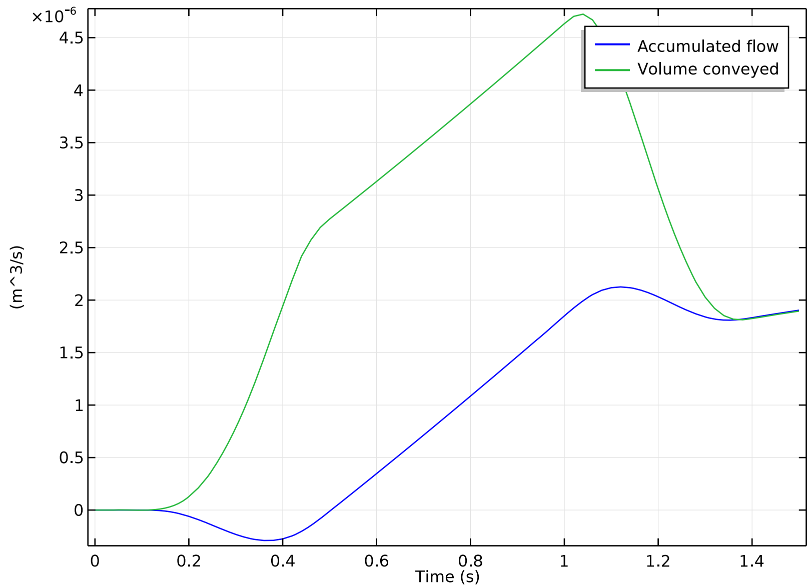


Figure 5: Accumulated flow (m^3) through the pump and volume of fluid conveyed out of the outlet versus time (s).

Notes About the COMSOL Implementation

This example is primarily intended to demonstrate the use of the Fluid-Structure Interaction interface, but it also shows some features for results analysis. Thus, it defines integration coupling operators to calculate the flow rate. An ordinary differential equation is used for calculating the accumulated fluid volume that has passed through the pump at certain points in time. The smooth step function used in this example is called `flc2hs` (a C^2-continuous step).

Application Library path: `Structural_Mechanics_Module/Fluid-Structure_Interaction/peristaltic_pump`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **2D Axisymmetric**.
2 In the **Select Physics** tree, select **Fluid Flow>Fluid-Structure Interaction (fsi)**.
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 `0.01`.
4 In the **Height** text field, type `0.1`.
5 Click **Build All Objects**.

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 `5e-3`.
4 In the **Height** text field, type `0.1`.
5 Locate the **Position** section. In the **r** text field, type `0.01`.
6 Click **Build All Objects**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, 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
t_on	0.3[s]	0.3 s	Time when roll is engaged
t_off	1.2[s]	1.2 s	Time when roll is disengaged
dt	0.2[s]	0.2 s	Time to reach full force
z0	0.03[m]	0.03 m	z coordinate where roll starts
v0	0.03[m/s]	0.03 m/s	Vertical velocity of roll
width	0.01[m]	0.01 m	Width of Gaussian force distribution
Ttot	1.5[s]	1.5 s	Total time for a pump cycle
Lmax	1.5e8[N/m^2]	1.5E8 N/m²	Max load

DEFINITIONS

To define the force density for load applied to the outer wall of the tube, follow the steps given below.

Analytic 1 (an1)

1. On the **Home** toolbar, click **Functions** and choose **Local>Analytic**.
2. In the **Settings** window for Analytic, type `load` in the **Function name** text field.
3. Locate the **Definition** section. In the **Expression** text field, type `flc2hs(t_off/dt-ts,1)*flc2hs(ts-t_on/dt,1)*exp(-(zs-(z0+v0*ts*dt)/width)^2/2)`.
4. In the **Arguments** text field, type `zs,ts`.

   Note that the function arguments are made dimensionless by zs = z/width and ts = t/dt.

To compute inflow/outflow rates, define the integration over the relevant boundaries.

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 2 only.

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 3 only.

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
inflow	intop1(2*pi*r*w_fluid)	m^3/s	Inflow
outflow	intop2(2*pi*r*w_fluid)	m^3/s	Outflow

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>Nylon**.

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

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

MATERIALS

Nylon (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Nylon (mat1)**.

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	Name	Value	Unit	Property group
Dynamic viscosity	mu	0.33	Pa·s	Basic

Material 2 (mat2)

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

2 Select Domain 1 only.

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

4 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Density	rho	1e3	kg/m³	Basic
Dynamic viscosity	mu	5e-3	Pa·s	Basic

FLUID-STRUCTURE INTERACTION (FSI)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Fluid-Structure Interaction (fsi)**.

2 In the **Settings** window for Fluid-Structure Interaction, locate the **Physical Model** section.

3 From the **Compressibility** list, choose **Incompressible flow**.

Linear Elastic Material 1

1 In the **Model Builder** window, expand the **Fluid-Structure Interaction (fsi)** node, then click **Linear Elastic Material 1**.

2 Select Domain 2 only.

Damping 1

1 On the **Physics** toolbar, click **Attributes** and choose **Damping**.

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

3 In the $\alpha_{\mathrm{d}M}$ text field, type `1e-2`.

4 In the $\beta_{\mathrm{d}K}$ text field, type `1e-3`.

Boundary Load 1

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

2 Select Boundary 7 only.

3 In the **Settings** window for Boundary Load, locate the **Force** section.

4 Specify the $\mathbf{F}_\mathrm{A}$ vector as

`-Lmax*load(z/width,t/dt)`	r
`0`	z

Fixed Constraint 1

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

2 Select Boundaries 5 and 6 only.

Open Boundary 1

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

2 Select Boundaries 2 and 3 only.

Prescribed Mesh Displacement 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Mesh Displacement**.

2 Select Boundaries 2 and 3 only.

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

4 Clear the **Prescribed r displacement** check box.

Prescribed Mesh Displacement 3

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Mesh Displacement**.

2 Select Boundary 1 only.

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

4 Clear the **Prescribed z displacement** check box.

Define the ordinary differential equations to calculate volume of the pumped fluid and the accumulated flow.

5 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)**	**Description**
`netflow`	`netflowt-(outflow+inflow)/2`	`0`	`0`	`Accumulated flow`

4 Locate the **Units** section. Find the **Dependent variable quantity** subsection. From the list, choose **Volume (m^3)**.

5 Find the **Source term quantity** subsection. From the list, choose **Volume per time (m^3/s)**.

Global Equations 2

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)	Description
`Vpump`	`Vpumpt-outflow`	`0`	`0`	`Volume conveyed`

4 Locate the **Units** section. Find the **Dependent variable quantity** subsection. From the list, choose **Volume (m^3)**.

5 Find the **Source term quantity** subsection. From the list, choose **Volume per time (m^3/s)**.

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(0,0.02,1.5)`.

Get the initial values, which will also generate the default plot to be shown while solving.

4 Right-click **Study 1>Step 1: Time Dependent** and choose **Get Initial Value for Step**.

5 In the **Settings** window for Time Dependent, click to expand the **Results while solving** section.

6 Locate the **Results While Solving** section. Select the **Plot** check box.

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

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)** node.

The problem size is relatively small, so that it can be solved efficiently using a fully coupled solver.

2 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** node.

3 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** and choose **Fully Coupled**.

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

RESULTS

The first default plot shows the stress distribution together with the velocity field t = 1.5 s. To plot the total displacement at t = 0.7 s (Figure 1), follow these steps:

2D Plot Group 4

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

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

3 From the **Time (s)** list, choose **0.7**.

Surface 1

1 Right-click **2D Plot Group 4** and choose **Surface**.

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

3 In the **Expression** text field, type `fsi.disp`.

4 From the **Unit** list, choose **mm**.

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

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

Flow and Stress (fsi)

To animate the velocity field as a function of time, as shown in Figure 2, proceed with the following steps:

Surface 1

In the **Model Builder** window, expand the **Flow and Stress (fsi)** node.

Flow and Stress (fsi)

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

Run the player to animate the velocity field as a function of time, as shown in Figure 2:

2 In the **Model Builder** window, under **Results** click **Flow and Stress (fsi)**.

3 On the **Flow and Stress (fsi)** toolbar, click **Animation** and choose **Player**.

Derived Values

To plot the total volume of fluid contained in the pump (Figure 3), follow the steps given below.

Surface Integration 1

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

2 Select Domain 1 only.

3 In the **Settings** window for Surface Integration, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
1	m^3	Inner volume

5 Locate the **Integration Settings** section. Select the **Compute volume integral** check box.

6 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

2 Click the right end of the **Display Table 1 - Surface Integration 1 (1)** split button in the window toolbar.

3 From the menu, choose **Table Graph**.

RESULTS

Table Graph 1

To plot the inlet and outlet flow rates (Figure 4), accumulated flow through the pump and volume of fluid conveyed out of the outlet(Figure 5), follow the steps given below.

1D Plot Group 6

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

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 `(m^3/s)`.

Global 1

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

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

3 In the table, enter the following settings:

Expression	Unit	Description
inflow	m^3/s	Inflow
outflow	m^3/s	Outflow

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

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

1D Plot Group 7

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 **Plot Settings** section.
3. Select the **y-axis label** check box.
4. In the associated text field, type `(m^3/s)`.

Global 1

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

Expression	Unit	Description
`netflow`	`m^3`	`Accumulated flow`
`Vpump`	`m^3`	`Volume conveyed`

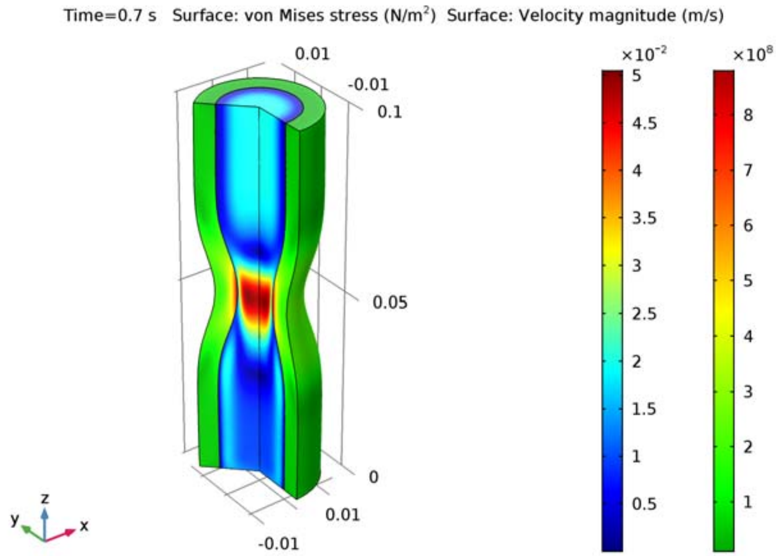
4. Click to expand the **Title** section. From the **Title type** list, choose **None**.
5. On the **1D Plot Group 7** toolbar, click **Plot**.

Flow and Stress, 3D (fsi)

1. In the **Model Builder** window, under **Results** click **Flow and Stress, 3D (fsi)**.
2. In the **Settings** window for 3D Plot Group, locate the **Data** section.
3. From the **Time (s)** list, choose **0.7**.
4. On the **Flow and Stress, 3D (fsi)** toolbar, click **Plot**.

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

The resulting plot should be similar to the one shown in the following figure:



Created in COMSOL Multiphysics 5.2a

Pratt Truss Bridge

Introduction

This example is inspired by a classic bridge type called a Pratt truss bridge. You can identify a Pratt truss by its diagonal members, which (except for the very end ones) all slant down and in toward the center of the span. All the diagonal members are subject to tension forces only, while the shorter vertical members handle the compressive forces. Since the tension removes the buckling risk, this allows for thinner diagonal members resulting in a more economic design.

A *truss structure* supports only tension and compression forces in its members and you would normally model it using bars, but as this model uses 3D beams it also includes bending moments to some extent in a *frame structure*. In the model, shell elements represent the roadway.

Model Definition

BASIC DIMENSIONS

The length of the bridge is 40 m, and the width of the roadway is 7 m. The main distance between the truss members is 5 m.

ANALYSIS TYPES

The model includes two different analyses of the bridge:

- The goal of the first analysis is to evaluate the stress and deflection fields of the bridge when exposed to a pure gravity load and also when a load corresponding to one or two trucks cross the bridge.
- Finally, an eigenfrequency analysis shows the eigenfrequencies and eigenmodes of the bridge.

LOADS AND CONSTRAINTS

To prevent rigid body motion of the bridge, it is important to constrain it properly. All translational degrees of freedom are constrained at the left-most horizontal edge. Constraints at the right-most horizontal edge prevent it from moving in the vertical and transversal directions but allow the bridge to expand or contract in the axial direction. This difference would however only be important if thermal expansion was studied.

Figure 1 shows the bridge geometry.

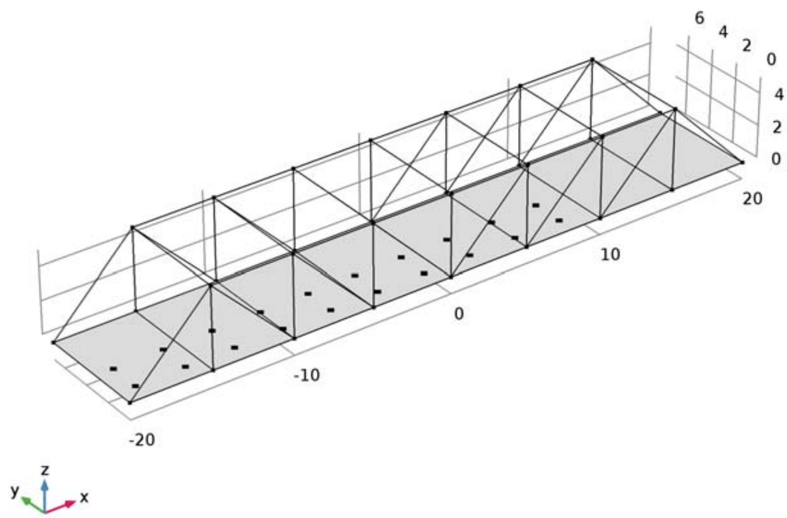


Figure 1: The geometry of the bridge

The first study uses several load cases. In the first load case the effects of self weight are analyzed. The following load cases compute the solution when two trucks are moving over the bridge. The weight of each truck is 12,000 kg, the wheelbase is 6 m, the axle track is 2 m, and the weight is distributed with one third on the front axle and two thirds on the rear axle. The right side wheels of the truck are 1 m from the edge of the bridge.

In the second study the natural frequencies of the bridge are computed.

MATERIAL PROPERTIES AND CROSS SECTION DATA

The material in the frame structure is structural steel. The roadway material is concrete; the effect of reinforcement is ignored. The frame members have different cross sections:

- The main beams along the bridge have square box profiles with height 200 mm and thickness 16 mm. This is also true for the outermost diagonal members.
- The diagonal and vertical members have a rectangular box section 200x100 mm, with 12.5 mm thickness. The large dimension is in the transverse direction of the bridge.

- The transverse horizontal members supporting the roadway (floor beams) are standard HEA100 profiles.
- The transverse horizontal members at the top of the truss (struts) are made from solid rectangular sections with dimension 100x25 mm. The large dimension is in the horizontal direction.

Results and Discussion

Figure 2 and Figure 3 illustrate the result. Figure 2 shows the displacements, and it can be seen that the maximum deflection amounts to 3 cm on the roadway. The distribution of axial forces (Figure 3) demonstrates the function of the frame: The interplay of members in tension and compression contribute to the load carrying function. The upper horizontal members are in compression and the lower in tension. The force in the lower members is much smaller, since the load is also shared by the roadway in this example. The diagonal members are subject to tension forces only, while the shorter vertical members handle the compressive forces.

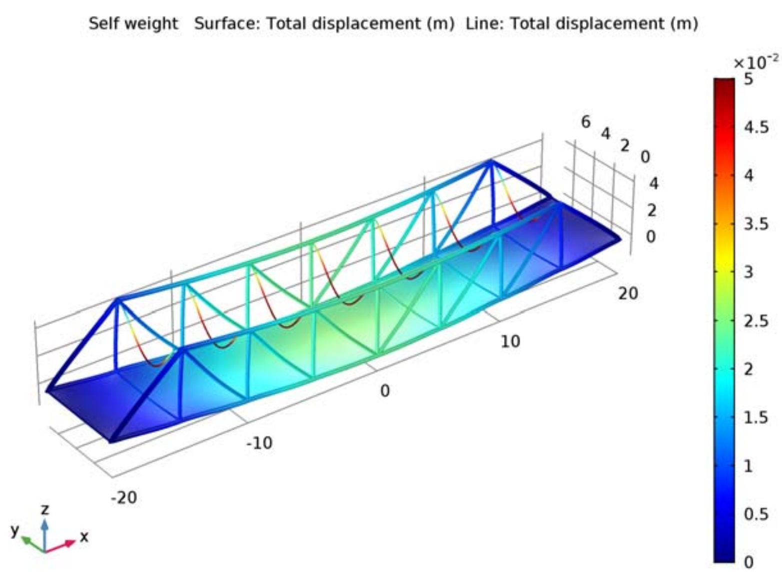


Figure 2: Deformation under self weight.

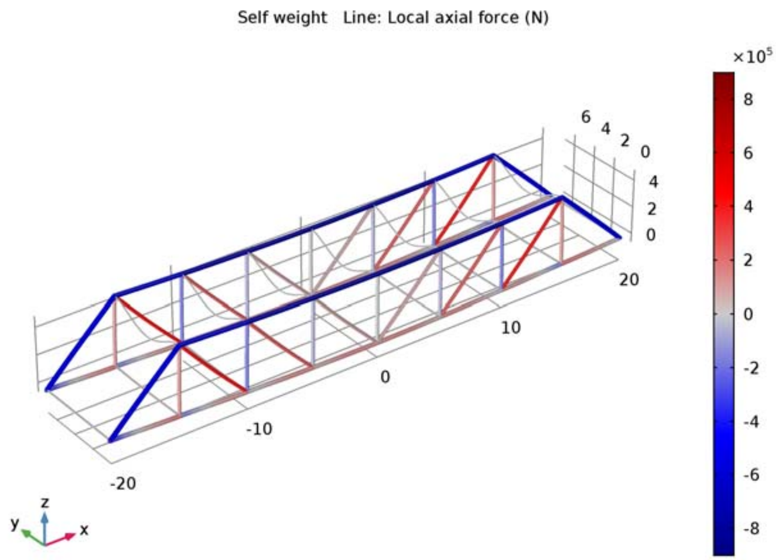


Figure 3: The axial forces in the beams. Red is tension and blue is compression.

To study the effects of trucks moving over the bridge, several load cases represent the position of the trucks. The trucks are moved 3 m along the bridge for each load case. Figure 4 shows the stress distribution in the roadway when the first truck has passed the bridge center and the second truck has entered the bridge deck.

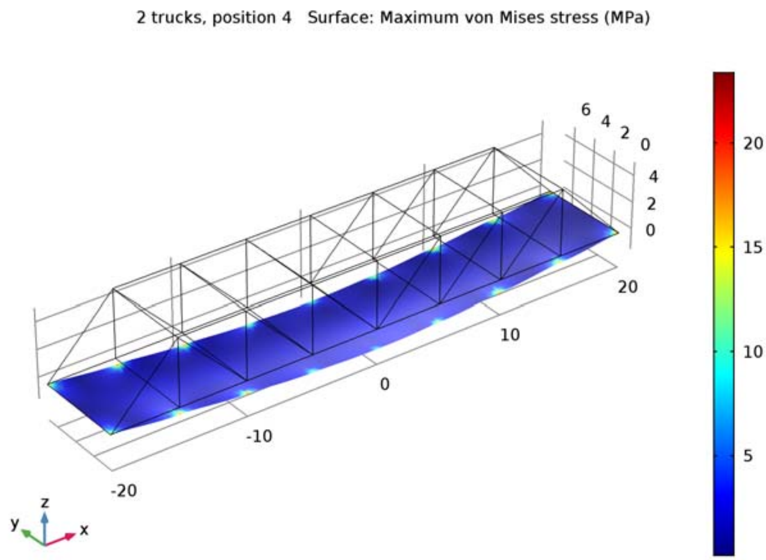


Figure 4: Truck load analysis: Stresses in the bridge deck with two trucks are on the bridge.

The study of eigenfrequencies is important with respect to the excitation and frequency content from various loads such as wind loads and earthquakes.

Figure 5 shows the 10^{th} eigenmode of the bridge, which is the fundamental mode for the roadway. The first eight eigenmodes only involve displacements of the weak struts at the top of the truss.

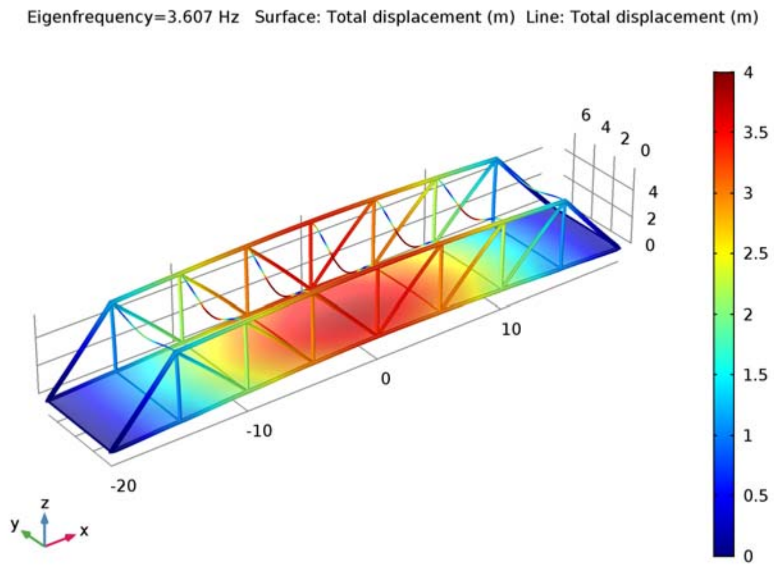


Figure 5: The 10^{th} eigenmode.

Notes About the COMSOL Implementation

You can define load cases to activate and deactivate loads within a study. All the loads need to be defined in the Model Builder. Under the Stationary Study node, you can then select which load or constraint to activate for a specific load case. Moreover for each load case you can modify the value of the applied load by changing its weight factor.


Define load cases

Load case	lg1	Weight	lg2	Weight	lg3	Weight	lg4	Weight	lg5	Weight
Self weight	☐	1.0	☐	1.0	☐	1.0	☐	1.0	☐	1.0
1 truck, position 1	☑	2.0	☐	1.0	☑	1.0	☐	1.0	☐	1.0
1 truck, position 2	☐	1.0	☑	2.0	☐	1.0	☑	1.0	☐	1.0
1 truck, position 3	☐	1.0	☐	1.0	☑	2.0	☐	1.0	☑	1.0

When combining two different physics interfaces, each have individual sets of degrees of freedom as a default. In structural mechanics, you usually want these to be equal. You can set such connections across various structural mechanics interfaces using built in connection features. In this particular model **Shell Connection** feature in **Beam** interface and **Beam Connection** feature in **Shell** interface are used to set up the connection between the two.

Application Library path: `Structural_Mechanics_Module/Civil_Engineering/pratt_truss_bridge`

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>Shell (shell)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Structural Mechanics>Beam (beam)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.

8 Click **Done**.

GLOBAL DEFINITIONS

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Load Group**.

2 Repeat this nine times so that you get ten load groups.

GEOMETRY 1

The geometry sequence for the model (see Figure 1) is available in a file. If you want to create it from scratch yourself, you can follow the instructions in the Appendix—Geometry Modeling Instructions. section. Otherwise, insert the geometry sequence as follows:

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

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

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

GLOBAL DEFINITIONS

Parameters

Add the non geometrical 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
`truck_weight`	`12000[kg]`	1.2E4 kg	`Total truck weight`

DEFINITIONS

Create groups for the different beam sections.

Box 1

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

2 In the **Settings** window for Box, type `BeamsTransvBelow` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.

4 Locate the **Box Limits** section. In the **x minimum** text field, type `-(length/2+1)`.

5 In the **x maximum** text field, type `length/2+1`.

6 In the **y minimum** text field, type `1`.

7 In the **y maximum** text field, type `width-1`.

8 In the **z minimum** text field, type `-1`.

9 In the **z maximum** text field, type `1`.

BeamsTransvBelow 1

1 Right-click **BeamsTransvBelow** and choose **Duplicate**.

2 In the **Settings** window for Box, type `BeamsAllBelow` in the **Label** text field.

3 Locate the **Box Limits** section. In the **y minimum** text field, type `-1`.

4 In the **y maximum** text field, type `width+1`.

5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Box 3

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

2 In the **Settings** window for Box, type `BeamsTransvAbove` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.

4 Locate the **Box Limits** section. In the **x minimum** text field, type `-(length/2+1)`.

5 In the **x maximum** text field, type `length/2+1`.

6 In the **y minimum** text field, type `1`.

7 In the **y maximum** text field, type `width-1`.

8 In the **z minimum** text field, type `height-1`.

9 In the **z maximum** text field, type `height+1`.

Box 4

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

2 In the **Settings** window for Box, type `BeamsDiag` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.

4 Locate the **Box Limits** section. In the **x minimum** text field, type `-(length/2-spacing+1)`.

5 In the **x maximum** text field, type `length/2-spacing+1`.

6 In the **y minimum** text field, type `-1`.

7 In the **y maximum** text field, type `width+1`.

8 In the **z minimum** text field, type `1`.

9 In the **z maximum** text field, type `2`.

Explicit 1

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

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

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

4 Select the **All edges** check box.

Difference 1

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

2 In the **Settings** window for Difference, type `BeamsMain` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, select **AllBeams** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for Difference, locate the **Input Entities** section.

8 Under **Selections to subtract**, click **Add**.

9 In the **Add** dialog box, In the **Selections to subtract** list, choose **BeamsTransvBelow**, **BeamsTransvAbove**, and **BeamsDiag**.

10 Click **OK**.

Add the materials.

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>Concrete**.

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

ADD MATERIAL

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

2 In the tree, select **Built-In>Structural steel**.

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

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

MATERIALS

Structural steel (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Structural steel (mat2)**.

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

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

4 From the **Selection** list, choose **All edges**.

SHELL (SHELL)

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

2 In the **Settings** window for Shell, locate the **Thickness** section.

3 In the d text field, type `0.25`.

Add self weight for the bridge deck.

Gravity 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Gravity**.
2. In the **Settings** window for Gravity, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **All boundaries**.

Pinned 1

1. On the **Physics** toolbar, click **Edges** and choose **Pinned**.
2. Select Edge 1 only.

Prescribed Displacement/Rotation 1 1

1. On the **Physics** toolbar, click **Edges** and choose **Prescribed Displacement/Rotation**.
2. Select Edge 78 only.
3. In the **Settings** window for Prescribed Displacement/Rotation, locate the **Prescribed Displacement** section.
4. Select the **Prescribed in y direction** check box.
5. Select the **Prescribed in z direction** check box.

Add the possible loads from the truck wheels.

Point Load 1

1. On the **Physics** toolbar, click **Points** and choose **Point Load**.
2. Click **Load Group** and choose **Load Group 1**.
3. In the **Model Builder** window, click **Point Load 1**.
4. Select Points 3 and 4 only.
5. In the **Settings** window for Point Load, locate the **Force** section.
6. Specify the $\mathbf{F}_\mathrm{P}$ vector as

0	x
0	y
-truck_weight*g_const/6	z

Point Load 2

1. Right-click **Point Load 1** and choose **Duplicate**.
2. On the **Physics** toolbar, click **Load Group** and choose **Load Group 2**.
3. In the **Model Builder** window, under **Component 1 (comp1)>Shell (shell)** click **Point Load 2**.
4. In the **Settings** window for Point Load, locate the **Point Selection** section.
5. Click **Clear Selection**.

6 Select Points 5 and 6 only.

7 Repeat this duplication procedure so that you get ten **Point Load** features. The loaded points and corresponding load groups are summarized in the table below:

Point load	Points	Load goup
Point Load 3	`11, 12`	`Load Group 3`
Point Load 4	`15, 16`	`Load Group 4`
Point Load 5	`19, 20`	`Load Group 5`
Point Load 6	`25, 26`	`Load Group 6`
Point Load 7	`27, 28`	`Load Group 7`
Point Load 8	`33, 34`	`Load Group 8`
Point Load 9	`37, 38`	`Load Group 9`
Point Load 10	`41, 42`	`Load Group 10`

BEAM (BEAM)

On the **Physics** toolbar, click **Shell (shell)** and choose **Beam (beam)**.

Set the cross-section data of the different beam types.

Cross Section Data 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Beam (beam)** click **Cross Section Data 1**.

2 In the **Settings** window for Cross Section Data, type `Cross Section Main` in the **Label** text field.

3 Locate the **Cross Section Definition** section. From the list, choose **Common sections**.

4 From the **Section type** list, choose **Box**.

5 In the h_y text field, type `200[mm]`.

6 In the h_z text field, type `200[mm]`.

7 In the t_y text field, type `16[mm]`.

8 In the t_z text field, type `16[mm]`.

Section Orientation 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Beam (beam)>Cross Section Main** node, then click **Section Orientation 1**.

2 In the **Settings** window for Section Orientation, locate the **Section Orientation** section.

3 From the **Orientation method** list, choose **Orientation vector**.

4 Specify the V vector as

0	x
1	y
0	z

5 In the **Model Builder** window, click **Beam (beam)**.

Cross Section Data 2

1 On the **Physics** toolbar, click **Edges** and choose **Cross Section Data**.

2 In the **Settings** window for Cross Section Data, type `Cross Section Diagonals` in the **Label** text field.

3 Locate the **Edge Selection** section. From the **Selection** list, choose **BeamsDiag**.

4 Locate the **Cross Section Definition** section. From the list, choose **Common sections**.

5 From the **Section type** list, choose **Box**.

6 In the h_y text field, type `200[mm]`.

7 In the h_z text field, type `100[mm]`.

8 In the t_y text field, type `12.5[mm]`.

9 In the t_z text field, type `12.5[mm]`.

Section Orientation 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Beam (beam)>Cross Section Diagonals** node, then click **Section Orientation 1**.

2 In the **Settings** window for Section Orientation, locate the **Section Orientation** section.

3 From the **Orientation method** list, choose **Orientation vector**.

4 Specify the V vector as

0	x
1	y
0	z

Cross Section Data 3

1 On the **Physics** toolbar, click **Edges** and choose **Cross Section Data**.

2 In the **Settings** window for Cross Section Data, type `Cross Section Transv Below` in the **Label** text field.

3 Locate the **Edge Selection** section. From the **Selection** list, choose **BeamsTransvBelow**.

4 Locate the **Cross Section Definition** section. From the list, choose **Common sections**.

5 From the **Section type** list, choose **H-profile**.

6 In the h_y text field, type `96[mm]`.

7 In the h_z text field, type `100[mm]`.

8 In the t_y text field, type `8[mm]`.

9 In the t_z text field, type `5[mm]`.

Section Orientation 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Beam (beam)>Cross Section Transv Below** node, then click **Section Orientation 1**.

2 In the **Settings** window for Section Orientation, locate the **Section Orientation** section.

3 From the **Orientation method** list, choose **Orientation vector**.

4 Specify the V vector as

0	x
0	y
1	z

Cross Section Data 4

1 On the **Physics** toolbar, click **Edges** and choose **Cross Section Data**.

2 In the **Settings** window for Cross Section Data, type `Cross Section Transv Above` in the **Label** text field.

3 Locate the **Edge Selection** section. From the **Selection** list, choose **BeamsTransvAbove**.

4 Locate the **Cross Section Definition** section. From the list, choose **Common sections**.

5 In the h_y text field, type `100[mm]`.

6 In the h_z text field, type `25[mm]`.

Section Orientation 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Beam (beam)>Cross Section Transv Above** node, then click **Section Orientation 1**.

2 In the **Settings** window for Section Orientation, locate the **Section Orientation** section.

3 From the **Orientation method** list, choose **Orientation vector**.

4 Specify the V vector as

1	x
0	y
0	z

Add the self weight of the beams.

Gravity 1

1 On the **Physics** toolbar, click **Edges** and choose **Gravity**.

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

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

Create connections between beams and shells.

Shell Connection 1

1 On the **Physics** toolbar, click **Edges** and choose **Shell Connection**.

2 Select Edges 2, 8, 18, 30, 40, 50, 62, and 72 only.

Shell Connection 2

1 On the **Physics** toolbar, click **Edges** and choose **Shell Connection**.

2 Select Edges 4, 13, 25, 35, 45, 57, 67, and 76 only.

Shell Connection 3

1 On the **Physics** toolbar, click **Edges** and choose **Shell Connection**.

2 Select Edges 7, 17, 22, 23, 29, 39, 49, 54, 55, 61, and 71 only.

Shell Connection 4

1 On the **Physics** toolbar, click **Edges** and choose **Shell Connection**.

2 Select Edge 1 only.

Shell Connection 5

1 On the **Physics** toolbar, click **Edges** and choose **Shell Connection**.

2 Select Edge 78 only.

SHELL (SHELL)

In the **Model Builder** window, expand the **Component 1 (comp1)>Shell (shell)** node, then click **Shell (shell)**.

Beam Connection 1

1 On the **Physics** toolbar, click **Edges** and choose **Beam Connection**.

2 Select Edges 2, 8, 18, 30, 40, 50, 62, and 72 only.

3 In the **Settings** window for Beam Connection, locate the **Beam Connection** section.

4 From the list, choose **Shell Connection 1 (beam)**.

5 From the **Offset definition** list, choose **Offset vector**.

6 Specify the $\mathbf{d}_0$ vector as

0	x
beam.hy_box/2	y
-beam.hz_box/2	z

Beam Connection 2

1 On the **Physics** toolbar, click **Edges** and choose **Beam Connection**.

2 Select Edges 4, 13, 25, 35, 45, 57, 67, and 76 only.

3 In the **Settings** window for Beam Connection, locate the **Beam Connection** section.

4 From the list, choose **Shell Connection 2 (beam)**.

5 From the **Offset definition** list, choose **Offset vector**.

6 Specify the $\mathbf{d}_0$ vector as

0	x
-beam.hy_box/2	y
-beam.hz_box/2	z

Beam Connection 3

1 On the **Physics** toolbar, click **Edges** and choose **Beam Connection**.

2 Select Edges 7, 17, 22, 23, 29, 39, 49, 54, 55, 61, and 71 only.

3 In the **Settings** window for Beam Connection, locate the **Beam Connection** section.

4 From the list, choose **Shell Connection 3 (beam)**.

5 From the **Offset definition** list, choose **Offset vector**.

6 Specify the $\mathbf{d}_0$ vector as

0	x
0	y
-beam.hy_H/2	z

Beam Connection 4

1 On the **Physics** toolbar, click **Edges** and choose **Beam Connection**.

2 Select Edge 1 only.

3 In the **Settings** window for Beam Connection, locate the **Beam Connection** section.

4 From the list, choose **Shell Connection 4 (beam)**.

5 From the **Offset definition** list, choose **Offset vector**.

6 Specify the $\mathbf{d}_0$ vector as

`beam.hz_H/2`	x
`0`	y
`-beam.hy_H/2`	z

Beam Connection 5

1 On the **Physics** toolbar, click **Edges** and choose **Beam Connection**.

2 Select Edge 78 only.

3 In the **Settings** window for Beam Connection, locate the **Beam Connection** section.

4 From the list, choose **Shell Connection 5 (beam)**.

5 From the **Offset definition** list, choose **Offset vector**.

6 Specify the $\mathbf{d}_0$ vector as

`-beam.hz_H/2`	x
`0`	y
`-beam.hy_H/2`	z

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 **Extremely fine**.

4 Click **Build All**.

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

STUDY 1

Step 1: Stationary

1 In the **Settings** window for Stationary, click to expand the **Study extensions** section.

2 Locate the **Study Extensions** section. Select the **Define load cases** check box.

3 Click **Add** nine times.

4 In the table, select the load cases and set the related weight according to the table:

Load case	Active load groups and weights
`Self Weight`	
`1 truck, position 1`	`lg1: 2.0; lg3: 1.0`

Load case	Active load groups and weights
`1 truck, position 2`	`lg2: 2.0; lg4: 1.0`
`1 truck, position 3`	`lg3: 2.0; lg5: 1.0`
`1 truck, position 4`	`lg4: 2.0; lg6: 1.0`
`2 trucks, position 1`	`lg1: 1.0; lg5: 2.0; lg7: 1.0`
`2 trucks, position 2`	`lg2: 1.0; lg6: 2.0; lg8: 1.0`
`2 trucks, position 3`	`lg1: 2.0; lg3: 1.0; lg7: 2.0; lg9: 1.0`
`2 trucks, position 4`	`lg2: 2.0; lg4: 1.0; lg8: 2.0; lg10: 1.0`

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

RESULTS

Stress (shell)

The default plot is the stress plot for the shells using the last load case, see Figure 4.

Surface 1

1 In the **Model Builder** window, expand the **Stress (shell)** node, then click **Surface 1**.

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

3 From the **Unit** list, choose **MPa**.

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

5 On the **Stress (shell)** toolbar, click **Plot**.

Add a new plot containing both shell and beam results, and examine the self weight load case to reproduce Figure 2.

3D Plot Group 11

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

2 In the **Settings** window for 3D Plot Group, type `Displacement` in the **Label** text field.

3 Locate the **Data** section. From the **Load case** list, choose **Self weight**.

4 Right-click **Displacement** and choose **Surface**.

Surface 1

In the **Model Builder** window, under **Results>Displacement** right-click **Surface 1** and choose **Deformation**.

Displacement

In the **Model Builder** window, under **Results** right-click **Displacement** and choose **Line**.

Line 1

1 In the **Settings** window for Line, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Beam>Displacement> beam.disp - Total displacement**.

You can indicate the dimensions of the beams by drawing them with a size depending on the radius of gyration.

2 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.

3 In the **Tube radius expression** text field, type `comp1.beam.re`.

4 Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Surface 1

1 In the **Model Builder** window, under **Results>Displacement** click **Surface 1**.

2 In the **Settings** window for Surface, click to expand the **Range** section.

3 Select the **Manual color range** check box.

4 In the **Maximum** text field, type `0.05`.

Line 1

In the **Model Builder** window, under **Results>Displacement** right-click **Line 1** and choose **Deformation**.

Deformation 1

On the **Displacement** toolbar, click **Plot**.

Displacement

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

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

3 Clear the **Plot data set edges** check box.

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

5 On the **Displacement** toolbar, click **Plot**.

Now plot the axial force in beams like in Figure 3.

6 Right-click **Results>Displacement** and choose **Duplicate**.

Displacement 1

In the **Settings** window for 3D Plot Group, type `Beam force` in the **Label** text field.

Line 1

1 In the **Model Builder** window, expand the **Results>Beam force** node, then click **Line 1**.

2 In the **Settings** window for Line, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Beam>Section forces>beam.Nxl - Local axial force**.

3 Locate the **Inherit Style** section. From the **Plot** list, choose **None**.

4 Click to expand the **Range** section. Select the **Manual color range** check box.

5 In the **Minimum** text field, type `-9e5`.

6 In the **Maximum** text field, type `9e5`.

7 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

Surface 1

In the **Model Builder** window, under **Results>Beam force** right-click **Surface 1** and choose **Disable**.

Beam force

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

2 In the **Model Builder** window, under **Results** click **Beam force**.

3 On the **Beam force** toolbar, click **Plot**.

Stress (shell)

1 In the **Model Builder** window, under **Results** click **Stress (shell)**.

2 On the **Stress (shell)** toolbar, click **Plot**.

Create an animation of the trucks passing the bridge.

Animation 1

1 On the **Results** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for Animation, click **Show Frame**.

3 Locate the **Frames** section. In the **Number of frames** text field, type `9`.

4 In the **Frame number** text field, type `9`.

You can easily remove unused plots to clean up the structure in the **Results** tree. An alternative could have been to deselect **Generate default plots** in the Study feature, but then you would have needed to create the current plot manually.

5 In the **Model Builder** window, select **Results>Stress Bottom (shell) 1**, hold down the Shift key, and then select **Results>Torsion Moment (beam) 1**. Right click and choose **Delete**.

Now add an eigenfrequency study.

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>Eigenfrequency**.

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

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

STUDY 2

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 2** 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 12.

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

RESULTS

Mode Shape (shell)

Select the first mode involving the roadway, see Figure 5.

1 In the **Model Builder** window, click **Mode Shape (shell)**.

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

3 From the **Eigenfrequency (Hz)** list, choose **3.607**.

4 Locate the **Plot Settings** section. Clear the **Plot data set edges** check box.

5 In the **Model Builder** window, expand the **Mode Shape (shell)** node.

Line 1

1 In the **Model Builder** window, expand the **Results>Mode Shape (beam)** node.

2 Right-click **Line 1** and choose **Copy**.

Mode Shape (shell)

In the **Model Builder** window, under **Results** right-click **Mode Shape (shell)** and choose **Paste Line**.

Line 1

1 In the **Settings** window for Line, locate the **Inherit Style** section.

2 From the **Plot** list, choose **Surface 1**.

Deformation

1 In the **Model Builder** window, expand the **Results>Mode Shape (shell)>Surface 1** node, then click **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 `0.2`.

Surface 1

1 In the **Model Builder** window, under **Results>Mode Shape (shell)** click **Surface 1**.

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

3 Select the **Manual color range** check box.

4 In the **Maximum** text field, type `4`.

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

6 On the **Mode Shape (shell)** toolbar, click **Plot**.

Appendix—Geometry Modeling Instructions.

If you wish to create the geometry yourself, follow these steps.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, 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
width	7[m]	7 m	Width of bridge
height	5[m]	5 m	Height of bridge
spacing	5[m]	5 m	Spacing between members along the bridge
length	40[m]	40 m	Total bridge length

GEOMETRY 1

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for Work Plane, click **Show Work Plane**.

Plane Geometry

Create the bridge deck.

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 `spacing`.

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

5 Locate the **Position** section. In the **xw** text field, type `-length/2`.

6 On the **Work Plane** toolbar, click **Build All**.

Array 1 (arr1)

1 On the **Work Plane** toolbar, click **Transforms** and choose **Array**.

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

3 From the **Array type** list, choose **Linear**.

4 Select the object **r1** only.

5 In the **Size** text field, type `length/spacing`.

6 Locate the **Displacement** section. In the **xw** text field, type `spacing`.

7 On the **Work Plane** toolbar, click **Build All**.

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

Plane Geometry

Make it possible to create beams along the edges.

Convert to Curve 1 (ccur1)

1 On the **Work Plane** toolbar, click **Conversions** and choose **Convert to Curve**.

2 In the **Settings** window for Convert to Curve, locate the **Input** section.

3 Select the **Keep input objects** check box.

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

5 On the **Work Plane** toolbar, click **Build All**.

6 In the **Model Builder** window, click **Geometry 1**.

7 On the **Home** toolbar, click **Build All**.

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

Start creating the truss.

Work Plane 2 (wp2)

1 On the **Geometry** toolbar, click **Work Plane**.
2 In the **Settings** window for Work Plane, locate the **Plane Definition** section.
3 From the **Plane** list, choose **xz-plane**.
4 Click **Show Work Plane**.

Bézier Polygon 1 (b1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.
2 In the **Settings** window for Bézier Polygon, locate the **General** section.
3 From the **Type** list, choose **Open curve**.
4 Locate the **Polygon Segments** section. Find the **Added segments** subsection. Click **Add Linear**.
5 Find the **Control points** subsection. In row **2**, set **xw** to `spacing` and **yw** to `height`.
6 Find the **Added segments** subsection. Click **Add Linear**.
7 Find the **Control points** subsection. In row **2**, set **yw** to `0`.
8 On the **Work Plane** toolbar, click **Build All**.

Bézier Polygon 2 (b2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.
2 In the **Settings** window for Bézier Polygon, locate the **General** section.
3 From the **Type** list, choose **Open curve**.
4 Locate the **Polygon Segments** section. Find the **Added segments** subsection. Click **Add Linear**.
5 Find the **Control points** subsection. In row **1**, set **yw** to `height`.
6 In row **2**, set **xw** to `spacing` and **yw** to `height`.
7 On the **Work Plane** toolbar, click **Build All**.

Array 1 (arr1)

1 On the **Work Plane** toolbar, click **Transforms** and choose **Array**.
2 In the **Settings** window for Array, locate the **Size** section.
3 From the **Array type** list, choose **Linear**.
4 Click in the **Graphics** window and then press Ctrl+A to select both objects.
5 In the **Size** text field, type `length/(2*spacing)-1`.
6 Locate the **Displacement** section. In the **xw** text field, type `spacing`.
7 On the **Work Plane** toolbar, click **Build All**.

Bézier Polygon 3 (b3)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.
2 In the **Settings** window for Bézier Polygon, locate the **General** section.
3 From the **Type** list, choose **Open curve**.
4 Locate the **Polygon Segments** section. Find the **Added segments** subsection. Click **Add Linear**.
5 Find the **Control points** subsection. In row **1**, set **xw** to `length/2-spacing` and **yw** to `height`.
6 In row **2**, set **xw** to `length/2`.
7 On the **Work Plane** toolbar, click **Build All**.

Mirror 1 (mir1)

1 On the **Work Plane** toolbar, click **Transforms** and choose **Mirror**.
2 Click in the **Graphics** window and then press Ctrl+A to select all objects.
3 In the **Settings** window for Mirror, locate the **Input** section.
4 Select the **Keep input objects** check box.
5 On the **Work Plane** toolbar, click **Build All**.
6 Click the **Zoom Extents** button on the **Graphics** toolbar.

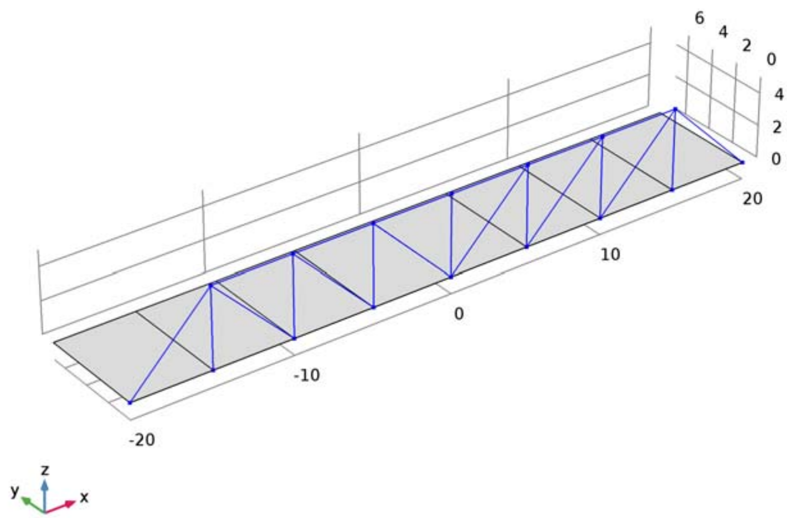
Bézier Polygon 4 (b4)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.
2 In the **Settings** window for Bézier Polygon, locate the **General** section.
3 From the **Type** list, choose **Open curve**.
4 Locate the **Polygon Segments** section. Find the **Added segments** subsection. Click **Add Linear**.
5 Find the **Control points** subsection. In row **2**, set **yw** to `height`.
6 On the **Work Plane** toolbar, click **Build All**.
7 In the **Model Builder** window, click **Geometry 1**.
8 On the **Home** toolbar, click **Build All**.

Copy 1 (copy1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Copy**.
2 In the **Settings** window for Copy, locate the **Displacement** section.
3 In the **y** text field, type `width`.

4 Select the object **wp2** only.



5 Click **Build All Objects**.

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **More 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 **1**, set **x** to `-length/2+spacing` and **z** to `height`.

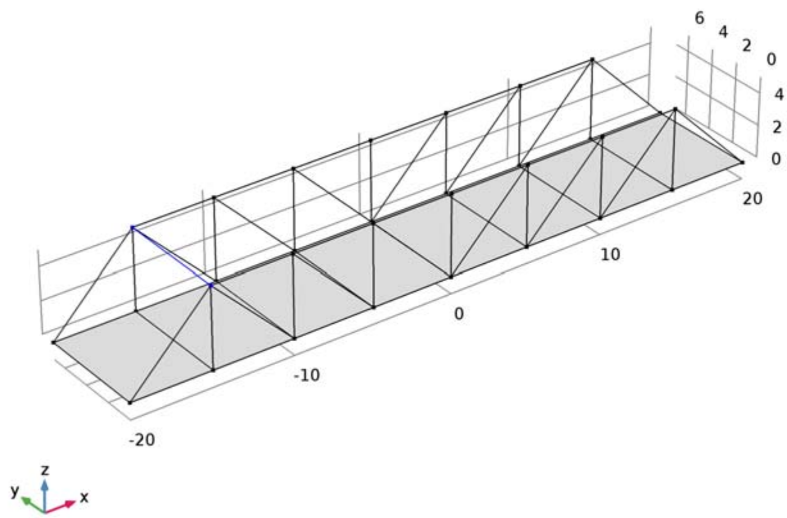
5 In row **2**, set **x** to `-length/2+spacing`, **y** to `width`, and **z** to `height`.

6 Click **Build All Objects**.

Array 1 (arr1)

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

2 Select the object **b1** only.



3 In the **Settings** window for Array, locate the **Size** section.

4 From the **Array type** list, choose **Linear**.

5 Locate the **Displacement** section. In the **x** text field, type `spacing`.

6 Locate the **Size** section. In the **Size** text field, type `length/spacing-1`.

Create points in the positions where the loads from the truck wheels are to be applied.

Point 1 (pt1)

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

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

3 In the **x** text field, type `-19`.

4 In the **y** text field, type `1`.

Array 2 (arr2)

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

2 Select the object **pt1** only.

3 In the **Settings** window for Array, locate the **Size** section.

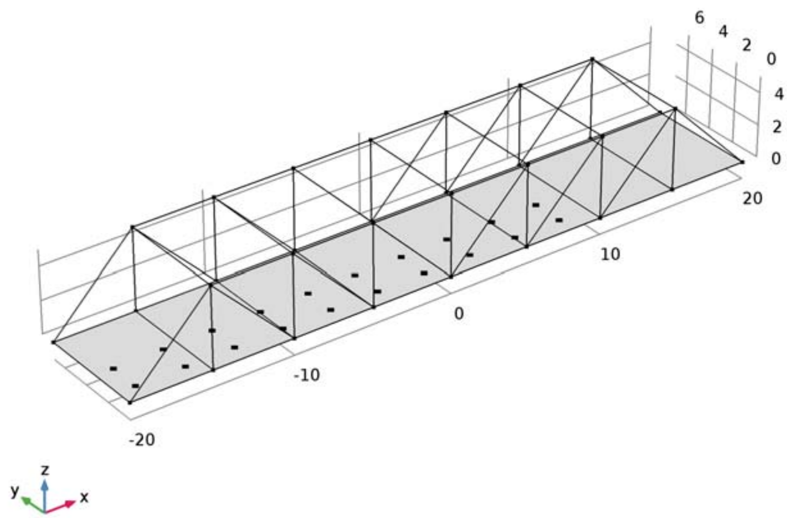
4 In the **x size** text field, type `10`.

5 In the **y size** text field, type `2`.

6 Locate the **Displacement** section. In the **x** text field, type 3.

7 In the **y** text field, type 2.

8 Click **Build All Objects**.



MEMS Pressure Sensor Drift Due to Hygroscopic Swelling

Introduction

For their integration in microelectronic circuits, MEMS and other devices are often overmolded with an *epoxy mold compound* (EMC) to protect the devices and their interconnects with the board. The epoxy polymers used for such applications are subject to moisture absorption and hygroscopic swelling, which can lead to delamination between the EMC and the board or to incorrect behavior of MEMS components. This example studies how the moisture absorption of an EMC affects the response of a MEMS pressure sensor over a one-year time period.

Model Definition

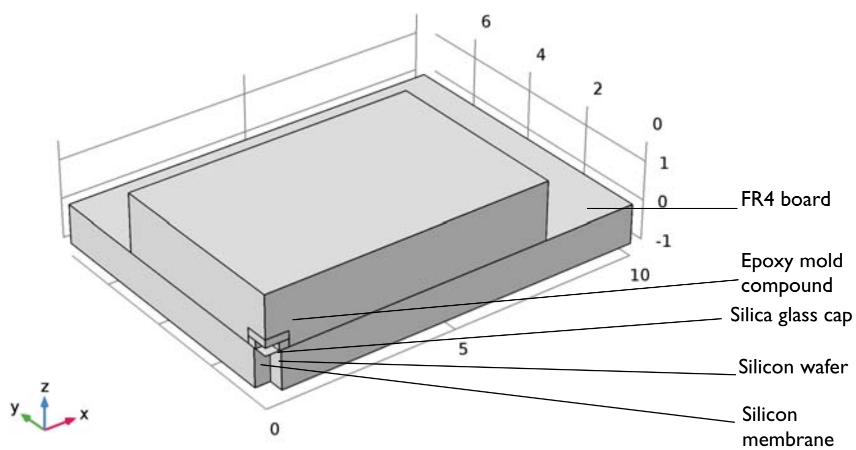


Figure 1: Component geometry.

It is sufficient to model a quarter of the whole structure due to the symmetry (Figure 1). The geometry is composed of:

- An FR4 board, on which the die is glued.

- The pressure sensor die made of:
  - A silicon component with a processed membrane. The membrane is modeled with a shell interface. The strain on the membrane surface is used to measure the pressure.
  - A silica glass capping
- An EMC that covers the die and a large part of the board.

When external pressure is applied on the bottom face of the membrane, the membrane deforms, and the strain is measured by means of a Wheatstone bridge made of piezoresistors. The measure of strain on the X- and Y-axes makes it possible to calculate the pressure. The membrane is modeled with a shell interface that is connected to the silicon domains via a shell-solid connection.

The moisture transport in the EMC is governed by the diffusion equation:

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

The moisture diffusion coefficient is temperature dependent:

$$D = D_0 \exp\left(-\frac{U}{kT}\right)$$

Here U is the activation energy and k is the Boltzmann's constant. For a typical EMC, $D_0 = 7.35 \cdot 10^{-6}$ m^2/s, $U = 0.43$ eV, and the diffusion coefficient at 25°C is around $4 \cdot 10^{-13}$ m^2/s.

The boundary conditions on the exterior faces of the EMC should be a flux of moisture concentration. However, given the long simulation time (one year) a concentration constraint can be assumed. The concentration applied on the boundaries is the saturation concentration of the material at a given temperature and humidity conditions:

$$C_{\mathrm{sat}} = SP_{\mathrm{sat}}\varphi$$

where S is the solubility on the water in the material a given condition, P_{sat} is the vapor saturation pressure of water, and φ is the relative humidity. The product of solubility and saturation pressure is supposed to be temperature-independent, thus the saturation concentration in the material depends only on the relative humidity. At 60% humidity, the saturation concentration is 140 mol/m^3.

The initial moisture concentration after molding is set to 40 mol/m^3. This value can be also taken as reference for hygroscopic swelling because all the stresses are assumed relaxed just after molding.

In order to avoid problems that can be caused by the discontinuity of concentration at initial state, the concentration boundary condition is applied smoothly, and a boundary layer type mesh is used near those boundaries (Figure 2).

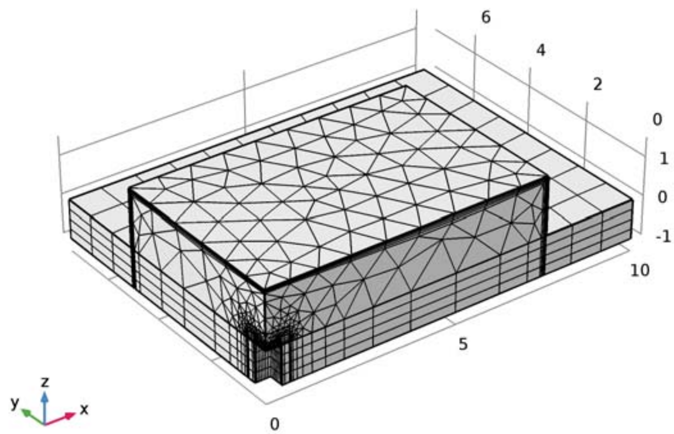


Figure 2: Mesh of the device.

As hygroscopic swelling induces a one-way coupling between concentration and mechanics, the concentration is calculated in a first time-dependent study, and then the structural domains are computed in a stationary study. This sequential approach reduces the computation time compared to a single solution including all physical interfaces.

Results and Discussion

The moisture diffuses progressively in the EMC. After 6 days, the moisture has already partially reached the top face of the die (Figure 3).

Figure 4 shows that the concentration at the die location starts to increase after 2 days until approximately 100 days. This is confirmed by the mass uptake shown in Figure 5, where the maximum value is reached after the same period of time.

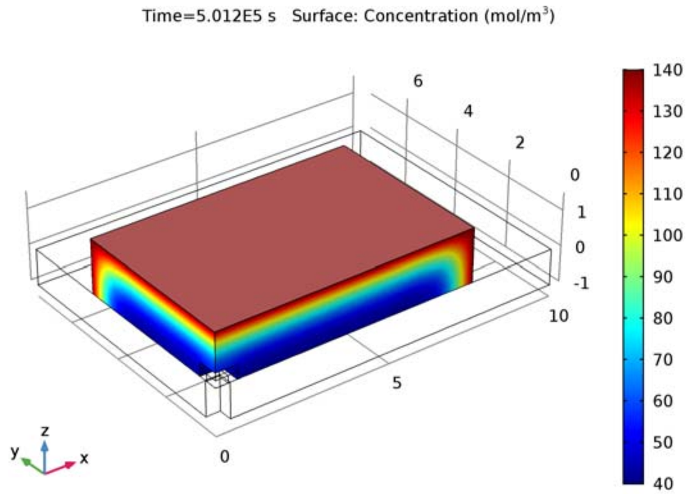


Figure 3: Moisture concentration in the EMC after 6 days.

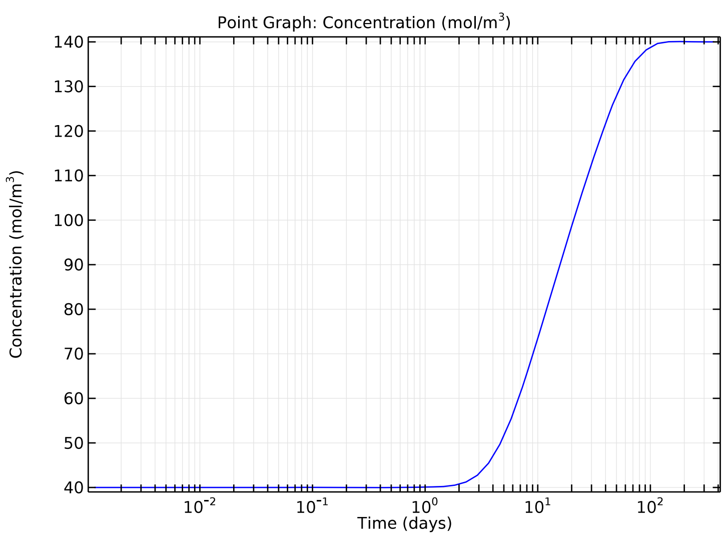


Figure 4: Moisture concentration at die location over time.

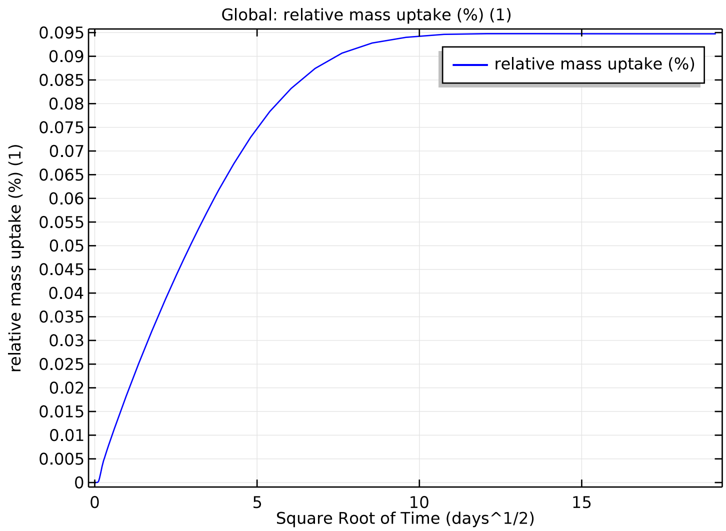


Figure 5: Total mass uptake in the EMC.

The progressive moisture diffusion is also noticed on displacement plots after hygroscopic swelling calculation: the EMC swells only on its boundaries during the first days (Figure 6), and it swells everywhere after one year (Figure 7). During the first time, the expansion on the exterior boundaries implies a stretching on the membrane and thus an increase of the measured strain. Then, the expansion of the center implies compression on the die and a decrease of the strain along the axes; see Figure 8.

The moisture absorption and hygroscopic swelling have significant effect on the sensor sensibility, which have to be taken in account during the measurements, or when designing the sensor.

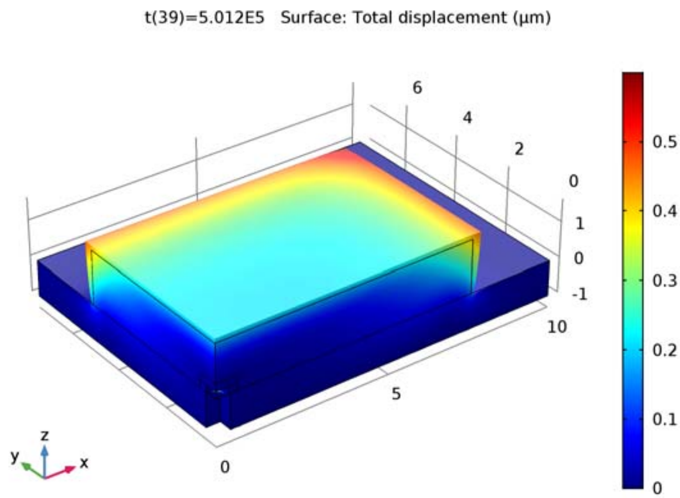


Figure 6: Displacement after 6 days.

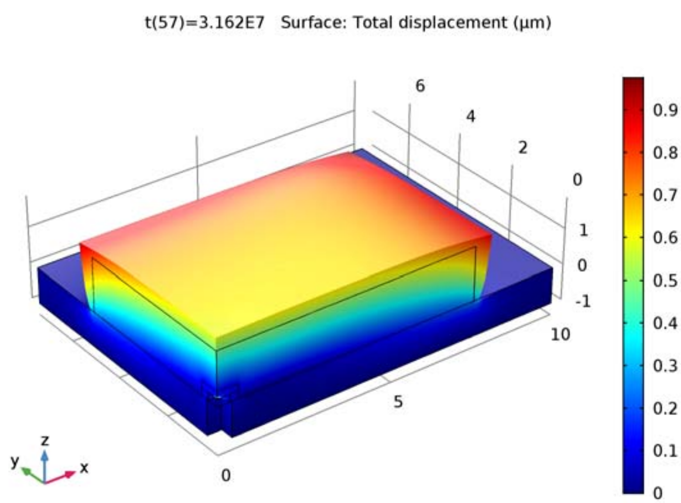


Figure 7: Displacement after 1 year.

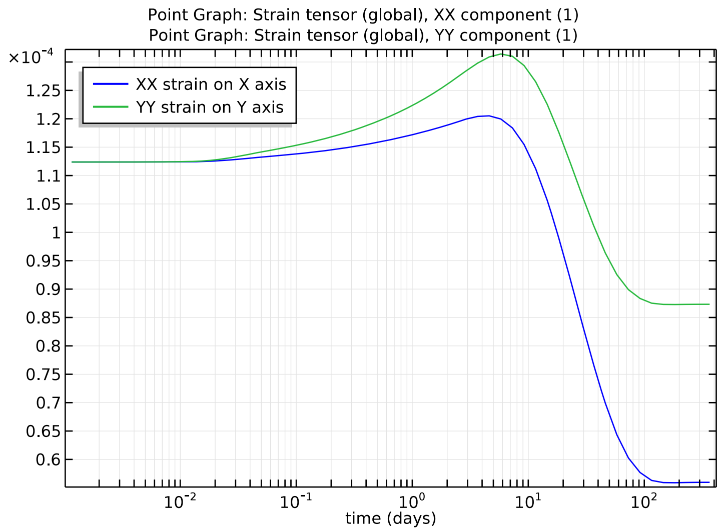


Figure 8: Evolution of measured strain on membrane axes.

Application Library path: `Structural_Mechanics_Module/Hygroscopic_Swelling/pressure_sensor_hygroscopic_swelling`

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 In the **Select Physics** tree, select **Structural Mechanics>Shell (shell)**.

5 Click **Add**.

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

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 On the **Home** toolbar, 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	20[mm]	0.02 m	length of the device
w	15[mm]	0.015 m	width of the device
l_MC	15[mm]	0.015 m	length of the eopxy mold compound
w_MC	10[mm]	0.01 m	width of the epoxy mold compound
l_die	1.2[mm]	0.0012 m	side length of the pressure sensor
l_memb	700[µm]	7E-4 m	side length of the silicon membrane
l_hole	800[µm]	8E-4 m	side length of the hole in FR4
t_FR4	1[mm]	0.001 m	thickness of the FR4
t_Si	200[µm]	2E-4 m	thickness of the silicon
t_memb	20[µm]	2E-5 m	thickness of the silicon membrane
t_glass	200[µm]	2E-4 m	thickness of the silica glass wafer
t_MC	1.5[mm]	0.0015 m	thickness of the mold compound
cmax	140[mol/m^3]	140 mol/m³	saturated concentration of the mold compound

Name	Expression	Value	Description
cini	40[mol/m^3]	40 mol/m³	initial moisture concentration
pext	1[bar]	1E5 Pa	external pressure
t	0[s]	0 s	time used for parametric sweep

GEOMETRY 1

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

Create a block for silicon wafer and glass cap.

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 `l_die/2`.
4. In the **Depth** text field, type `l_die/2`.
5. In the **Height** text field, type `t_Si+t_glass`.
6. Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	t_Si

Create a block for the EMC.

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 **Width** text field, type `l_MC/2`.
4. In the **Depth** text field, type `w_MC/2`.
5. In the **Height** text field, type `t_MC`.

Create a block and substract it to make the cavity in the wafer.

Block 3 (blk3)

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 `l_memb/2`.

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

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

Difference 1 (dif1)

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

2 Select the objects **blk1** and **blk2** 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 **blk3** only.

6 Click **Build All Objects**.

Create a block for the board.

Block 4 (blk4)

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 `l/2`.

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

5 In the **Height** text field, type `t_FR4`.

6 Locate the **Position** section. In the **z** text field, type `-t_FR4`.

Create a block and substract it to make a hole in the board.

Block 5 (blk5)

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 `l_hole/2`.

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

5 In the **Height** text field, type `t_FR4`.

6 Locate the **Position** section. In the **z** text field, type `-t_FR4`.

Difference 2 (dif2)

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

2 Select the object **blk4** 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 **blk5** only.

6 Click **Build All Objects**.

Create rectangle in a workplane to build the membrane.

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 `l_memb/2`.

4 In the **Height** text field, type `l_memb/2`.

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

6 In the **Model Builder** window, click **Geometry 1**.

7 On the **Home** toolbar, click **Build All**.

8 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

DEFINITIONS

Create selections to select domains easily in the following steps.

Explicit 1

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

2 Select Domain 4 only.

3 In the **Settings** window for Explicit, type `FR4` in the **Label** text field.

Explicit 2

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

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

3 Select Domain 3 only.

Explicit 3

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

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

3 Select Domain 1 only.

Explicit 4

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

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

3 Select Domain 2 only.

Explicit 5

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

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

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

4 Select Boundary 1 only.

SHELL (SHELL)

Set the shell thickness and the offset so that the bottom face is at z = 0, and set the height of evaluation so that the results are calculated on the bottom face.

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

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

3 From the **Selection** list, choose **Membrane**.

4 Locate the **Thickness** section. In the d text field, type `t_memb`.

5 From the **Offset definition** list, choose **Relative offset**.

6 In the $z_{\mathrm{reloffset}}$ text field, type `1`.

7 Click to expand the **Default through-thickness result location** section. Locate the **Default Through-Thickness Result Location** section. In the z text field, type `-1`.

Solid Connection 1

1 On the **Physics** toolbar, click **Edges** and choose **Solid Connection**.

2 Select Edges 13 and 35 only.

3 In the **Settings** window for Solid Connection, locate the **Solid Connection** section.

4 From the **Connection type** list, choose **Simplified**.

Use symmetry on the shell edges. The normal of symmetry plane is the second axis of the local edge system, which is orthogonal to the edge and to the shell normal.

Symmetry 11

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

2 Select Edges 1 and 2 only.

Face Load 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Face Load**.
2 In the **Settings** window for Face Load, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Membrane**.
4 Locate the **Force** section. From the **Load type** list, choose **Pressure**.
5 In the p text field, type `-pext`.

SOLID MECHANICS (SOLID)

First, set the discretization to **Quadratic** in **Solid Mechanics** in order to fit the discretization of **Shell**.

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Discretization** in the menu.
2 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.
3 In the **Settings** window for Solid Mechanics, click to expand the **Discretization** section.
4 From the **Displacement field** list, choose **Quadratic**.

Fixed Constraint 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.
2 Select Boundary 15 only.

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
2 Select Boundaries 2, 3, 5, 6, 9, 13, 24, and 26 only.

Shell Connection 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Shell Connection**.
2 Select Boundaries 10 and 23 only.
3 In the **Settings** window for Shell Connection, locate the **Shell Connection** section.
4 From the list, choose **Solid Connection 1 (shell)**.

DEFINITIONS

Create a step function in order to apply the concentration boundary condition progressively.

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 `0.5`.

4 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `1`.

5 Click **Plot**.

TRANSPORT OF DILUTED SPECIES (TDS)

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

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

3 From the **Selection** list, choose **Mold Compound**.

4 Locate the **Transport Mechanisms** section. Clear the **Convection** check box.

5 Click to expand the **Discretization** section. From the **Concentration** list, choose **Quadratic**.

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 `4e-13[m^2/s]`.

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 `cini`.

4 In the **Model Builder** window, click **Transport of Diluted Species (tds)**.

Symmetry 1

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

2 Select Boundaries 5 and 6 only.

Concentration 1

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

2 Select Boundaries 8, 20, and 29 only.

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

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

5 In the $c_{0,c}$ text field, type `cini+(cmax-cini)*step1(t[1/s]/3600)`.

MULTIPHYSICS

Add a multiphysics node to model hygroscopic swelling.

Hygroscopic Swelling 1 (hs1)

1. On the **Physics** toolbar, click **Multiphysics** and choose **Domain>Hygroscopic Swelling**.
2. In the **Settings** window for Hygroscopic Swelling, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Mold Compound**.
4. Locate the **Hygroscopic Swelling Properties** section. In the $c_{\mathrm{mo,\ ref}}$ text field, type `cini`.

DEFINITIONS

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Mass Properties**.
2. In the **Settings** window for Mass Properties, locate the **Source Selection** section.
3. From the **Selection** list, choose **Mold Compound**.
4. Locate the **Density** section. From the **Density source** list, choose **From physics interface**.

MATERIALS

Add a material for each domain and for the membrane.

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>FR4 (Circuit Board)**.
4. Click **Add to Selection** in the window toolbar.

MATERIALS

FR4 (Circuit Board) (mat1)

1. In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **FR4 (Circuit Board) (mat1)**.
2. In the **Settings** window for Material, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **FR4**.

ADD MATERIAL

1. Go to the **Add Material** window.
2. In the tree, select **Built-In>Silicon**.
3. Click **Add to Selection** in the window toolbar.

MATERIALS

Silicon (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Silicon (mat2)**.
2 In the **Settings** window for Material, locate the **Geometric Entity Selection** section.
3 From the **Selection** list, choose **Silicon**.
4 Click **Add to Selection** in the window toolbar.

Silicon 1 (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Silicon 1 (mat3)**.
2 In the **Settings** window for Material, type `Silicon (membrane)` in the **Label** text field.
3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4 From the **Selection** list, choose **Membrane**.

ADD MATERIAL

1 Go to the **Add Material** window.
2 In the tree, select **Built-In>Silica glass**.
3 Click **Add to Selection** in the window toolbar.

MATERIALS

Silica glass (mat4)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Silica glass (mat4)**.
2 In the **Settings** window for Material, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 From the **Selection** list, choose **Glass**.
5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

Material 5 (mat5)

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 `Mold Compound` in the **Label** text field.
3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Mold Compound**.

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

Property	**Name**	**Value**	**Unit**	**Property group**
Young's modulus	E	22[GPa]	Pa	Basic
Poisson's ratio	nu	0.4	1	Basic
Density	rho	1900	kg/m³	Basic
Coefficient of hygroscopic swelling	beta_h	1.1e-4	m³/kg	Basic

MESH 1

Mesh the membrane using a 2D mapped mesh.

Mapped 1

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

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

3 From the **Selection** list, choose **Membrane**.

Distribution 1

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

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

3 In the **Number of elements** text field, type 10.

4 Locate the **Edge Selection** section. From the **Selection** list, choose **All edges**.

Free Triangular 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2 Select Boundaries 11 and 16 only.

Use a swept mesh in the wafer domain to avoid stress singularities near the solid-shell connection.

Swept 1

1 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 From the **Selection** list, choose **Silicon**.

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

4 In the **Element ratio** text field, type 5.

Convert rectangles to triangles on the exterior boundaries to enable tetraedral meshing.

Convert 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Convert**.

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

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

4 Select Boundaries 17 and 27 only.

Free Tetrahedral 1

1 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 **Glass**.

Add boundary layer meshing on the exterior faces in order to smooth the initial concentration discontinuity.

Boundary Layers 1

1 Right-click **Mesh 1** and choose **Boundary Layers**.

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

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

4 From the **Selection** list, choose **Mold Compound**.

Boundary Layer Properties

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Boundary Layers 1** click **Boundary Layer Properties**.

2 Select Boundaries 8, 20, and 29 only.

3 In the **Settings** window for Boundary Layer Properties, locate the **Boundary Layer Properties** section.

4 In the **Number of boundary layers** text field, type 4.

5 In the **Thickness adjustment factor** text field, type 1.

Swept 2

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 From the **Selection** list, choose **FR4**.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Swept 2** and choose **Distribution**.
2 In the **Settings** window for Distribution, locate the **Distribution** section.
3 In the **Number of elements** text field, type `4`.
4 Click **Build All**.
5 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

STUDY 1

Step 1: Time Dependent

Since the moisture diffusion is independent of the structural behavior, compute only transport of diluted species in the time dependent analysis.

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 `0 10^range(2,0.1,7.5)`.
4 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Solid Mechanics (solid)** and **Shell (shell)**.

Prepare a plot to visualize the concentration during the computation.

Solution 1 (sol1)

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

RESULTS

3D Plot Group 1

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2 In the **Settings** window for 3D Plot Group, type `Concentration` in the **Label** text field.

Surface 1

1 Right-click **Concentration** 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 (comp1)>Transport of Diluted Species>c - Concentration**.

3 Click to expand the **Range** section. Select the **Manual color range** check box.

4 In the **Minimum** text field, type `40`.

5 In the **Maximum** text field, type `140`.

STUDY 1

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, click to expand the **Results while solving** section.

3 Locate the **Results While Solving** section. Select the **Plot** check box.

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

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

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

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

RESULTS

Concentration

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

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

3 From the **Time (s)** list, choose **5.012E5**.

4 On the **Concentration** toolbar, click **Plot**.

5 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

1D Plot Group 2

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

2 In the **Settings** window for 1D Plot Group, type `Concentration at Die Location` in the **Label** text field.

Point Graph 1

1 On the **Concentration at Die Location** toolbar, click **Point Graph**.

2 Select Point 3 only.

3 In the **Settings** window for Point Graph, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Transport of Diluted Species>c - Concentration**.

4 Locate the **x-Axis Data** section. From the **Unit** list, choose **d**.

Concentration at Die Location

1 In the **Model Builder** window, under **Results** click **Concentration at Die Location**.

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

3 Select the **x-axis log scale** check box.

4 On the **Concentration at Die Location** toolbar, click **Plot**.

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

6 In the associated text field, type `Time (days)`.

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

1D Plot Group 3

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

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

Global 1

1 On the **Mass Uptake** toolbar, click **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`(mass1.mass-at(0, mass1.mass))/at(0, mass1.mass)*100`	`1`	`relative mass uptake (%)`

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

5 In the **Expression** text field, type `sqrt(t[1/d])`.

6 Select the **Description** check box.

7 In the associated text field, type `(time)^(1/2) [d^(1/2)]`.

Mass Uptake

1 In the **Model Builder** window, under **Results** click **Mass Uptake**.

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

3 Select the **x-axis label** check box.

4 In the associated text field, type `Square Root of Time (days^1/2)`.

5 Locate the **Axis** section. Select the **Manual axis limits** check box.

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

7 In the **x maximum** text field, type `15`.

8 On the **Mass Uptake** toolbar, click **Plot**.

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

Add a stationary study with a parametric sweep to compute the mechanical behavior.

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**.

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

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

STUDY 2

Step 1: Stationary

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

2 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

3 In the **Settings** window for Stationary, locate the **Physics and Variables Selection** section.

4 In the table, clear the **Solve for** check box for **Transport of Diluted Species (tds)**.

5 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

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

7 From the **Study** list, choose **Study 1, Time Dependent**.

8 From the **Time (s)** list, choose **All**.

9 Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

10 Click **Add**.

11 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
`t`	`0 10^range(2,0.1,7.5)`	

Solution 2 (sol2)

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** node.

4 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** and choose **Fully Coupled**.

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

Plot stress of solid and shell in the same plot group.

RESULTS

Surface 1

In the **Model Builder** window, expand the **Results>Stress (shell)** node.

Stress (solid)

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

2 In the **Model Builder** window, under **Results** click **Stress (solid)**.

3 In the **Settings** window for 3D Plot Group, type `Stress` in the **Label** text field.

Surface 2

1 Right-click **Results>Stress** and choose **Paste Surface**.

2 In the **Settings** window for Surface, click to expand the **Title** section.

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

4 Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Deformation

1 In the **Model Builder** window, expand the **Results>Stress>Surface 1** node, then click **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 `600`.

5 On the **Stress** toolbar, click **Plot**.

Stress (shell)

In the **Model Builder** window, under **Results** right-click **Stress (shell)** and choose **Delete**.

RESULTS

Undeformed Geometry (shell)

1. In the **Model Builder** window, expand the **Undeformed Geometry (shell)** node.
2. Right-click **Results>Undeformed Geometry (shell)** and choose **Delete**.

RESULTS

Plot displacement of solid and shell in the same plot group.

Stress

In the **Model Builder** window, under **Results** right-click **Stress** and choose **Duplicate**.

Stress 1

In the **Settings** window for 3D Plot Group, type `Displacement` in the **Label** text field.

Surface 1

1. In the **Model Builder** window, expand the **Results>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>solid.disp - Total displacement**.
3. Locate the **Expression** section. From the **Unit** list, choose **µm**.

Surface 2

1. In the **Model Builder** window, under **Results>Displacement** click **Surface 2**.
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>Shell>Displacement> shell.disp - Total displacement**.
3. Locate the **Expression** section. From the **Unit** list, choose **µm**.
4. On the **Displacement** toolbar, click **Plot**.
5. Click the **Go to Default 3D View** button on the **Graphics** toolbar.

Displacement

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 **Time** list, choose **5.012E5**.
4. On the **Displacement** toolbar, click **Plot**.

Plot the strain on the bottom face of the membrane. To do so, add 3D cut points on the x and y axes.

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 `Strain` in the **Label** text field.

Cut Point 3D 1

1 On the **Results** toolbar, click **Cut Point 3D**.

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

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

4 Locate the **Point Data** section. In the **X** text field, type `l_memb/2-30[µm]`.

5 In the **Y** text field, type `0`.

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

7 Select the **Snap to closest boundary** check box.

Cut Point 3D 2

1 On the **Results** toolbar, click **Cut Point 3D**.

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

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

4 Locate the **Point Data** section. In the **X** text field, type `0`.

5 In the **Y** text field, type `l_memb/2-30[µm]`.

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

7 Select the **Snap to closest boundary** check box.

8 Click **Plot**.

Strain

In the **Model Builder** window, under **Results** click **Strain**.

Point Graph 1

1 On the **Strain** toolbar, click **Point Graph**.

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

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

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `shell.eXX`.

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

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

7 From the **Unit** list, choose **d**.

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

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

10 In the table, enter the following settings:

Legends
`XX strain on X axis`

Strain

In the **Model Builder** window, under **Results** click **Strain**.

Point Graph 2

1 On the **Strain** toolbar, click **Point Graph**.

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

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

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `shell.eYY`.

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

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

7 From the **Unit** list, choose **d**.

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

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

10 In the table, enter the following settings:

Legends
`YY strain on Y axis`

Strain

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

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

3 Select the **x-axis log scale** check box.

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

5 In the associated text field, type `time (days)`.

6 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

7 On the **Strain** toolbar, click **Plot**.

Submodel in a Wheel Rim

Introduction

In stress analysis, it is common that the regions with high stresses are small when compared to the whole structure. Sometimes it is not feasible to have a mesh that at the same time captures the global behavior and resolves the stress concentrations with high accuracy. This is especially true in nonlinear or dynamic problems.

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.

This example shows how to perform submodel analysis in COMSOL Multiphysics.

Model Definition

The wheel rim for this analysis is a ten-spoke example where the design elements of the geometry cause the finite element mesh to become quite large. The loading on the tire is composed of both the tire pressure and a load transferred from the road via the tire to the rim.

In the submodel (shown in Figure 1), you cut out a small region around the hot spot using an intersection between the rim geometry and a 70 mm-by-70 mm-by-60 mm block.

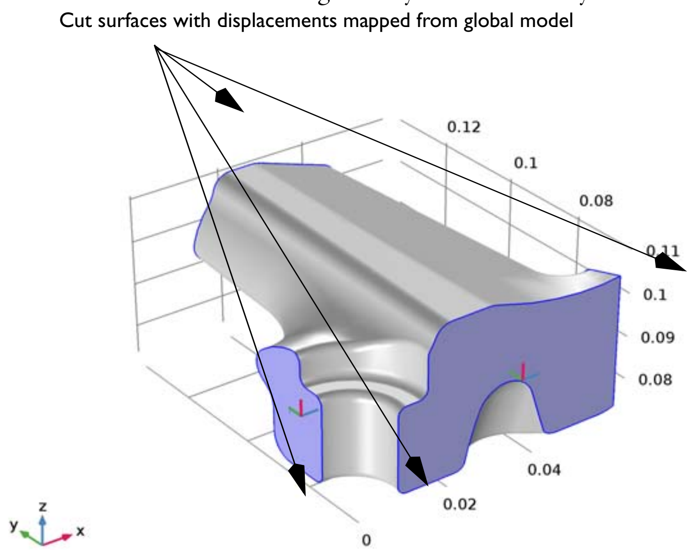


Figure 1: The submodel geometry.

MATERIAL

Aluminum with E = 70 GPa, ν = 0.33.

CONSTRAINTS

- A region around each bolt hole where the wheel is rim is attached to the wheel hub is fixed.

LOADS

- Tire pressure: The overpressure is 2 bar = 200 kPa.

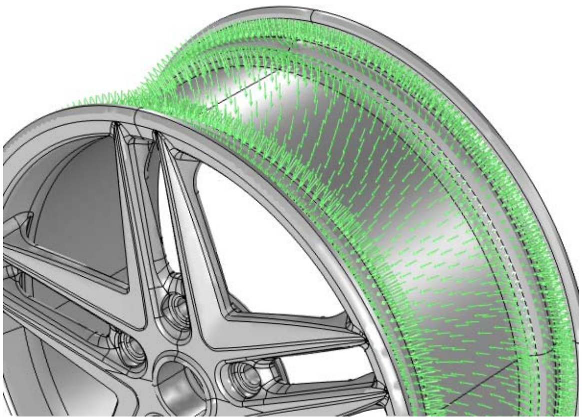


Figure 2: Tire pressure distribution

- The total load carried by the wheel corresponds to a weight of 1120 kg. It is applied as a pressure on the rim surfaces where the tire is in contact. Assume that the load distribution in the circumferential direction can be approximated as $p = p_0\cos(3\vartheta)$, where ϑ is the angle from the point of contact between the road and the tire. The loaded area thus extends 30° in each direction from the peak of the load. Four different load cases are analyzed, where the center of the peak load is rotated 18° each time. In this way the whole load cycle for the rotating wheel can be covered. The load distribution is shown in Figure 3.

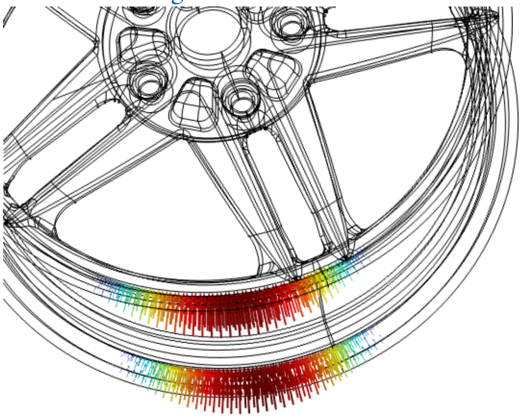


Figure 3: Tire load when rotated 18° from the center of the first pair of spokes

- In the submodel, the stress history for a full revolution of the wheel is computed. This is possible, since results from different spokes are applied to the submodel sequentially.

Results and Discussion

The highest stresses occur in the fillet where the spoke connects to the hub. In the global model the maximum effective stress is completely mesh dependent, and not reliable. In the submodel, where the resolution is good, the effective stress is about 96 MPa. It occurs when the load is rotated 18° from the reference angle. In a fatigue analysis where the lifetime could vary as the fifth power of the stress it is essential to get this level of accuracy in the critical regions.

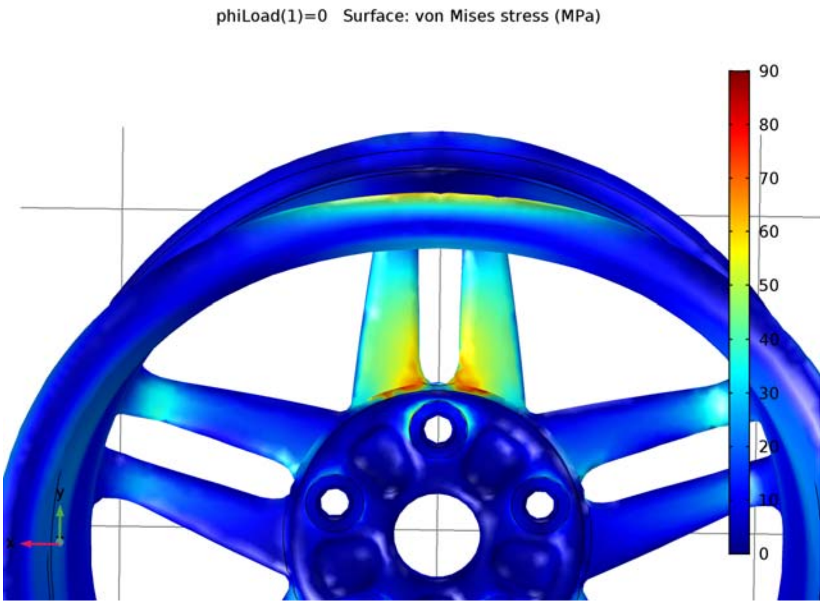


Figure 4: Stresses in the global model.

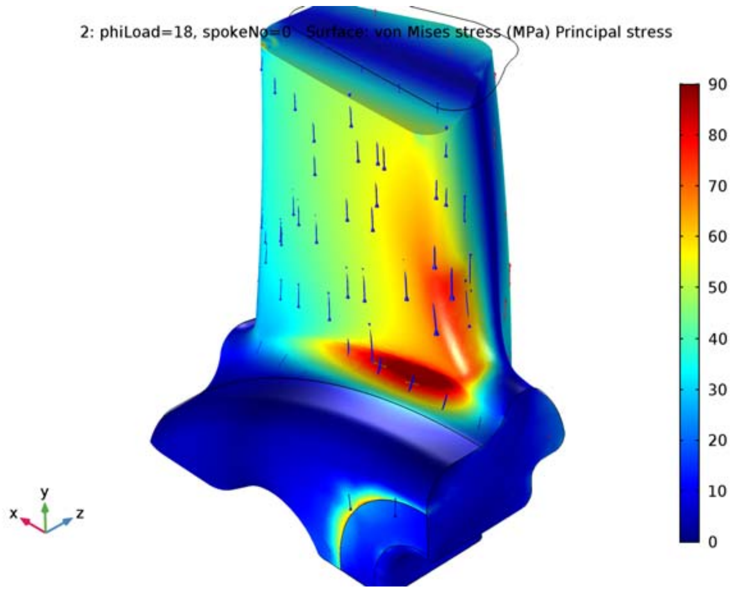


Figure 5: Stresses in the submodel at the load position giving the peak effective stress

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 general extrusion is parametrized so that displacements form different spokes can be applied to the submodel.

The pressure distribution from the tire must be adjusted so that its resultant is the intended (about 11 kN). This can be done in different ways. One simple possibility is to run a separate analysis with the tire load as the only loadcase and use an arbitrary load amplitude. The total reaction force is then computed, and the load amplitude is rescaled based on the result. In this model a more sophisticated method is used: An extra Global Equation is added, in which the integral of the distribution load is set equal to the known force. Thus the load amplitude is solved simultaneously with the rest of the problem.

Application Library path: `Structural_Mechanics_Module/Tutorials/rim_submodel`

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 On the **Home** toolbar, 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**
pInflation	2[bar]	2E5 Pa	Inflation pressure
tireLoad	1120[kg]*g_const	1.098E4 N	Load on wheel
spokeNo	0	0	Spoke selection
spokeAngle	spokeNo*2*pi[rad]/5	0 rad	Rotation angle to selected spoke
phiLoad	0	0	Peak load angle
numLpos	4	4	Number of load positions in first sector
angleStep	360/(5*numLpos)	18	Step in peak load angle [deg]
angleLast	angleStep*(numLpos-1)	54	Last peak load angle [deg]

GEOMETRY 1

1. On the **Geometry** toolbar, click **Insert Sequence**.
2. Browse to the application's Application Libraries folder and double-click the file `wheel_rim_geom_sequence.mph`.

Form Composite Domains 1 (cmd1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Composite Domains 1 (cmd1)** and choose **Build Selected**.

DEFINITIONS

Explicit 1

1. On the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for Explicit, type `TireAttachment` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 2–4 and 6 only.

Explicit 2

1. On the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for Explicit, type `PressureSurface` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 2–6 only.

Explicit 3

1. On the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for Explicit, type `FixedToHub` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 8–12 only.

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>Aluminum**.
4. Click **Add to Component** in the window toolbar.
5. On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Aluminum (mat1)

In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

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 **FixedToHub**.

Boundary Load 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
2. In the **Settings** window for Boundary Load, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **PressureSurface**.
4. Locate the **Force** section. From the **Load type** list, choose **Pressure**.
5. In the p text field, type `pInflation`.

DEFINITIONS

Analytic 1 (an1)

1. On the **Home** toolbar, click **Functions** and choose **Local>Analytic**.
2. In the **Settings** window for Analytic, locate the **Definition** section.
3. In the **Expression** text field, type `(abs(atan2(x,y)-z*pi/180)<pi/6)*cos(3*(atan2(x,y)-z*pi/180))`.
4. In the **Arguments** text field, type `x, y, z`.
5. Locate the **Units** section. In the **Arguments** text field, type `m,m,1`.
6. In the **Function** text field, type `Pa`.
7. In the **Function name** text field, type `loadDistr`.

Cylindrical System 2 (sys2)

On the **Definitions** toolbar, click **Coordinate Systems** and choose **Cylindrical System**.

SOLID MECHANICS (SOLID)

Boundary Load 2

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

2 In the **Settings** window for Boundary Load, locate the **Coordinate System Selection** section.

3 From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **TireAttachment**.

5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

-loadAmpl*loadDistr(X,Y,phiLoad)	r
0	phi
0.2*loadAmpl*loadDistr(X,Y,phiLoad)*(2*(Z>0)-1)	a

DEFINITIONS

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 From the **Selection** list, choose **TireAttachment**.

SOLID MECHANICS (SOLID)

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)**	**Description**
loadAmpl	loadAmpl* intop1(loadDistr(X,Y,0)* cos(atan2(X, Y)))-tireLoad	0	0	

4 Locate the **Units** section. Find the **Source term quantity** subsection. From the list, choose **Force load (N)**.

MESH 1

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

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 `0.07`.
5 In the **Minimum element size** text field, type `0.007`.
6 In the **Maximum element growth rate** text field, type `1.8`.
7 Click **Build All**.

STUDY 1

Step 1: Stationary

1 In the **Settings** window for Stationary, click to expand the **Study extensions** section.
2 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
3 Click **Add**.
4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
phiLoad	`range(0,angleStep,angleLast)`	

Solution 1 (sol1)

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

Because of the model's considerable size, you set up an iterative solver that can significantly save on the memory needed for the computations. Use the GMRES iterative solver with Geometric Multigrid as a preconditioner. By default, the solver lowers the order in the discretization of the displacement variables from the default quadratic elements to linear elements. Because the model geometry contains slender elements, use SOR Line as presmoother and postsmoother. SOR Line can handle slender geometries better than the SOR presmoother used by default. Note however that it cannot make full use of the matrix symmetry.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.
3 Right-click **Stationary Solver 1** and choose **Iterative**.
4 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1>Iterative 1** and choose **Multigrid**.

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

6 Right-click **Presmoother** and choose **SOR Line**.

7 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1>Iterative 1>Multigrid 1>Postsmoother** and choose **SOR Line**.

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

RESULTS

Stress (solid)

1 In the **Model Builder** window, click **Stress (solid)**.

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

3 From the **Parameter value (phiLoad)** list, choose **0**.

Surface 1

1 In the **Model Builder** window, under **Results>Stress (solid)** click **Surface 1**.

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

3 From the **Unit** list, choose **MPa**.

4 Click to expand the **Range** section. Select the **Manual color range** check box.

5 In the **Minimum** text field, type `0`.

6 In the **Maximum** text field, type `90`.

To get a better view of the region with the highest stresses (compare with Figure 4), rotate the geometry in the Graphics window and use the Zoom Box tool to obtain a close-up.

DEFINITIONS

View 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **View 2**.

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

3 Select the **Lock camera** check box.

RESULTS

Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

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

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

4 In the **Model Builder** window, click **Stress (solid)**.
5 In the **Settings** window for 3D Plot Group, locate the **Plot Settings** section.
6 From the **View** list, choose **View 2**.
7 On the **Stress (solid)** toolbar, click **Plot**.

Start creating the submodel.

ROOT

On the **Home** toolbar, click **Add Component** and choose **3D**.

GEOMETRY 2

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.
2 On the **Geometry** toolbar, click **Insert Sequence**.
3 Browse to the application's Application Libraries folder and double-click the file `wheel_rim_geom_sequence.mph`.

Form Composite Domains 1 (cmd1)

In the **Model Builder** window, under **Component 2 (comp2)>Geometry 2** right-click **Form Composite Domains 1 (cmd1)** and choose **Build Selected**.

Rotate 1 (rot1)

In the **Model Builder** window, under **Component 2 (comp2)>Geometry 2** right-click **Rotate 1 (rot1)** and choose **Disable**.

Form Composite Domains 1 (cmd1)

In the **Model Builder** window, under **Component 2 (comp2)>Geometry 2** right-click **Form Composite Domains 1 (cmd1)** and choose **Build Selected**.

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 `6e-2`.
4 In the **Depth** text field, type `7e-2`.
5 In the **Height** text field, type `6e-2`.
6 Locate the **Position** section. In the **y** text field, type `6.5e-2`.
7 In the **z** text field, type `6e-2`.

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.

Form Composite Domains 1 (cmd1)

1 Right-click **Form Composite Domains 1 (cmd1)** and choose **Build Selected**.

2 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>Aluminum**.

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

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

DEFINITIONS

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

General Extrusion 1 (genext1)

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

2 In the **Settings** window for General Extrusion, type `from_global` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Selection** list, choose **All domains**.

4 Locate the **Source** section. From the **Source frame** list, choose **Material (X, Y, Z)**.

5 Locate the **Destination Map** section. In the **X-expression** text field, type `X*cos(spokeAngle)-Y*sin(spokeAngle)`.

6 In the **Y-expression** text field, type `Y*cos(spokeAngle)+X*sin(spokeAngle)`.

7 In the **Z-expression** text field, type `Z`.

8 Click to expand the **Advanced** section. From the **Mesh search method** list, choose **Closest point**.

COMPONENT 2 (COMP2)

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

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

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

SOLID MECHANICS 2 (SOLID2)

Fixed Constraint 1

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

2 Select Boundary 3 only.

Prescribed Displacement 1

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

2 Select Boundaries 1 and 6–8 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.from_global(comp1.u*cos(spokeAngle)+ comp1.v*sin(spokeAngle))`.

6 Select the **Prescribed in y direction** check box.

7 In the u_{0y} text field, type `comp1.from_global(comp1.v* cos(spokeAngle)-comp1.u*sin(spokeAngle))`.

8 Select the **Prescribed in z direction** check box.

9 In the u_{0z} text field, type `comp1.from_global(comp1.w)`.

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 **Fine**.

4 From the **Sequence type** list, choose **User-controlled mesh**.

Free Tetrahedral 1

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

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 Select Boundary 5 only.

4 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely fine**.

5 Click **Build All**.

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

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

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

Fetch the displacements from the solution of the global model.

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 **Study 1, Stationary**.

6 From the **Parameter value (phiLoad)** list, choose **All**.

7 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

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

9 Click **Add**.

10 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
spokeNo		

11 Click **Add**.

12 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
spokeNo	`range(0,1,4)`	
phiLoad	`range(0,angleStep,angleLast)`	

Solution 2 (sol2)

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

Solution 2 (sol2)

1 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)** node.

   For the submodel, you also use an iterative solver with GMG as preconditioner. You select the Conjugate gradients solver because it can make full use of the matrix symmetry, thus saving memory.

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

3 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** and choose **Iterative**.

4 In the **Settings** window for Iterative, locate the **General** section.

5 From the **Solver** list, choose **Conjugate gradients**.

6 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1>Iterative 1** and choose **Multigrid**.

7 In the **Settings** window for Multigrid, locate the **General** section.

8 In the **Use hierarchy in geometries** list, select **Geometry 1**.

9 Under **Use hierarchy in geometries**, click **Delete**.

STUDY 1

Solution 1 (sol1)

1 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1>Iterative 1** click **Multigrid 1**.

2 In the **Settings** window for Multigrid, locate the **General** section.

3 In the **Use hierarchy in geometries** list, select **Geometry 2**.

4 Under **Use hierarchy in geometries**, click **Delete**.

Avoid saving a lot of duplicate results for the full geometry.

STUDY 2

Solution 2 (sol2)

1 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** node, then click **Displacement field (Material) (comp1.u)**.
2 In the **Settings** window for Field, locate the **General** section.
3 Clear the **Store in output** check box.
4 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** click **State variable loadAmpl (comp1.ODE1)**.
5 In the **Settings** window for State, locate the **General** section.
6 Clear the **Store in output** check box.
7 On the **Study** toolbar, click **Compute**.

RESULTS

Surface 1

1 In the **Model Builder** window, expand the **Stress (solid2)** node, then click **Surface 1**.
2 In the **Settings** window for Surface, locate the **Expression** section.
3 From the **Unit** list, choose **MPa**.
4 Locate the **Range** section. Select the **Manual color range** check box.
5 In the **Minimum** text field, type `0`.
6 In the **Maximum** text field, type `90`.

Stress (solid2)

1 In the **Model Builder** window, under **Results** click **Stress (solid2)**.
2 In the **Settings** window for 3D Plot Group, locate the **Data** section.
3 From the **Parameter value (spokeNo)** list, choose **0**.
4 From the **Parameter value (phiLoad)** list, choose **18**.

Principal Stress Surface 1

1 On the **Stress (solid2)** toolbar, click **More Plots** and choose **Principal Stress Surface**.
2 In the **Model Builder** window, right-click **Principal Stress Surface 1** and choose **Deformation**.

Principal Stress Surface 1

1 In the **Model Builder** window, under **Results>Stress (solid2)** click **Principal Stress Surface 1**.

2 In the **Settings** window for Principal Stress Surface, click to expand the **Inherit style** section.

3 Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

4 Clear the **Arrow scale factor** check box.

5 Clear the **Color** check box.

6 Clear the **Color and data range** check box.

Stress (solid2)

1 In the **Model Builder** window, under **Results** click **Stress (solid2)**.

2 In the **Settings** window for 3D Plot Group, type `Stress in Submodel` in the **Label** text field.

Again, create a View 3D feature node for the plot

DEFINITIONS

In the **Model Builder** window, under **Component 2 (comp2)** right-click **Definitions** and choose **View**.

View 4

1 In the **Settings** window for View, locate the **View** section.

2 Clear the **Show grid** check box.

In the graphics window rotate the model and capture the highest stressed part.

3 In the **Model Builder** window, click **View 4**.

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

5 Select the **Lock camera** check box.

Apply the view to 3D Plot Group 3.

RESULTS

Stress in Submodel

1 In the **Model Builder** window, under **Results** click **Stress in Submodel**.

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

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

4 On the **Stress in Submodel** toolbar, click **Plot**.

Finally, create an animation, showing the stress history for a complete revolution of the wheel.

Animation 1

1 On the **Results** toolbar, click **Animation** and choose **File**.

2 In the **Settings** window for Animation, locate the **Target** section.

3 From the **Target** list, choose **Player**.

4 Locate the **Scene** section. From the **Subject** list, choose **Stress in Submodel**.

5 Locate the **Animation Editing** section. From the **Loop over** list, choose **All solutions**.

Replace next instruction with: In the **Parameter values** list, chose all values.

6 In the **Parameter values (phiLoad,spokeNo)** list, choose **1: phiLoad=0, spokeNo=0**, **2: phiLoad=18, spokeNo=0**, **3: phiLoad=36, spokeNo=0**, **4: phiLoad=54, spokeNo=0**, **5: phiLoad=0, spokeNo=1**, **6: phiLoad=18, spokeNo=1**, **7: phiLoad=36, spokeNo=1**, **8: phiLoad=54, spokeNo=1**, **9: phiLoad=0, spokeNo=2**, **10: phiLoad=18, spokeNo=2**, **11: phiLoad=36, spokeNo=2**, **12: phiLoad=54, spokeNo=2**, **13: phiLoad=0, spokeNo=3**, **14: phiLoad=18, spokeNo=3**, **15: phiLoad=36, spokeNo=3**, **16: phiLoad=54, spokeNo=3**, **17: phiLoad=0, spokeNo=4**, **18: phiLoad=18, spokeNo=4**, **19: phiLoad=36, spokeNo=4**, and **20: phiLoad=54, spokeNo=4**.

7 Locate the **Frames** section. From the **Frame selection** list, choose **All**.

8 Right-click **Animation 1** and choose **Play**.

STUDY 1

Step 1: Stationary

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

2 In the **Settings** window for Stationary, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for the **Solid Mechanics 2 (solid2)** interface.

Fundamental Eigenfrequency of a Rotating Blade

Introduction

High rotational speed in, for example, gas turbine machinery can result in centrifugal forces of considerable magnitude. This example studies the effects of such forces on the natural frequencies of the structure.

The fictitious forces induced by the rotation give rise to two counteracting effects: *stress-stiffening* and *spin-softening* (or *centrifugal softening*). The former is caused by the stationary stress field created by the centrifugal force and acts to increase the stiffness of the body, and so increase its resonance frequencies. At the same time, any radial displacement away from the axis of rotation increases the centrifugal force, while motion toward the axis decreases it. This effect therefore tends to amplify any radial motion, which is the opposite of stiffening—hence *spin-softening*. Which mechanism is dominating depends on the shape of the particular mode.

In some rotating systems, *Coriolis forces* can also play an important role. These apparent forces split some natural modes of vibration into one co-rotating and one counter-rotating *precessing* mode. In particular, this happens for natural modes that include bending of the axis of rotation. For most modes of vibration, however, the Coriolis frequency shifts are small.

Model Definition

In the first part of the modeling, you conduct a modal analysis of a blade mounted on a rigid rotating cylinder. The rotational speed is 3000 rpm (100π rad/s) about the global y-axis. The goal is to compare the fundamental frequency in three cases:

- A basic modal analysis at 0 rpm, thus excluding any fictitious force effects.
- A modal analysis including stress-stiffening and spin-softening at 3000 rpm.
- A complete analysis also including Coriolis effects at 3000 rpm.

The result in the first two cases can be compared directly to an analytical reference solution. Once you are satisfied that the solutions are accurate in all cases, the second step proceeds to compute the first few natural frequencies for a large range of rotational speeds.

FICTITIOUS FORCES IN ROTATING COORDINATE SYSTEMS

The spin-softening and Coriolis effects both ultimately arise because the standard form of the laws governing the deformation of solid objects only hold in *inertial*, that is, nonaccelerating, coordinate systems. When you build and simulate a model in a rotating coordinate system, you must extend the basic laws of motion to account for the acceleration of the system itself.

In COMSOL Multiphysics, you can model the frame acceleration effects by using the Rotating Frame domain feature.

In this particular case, the axis of rotation is the y-axis, see Figure 1. The only explicit boundary condition in the model is on the blade's base, which is fixed to the axis.

STRESS-STIFFENING AND NONLINEAR EFFECTS

To include the stress-stiffening effects in the model, you activate the large deformation option. This redefines the strain measure to include second-order terms, which make the strain-displacement relation nonlinear. The resulting strain measure is called Green strain.

For the stress stiffening to have any effect on the natural frequencies, you must first solve for the stationary stresses from the centrifugal loading in a stationary analysis. Then you perform the modal analysis using the static solution as a linearization point.

Note that the spin-softening affects not only the modal solution but also adds a positive feedback to the stationary nonlinear solution. If the structure has a natural mode for which spin-softening dominates over the stress-stiffening effect, the natural frequency of this mode becomes zero for some rotational frequency. This means that the structure looses all stiffness, and that no stable solution exists for higher angular velocities. So if the nonlinear solver does not converge for a very fast rotation, it should come as no surprise.

MATERIAL AND GEOMETRICAL PROPERTIES

The blade's material and geometrical properties are given in the table below (see also Figure 1):

MATERIAL PROPERTIES			GEOMETRIC PROPERTIES		
E	$221 \cdot 10^9$ Pa	Young's modulus	r	10 in	Cylinder radius
ν	0	Poisson's ratio	L	5 in	Blade length
ρ	7850 kg/m^3	Density	d	0.0625 in	Blade thickness
			b	1 in	Blade width

The approximate equation for the natural frequency in Ref. 1 does not include the Poisson's ratio. To facilitate easy comparison, the Poisson's ratio is set to zero in the modeling.

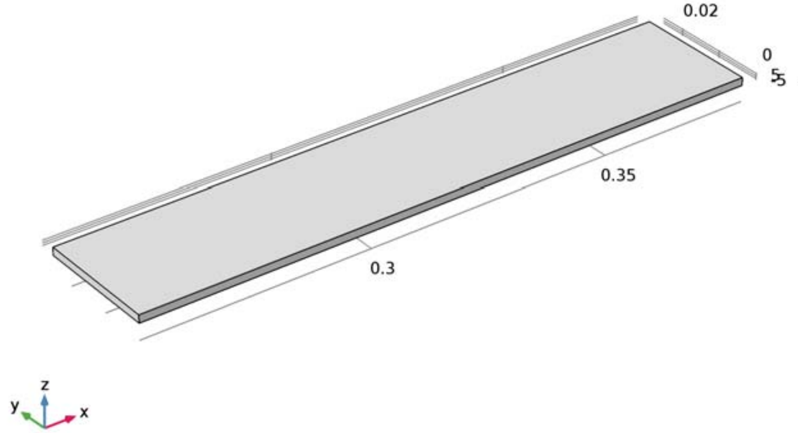


Figure 1: Blade geometry.

ANALYTICAL SOLUTIONS

You can compare COMSOL Multiphysics' results to an accurate analytical approximation. Using Euler beam theory, which is sufficiently accurate for the present geometry, the fundamental angular frequency for the nonrotating case is given by

$$w_0 = 1.875^2\sqrt{\frac{EI}{mL^4}}$$

where E is the Young's modulus, I is the area moment of inertia, m is the mass per unit length, and L is the blade length.

For a rotating blade attached to a rigid axis with radius r and rotating with angular frequency Ω, the fundamental angular frequency according to Ref. 1 is

$$w_\Omega = w_0\sqrt{1+\frac{(m\Omega^2r^4)}{EI}\cdot\left[\frac{1}{8}\left(\frac{L}{r}\right)^3+\frac{1}{10.6}\left(\frac{L}{r}\right)^4-\frac{(\cos\phi)^2}{12.45}\left(\frac{L}{r}\right)^4\right]}$$

This formula assumes a blade of uniform cross section slanted at the so-called stagger angle ϕ. In the present case, $\phi = 0$. The author states without proof that the Coriolis effect is negligible. You can check the validity of this statement by comparing the finite element solutions with and without the Coriolis term included.

Results and Discussion

The following table shows a comparison between the analytical results and the results computed in COMSOL Multiphysics:

Table 0-1:

CASE	ANALYTICAL RESULT	COMSOL RESULTS
0 rpm	84.4 Hz	84.4 Hz
3000 rpm, no Coriolis effect	123.6 Hz	124 Hz
3000 rpm, with Coriolis effect	-	124 Hz

Apparently, the Coriolis effect is negligible as stated in Ref. 1. Because including the Coriolis terms makes the system of equations nonsymmetric—and therefore more expensive to solve—there is, in general, no need to include the Coriolis effect when modeling systems with a rigid axis.

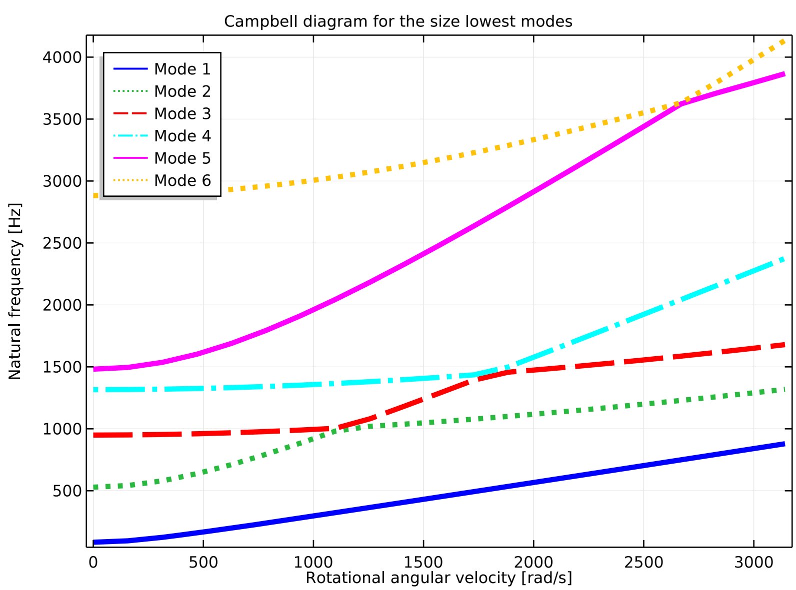


Figure 2: Campbell diagram created from the parametric sweep output file. Notice the crossing modes, which are misinterpreted because of the sorting of the natural frequencies.

The Campbell diagram, see Figure 2, shows that the first 6 natural frequencies all increase with increasing rotational speed. However, the balance between stress stiffening and spin softening is different for the different modes. Stress stiffening is more pronounced for modes 2 and 5 (referring to the mode order at zero rotational frequency) compared to the others. Therefore, the order of the modes in the result file—which is sorted on mode frequency—changes with increasing rotational velocity.

Notes About the COMSOL Implementation

Note that the eigenvalue solver always finds the requested number of modes but not necessarily the ones with lowest eigenfrequencies. Sometimes it can converge on a higher-order mode instead. A jagged appearance of the highest frequency mode in the Campbell diagram is a result of the solver's picking a higher mode instead of requested highest mode for some rotational frequencies. In order to avoid this behavior it is recommended to solve for one additional eigenfrequency in the study and display results for all eigenfrequencies except the one of the highest mode.

When eigenfrequencies are calculated, numerical noise can occasionally be seen via the imaginary part of the eigenfrequency. If the imaginary part is several orders of magnitude smaller than the real part then use the real part as the eigenfrequency and disregard the imaginary part.

Reference

1. W. Carnegie, "Vibrations of Rotating Cantilever Blading," *J. Mech. Engrg Sci.*, vol. 1, no. 3, London, 1959.

Application Library path: `Structural_Mechanics_Module/Dynamics_and_Vibration/rotating_blade`

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>Prestressed Analysis, Eigenfrequency**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, 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
Omega	0*pi[rad/s]	0 rad/s	Angular velocity
L	5[in]	0.127 m	Blade length
b	1[in]	0.0254 m	Blade width
d	0.0625[in]	0.001588 m	Blade thickness
E0	221[GPa]	2.21E11 Pa	Young's modulus
rho0	7850[kg/m^3]	7850 kg/m³	Density
r	10[in]	0.254 m	Inner radius
mpl	rho0*b*d	0.3165 kg/m	Mass per unit length
I	b*d^3/12	8.468E-12 m^4	Area moment of inertia
f_0	1.875^2*sqrt(E0*I/(mpl*L^4))/(2*pi)	84.35 1/s	Euler beam resonance frequency
f_ref	f_0*sqrt(1+mpl*Omega^2*r^4/(E0*I)*((L/r)^3/8+(L/r)^4*(1/10.6-1/12.45)))	84.35 rad/s	Reference resonance frequency

GEOMETRY 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 **Width** text field, type `L`.

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

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

6 Locate the **Position** section. In the **x** text field, type `r`.

7 In the **z** text field, type `-d/2`.

8 Right-click **Block 1 (blk1)** and choose **Build Selected**.

Form Union (fin)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

MATERIALS

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

Material 1 (mat1)

1 Select Domain 1 only.

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

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	E0	Pa	Basic
Poisson's ratio	nu	0	1	Basic
Density	rho	rho0	kg/m³	Basic

SOLID MECHANICS (SOLID)

Rotating Frame 1

1 On the **Physics** toolbar, click **Domains** and choose **Rotating Frame**.

2 Select Domain 1 only.

3 In the **Settings** window for Rotating Frame, locate the **Rotating Frame** section.

4 From the **Axis of rotation** list, choose **y-axis**.

5 In the Ω text field, type `Omega`.

Fixed Constraint 1

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

2 Select Boundary 1 only.

MESH 1

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

Mapped 1

1 Select Boundary 4 only.

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

Distribution 1

Select Edge 4 only.

Mapped 1

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

Distribution 2

1 Select Edge 5 only.

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

3 In the **Number of elements** text field, type 25.

Mapped 1

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

2 In the **Settings** window for Mapped, click **Build Selected**.

3 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

Swept 1

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

Distribution 1

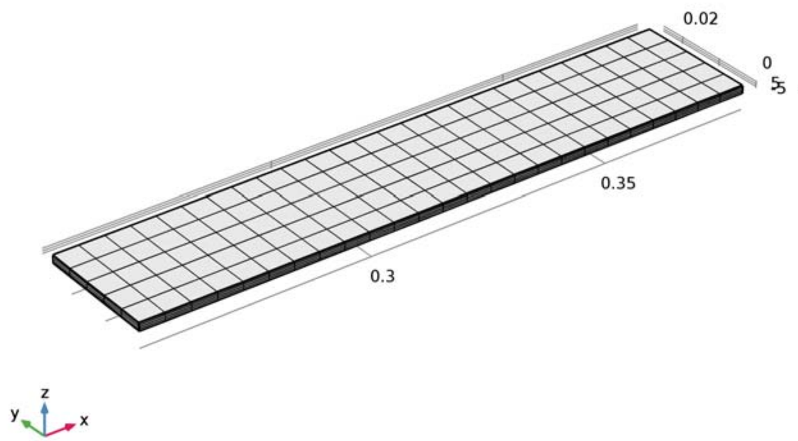
1 Select Domain 1 only.

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

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

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

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



STUDY 1

The default solver settings consists of a sequence of a stationary study step followed by an eigenfrequency analysis. COMSOL Multiphysics stores the solution from the stationary solver and uses it as the linearization point for the eigenfrequency analysis. Together with the fact that a geometrically nonlinear analysis is pre-selected, this ensures that effects of stress-stiffening and spin-softening are included in the calculated eigenfrequencies.

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

RESULTS

Surface 1

The default plot shows the scaled deformation for the first mode of vibration for the nonrotating blade.

The first mode occurs at 84.4 Hz. This agrees well with the analytical solution, which you can compute as described below.

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 **Eigenfrequency selection** list, choose **First**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Global Definitions>Parameters>f_ref - Reference resonance frequency**.

5 Click **Evaluate**.

The table in the Table window shows that the analytical value of the first mode is also 84.4 Hz.

Next, solve for the eigenfrequencies of the blade rotating at 3000 RPM.

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
Omega	100*pi[rad/s]	314.2 rad/s	Angular velocity

STUDY 1

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

RESULTS

Mode Shape (solid)

The default plot now shows the scaled deformation for the first mode of vibration when the blade is rotating at 3000 RPM. In this case the first eigenmode occurs at 124 Hz. This result clearly shows the effect of stress-stiffening.

Global Evaluation 1

1 In the **Model Builder** window, under **Results>Derived Values** click **Global Evaluation 1**.

2 In the **Settings** window for Global Evaluation, click **Evaluate**.

The table in the Table window shows that the analytical value of the first mode is 123.6 Hz.

In the next section, you will also include the Coriolis effect and solve for the eigenfrequencies of the blade rotating at 3000 RPM.

SOLID MECHANICS (SOLID)

Rotating Frame 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Rotating Frame 1**.
2 In the **Settings** window for Rotating Frame, locate the **Rotating Frame** section.
3 Select the **Coriolis force** check box.

STUDY 1

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

RESULTS

Mode Shape (solid)

The default plot now shows the scaled deformation for the first mode of vibration when the blade is rotating at 3000 RPM. In this case the first eigenmode also occurs at 124 Hz. This result shows that enabling the Coriolis effect in the equations does not change the first natural frequency appreciably compared to the previous case.

In the next section, you perform a parametric study of the effect of rotational velocity and use the results to generate a Campbell diagram.

SOLID MECHANICS (SOLID)

Rotating Frame 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Rotating Frame 1**.
2 In the **Settings** window for Rotating Frame, locate the **Rotating Frame** section.
3 Clear the **Coriolis force** check box.

STUDY 1

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
Omega	pi*range(0,50,1000)	

Step 2: Eigenfrequency

1 In the **Model Builder** window, under **Study 1** click **Step 2: Eigenfrequency**.

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

3 In the **Search for eigenfrequencies around** text field, type `1000`.

Compute one additional eigenfrequency to avoid the possibility that the solver converges on a higher-order mode.

4 Select the **Desired number of eigenfrequencies** check box.

5 In the associated text field, type `7`.

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

RESULTS

Mode Shape (solid) 1

The default plot shows the scaled total displacement for the first mode for a rotational velocity of 30,000 RPM as a surface plot in Mode Shape (solid) 1. This mode occurs at a frequency of 723 Hz.

You can use the drop-down menu for Parameter value (Omega) and Eigenfrequency to find out the six lowest eigenfrequencies for each rotational velocity. A more elegant way to visualize this result is by creating a Campbell diagram.

1D Plot Group 3

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 **Study 1/Parametric Solutions 1 (sol3)**.

4 From the **Eigenfrequency selection** list, choose **Manual**.

Display 6 lowest eigenfrequencies.

5 In the **Eigenfrequency (Hz) (1-7)** text field, type `range(1,1,6)`.

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

7 In the **Title** text area, type `Campbell diagram for the size lowest modes`.

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

9 In the associated text field, type `Rotational angular velocity [rad/s]`.

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

11 In the associated text field, type `Natural frequency [Hz]`.

12 Locate the **Axis** section. Select the **Manual axis limits** check box.

13 In the **x minimum** text field, type `-100`.

14 In the **x maximum** text field, type `4000`.

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

16 In the **y maximum** text field, type `4500`.

Global 1

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

2 In the **Settings** window for Global, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Global>solid.freq - Frequency**.

3 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Outer solutions**.

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 **Cycle**.

5 In the **Width** text field, type `4`.

6 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
`Mode 1`
`Mode 2`
`Mode 3`
`Mode 4`
`Mode 5`
`Mode 6`

1D Plot Group 3

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

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

3 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

4 On the **Campbell diagram** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.2a

Piezoelectric Shear-Actuated Beam

Introduction

This example performs a static analysis on a piezoelectric actuator based on the movement of a cantilever beam, using the Piezoelectric Devices predefined multiphysics interface. Inspired by work done by V. Piefort (Ref. 1) and A. Benjeddou (Ref. 2), it models a sandwich beam using the shear mode of the piezoelectric material to deflect the tip.

Model Definition

GEOMETRY

The model consists of a 100-mm long sandwiched cantilever beam (Figure 1).

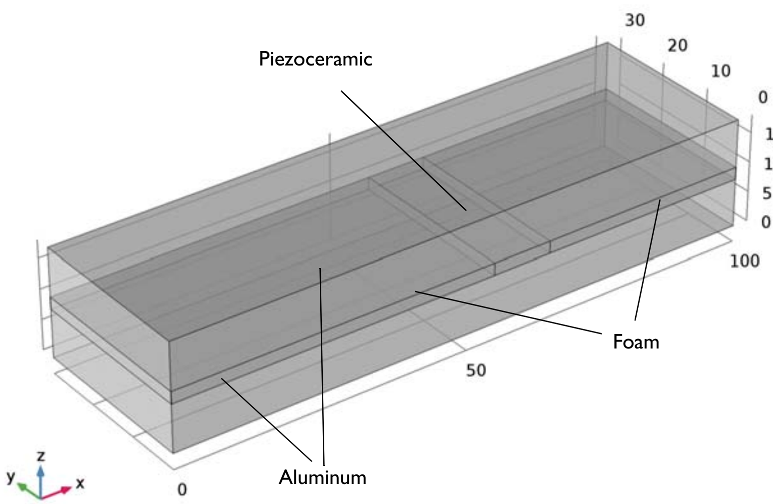


Figure 1: The shear bender geometry. Note that a piezoceramic material replaces part of the foam core.

This beam is composed of a 2-mm thick flexible foam core sandwiched by two 8-mm thick aluminum layers. Further, the device replaces part of the foam core with a 10-mm long piezoceramic actuator that is positioned between x = 55 mm and x = 65 mm. The cantilever beam is orientated along the global x-axis.

BOUNDARY CONDITIONS

- *Solid Mechanics:* the cantilever beam is fixed at its surfaces at $x = 0$; all other surfaces are free.
- *Electrostatics:* The system applies a 20 V potential difference between the top and bottom surfaces of the piezoceramic domain (Figure 2). This gives rise to an electric field perpendicular to the poling direction (x direction) and thus induces a transverse shear strain.

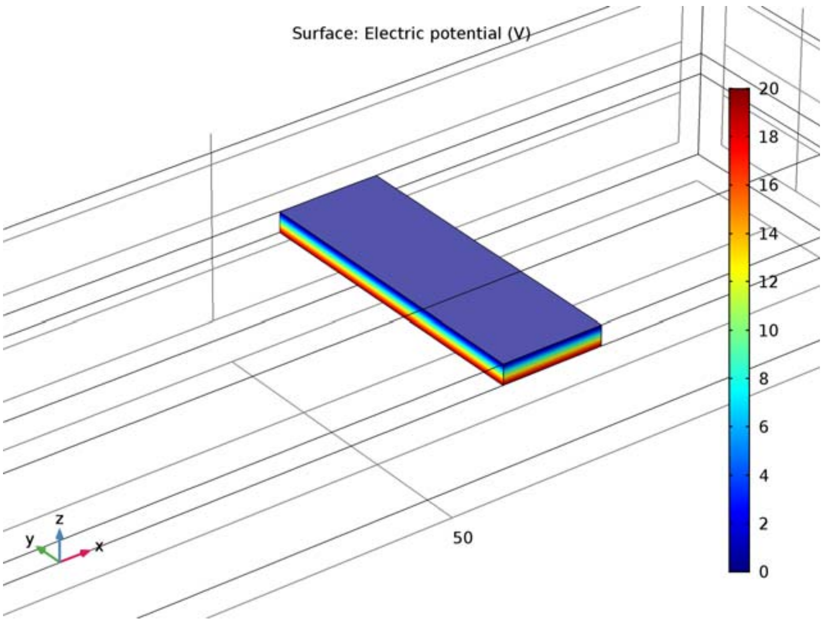


Figure 2: Applied voltage through the piezoelectric material

MATERIAL PROPERTIES

The following table lists the material properties for the aluminum layers and the foam core:

PROPERTY	ALUMINUM	FOAM	PIEZOCERAMIC
E	70 GPa	35.3 MPa	-
ν	0.35	0.383	-
ρ	2700 kg/m^3	32 kg/m^3	7500 kg/m^3

Aluminum is available as a predefined material, whereas you must define the foam material manually.

The piezoceramic material in the actuator, PZT-5H, is already defined in the material library. Thus, you do not need to enter the components of the elasticity matrix, c_E, the piezoelectric coupling matrix, e, or the relative permittivity matrix, ε_{rS}.

Results

The shear deformation of the piezoceramic core layer and the flexible foam layer induce a bending action. Figure 3 shows the resulting tip deflection. The model calculates this deflection as 83 nm, a result that agrees well with those of Ref. 1 and Ref. 2.

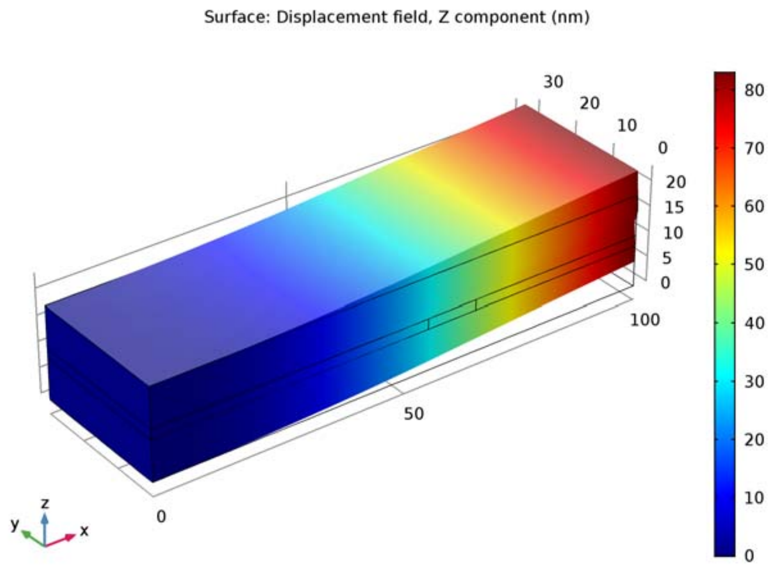


Figure 3: Tip deflection with the piezoceramic positioned at x = 60 mm.

Notes About the COMSOL Implementation

The matrix components for the piezoelectric material properties refer to a coordinate system, where the poling direction is the z direction. Because the poling direction of the piezoceramic actuator in this model is aligned with the x-axis, you need to use a local coordinate system in the material settings to rotate the piezoceramic material.

More specifically, you define a local coordinate system that is rotated 90 degrees about the global y-axis. Then, you use this coordinate system in the piezoelectric material settings to rotate the material so that the polarization direction is aligned with the x-axis (Figure 4).

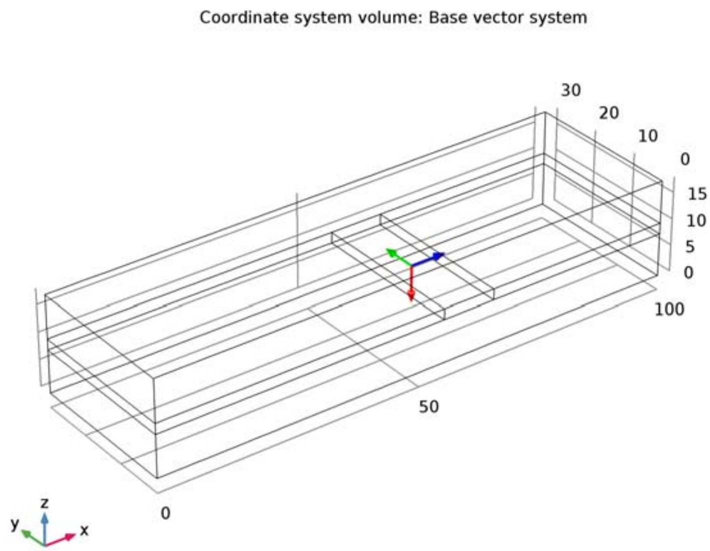


Figure 4: Definition of local coordinate system to define the piezoelectric orientation. The material is poled along the local x3 direction (blue arrow).

References

1. V. Piefort, *Finite Element Modelling of Piezoelectric Active Structures*, Ph.D. thesis, Université Libre de Bruxelles, Belgium, Dept. Mechanical Engineering and Robotics, 2001.

2. A. Benjeddou, M.A. Trindade, and R. Ohayon, *A Unified Beam Finite Element Model for Extension and Shear Piezoelectric Actuation Mechanisms*, CNAM (Paris, France), Structural Mechanics and Coupled Systems Laboratory, 1997.

Application Library path: `Structural_Mechanics_Module/Piezoelectric_Effects/shear_bender`

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>Piezoelectric Devices**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.

6 Click **Done**.

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**.

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 100.

4 In the **Depth** text field, type 30.

5 In the **Height** text field, type 18.

Block 2 (blk2)

1 Right-click **Block 1 (blk1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Block**.

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

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

5 In the **Depth** text field, type 30.

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

7 Locate the **Position** section. In the **z** text field, type 8.

8 Click to expand the **Layers** section. Find the **Layer position** subsection. Select the **Left** check box.

9 Clear the **Bottom** check box.

10 In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	55
Layer 2	10

11 Click **Build All Objects**.

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

The model geometry is now complete.

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

The geometry in the Graphics window should now look like that in Figure 1.

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

DEFINITIONS

Define a coordinate system whose third axis is aligned with the global *x*-axis, that is, the polarization direction of the piezoceramic material. Choose the second axis to be parallel to the global y-axis.

Base Vector System 2 (sys2)

1 On the **Definitions** toolbar, click **Coordinate Systems** and choose **Base Vector System**.

2 In the **Settings** window for Base Vector System, locate the **Settings** section.

3 Find the **Base vectors** subsection. In the table, enter the following settings:

	x	y	z
x1	0	0	-1
x3	1	0	0

Leave the other components at their default values. You will use this coordinate system in the piezoelectric material settings.

4 Find the **Simplifications** subsection. Select the **Assume orthonormal** check box.

ELECTROSTATICS (ES)

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

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

3 Click **Clear Selection**.

4 Select Domain 4 only.

SOLID MECHANICS (SOLID)

On the **Physics** toolbar, click **Electrostatics (es)** and choose **Solid Mechanics (solid)**.

Piezoelectric Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Piezoelectric Material 1**.

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

3 Click **Clear Selection**.

4 Select Domain 4 only.

5 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Base Vector System 2 (sys2)**.

MATERIALS

For the aluminum layers, use a library material.

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 **MEMS>Metals>Al - Aluminum / Aluminium**.

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

MATERIALS

Al - Aluminum / Aluminium (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Al - Aluminum / Aluminium (mat1)**.

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

3 Click **Clear Selection**.

4 Select Domains 1 and 3 only.

For the foam core, specify the material properties by hand.

Material 2 (mat2)

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 `Foam` in the **Label** text field.

3 Select Domains 2 and 5 only.

Foam (mat2)

Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	`35.3[MPa]`	Pa	Basic
Poisson's ratio	nu	`0.383`	1	Basic
Density	rho	32	kg/m³	Basic

ADD MATERIAL

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

The piezoceramic PZT-5H is available as a predefined material.

2 In the tree, select **Piezoelectric>Lead Zirconate Titanate (PZT-5H)**.

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

MATERIALS

Lead Zirconate Titanate (PZT-5H) (mat3)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Materials>Foam (mat2)** node, then click **Component 1 (comp1)>Materials>Lead Zirconate Titanate (PZT-5H) (mat3)**.

2 Select Domain 4 only.

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

SOLID MECHANICS (SOLID)

Fixed Constraint 1

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

2 Select Boundaries 1, 4, and 7 only.

ELECTROSTATICS (ES)

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

Electric Potential 1

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

2 Select Boundary 16 only.

3 In the **Settings** window for Electric Potential, locate the **Electric Potential** section.

4 In the V_0 text field, type 20.

Ground 1

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

2 Select Boundary 17 only.

MESH 1

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

Swept 1

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

Distribution 1

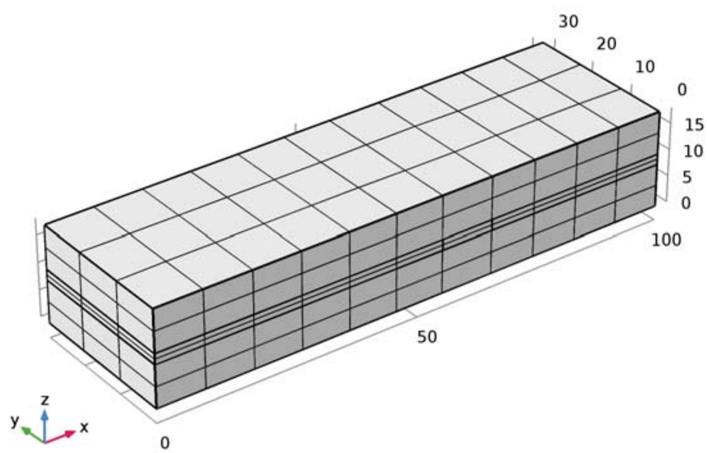
1 In the **Settings** window for Distribution, locate the **Distribution** section.

2 In the **Number of elements** text field, type 2.

3 Click **Build All**.

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

The mesh consists of 198 hexahedral elements.



STUDY 1

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

RESULTS

Stress (solid)

Replace the default stress plot by displacement to reproduce the plot shown in Figure 3.

1. In the **Model Builder** window, under **Results** click **Stress (solid)**.
2. In the **Settings** window for 3D Plot Group, type `Displacement (solid)` in the **Label** text field.

Surface 1

1. In the **Model Builder** window, expand the **Results>Displacement (solid)** 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)>w - Displacement field, Z component**.
3. Locate the **Expression** section. From the **Unit** list, choose **nm**.
4. On the **Displacement (solid)** toolbar, click **Plot**.
5. Click the **Go to Default 3D View** button on the **Graphics** toolbar.

Multislice 1

1. In the **Model Builder** window, expand the **Electric Potential (es)** node.
2. Right-click **Multislice 1** and choose **Delete**.

Electric Potential (es)

In the **Model Builder** window, under **Results** right-click **Electric Potential (es)** and choose **Surface**.

Surface 1

1. 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>Electric>V - Electric potential**.
2. On the **Electric Potential (es)** toolbar, click **Plot**.

   Zoom in to find a plot similar to Figure 2.
3. Click the **Zoom In** button on the **Graphics** toolbar.

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

Show the base vector that defines the polarization of the piezoelectric material, shown on Figure 4.

3D Plot Group 3

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

2 In the **Settings** window for 3D Plot Group, type `PZT coordinate system` in the **Label** text field.

Coordinate System Volume 1

1 On the **PZT coordinate system** toolbar, click **More Plots** and choose **Coordinate System Volume**.

2 In the **Settings** window for Coordinate System Volume, locate the **Coordinate System** section.

3 From the **Coordinate system** list, choose **Base Vector System 2 (sys2)**.

4 Locate the **Positioning** section. Find the **x grid points** subsection. From the **Entry method** list, choose **Coordinates**.

5 In the **Coordinates** text field, type `60`.

6 Find the **y grid points** subsection. In the **Points** text field, type `1`.

7 Find the **z grid points** subsection. In the **Points** text field, type `1`.

8 On the **PZT coordinate system** toolbar, click **Plot**.

Contact Analysis of a Snap Hook Using a Penalty Formulation

Introduction

Several numerical algorithms exist to handle structural contact problems. The one used as default in COMSOL Multiphysics is based on the so-called Augmented Lagrangian method.

Alternatively, you can use the Penalty method. It is based on the penalty factor formulation related to the penetration of the solid parts in contact. In some cases, this approach can result in faster computations and provide smoother convergence when compared to the Augmented Lagrangian method. The penalty method is however not suitable if accurate values of the contact pressure are important.

In this example, the purpose of the contact modeling is only to transmit correct contact forces in a global sense into the model, which is a situation well suited for using the penalty method.

Model Definition

The mechanical problem consists of the insertion of a snap hook into its slot. The objective is to compute the force needed to place the hook in the slot and then to remove it.

The geometry is shown in Figure 1. Due to the symmetry, you can study only a half of the original snap hook geometry.

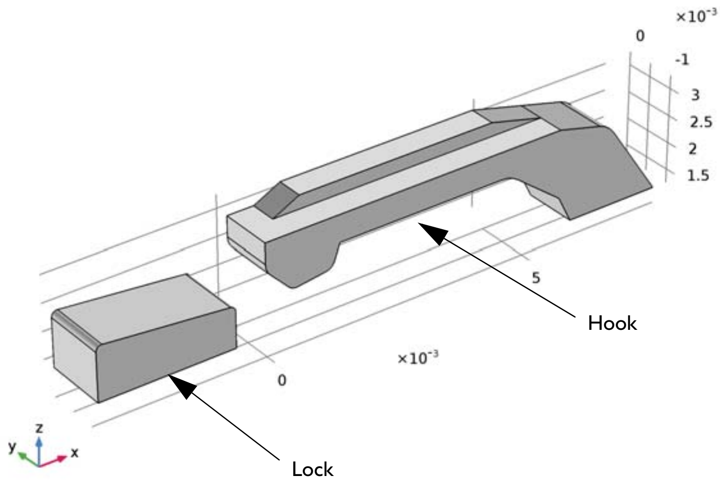


Figure 1: Geometry of the modeled half of the snap hook and locking mechanism.

MATERIAL PROPERTIES

The locking mechanism is made of steel. The hook is made of a modified nylon material; the material parameters are given in the following table:

MATERIAL PARAMETER	VALUE
Young's modulus	10 GPa
Poisson's ratio	0.35

Since the steel is much stiffer than the nylon, it can be treated as rigid. Therefore, you use a linear elastic material model for the hook and rigid domain material model for the locking mechanism.

BOUNDARY CONDITIONS

- The locking mechanism is fixed.

A prescribed displacement boundary condition is applied at the bottom surface of the hook. The displacement in the x direction is gradually changed by using the parametric solver; the other two displacement components are zero.

- Two side boundaries within the xz plane use symmetry boundary conditions.

- All the other boundaries are free boundaries. However, several of them are selected as parts of a contact pair with the destination side being on the hook surface.

Results

Figure 2 shows that the maximum penetration is less than 8 microns during the entire analysis. This is a good accuracy when compared to the geometry size.

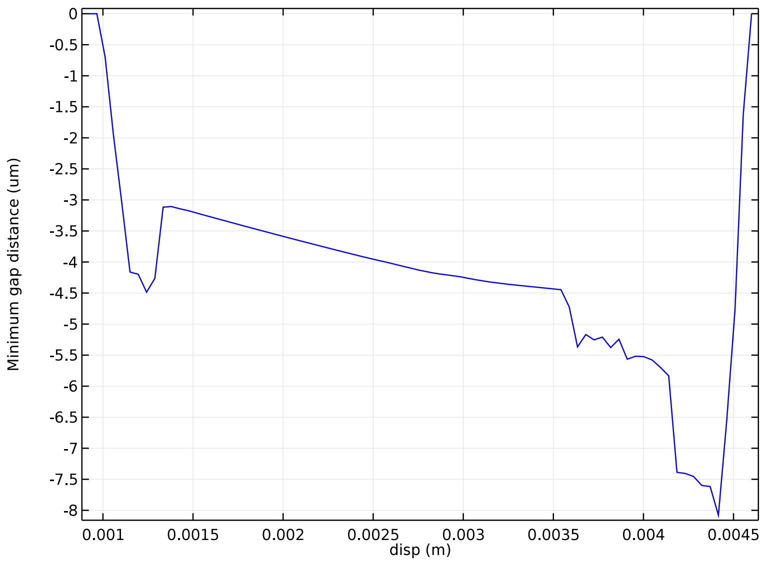


Figure 2: Evolution of the maximum penetration for all position of the hook.

Note that the penetration is not constant since the contact force varies depending on the position of the hook

The maximum effective stress levels are found at parameter step 0.83, which is just before the hook enters the slot, see Figure 3.

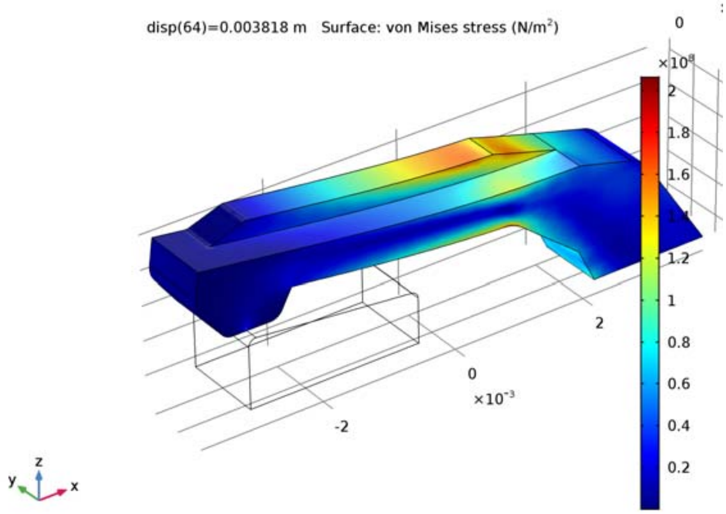


Figure 3: The effective stress levels in the hook just before it enters the slot.

Figure 4 shows the force required for the insertion and removal of the hook versus the parameter value. The hook is in its slot at the end of the simulation.

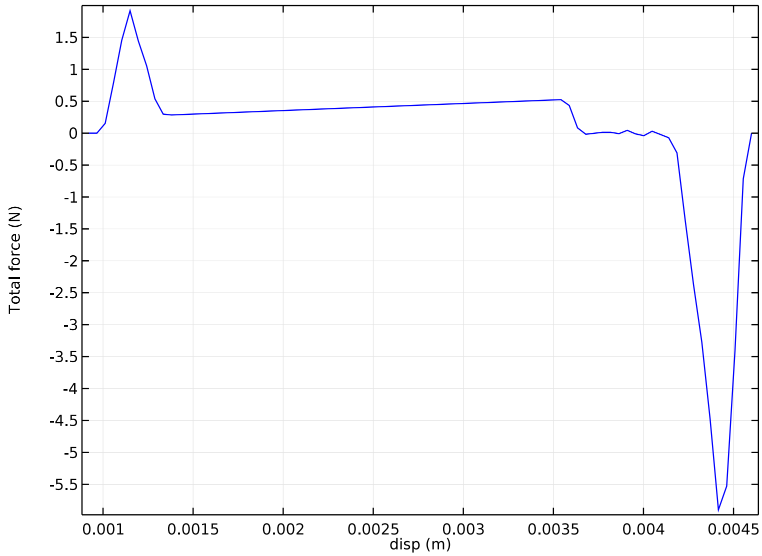


Figure 4: The mounting force as a function of the parameter step.

Application Library path: `Structural_Mechanics_Module/Contact_and_Friction/snap_hook_penalty`

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 On the **Home** toolbar, 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
`Displ_max`	`4.6[mm]`	0.0046 m	`Maximum hook displacement`
`disp`	`0`	0	`Prescribed hook displacement`

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 application's Application Libraries folder and double-click the file `snap_hook_penalty.mphbin`.

5 Click **Import**.

Rotate 1 (rot1)

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

2 Select the object **imp1** only.

3 In the **Settings** window for Rotate, locate the **Rotation Angle** section.

4 In the **Rotation** text field, type `90`.

5 Locate the **Axis of Rotation** section. From the **Axis type** list, choose **Cartesian**.

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

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

8 Right-click **Rotate 1 (rot1)** and choose **Build Selected**.

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

DEFINITIONS

Contact Pair 1 (p1)

1 On the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 Select Boundaries 4, 6, and 7 only.

3 In the **Settings** window for Pair, locate the **Destination Boundaries** section.

4 Select the **Active** toggle button.

5 Select Boundaries 14, 15, 20, 22, and 23 only.

MATERIALS

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

Material 1 (mat1)

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

2 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	`10[GPa]`	Pa	Basic
Poisson's ratio	nu	`0.35`	1	Basic
Density	rho	`1150[kg/m^3]`	kg/m^3	Basic

SOLID MECHANICS (SOLID)

Rigid Domain 1

1 On the **Physics** toolbar, click **Domains** and choose **Rigid Domain**.

2 Select Domain 1 only.

3 Right-click **Rigid Domain 1** and choose **Fixed Constraint**.

Prescribed Displacement 1

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

2 Select Boundary 30 only.

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

4 Select the **Prescribed in x direction** check box.

5 Select the **Prescribed in y direction** check box.

6 Select the **Prescribed in z direction** check box.

7 In the u_{0x} text field, type `-disp`.

Symmetry 1

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

2 Select Boundaries 5, 13, and 19 only.

Contact 1

1 On the **Physics** toolbar, in the Boundary section, click **Pairs** and choose **Contact**.

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

3 In the **Pairs** list, select **Contact Pair 1 (p1)**.

4 Locate the **Contact Pressure Method** section. From the list, choose **Penalty**.

5 Locate the **Penalty Factor** section. In the p_n text field, type `solid.cnt1.E_char/solid.hmin_dst/10`.

MESH 1

Add a structured mesh on the contact destination boundaries.

Mapped 1

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

2 Select Boundaries 14, 15, 20, 22, and 23 only.

Distribution 1

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

2 Select Edges 27, 28, 39, and 48 only.

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

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

Mapped 1

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

Distribution 2

1 Select Edges 31 and 45 only.

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

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

4 Click **Build Selected**.

Convert the quad mesh to triangles so that the rest of the geometry can be meshed using the free tetrahedral method.

Convert 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Convert**.
2 In the **Settings** window for Convert, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundaries 14, 15, 20, 22, and 23 only.
5 Click **Build Selected**.

Free Tetrahedral 1

1 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 Select Domains 2 and 3 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 Click the **Custom** button.
4 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
5 In the associated text field, type `4e-4`.

Free Tetrahedral 1

Right-click **Free Tetrahedral 1** and choose **Size**.

Size 2

Refine the mesh on boundary where high stresses are expected.

1 In the **Settings** window for Size, locate the **Geometric Entity Selection** section.
2 From the **Geometric entity level** list, choose **Boundary**.
3 Select Boundary 25 only.
4 Locate the **Element Size** section. Click the **Custom** button.
5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
6 In the associated text field, type `1e-4`.
7 Click **Build Selected**.

   As the groove is modeled as rigid, you only need to represent the surface that is in contact with the hook accurately.

Free Triangular 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2 Select Boundary 2 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1** and choose **Size**.

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

3 From the **Predefined** list, choose **Extremely coarse**.

Free Triangular 1

Right-click **Free Triangular 1** and choose **Distribution**.

Distribution 1

1 Select Edges 5 and 13 only.

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

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

4 Click **Build Selected**.

5 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

Swept 1

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

Distribution 1

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

2 In the **Number of elements** text field, type `1`.

3 Click **Build All**.

STUDY 1

Step 1: Stationary

Set up an auxiliary continuation sweep for the `disp` parameter.

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

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

3 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
disp	range(0.2,1e-2,1)*Displ_max	

6 Click to expand the **Results while solving** section. Locate the **Results While Solving** section. Select the **Plot** check box.

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, then click **Parametric 1**.

4 In the **Settings** window for Parametric, click to expand the **Continuation** section.

5 From the **Predictor** list, choose **Linear**.

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

RESULTS

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
solid.gapmin_p1*(solid.gapmin_p1<0)	um	Minimum gap distance

4 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

2 Click the right end of the **Display Table 1 - Global Evaluation 1 (solid.gapmin_p1*(solid.gapmin_p1<0))** split button in the window toolbar.

3 From the menu, choose **Table Graph**.

RESULTS

Surface Integration 1

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

2 Select Boundary 30 only.

3 In the **Settings** window for Surface Integration, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
`-2*solid.RFx`	`N`	`Total force`

5 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

2 Click the right end of the **Display Table 1 - Global Evaluation 1 (solid.gapmin_p1*(solid.gapmin_p1<0))** split button in the window toolbar.

3 From the menu, choose **Table Graph**.

Created in COMSOL Multiphysics 5.2a

Thermal Microactuator

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.

Model Definition

Figure 1 shows the actuator's parts and dimensions as well as its position on top of a substrate surface.

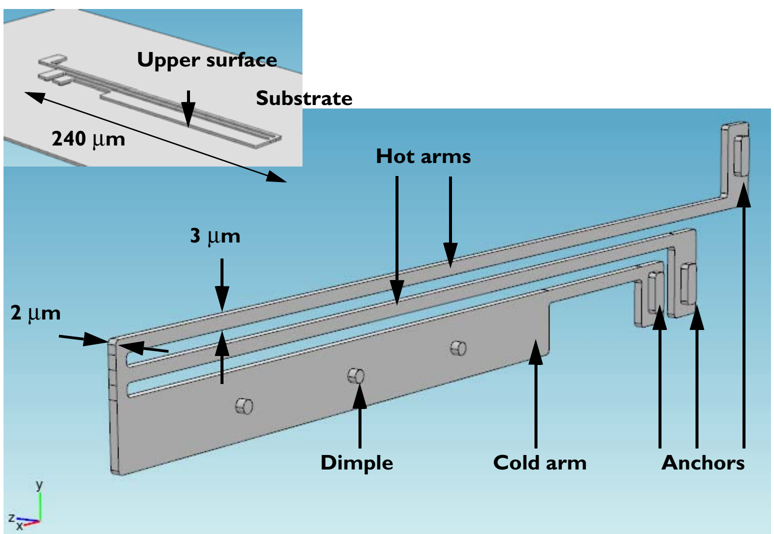


Figure 1: The thermal microactuator.

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.

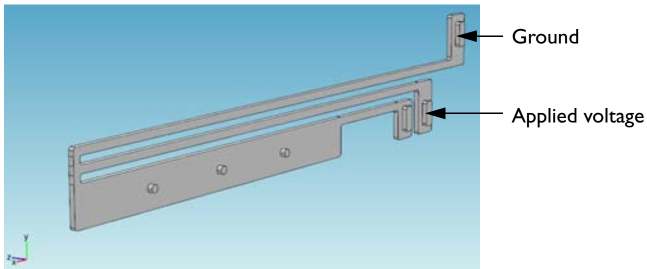


Figure 2: Electrical boundary conditions.

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. This can be implemented as thermal contact conditions or "Convective heat flux" condition where the heat flux coefficient represent one over the thermal resistance. In this model we chose to use a heat flux condition. 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.

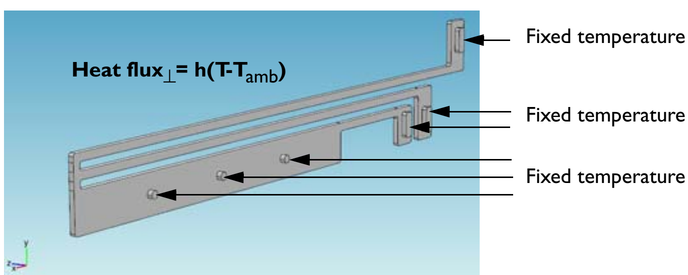


Figure 3: Heat-transfer boundary conditions.

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).

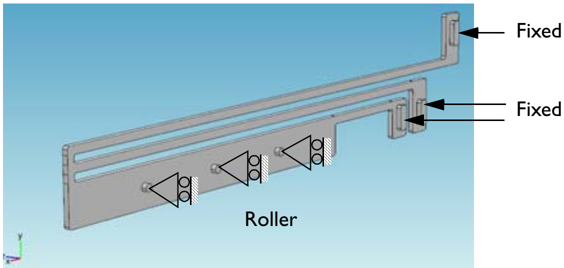


Figure 4: Structural boundary conditions and constraints.

Results

Figure 5 shows the surface temperature distribution for the actuator. It also illustrates the displacement field through a deformation plot.

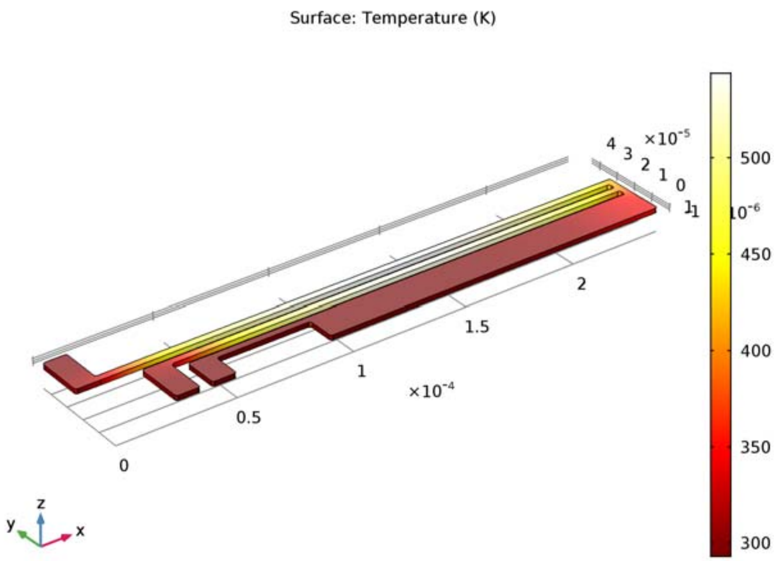


Figure 5: Temperature (surface) and displacement (deformation).

Application Library path: `Structural_Mechanics_Module/Thermal-Structure_Interaction/thermal_actuator_tem_parameterized`

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>Joule Heating and Thermal Expansion**.

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 On the **Home** toolbar, 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]	2E4 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

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Model Builder** window, right-click **Work Plane 1 (wp1)** and choose **Build Selected**.

3 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`.

6 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 `L4`.

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

5 Locate the **Position** section. In the **xw** text field, type `L-L3-L4`.

6 In the **yw** text field, type `dw-d`.

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

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

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`.

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

5 Locate the **Position** section. In the **xw** text field, type `L-L3-L4-wb`.

6 Right-click **Rectangle 3 (r3)** and choose **Build Selected**.

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

Rectangle 4 (r4)

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 `L2`.

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

5 Locate the **Position** section. In the **xw** text field, type `L-L2`.

6 In the **yw** text field, type `dw+gap`.

7 Right-click **Rectangle 4 (r4)** and choose **Build Selected**.

Rectangle 5 (r5)

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`.

4 In the **Height** text field, type `dw+gap+d`.

5 Locate the **Position** section. In the **xw** text field, type `L-L2-wb`.

6 Right-click **Rectangle 5 (r5)** and choose **Build Selected**.

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

Rectangle 6 (r6)

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 `L1`.

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

5 Locate the **Position** section. In the **xw** text field, type `L-L1`.

6 In the **yw** text field, type `dw+d+2*gap`.

7 Right-click **Rectangle 6 (r6)** and choose **Build Selected**.

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

Rectangle 7 (r7)

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`.

4 In the **Height** text field, type `dw+gap+d`.

5 Locate the **Position** section. In the **yw** text field, type `dw+d+2*gap`.

6 Right-click **Rectangle 7 (r7)** and choose **Build Selected**.

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

Rectangle 8 (r8)

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 `d`.

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

5 Locate the **Position** section. In the **xw** text field, type `L-d`.

6 In the **yw** text field, type `dw+gap+d`.

7 Right-click **Rectangle 8 (r8)** and choose **Build Selected**.

Rectangle 9 (r9)

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 `d`.

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

5 Locate the **Position** section. In the **xw** text field, type `L-d`.

6 In the **yw** text field, type `dw`.

7 Right-click **Rectangle 9 (r9)** 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**.

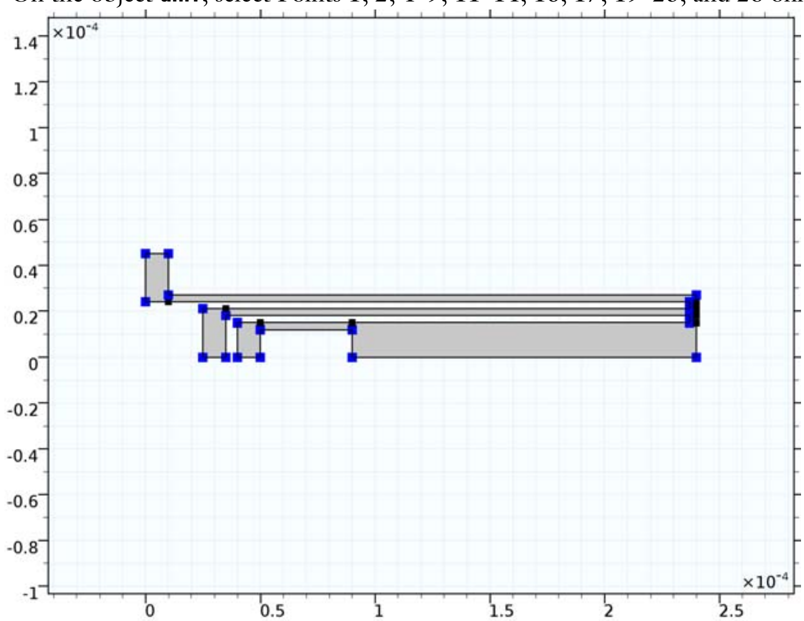
Fillet 1 (fil1)

1 On the **Work Plane** toolbar, click **Fillet**.

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

3 In the **Radius** text field, type `d/3`.

4 On the object **uni1**, select Points 1, 2, 4–9, 11–14, 16, 17, 19–23, and 28 only.



5 Right-click **Fillet 1 (fil1)** and choose **Build Selected**.

Work Plane 1 (wp1)

In the **Model Builder** window, under **Thermal Actuator (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 from Plane** section.

3 In the table, enter the following settings:

Distances (m)
2e-6

4 Right-click **Extrude 1 (ext1)** and choose **Build Selected**.

5 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

Work Plane 2 (wp2)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Model Builder** window, right-click **Work Plane 2 (wp2)** and choose **Build Selected**.

3 In the **Settings** window for Work Plane, click **Show Work Plane**.

Plane Geometry

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

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`.

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 `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`.

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 `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`.

7 Right-click **Rectangle 3 (r3)** and choose **Build Selected**.

Fillet 1 (fil1)

1 On the **Work Plane** toolbar, click **Fillet**.

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

3 In the **Radius** text field, type `d/3`.

4 Select all four vertices for all three rectangles as follows:

5 Click the **Select Box** button on the **Graphics** toolbar.

6 In the Graphics window, draw a box encompassing the three rectangles you just created, then right-click to confirm the selection.

7 Right-click **Fillet 1 (fil1)** and choose **Build Selected**.

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`.

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

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`.

6 Right-click **Circle 2 (c2)** and choose **Build Selected**.

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 Right-click **Circle 3 (c3)** and choose **Build Selected**.

Work Plane 2 (wp2)

In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** click **Work Plane 2 (wp2)**.

Extrude 2 (ext2)

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

2 In the **Settings** window for Extrude, locate the **Distances from Plane** section.

3 In the table, enter the following settings:

Distances (m)
2e-6

4 Select the **Reverse direction** check box.

Union 1 (uni1)

1 Right-click **Extrude 2 (ext2)** and choose **Build Selected**.

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

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

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

5 In the **Settings** window for Union, click **Build All Objects**.

DEFINITIONS

Explicit 1

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 Select Boundaries 10, 30, 50, 70, 76, and 82 only.

5 Right-click **Explicit 1** and choose **Rename**.

6 In the **Rename Explicit** dialog box, type `substrate contact` in the **New label** text field.

7 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 **MEMS>Semiconductors>Si - Polycrystalline Silicon**.

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

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

MATERIALS

Si - Polycrystalline Silicon (mat1)

By default, the first material you add applies on all domains so you can keep the **Geometric Entity Selection** settings.

1 In the **Model Builder** window, under **Thermal Actuator (comp1)>Materials** click **Si - Polycrystalline Silicon (mat1)**.

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

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Electrical conductivity	sigma	5e4	S/m	Basic

SOLID MECHANICS (SOLID)

Fixed Constraint 1

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

2 Select Boundaries 10, 30, and 50 only.

Roller 1

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

2 Select Boundaries 70, 76, and 82 only.

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**.

ELECTRIC CURRENTS (EC)

In the **Model Builder** window, under **Thermal Actuator (comp1)** click **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`.

MESH 1

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

Free Tetrahedral 1

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

Size

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

2 From the **Predefined** list, choose **Fine**.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element growth rate** text field, type `1.2`.

   This setting makes the mesh more robust for parametric sweeps over the geometry length parameter `L`.

Size 1

1 In the **Model Builder** window, under **Thermal Actuator (comp1)>Mesh 1>Free Tetrahedral 1** click **Size 1**.

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

3 From the **Predefined** list, choose **Finer**.

4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundaries 86–91 only.

6 Click **Build All**.

STUDY 1

Step 1: Stationary

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

2 Select the **Include geometric nonlinearity** check box.

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

RESULTS

Temperature (ht)

1 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

The second default plot shows the temperature field.

Created in COMSOL Multiphysics 5.2a

Piezoelectric Tonpilz Transducer with a Prestressed Bolt

Introduction

A tonpilz transducer, such as the one shown in Figure 1, is used for relatively low frequency, high power sound emission. It is one of the popular transducer configurations for SONAR applications. The transducer consists of piezoceramic rings stacked between a head mass and a tail mass which are connected by a central bolt. This example shows how to incorporate the effect of a pre-tension in the bolt. The bolt geometry is imported from the Part Libraries. The frequency response of the transducer is studied to determine structural and acoustic response of the device such as deformation, stresses, radiated power, sound pressure level, the transmitting voltage response (TVR) curve, and the directivity index (DI) of the sound beam.

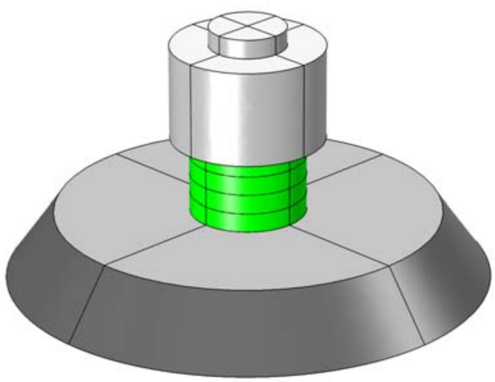


Figure 1: A tonpilz transducer. The aluminum head mass is shown in dark gray, the central steel bolt and steel tail mass are shown in light gray and the piezostack actuator with four disks of PZT-4 is shown in green.

Note: This application requires both the Acoustics Module and the Structural Mechanics Module.

A different version of this tutorial entitled *Piezoelectric Tonpilz Transducer*, is available in the Application Libraries under Acoustics Module only. That version of the tutorial does not implement the pre-tension in the bolt and hence does not require the Structural Mechanics Module. For additional details related to different customized settings and user-defined variables used in both the tonpilz transducer models, you are encouraged to read the documentation of the *Piezoelectric Tonpilz Transducer* tutorial.

Model Definition

The basic working principle involved in the operation of this transducer is that an AC electrical signal applied to the piezostack actuator produces vibration in the entire transducer which in turn produces sound waves in the surrounding fluid. Thus modeling the operation of the transducer requires coupling electrical, structural and acoustic phenomena.

In this tutorial we will particularly emphasize on the implementation of pre-tension in the central bolt of the transducer and associated solver settings that allow us to model the effect of prestress on the frequency response characteristics of the transducer.

The parameters used in this model are shown in Table 1.

TABLE 1: LIST OF MODELING PARAMETERS

NAME	EXPRESSION	DESCRIPTION
Rwater	40[mm]	Water domain radius
Rpml	10[mm]	PML layer thickness
a	25[mm]	Piston head radius
Zeval	-10[m]	Directivity evaluation distance
Vrms	1[V]	RMS drive voltage
V0	sqrt(2)*Vrms	Zero-to-peak drive voltage
f0min	1[kHz]	Minimum operating frequency
f0max	40[kHz]	Maximum operating frequency
f0step	1[kHz]	Frequency step
F_pre	1[kN]	Bolt pre-stress force

PHYSICS IMPLEMENTATION

The Acoustic-Piezoelectric Interaction, Frequency Domain multiphysics interface available in the Acoustics Module is used for simulating the multiphysics interactions. This predefined interface includes the necessary fundamental physics which are Pressure Acoustics, Solid Mechanics and Electrostatics. The Pressure Acoustics interface is used to solve for the wave equation in the water domain. The Solid Mechanics physics is solved on all structural materials including the PZT-4 disks. The Electrostatics physics is solved on the PZT-4 disks. The multiphysics couplings necessary to model this system are available as predefined nodes under the Multiphysics branch. These couplings are:

Acoustic-Structure Boundary: This node is active on the boundaries that are at the interface of the water domain and transducer head mass. On these boundaries a bidirectional coupling is automatically set up. The fluid pressure evaluated by the Pressure Acoustics physics is applied as a mechanical load in the Solid Mechanics physics. Furthermore, the normal component of the structural acceleration is used as a sound source.

Piezoelectric Effect: This node is active on the PZT-4 domains only and couple the Solid Mechanics and Electrostatics equations solved in these domains via the linear constitutive equations that model the piezoelectric effect by coupling stresses and strains with electric field and electric displacement.

BOUNDARY SETTINGS

The outer curved surface of the steel tail mass is assumed to be fixed. Each of the piezo disks are excited with a 1 V RMS electrical signal.

The head mass is exposed to an unbounded region of water. A Perfectly Matched Layer (PML) is used to model the absorption of sound waves as they propagate far away from the sound source. Although PML is strictly not a boundary condition, it is used to imitate the effect of an open boundary.

A far-field calculation is set up on the interface boundaries between the inner water domain and the PML domains. The far-field integral type is set to *Full integral* which allows computation of both amplitude and phase of the acoustic pressure and sound pressure level (SPL) at any point in space outside the computational domain. The far field pressure variable is used to compute the TVR and DI.

The user-defined variables used to compute the transducer characteristics are shown in Table 2.

TABLE 2: LIST OF VARIABLES

NAME	EXPRESSION	DESCRIPTION
rho0	intop3(acpr.rho)	Density of water at room temperature
c0	intop3(acpr.c)	Speed of sound in water at room temperature
Zaco	intop2(p)/intop2(acpr.iomega*(w+eps))/(rho0*c0)	Specific acoustic impedance
pfar_1	pfar(0,0,-1)	Far field pressure at 1 m
prms	sqrt(0.5*pfar_1*conj(pfar_1))[Pa]	RMS pressure at 1 m

TABLE 2: LIST OF VARIABLES

NAME	EXPRESSION	DESCRIPTION
TVR	20*log10(prms/Vrms/1[uPa/V])	Transmitting Voltage Response (TVR)
pfar_Zeval	pfar(0,0,Zeval[1/m])	Far field pressure at Zeval
Ifront	0.5*pfar_Zeval*conj(pfar_Zeval)[Pa^2]/(rho0*c0)	On-axis intensity at Zeval
Ptot	intop1(down(acpr.Ix)*acpr.nx+down(acpr.Iy)*acpr.ny+down(acpr.Iz)*acpr.nz)	Total radiated power
Iave	Ptot/(4*pi*Zeval^2)	Average intensity of monopole source at Zeval
Di	Ifront/Iave	Intensity directivity
DI	10*log10(Di)	Directivity index of Tonpilz transducer
k0	2*pi*freq/c0	Wave number
DI_fl_pist	10*log10((k0*a)^2/(1-2*besselj(1,2*k0*a)/(2*k0*a)))	Directivity index of flanged piston

MODELING A BOLT WITH PRE-TENSION

When a bolt is mounted on a device to clamp the components, it is tightened by twisting the bolt head. As a reaction to the tightening process, a pre-tension force is experienced by the bolt. This force produces a prestress that helps to hold the bolt in place during regular operation of the device. This also ensures that when additional stresses develop in the bolt during operation of the device, it should not become loose. Note that the tightening of the bolt also produces stresses in materials that are touching the bolt. This is why accounting for the pre-tension force in the bolt would give us an accurate picture of the pre-stress distribution not only in the bolt but also in the entire device.

COMSOL's Structural Mechanics Module provides a Bolt Pre-Tension feature that can be used to implement a desired pre-tension or prestress in bolted joints. You can import the bolt geometry from the Part Libraries. These bolt geometries are created in a certain way so that we can directly use the Bolt Pre-Tension feature on them. In order to use this feature, there should be a cross section surface passing through the shank of the bolt. This surface needs to be associated with the Bolt Selection subnode under the Bolt Pre-Tension node. COMSOL sets up an additional equation for each bolt that computes the pre-deformation of the bolt, the pre-tension force as well as the shear force in the bolt. For example, in this model, on application of 1 kN pre-tension, we get a pre-deformation of 12 μm.

Note that the deformation along the longitudinal axis of the bolt is discontinuous at the surface assigned to the Bolt Selection subnode but the stresses and strains are continuous. For more details on implementation of the Bolt Pre-Tension feature, you can refer to the section on *Using Pre-tensioned Bolts* in the Structural Mechanics Module User's Guide.

As a result of the prestress in the device, if we want to solve a vibrations problem in frequency domain, we need to account for the fact that the harmonic variation of stress and other physical quantities during vibration takes place on top of the static bias stress. Hence we need to solve this model using a two-step approach where the first step involves solving for the static stress distribution using a Stationary study step. The solution from this step is then used as a linearization point for solving the vibration problem in Frequency Domain Perturbation study step.

Note that this workflow is valid only for small perturbations about the static solution. Hence we should only use this technique if the magnitude of the stress and other physical quantities from the frequency domain problem is significantly smaller than the magnitude of the same quantities obtained from the static problem.

The AC voltage signal applied to the piezostack actuator is specified using the `linper()` operator. This operator ensures that the numerical input is used only in the Frequency Domain, Perturbation step and not in the Stationary step when solving the model. Furthermore, when solving the vibrations problem in frequency domain, you only want to account for the stress generated in the device as a result of its operation and hence you do not want to solve the pre-deformation variable in the bolt. This is ensured by changing the solver settings for the frequency domain part and instructing the solver to not solve for the pre-deformation variable.

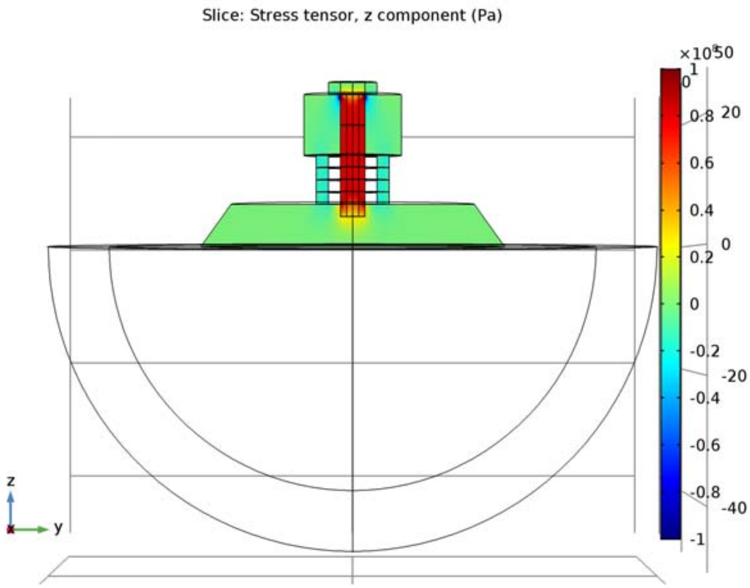


Figure 2: A slice plot showing the z-component of the prestress in the central bolt and other parts of the transducer. This stress arises due to the pre-tension in the bolt.

Figure 2 shows the z-component of the prestress in the transducer. The prestress is fairly uniform in the bolt. Note that the prestress exists mainly in the shank of the bolt and in the central part of the Aluminum head mass which is located directly below the bolt.

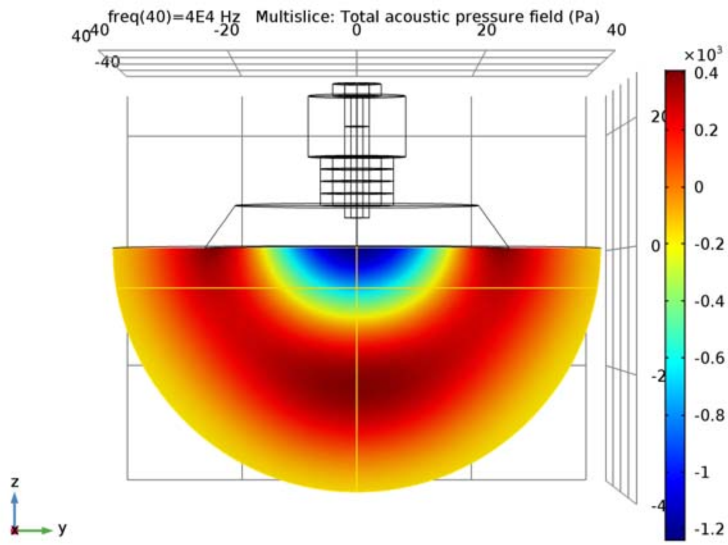


Figure 3: A slice plot showing the total acoustic pressure variation in the water domain at 40 kHz.

Figure 3 shows the total acoustic pressure in the water domain for 40 kHz excitation. The inner water domain in the model captures roughly half a wavelength at this frequency.

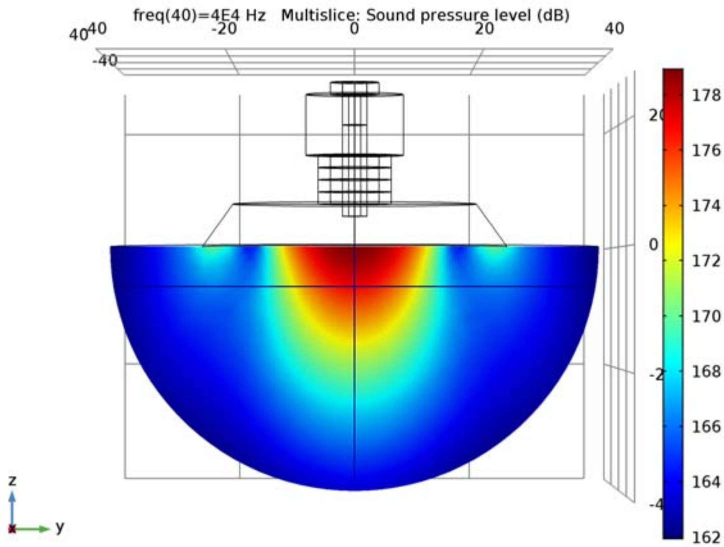


Figure 4: A slice plot showing the variation in sound pressure level (in dB) in the water domain at 40 kHz.

Figure 4 shows the sound pressure level (SPL) in the water domain for 40 kHz excitation. The SPL is highest near the transducer head mass. The variation in the SPL around the transducer would depend on the operating frequency and the dominant mode in which the transducer vibrates.

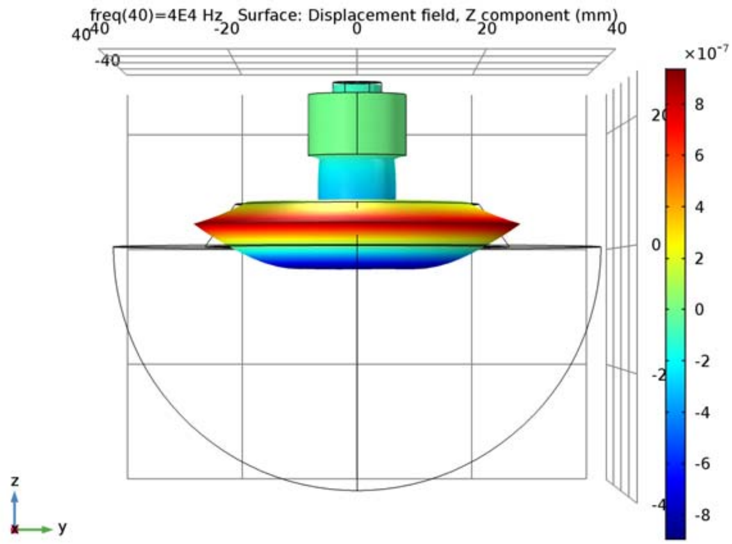


Figure 5: Total structural displacement of the transducer at 40 kHz. An exaggerated deformation has been used for better visualization.

Figure 5 shows the total structural displacement of the tonpilz transducer at 40 kHz excitation. At this frequency, the head mass vibrates in a mode whose shape is somewhat toroidal. If you observe the vibration mode for relatively lower frequencies, say 1 kHz, you will observe that the head mass vibrates mainly along its axis similar to a flanged piston.

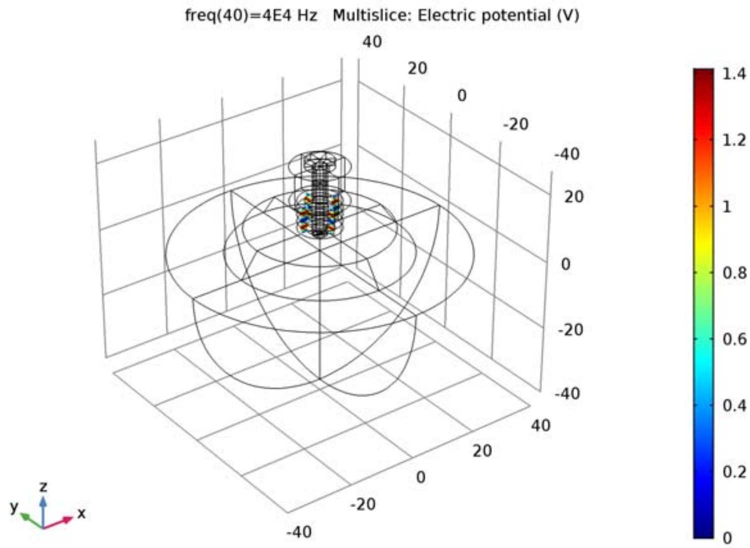


Figure 6: A multislice plot of the electric potential distribution within the four PZT-4 disks.

Figure 6 shows the electric potential distribution through the thickness of the PZT-4 disks. The piezoelectric disks are stacked in a way such that alternate disks are poled along opposite directions. This allows us to use a single electrical terminal at the interface of each pair of disks and obtain the piezoelectric actuation effect in each of the disks along the same direction. Having the piezoelectric strain in-phase in all the disks maximizes the actuation.

In this model, the PZT-4 disks actuate in the d_{33}-mode. Hence two of the disks are poled along the +Z-direction while the other two are poled along the -Z-direction. The default definition of the piezoelectric material properties in COMSOL's Global Coordinate System automatically creates a +Z polarization. In order to create a -Z polarization, a user-defined Rotated Coordinate System is used. In this coordinate system, the Euler angles are set to $\alpha = 0$, $\beta = \pi$ and $\gamma = 0$.

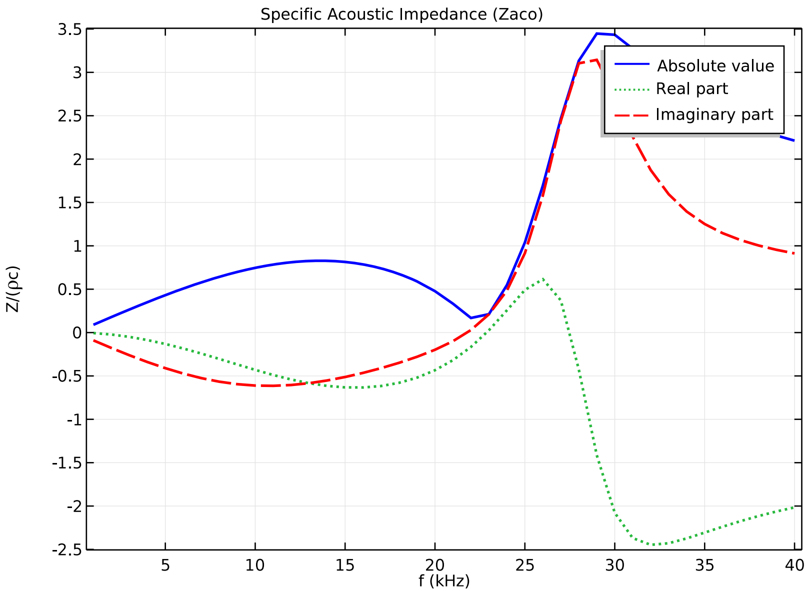


Figure 7: Frequency response plot of the absolute value, real and imaginary components of the specific acoustic impedance at the interface between the head mass and water.

Figure 7 shows the frequency response of the specific acoustic impedance of the head mass surface that is exposed to water. A resonance and anti-resonance is observed within the frequency range of 20 kHz and 35 kHz.

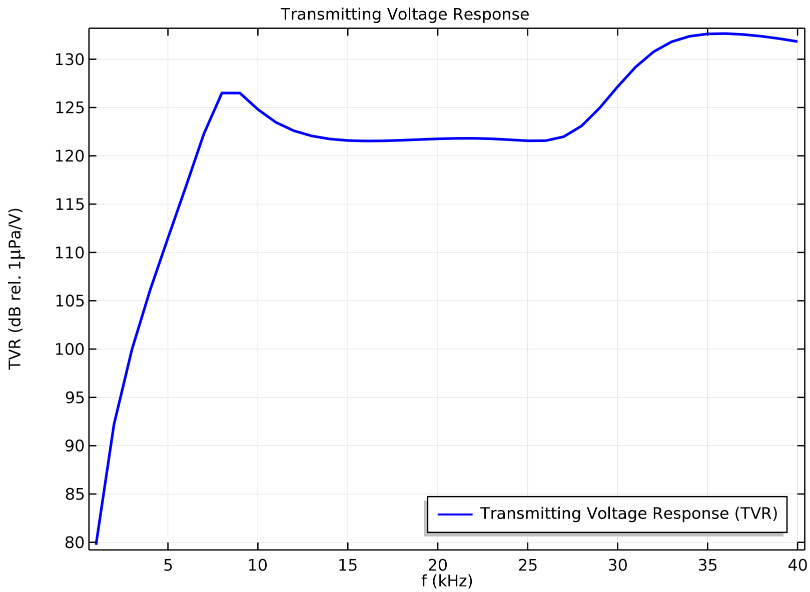


Figure 8: Transmitting Voltage Response (TVR) as a function of frequency obtained at an on-axis distance of 1 m ahead of the head mass and computed relative to 1 µ*Pa/V.*

Figure 8 shows the variation in the TVR of the transducer as a function of operating frequency. The fairly flat region between 15 kHz and 25 kHz can be particularly useful for sensing applications.

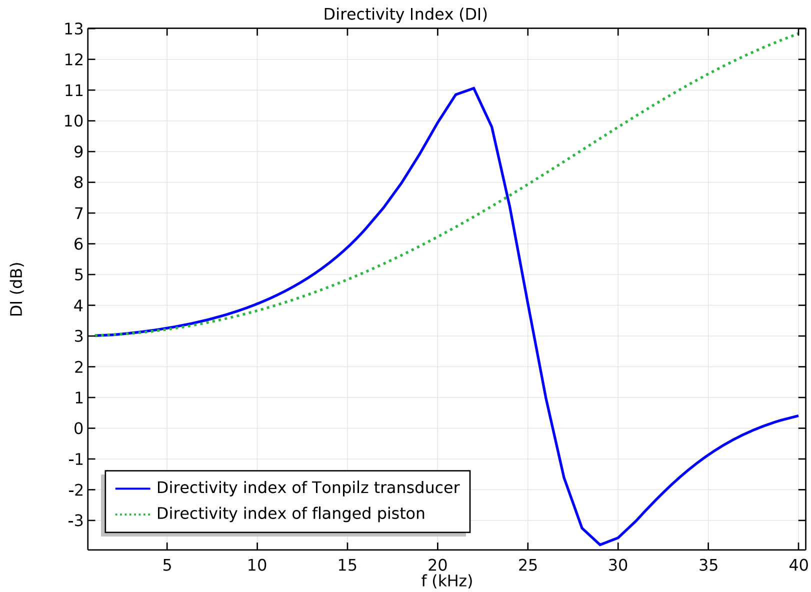


Figure 9: Frequency response of the Directivity Index (DI) computed at an on-axis distance of 10 m from the head mass. The DI of the tonpilz transducer is compared to that of a flanged piston.

Figure 9 shows the Directivity Index (DI) of the tonpilz transducer (blue curve) and compares it with the DI of a flanged piston (green curve). The latter can be computed from analytical expression as shown in Table 2. It is defined by the variable `DI_fl_pist`. Note that when the tonpilz transducer operates like a piston at lower frequencies, its DI becomes very similar to that of a flanged piston. Another feature worth noting is that within the range of 15 kHz and 25 kHz, the DI of the tonpilz transducer changes from nearly 4 dB to 11 dB while its TVR remains nearly constant. This can make the transducer quite versatile within this operating range.

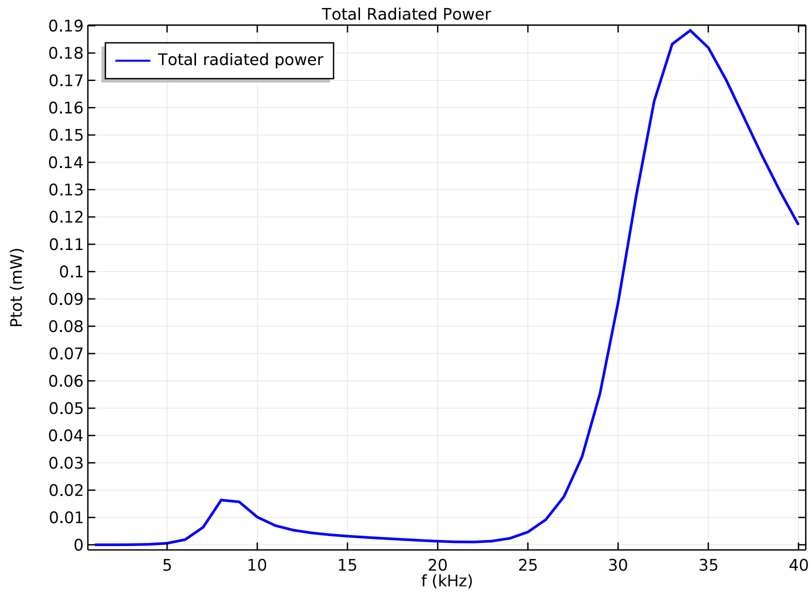


Figure 10: Total radiated power from the tonpilz transducer within the operating frequency range of 1 kHz to 40 kHz.

Figure 10 shows the total radiated power as a function of the operating frequency of the tonpilz transducer. Since the radiated power is proportional to the sound intensity, the peak in power at around 8 kHz and 34 kHz also indicate that peak sound intensity is obtained at these frequencies.

Application Library path: `Structural_Mechanics_Module/Piezoelectric_Effects/tonpilz_transducer_prestressed`

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>Acoustic-Structure Interaction> Acoustic-Piezoelectric Interaction, Frequency Domain**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Prestressed Analysis, Frequency Domain**.

6 Click **Done**.

GEOMETRY 1

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

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

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

Parameters

1 On the **Home** toolbar, click **Parameters**.

Import the file containing the model parameters.

GLOBAL DEFINITIONS

Parameters

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

2 Click **Load from File**.

3 Browse to the application's Application Libraries folder and double-click the file `tonpilz_transducer_prestressed_parameters.txt`.

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

The modeling geometry is created by first drawing the cross section on a workplane and then revolving this cross section to get the 3-dimensional geometry. The central bolt in the transducer is later added from the Part Libraries.

GEOMETRY 1

Work Plane 1 (wp1)

1 In the **Settings** window for Work Plane, locate the **Plane Definition** section.

2 From the **Plane** list, choose **xz-plane**.

3 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 `6`.

4 In the **Height** text field, type `10`.

5 Locate the **Position** section. In the **xw** text field, type `2`.

6 In the **yw** text field, type `15`.

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 `2`.

4 In the **Height** text field, type `8`.

5 Locate the **Position** section. In the **xw** text field, type `4`.

6 In the **yw** text field, type `7`.

7 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	2
Layer 2	2
Layer 3	2

Polygon 1 (pol1)

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

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

3 In the **xw** text field, type `0 0 2 2 20 a`.

4 In the **yw** text field, type `0 5 5 7 7 0`.

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 `Rwater+Rpml`.

4 In the **Sector angle** text field, type `90`.

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

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

Layer name	**Thickness (mm)**
`Layer 1`	`Rpml`

7 On the **Work Plane** toolbar, click **Build All**.

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

9 Click **Close**.

Work Plane 1 (wp1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 1 (wp1)**.

Revolve 1 (rev1)

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

2 In the **Settings** window for Revolve, locate the **Revolution Angles** section.

3 Click the **Angles** button.

4 In the **End angle** text field, type `90`.

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

Explicit Selection 1 (sel1)

1 Right-click **Revolve 1 (rev1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

Create domain and boundary selections that will be used in the model.

3 In the **Settings** window for Explicit Selection, type `Water domain - Inner` in the **Label** text field.

4 On the object **rev1**, select Domain 2 only.

Explicit Selection 2 (sel2)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for Explicit Selection, type `Water domain - PML` in the **Label** text field.

3 On the object **rev1**, select Domain 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 `Aluminum` in the **Label** text field.

3 On the object **rev1**, select Domain 3 only.

Explicit Selection 4 (sel4)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for Explicit Selection, type `Steel Part` in the **Label** text field.

3 On the object **rev1**, select Domain 4 only.

4 Locate the **Resulting Selection** section. Click **New**.

5 Clear the **Keep selection** check box.

6 In the **New Cumulative Selection** dialog box, type `Steel` in the **Name** text field.

7 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, type `+Z poled Piezo` in the **Label** text field.

3 On the object **rev1**, select Domains 5 and 7 only.

Explicit Selection 6 (sel6)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for Explicit Selection, type `-Z poled Piezo` in the **Label** text field.

3 On the object **rev1**, select Domains 6 and 8 only.

Explicit Selection 7 (sel7)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for Explicit Selection, type `Ground boundaries` in the **Label** text field.

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

4 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

5 On the object **rev1**, select Boundaries 19, 25, and 29 only.

Explicit Selection 8 (sel8)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for Explicit Selection, type `Voltage boundaries` in the **Label** text field.

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

4 On the object **rev1**, select Boundaries 22 and 28 only.

Explicit Selection 9 (sel9)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for Explicit Selection, type `Fixed boundaries` in the **Label** text field.

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

4 On the object **rev1**, select Boundary 36 only.

Explicit Selection 10 (sel10)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for Explicit Selection, type `Far Field boundaries` in the **Label** text field.

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

4 On the object **rev1**, select Boundary 6 only.

Rotate 1 (rot1)

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

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

3 From the **Input objects** list, choose **Revolve 1**.

4 Locate the **Rotation Angle** section. In the **Rotation** text field, type `range(0,90,270)`.

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

6 Click **Build All Objects**.

Union 1 (uni1)

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

The following Union operation combines all the geometry objects created so far. This step ensures that when later on an Assembly option is used for finalizing the geometry sequence, the process does not create additional and unnecessary boundaries at the interface of all the geometry objects.

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

3 From the **Input objects** list, choose **Rotate 1**.

The CAD geometry of a simple bolt with no thread is imported from the Part Libraries. The design parameters of the bolt are adjusted to position the bolt in the tonpilz transducer.

PART LIBRARIES

1 On the **Geometry** toolbar, click **Parts** and choose **Part Libraries**.

2 In the **Part Libraries** window, select **Structural Mechanics Module>Bolts>simple bolt no thread** in the tree.

3 click **Add to Geometry**.

GEOMETRY 1

Simple Bolt, No Thread 1 (pi1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Simple Bolt, No Thread 1 (pi1)**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
hdia	8[mm]	8 mm	Head diameter
hthic	2[mm]	2 mm	Head thickness
ndia	4[mm]	4 mm	Nominal diameter
blen	20[mm]	20 mm	Bolt length

4 Locate the **Position and Orientation of Output** section. Find the **Coordinate system in part** subsection. From the **Work plane in part** list, choose **Head inner plane (wp1)**.

5 Find the **Displacement** subsection. In the **zw** text field, type 25.

6 Click to expand the **Domain selections** section. Locate the **Domain Selections** section. Click to select row number 1 in the table.

7 In the table, enter the following settings:

Name	Contribute to	Keep	Physics
All	csel1		

8 Click to expand the **Boundary selections** section. Locate the **Boundary Selections** section. Click to select row number 5 in the table.

9 In the table, enter the following settings:

Name	Contribute to	Keep	Physics
Exterior	none		
Shank	none		
Head, free surface	none		
Head, contact surface	none		
Pre-tension cut	none		

10 Right-click **Component 1 (comp1)>Geometry 1>Simple Bolt, No Thread 1 (pi1)** and choose **Build Selected**.

The geometry finalization method is changed to Form an Assembly to ensure that the bolt is not glued or rigidly attached to the adjacent parts.

Form Union (fin)

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

2 In the **Settings** window for Form Union/Assembly, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Select the **Create imprints** check box.

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

DEFINITIONS

An Identity Pair is used to get continuity in solution between the external surfaces of the bolt and the surfaces of other materials touching them. This continuity in solution is applicable for the lower end of the shank that is bolted into the aluminum head mass and the lower surface of the bolt head which is resting on the steel tail mass. The outer surface of the shank should be allowed to slip through the hole in the tail mass. Hence those boundaries need to be removed from the Identity Pair by modifying the default Identity Pair that has been created.

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node, then click **Identity Pair 1 (ap1)**.

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

3 Select the **Manual control of selections** check box.

4 Locate the **Source Boundaries** section. Select the **Active** toggle button.

5 Click **Clear Selection**.

6 Click **Paste Selection**.

7 In the **Paste Selection** dialog box, type `59-66 71 72 75 76 112-114 126-129 135 160 163 164 167` in the **Selection** text field.

8 Click **OK**.

9 In the **Settings** window for Pair, locate the **Destination Boundaries** section.

10 Select the **Active** toggle button.

11 Click **Clear Selection**.

12 Click **Paste Selection**.

13 In the **Paste Selection** dialog box, type `174 175 178 183 185-188 196 198 206 207 216 219 220 230-232 245 249 251 255 256 259` in the **Selection** text field.

14 Click **OK**.

15 In the **Settings** window for Pair, locate the **Destination Boundaries** section.

16 Click **Create Selection**.

17 In the **Create Selection** dialog box, type `Destination boundaries` in the **Selection name** text field.

18 Click **OK**.

Union 1

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

Create additional domain selections that will be used in the model.

2 In the **Settings** window for Union, type `Water domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, In the **Selections to add** list, choose **Water domain - Inner** and **Water domain - PML**.

5 Click **OK**.

Union 2

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

2 In the **Settings** window for Union, type `Piezo domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, In the **Selections to add** list, choose **+Z poled Piezo** and **-Z poled Piezo**.

5 Click **OK**.

Complement 1

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

2 In the **Settings** window for Complement, type `Solid domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to invert**, click **Add**.

4 In the **Add** dialog box, select **Water domains** in the **Selections to invert** list.

5 Click **OK**.

Complement 2

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

2 In the **Settings** window for Complement, type `Non-PML domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to invert**, click **Add**.

4 In the **Add** dialog box, select **Water domain - PML** in the **Selections to invert** list.

5 Click **OK**.

Integration 1 (intop1)

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

Define an integration coupling operator on the far-field boundary.

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

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

4 From the **Selection** list, choose **Far Field boundaries**.

Integration 2 (intop2)

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

Define an integration coupling operator on the acoustic-structure interface.

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

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `14 15 87 125` in the **Selection** text field.

6 Click **OK**.

Integration 3 (intop3)

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

Define an integration coupling operator to retrieve values at the summit point of the water domain.

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

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `58` in the **Selection** text field.

6 Click **OK**.

Variables 1

1 On the **Definitions** toolbar, click **Local Variables**.

Import the file containing the variable definitions. These variables will mainly be used for post-processing calculations.

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

3 Click **Load from File**.

4 Browse to the application's Application Libraries folder and double-click the file `tonpilz_transducer_prestressed_variables.txt`.

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>Water, liquid**.

4 Click **Add to Component 1**.

MATERIALS

Water, liquid (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Water, liquid (mat1)**.

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

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

ADD MATERIAL

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

2 In the tree, select **Built-In>Aluminum**.

3 Click **Add to Component 1**.

MATERIALS

Aluminum (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Aluminum (mat2)**.

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

3 From the **Selection** list, choose **Aluminum**.

ADD MATERIAL

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

2 In the tree, select **Built-In>Steel AISI 4340**.

3 Click **Add to Component 1**.

MATERIALS

Steel AISI 4340 (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Steel AISI 4340 (mat3)**.

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

3 From the **Selection** list, choose **Steel**.

ADD MATERIAL

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

2 In the tree, select **Piezoelectric>Lead Zirconate Titanate (PZT-4)**.

3 Click **Add to Component 1**.

MATERIALS

Lead Zirconate Titanate (PZT-4) (mat4)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Lead Zirconate Titanate (PZT-4) (mat4)**.

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

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

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

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Frequency Domain (acpr)**.

2 In the **Settings** window for Pressure Acoustics, Frequency Domain, locate the **Domain Selection** section.

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

4 Locate the **Sound Pressure Level Settings** section. From the **Reference pressure for the sound pressure level** list, choose **Use reference pressure for water**.

5 Locate the **Typical Wave Speed for Perfectly Matched Layers** section. In the c_{ref} text field, type `1500[m/s]`.

Far-Field Calculation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Far-Field Calculation**.

2 In the **Settings** window for Far-Field Calculation, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Far Field boundaries**.

4 Locate the **Far-Field Calculation** section. Select the **Symmetry in the z=0 plane** check box.

5 From the **Type of integral** list, choose **Full integral**.

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 **Domain Selection** section.

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

Piezoelectric Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Piezoelectric Material 1**.

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

3 From the **Selection** list, choose **+Z poled Piezo**.

4 In the **Model Builder** window, click **Solid Mechanics (solid)**.

Piezoelectric Material 2

1 On the **Physics** toolbar, click **Domains** and choose **Piezoelectric Material**.

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

3 From the **Selection** list, choose **-Z poled Piezo**.

Define a rotated system that will be used for the poling of the -Z poled piezoelectric disks.

DEFINITIONS

Rotated System 2 (sys2)

1 On the **Definitions** toolbar, click **Coordinate Systems** and choose **Rotated System**.

2 In the **Settings** window for Rotated System, locate the **Settings** section.

3 Find the **Euler angles (Z-X-Z)** subsection. In the β text field, type `pi`.

SOLID MECHANICS (SOLID)

Piezoelectric Material 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Piezoelectric Material 2**.

2 In the **Settings** window for Piezoelectric Material, locate the **Coordinate System Selection** section.

3 From the **Coordinate system** list, choose **Rotated System 2 (sys2)**.

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 **Fixed boundaries**.

Bolt Pre-Tension 1

1 On the **Physics** toolbar, click **Global** and choose **Bolt Pre-Tension**.

2 In the **Settings** window for Bolt Pre-Tension, locate the **Bolt Pre-Tension** section.

3 In the F_p text field, type `F_pre`.

Bolt Selection 1

1 In the **Model Builder** window, expand the **Bolt Pre-Tension 1** node, then click **Bolt Selection 1**.

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

3 From the **Selection** list, choose **Pre-tension cut (Simple Bolt, No Thread 1)**.

Continuity 1

1 On the **Physics** toolbar, in the Boundary section, click **Pairs** and choose **Continuity**.

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

3 In the **Pairs** list, select **Identity Pair 1 (ap1)**.

ELECTROSTATICS (ES)

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

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

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

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 boundaries**.

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 **Voltage boundaries**.

4 Locate the **Electric Potential** section. In the V_0 text field, type `linper(V0)`.

The `linper()` operator ensures that the voltage V0 is only applied in the frequency domain perturbation study step and not during the stationary analysis. If you have the AC/DC Module you can right click and add the Harmonic Perturbation sub-feature, which will do the same.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

Insert a Perfectly Matched Layer to model the absorption of acoustic wave far away from the source.

2 In the **Settings** window for Perfectly Matched Layer, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Water domain - PML**.

4 Locate the **Geometry** section. From the **Type** list, choose **Spherical**.

5 Locate the **Scaling** section. From the **Coordinate stretching type** list, choose **Rational**.

6 In the **PML scaling factor** text field, type `0.5`.

7 In the **PML scaling curvature parameter** text field, type `5`.

Mesh the geometry; create a tetrahedral mesh in the solid and the water-inner domains and create a swept mesh in the PML.

MESH 1

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

Free Tetrahedral 1

1. In the **Settings** window for Free Tetrahedral, locate the **Domain Selection** section.
2. From the **Geometric entity level** list, choose **Domain**.
3. From the **Selection** list, choose **Non-PML domains**.

Size 1

1. Right-click **Component 1 (comp1)>Mesh 1>Free Tetrahedral 1** and choose **Size**.

   Define a mesh size in the water domain to ensure that the smallest wavelength is resolved by at least 5 elements.
2. In the **Settings** window for Size, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Water domains**.
4. Locate the **Element Size** section. Click the **Custom** button.
5. Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
6. In the associated text field, type `1500[m/s]/f0max/5`.

Free Tetrahedral 1

Right-click **Free Tetrahedral 1** and choose **Size**.

Size 2

Apply a mesh setting on the Destination boundaries of the Identity Pair such that the mesh on these surfaces is somewhat finer than the mesh on the Source boundaries of the Identity Pair.

1. In the **Settings** window for Size, locate the **Geometric Entity Selection** section.
2. From the **Geometric entity level** list, choose **Boundary**.
3. From the **Selection** list, choose **Destination boundaries**.
4. Locate the **Element Size** section. From the **Predefined** list, choose **Finer**.
5. In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

Swept 1

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

Distribution 1

1. In the **Settings** window for Distribution, click **Build All**.

   Create a boundary layer mesh at the external boundaries of the water domain. This will ensure numerically well defined normal gradients used in the far-field calculation feature.

Boundary Layers 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Boundary Layers**.

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

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

4 From the **Selection** list, choose **Water domain - Inner**.

Boundary Layer Properties

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Boundary Layers 1** click **Boundary Layer Properties**.

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

3 From the **Selection** list, choose **Far Field boundaries**.

4 Locate the **Boundary Layer Properties** section. In the **Number of boundary layers** text field, type `1`.

5 From the **Thickness of first layer** list, choose **Manual**.

6 In the **Thickness** text field, type `1500[m/s]/f0max/5/20`.

7 Click **Build All**.

The Stationary step solves for the effect of pre-tension in the bolt and hence does not need to solve for the Acoustics problem. The Frequency-Domain Perturbation step solves all the physics.

STUDY 1

Step 1: Stationary

1 In the **Settings** window for Stationary, locate the **Physics and Variables Selection** section.

2 In the table, clear the **Solve for** check box for the **Pressure Acoustics, Frequency Domain (acpr)** interface.

Step 2: Frequency-Domain Perturbation

1 In the **Model Builder** window, under **Study 1** click **Step 2: Frequency-Domain Perturbation**.

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

3 In the **Frequencies** text field, type `range(f0min,f0step,f0max)`.

The following steps are performed to ensure that the pre-deformation in the bolt as a result of the pre-tension is not computed in the Frequency-Domain Perturbation step.

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, under **Study 1>Solver Configurations>Solution 1 (sol1)** click **Dependent Variables 2**.

4 In the **Settings** window for Dependent Variables, locate the **General** section.

5 From the **Defined by study step** list, choose **User defined**.

6 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 2** click **Pre-deformation (comp1.solid.pblt1.sblt1.d_pre)**.

7 In the **Settings** window for State, locate the **General** section.

8 Clear the **Solve for this state** check box.

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

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

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

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

The following instructions describe how to create the plots shown in the **Results** section.

RESULTS

Data Sets

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

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **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 **Domain**.

4 From the **Selection** list, choose **Non-PML domains**.

5 Select the **Propagate to lower dimensions** check box.

3D Plot Group 1

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

2 In the **Settings** window for 3D Plot Group, type `Static stress from pre-tension` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1/Solution Store 1 (sol2)**.

4 Click the **Go to YZ View** button on the **Graphics** toolbar.

Slice 1

1 Right-click **Static stress from pre-tension** and choose **Slice**.

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

3 In the **Planes** text field, type `1`.

4 Locate the **Expression** section. In the **Expression** text field, type `solid.sz`.

5 Click to expand the **Range** section. Select the **Manual color range** check box.

6 In the **Minimum** text field, type `-1e8`.

7 In the **Maximum** text field, type `1e8`.

8 On the **Static stress from pre-tension** toolbar, click **Plot**.

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, type `Acoustic Pressure` in the **Label** text field.

Multislice 1

1 On the **Acoustic Pressure** toolbar, click **More Plots** and choose **Multislice**.

2 Click **Plot**.

Acoustic Pressure

In the **Model Builder** window, under **Results** right-click **Acoustic Pressure** and choose **Duplicate**.

Acoustic Pressure 1

In the **Settings** window for 3D Plot Group, type `Sound Pressure Level` in the **Label** text field.

Multislice 1

1 In the **Model Builder** window, expand the **Results>Sound Pressure Level** node, then click **Multislice 1**.

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

3 In the **Expression** text field, type `acpr.Lp`.

4 On the **Sound Pressure Level** toolbar, click **Plot**.

3D Plot Group 4

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

2 In the **Settings** window for 3D Plot Group, type `Displacement` in the **Label** text field.

Surface 1

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

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

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

4 Right-click **Results>Displacement>Surface 1** and choose **Deformation**.

5 On the **Displacement** toolbar, click **Plot**.

Sound Pressure Level

In the **Model Builder** window, under **Results** right-click **Sound Pressure Level** and choose **Duplicate**.

Sound Pressure Level 1

In the **Settings** window for 3D Plot Group, type `Electric Potential` in the **Label** text field.

Multislice 1

1 In the **Model Builder** window, expand the **Results>Electric Potential** node, then click **Multislice 1**.

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

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

4 On the **Electric Potential** toolbar, click **Plot**.

5 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

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 `Specific Acoustic Impedance` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Specific Acoustic Impedance (Zaco)`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `f (kHz)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Z/(\rho c)`.

Global 1

1 On the **Specific Acoustic Impedance** toolbar, click **Global**.

2 In the **Settings** window for Global, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
abs(Zaco)	1	
real(Zaco)	1	
imag(Zaco)	1	

4 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
Absolute value
Real part
Imaginary part

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** text field, type `freq`.

8 From the **Unit** list, choose **kHz**.

9 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type `2`.

10 Find the **Line style** subsection. From the **Line** list, choose **Cycle**.

11 On the **Specific Acoustic Impedance** toolbar, click **Plot**.

Specific Acoustic Impedance

In the **Model Builder** window, under **Results** right-click **Specific Acoustic Impedance** and choose **Duplicate**.

Specific Acoustic Impedance 1

1 In the **Settings** window for 1D Plot Group, type `Transmitting Voltage Response` in the **Label** text field.

2 Locate the **Title** section. In the **Title** text area, type `Transmitting Voltage Response`.

3 Locate the **Plot Settings** section. In the **y-axis label** text field, type `TVR (dB rel. 1\mu Pa/V)`.

4 Click to expand the **Legend** section. From the **Position** list, choose **Lower right**.

Global 1

1 In the **Model Builder** window, expand the **Results>Transmitting Voltage Response** 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>TVR - Transmitting Voltage Response (TVR)**.

3 Locate the **Legends** section. From the **Legends** list, choose **Automatic**.

4 On the **Transmitting Voltage Response** toolbar, click **Plot**.

Transmitting Voltage Response

In the **Model Builder** window, under **Results** right-click **Transmitting Voltage Response** and choose **Duplicate**.

Transmitting Voltage Response 1

1 In the **Settings** window for 1D Plot Group, type `Directivity Index (DI)` in the **Label** text field.

2 Locate the **Title** section. In the **Title** text area, type `Directivity Index (DI)`.

3 Locate the **Plot Settings** section. In the **y-axis label** text field, type `DI (dB)`.

4 Locate the **Legend** section. From the **Position** list, choose **Lower left**.

Global 1

1 In the **Model Builder** window, expand the **Results>Directivity Index (DI)** 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>DI - Directivity index of Tonpilz transducer**.

3 Click **Add Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Definitions>Variables>DI_fl_pist - Directivity index of flanged piston**.

4 On the **Directivity Index (DI)** toolbar, click **Plot**.

Directivity Index (DI)

In the **Model Builder** window, under **Results** right-click **Directivity Index (DI)** and choose **Duplicate**.

Directivity Index (DI) 1

1 In the **Settings** window for 1D Plot Group, type `Total Radiated Power` in the **Label** text field.

2 Locate the **Title** section. In the **Title** text area, type `Total Radiated Power`.

3 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Ptot (mW)`.

4 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Global 1

1 In the **Model Builder** window, expand the **Results>Total Radiated Power** 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>Ptot - Total radiated power**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
Ptot	mW	Total radiated power

4 On the **Total Radiated Power** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.2a

Linear Buckling Analysis of a Truss Tower

Introduction

Trusses are commonly used to create light structures that can support heavy loads. When designing such a structure, it is important to ensure its safety. For a tower made of bars, buckling can cause the structure to collapse. This example shows how to compute the critical buckling load using a linear buckling analysis. The solution is compared with an analytical expression for critical load estimation for Euler buckling.

Model Definition

The model geometry consists of a 19-meters tall truss tower with a rectangular section. The critical buckling load is computed using the linear buckling analysis available in the Truss interface.

The geometry is the periodic structure represented in Figure 1 below. It consists of 19 blocks of trusses. Each block has width of 0.45 m, depth of 0.40 m and height of 1.0 m. The trusses that are perpendicular to the ground are thicker and have outer radius of 15 cm and inner radius of 10 cm. The remaining trusses have outer radius of 10 cm and inner radius of 7 cm. The tower is made out of structural steel that is one of the predefined materials in the material library.

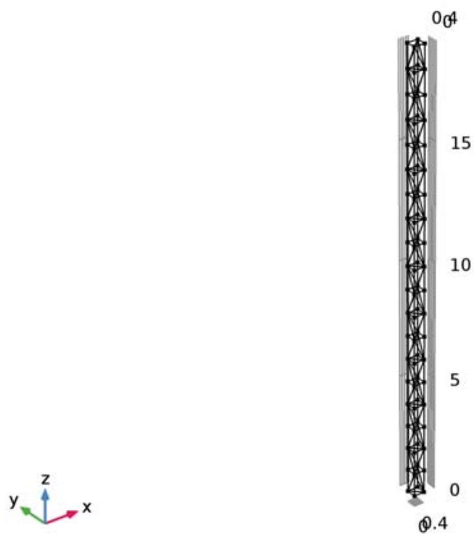


Figure 1: Geometry of the truss tower.

The tower is fixed at the ground level and a vertical load is applied at the top.

One fourth of the unit load is applied at each point of the tower top so that the critical load factor returned by the linear buckling analysis corresponds to the load that would cause the collapse of the structure.

Results and Discussion

For a simple column the critical buckling load is given by the Euler buckling formula

$$F_c = \frac{\pi^2 EI}{(KL)^2}$$

where E is the Young's modulus, I is the area moment of inertia, L is the unsupported length of the column and K is the column effective length factor.

For a column with one end fixed and the other end free to move laterally, $K = 2$.

For a tower like the one in this example with 4 main bars in the axial direction, the area of moment of inertia of the section can be computed as:

$$I = 4S\left(\frac{h}{2}\right)^2$$

where h is the distance between the vertical bars, and S the cross section area of the bars.

As the section is rectangular with different depth and width value, the tower has one weak direction. Here the depth is 40 cm and the width is 45 cm. This means that the first critical buckling load is expected to be about 8.6e4N in the depth direction (y-direction). In the width direction, which is expected to be stiffer, the critical buckling load is estimated to be about 1.1e5N.

The results obtained with the linear buckling analysis agree well with these values.

Figure 2 shows the value of the first critical buckling load and the deformation shape.

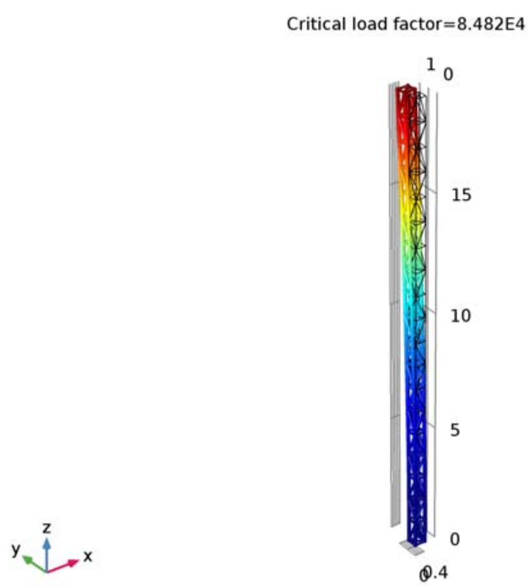


Figure 2: Deformation shape at the first critical buckling load

[Figure 3](#) shows the value of the second critical buckling load and the deformation shape.

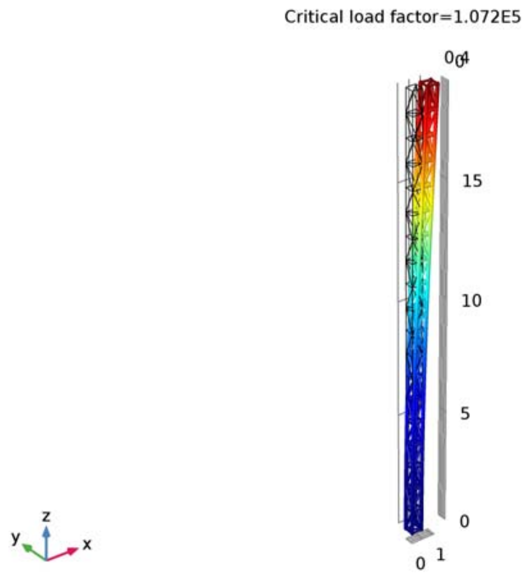


Figure 3: Deformation shape at the second critical buckling load

Note that the approximation given for the Euler buckling critical load is suitable for a tower structure when the height is significantly larger than the width or the depth.

Notes About the COMSOL Implementation

For a model with only a Truss interface you can improve the performance by deactivating the **Straight-edge constraint** node.

Application Library path: `Structural_Mechanics_Module/Buckling/truss_tower_buckling`

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>Truss (truss)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Linear Buckling**.
6. Click **Done**.

GLOBAL DEFINITIONS

Parameters

1. On the **Home** toolbar, 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
ro1	1.5[cm]	0.015 m	Outer radius tube 1
ri1	1[cm]	0.01 m	Inner radius tube 1
ro2	1[cm]	0.01 m	Outer radius tube 2
ri2	0.7[cm]	0.007 m	Inner radius tube 2
A1	pi*(ro1^2-ri1^2)	3.927E-4 m²	Area tube 1
A2	pi*(ro2^2-ri2^2)	1.602E-4 m²	Area tube 2
depth	0.4[m]	0.4 m	Depth of the tower
width	0.45[m]	0.45 m	Width of the tower
height	1[m]	1 m	Height of the tower
n	10	10	Number of sections
L	height*(2*n-1)	19 m	Total height of the tower
I1	4*A1*(depth/2)^2	6.283E-5 m^4	Area moment of inertia weak direction
Fc1	pi^2*200e9[Pa]*I1/(2*L)^2	8.589E4 N	First critical buckling load

Name	Expression	Value	Description
I2	4*A1*(width/2)^2	7.952E-5 m^4	Area moment of inertia stiffer direction
Fc2	pi^2*200e9[Pa]*I2/(2*L)^2	1.087E5 N	Second critical buckling load

GEOMETRY 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 **Width** text field, type `width`.

4 In the **Depth** text field, type `depth`.

5 In the **Height** text field, type `height`.

Bézier Polygon 1 (b1)

1 Right-click **Block 1 (blk1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **More Primitives** and choose **Bézier Polygon**.

3 In the **Settings** window for Bézier Polygon, locate the **Polygon Segments** section.

4 Find the **Added segments** subsection. Click **Add Linear**.

5 Find the **Control points** subsection. In row **1**, set **y** to `depth`.

6 In row **2**, set **z** to `height`.

7 Find the **Added segments** subsection. Click **Add Linear**.

8 Find the **Control points** subsection. In row **2**, set **x** to `width` and **z** to `0`.

9 Find the **Added segments** subsection. Click **Add Linear**.

10 Find the **Control points** subsection. In row **2**, set **y** to `depth` and **z** to `height`.

11 Find the **Added segments** subsection. Click **Add Linear**.

12 Find the **Control points** subsection. In row **2**, set **x** to `0` and **z** to `0`.

Bézier Polygon 2 (b2)

1 On the **Geometry** toolbar, click **More 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 **1**, set **y** to `depth`.

5 In row **2**, set **x** to `width`.

Bézier Polygon 3 (b3)

1 On the **Geometry** toolbar, click **More 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 **1**, set **z** to `height`.

5 In row **2**, set **x** to `width`, **y** to `depth`, and **z** to `height`.

Convert to Curve 1 (ccur1)

1 Right-click **Bézier Polygon 3 (b3)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Conversions** and choose **Convert to Curve**.

3 Click in the **Graphics** window and then press Ctrl+A to select all objects.

Mirror 1 (mir1)

1 Right-click **Convert to Curve 1 (ccur1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

3 Select the object **ccur1** only.

4 In the **Settings** window for Mirror, locate the **Input** section.

5 Select the **Keep input objects** check box.

6 Locate the **Point on Plane of Reflection** section. In the **z** text field, type `height`.

Array 1 (arr1)

1 Right-click **Mirror 1 (mir1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Transforms** and choose **Array**.

3 Select the object **ccur1** only.

4 In the **Settings** window for Array, locate the **Size** section.

5 In the **z size** text field, type `n`.

6 Locate the **Displacement** section. In the **z** text field, type `2*height`.

7 Right-click **Array 1 (arr1)** and choose **Build Selected**.

8 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

Array 2 (arr2)

1 On the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the object **mir1** only.

3 In the **Settings** window for Array, locate the **Size** section.

4 In the **z size** text field, type `n-1`.

5 Locate the **Displacement** section. In the **z** text field, type `2*height`.

6 Right-click **Array 2 (arr2)** and choose **Build Selected**.

7 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.

TRUSS (TRUSS)

Cross Section Data 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Truss (truss)** click **Cross Section Data 1**.

2 In the **Settings** window for Cross Section Data, locate the **Cross Section Data** section.

3 In the A text field, type `A2`.

Cross Section Data 2

1 On the **Physics** toolbar, click **Edges** and choose **Cross Section Data**.

2 Select all the vertical edges. You can use the Select box tool for easier selection.

3 In the **Settings** window for Cross Section Data, locate the **Cross Section Data** section.

4 In the A text field, type `A1`.

Pinned 1

1 On the **Physics** toolbar, click **Points** and choose **Pinned**.

2 Select Points 1, 21, 41, and 61, located at the bottom of the tower.

Point Load 1

1 On the **Physics** toolbar, click **Points** and choose **Point Load**.

2 Select Points 20, 40, 60 and 80 located at the top of the tower.

3 In the **Settings** window for Point Load, locate the **Force** section.

4 Specify the $\mathbf{F}_\mathrm{P}$ vector as

0	x

0	y
-1/4	z

STUDY 1

Step 2: Linear Buckling

1 In the **Model Builder** window, under **Study 1** click **Step 2: Linear Buckling**.

2 In the **Settings** window for Linear Buckling, locate the **Study Settings** section.

3 In the **Desired number of buckling modes** text field, type 2.

4 On the **Home** toolbar, click **Compute**.

RESULTS

Line

1 In the **Model Builder** window, expand the **Mode Shape (truss)** node, then click **Line**.

2 In the **Settings** window for Line, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

4 Locate the **Coloring and Style** section. Clear the **Color legend** check box.

5 On the **Mode Shape (truss)** toolbar, click **Plot**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

Mode Shape (truss)

1 In the **Model Builder** window, under **Results** click **Mode Shape (truss)**.

2 In the **Settings** window for 3D Plot Group, locate the **Data** section.

3 From the **Critical load factor** list, choose **1.072E5**.

4 On the **Mode Shape (truss)** toolbar, click **Plot**.

Prestressed Bolts in a Tube Connection

Introduction

A tube connection consisting of a flange with four prestressed bolts (see Figure 1) is subjected to an external bending moment. In this example, you study how the stress state in the tube and the bolts varies with the applied load.

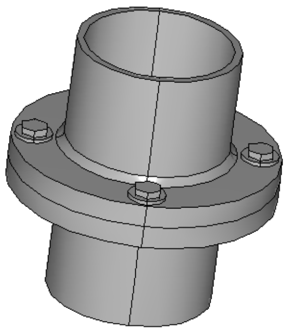


Figure 1: Tube connection.

Note: This application requires the Structural Mechanics Module and the CAD Import Module.

Model Definition

The tube is made of steel and has an outer diameter of 220 mm and an inner diameter of 200 mm. The flange has a diameter of 360 mm and a thickness of 30 mm. The connection consists of four prestressed M24 bolts. The bolts are prestressed to 75% of the yield strength. The total bending moment on the tube is increased from 0 to 26 kNm.

To compute the influence of the tensile force on the stress level in the bolt, the modeling includes a parametric analysis. Because of symmetry in both load and geometry, you only need to analyze one half of one of the flanges. The geometry has been created in the CAD software SOLIDWORKS® and is available as an IGES file.

Two contact regions are modeled. One contact pair acts between the bottom surface of the flange and the top surface of an additional fixed solid which supplies the symmetry condition with respect to contact. The other contact pair acts between the washers under the bolt heads and the flange. The possibility to automatically detect potential contact surfaces is used when creating the contact pairs.

Results and Discussion

After the pretension step, there is a tensile stress in the bolt, and compressive stress in the flange under the bolt. This is illustrated in Figure 2.

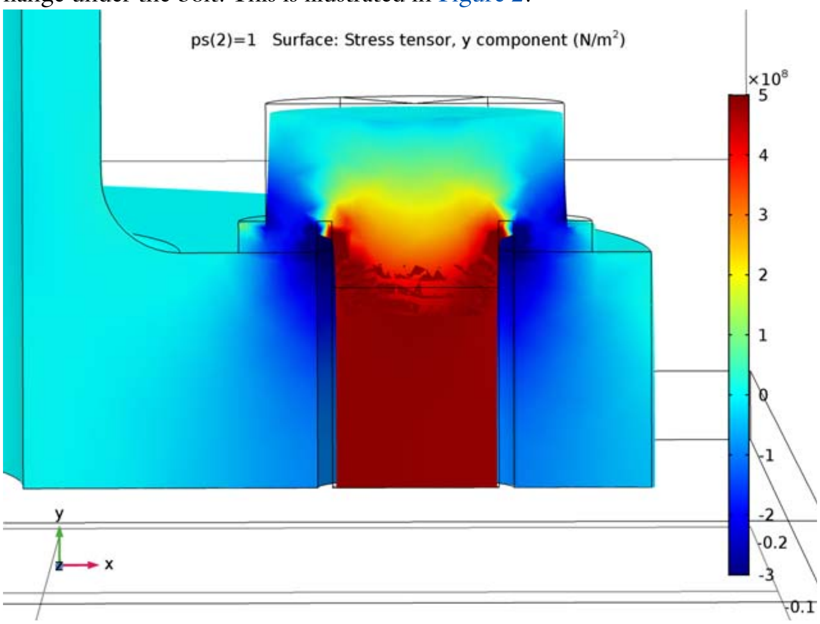


Figure 2: The axial stress after the pretension step.

The general stress state at maximum external load is shown in Figure 3. In addition to the stress that has increased in the fillet between tube and flange, additional features are visible. The stress state in the bolt on the tensile side has increased significantly and is no longer axisymmetric. Furthermore, a stress of the order of 300 MPa has developed in the fillet between the tube and the flange.This stress is caused by local bending, since the flange no longer is in contact with the mating surface at the tensile side.

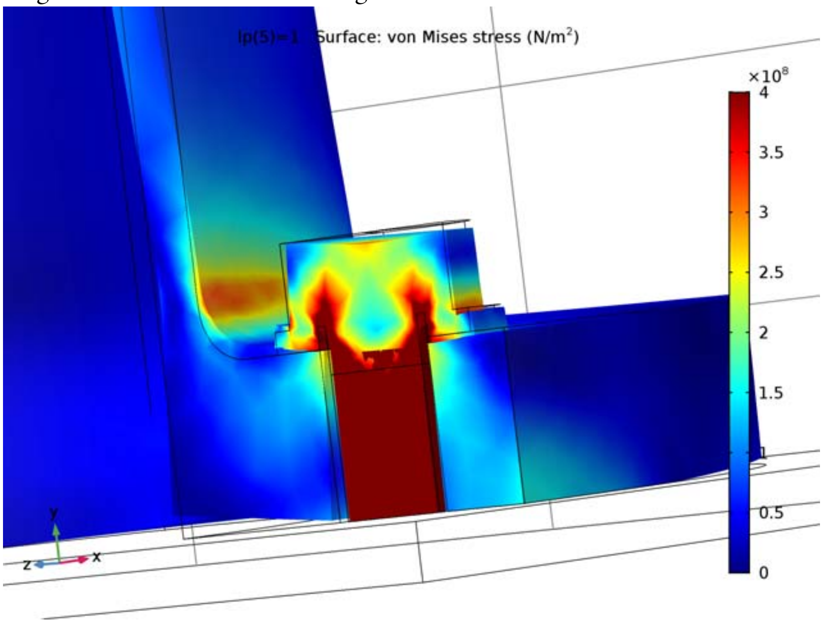


Figure 3: Effective stress at maximum external load.

The applied external load in this example overloads the bolted joint. Up to about half the full load, it works fairly well, but then the contact between the two mating flanges start to open up on the tensile side. This is displayed in Figure 4. Actually, the conditions are even worse than the average force indicates. The bolt is subjected to bending with a non-uniform stress distribution over the cross section. The maximum stress has increased from the prestress value of 75% of the yield stress to 88% of the yield stress, and the progression is fast. The development of the axial stress in two points on opposite sides of the bolt is displayed in Figure 5. The points are located in the x-z symmetry plane. One point is as close to the tube centerline as possible and the other as far out as possible.

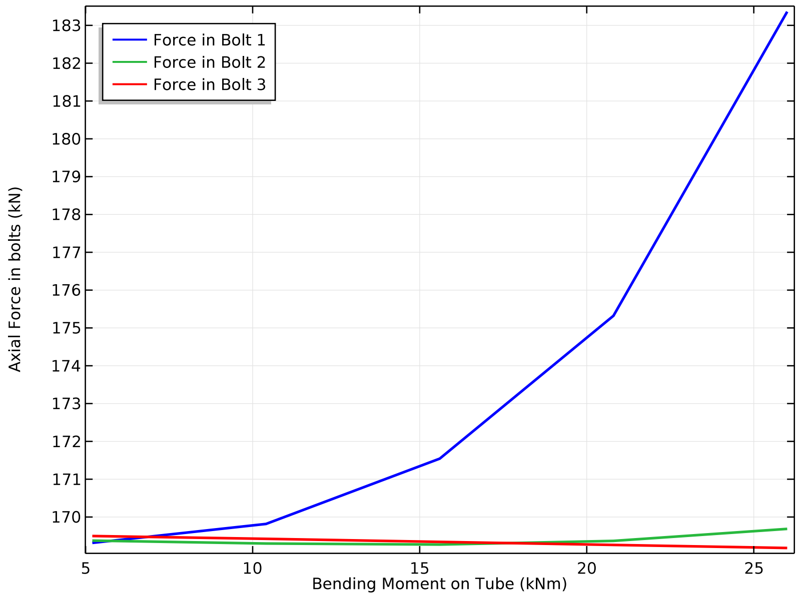


Figure 4: The bolt force as a function of the tensile force.

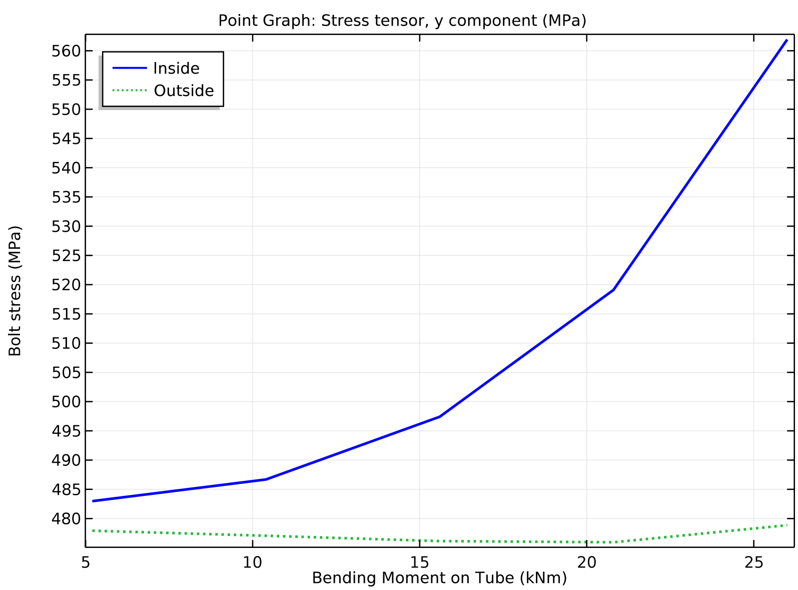


Figure 5: The development of the bolt stress at two different positions in the cross section.

The plots of the contact pressure between the mating flanges are shown in Figure 6 and Figure 7. By comparing these two figures, it is clear that the contact pressure shifts away from the initially prestressed area which at the peak load becomes almost stress free. This indicates that there are too few bolts that connect the two parts with each other.

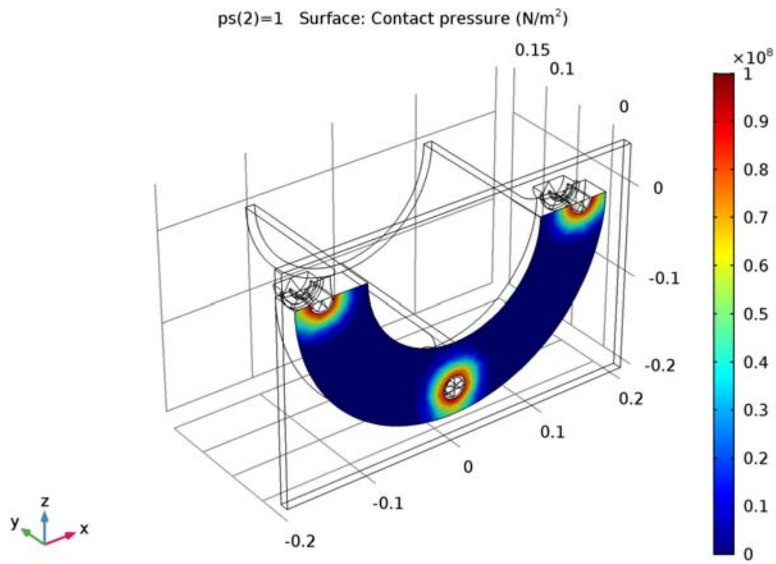


Figure 6: Contact pressure between flanges after pretensioning the bolts.

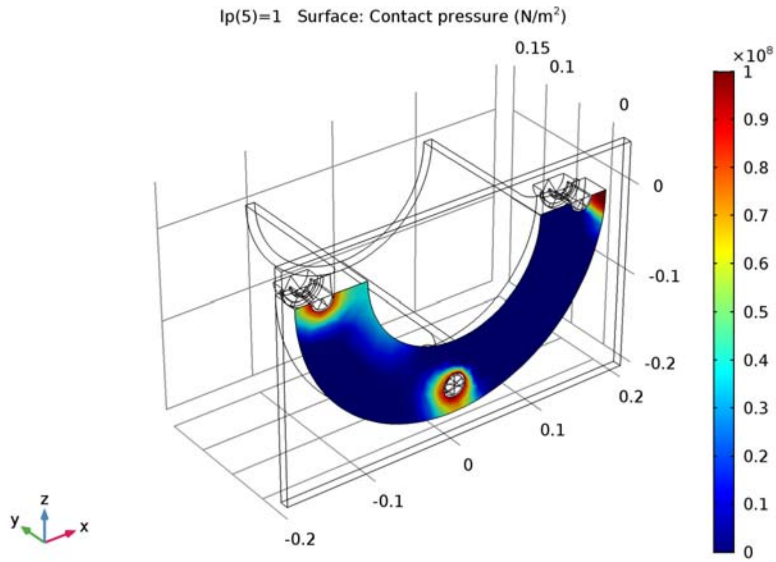


Figure 7: Contact pressure between the flanges at full external load.

Notes About the COMSOL Implementation

The analysis is performed in two steps and therefore represented by two studies. In the first step, the effects of pretensioning the bolt are computed, and in the second step the external load on the tube is applied as a parametric sweep.

The prestress in the bolts is introduced using the built-in Bolt Pre-tension feature. This feature creates one degree of freedom for each bolt, which can be interpreted as the shortening of the bolt caused by prestress. This degree of freedom is then kept fixed under service loads.

In order to keep the solution time down, a coarse mesh is used, and it probably needs a refinement if one needs to obtain accurate quantitative results for stresses.

Application Library path: `Structural_Mechanics_Module/Contact_and_Friction/tube_connection`

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

First, make sure that the CAD Import Module kernel is used.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for Geometry, locate the **Advanced** section.

3 From the **Geometry representation** list, choose **CAD kernel**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, 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
Di_wash	25[mm]	0.025 m	Washer inner diameter
Do_wash	45[mm]	0.045 m	Washer outer diameter
T_wash	4[mm]	0.004 m	Washer thickness
Do_bolthead	38[mm]	0.038 m	Bolthead diameter
H_bolthead	15[mm]	0.015 m	Bolthead height

Name	Expression	Value	Description
As_bolt	353[mm^2]	3.53E-4 m^2	Stress area of bolt
Ds_bolt	sqrt(4/pi*As_bolt)	0.0212 m	Effective bolt diameter
R_bc	150[mm]	0.15 m	Bolt circle radius
Do_pipe	0.22[m]	0.22 m	Pipe outer diameter
Di_pipe	0.20[m]	0.2 m	Pipe inner diameter
Wb	pi/32*(Do_pipe^4-Di_pipe^4)/Do_pipe	3.314E-4 m^3	Pipe bending resistance
M_appl	26E3[N*m]	2.6E4 N·m	Applied bending moment
ps	1	1	Parameter for spring relaxation
lp	0	0	Loading parameter

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 application's Application Libraries folder and double-click the file `tube_connection.igs`.
5. Click **Import**.

Use the bolts and washers from the part library.

PART LIBRARIES

1. On the **Home** toolbar, click **Windows** and choose **Part Libraries**.
2. In the **Part Libraries** window, select **Structural Mechanics Module>Bolts>flat washer** in the tree.
3. click **Add to Geometry**.

GEOMETRY 1

Flat Washer 1 (pi1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Flat Washer 1 (pi1)**.

2 In the **Settings** window for Part Instance, type `Washer 1` in the **Label** text field.

3 Locate the **Input Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
odia	`Do_wash`	0.045 m	Outer diameter
idia	`Di_wash`	0.025 m	Inner diameter
thickness	`T_wash`	0.004 m	Thickness

4 Locate the **Position and Orientation of Output** section. Find the **Displacement** subsection. In the **xw** text field, type `R_bc`.

5 Find the **Coordinate system in part** subsection. From the **Work plane in part** list, choose **Outer plane (wp2)**.

6 Find the **Rotation** subsection. From the **Axis type** list, choose **xw-axis**.

7 In the **Rotation angle** text field, type `90`.

8 Click to expand the **Boundary selections** section. Locate the **Boundary Selections** section. In the table, enter the following settings:

Name	Contribute to	Keep	Physics
Exterior	none		√
Inner face	none		√
Outer face	none	√	√
Hole faces	none		√

PART LIBRARIES

1 On the **Home** toolbar, click **Windows** and choose **Part Libraries**.

2 In the **Model Builder** window, click **Geometry 1**.

3 In the **Part Libraries** window, select **Structural Mechanics Module>Bolts>simple bolt no thread** in the tree.

4 click **Add to Geometry**.

GEOMETRY 1

Simple Bolt, No Thread 1 (pi2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Simple Bolt, No Thread 1 (pi2)**.

2 In the **Settings** window for Part Instance, type `Bolt 1` in the **Label** text field.

3 Locate the **Input Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
hdia	`Do_bolthead`	0.038 m	Head diameter
hthic	`H_bolthead`	0.015 m	Head thickness
ndia	`Ds_bolt`	0.0212 m	Nominal diameter
blen	`30[mm]+T_wash`	0.034 m	Bolt length

4 Locate the **Position and Orientation of Output** section. Find the **Coordinate system in part** subsection. From the **Work plane in part** list, choose **Head inner plane (wp1)**.

5 Find the **Coordinate system to match** subsection. From the **Take work plane from** list, choose **Washer 1 (pi1)**.

6 From the **Work plane** list, choose **Inner plane (wp1)**.

7 Find the **Rotation** subsection. From the **Axis type** list, choose **xw-axis**.

8 In the **Rotation angle** text field, type `180`.

9 Locate the **Boundary Selections** section. In the table, enter the following settings:

Name	Contribute to	Keep	Physics
Exterior	none		√
Shank	none		√
Head, free surface	none		√
Head, contact surface	none		√
Pre-tension cut	none	√	√

In the **Model Builder** window, select **Component 1 (comp1)>Geometry 1>Bolt 1 (pi2)** and **Component 1 (comp1)>Geometry 1>Washer 1 (pi1)**, right click and choose **Duplicate**.

Washer 1.1 (pi3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Washer 1.1 (pi3)**.

2 In the **Settings** window for Part Instance, type `Washer 2` in the **Label** text field.

3 Locate the **Position and Orientation of Output** section. Find the **Displacement** subsection. In the **xw** text field, type `0`.

4 In the **zw** text field, type `-R_bc`.

Bolt 1.1 (pi4)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Bolt 1.1 (pi4)**.

2 In the **Settings** window for Part Instance, type `Bolt 2` in the **Label** text field.

3 Locate the **Position and Orientation of Output** section. Find the **Coordinate system to match** subsection. From the **Take work plane from** list, choose **Washer 2 (pi3)**.

4 From the **Work plane** list, choose **Inner plane (wp1)**.

In the **Model Builder** window, select **Component 1 (comp1)>Geometry 1>Bolt 2 (pi4)** and **Component 1 (comp1)>Geometry 1>Washer 2 (pi3)**, right click and choose **Duplicate**.

Washer 2.1 (pi5)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Washer 2.1 (pi5)**.

2 In the **Settings** window for Part Instance, type `Washer 3` in the **Label** text field.

3 Locate the **Position and Orientation of Output** section. Find the **Displacement** subsection. In the **xw** text field, type `-R_bc`.

4 In the **zw** text field, type `0`.

Bolt 2.1 (pi6)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Bolt 2.1 (pi6)**.

2 In the **Settings** window for Part Instance, type `Bolt 3` in the **Label** text field.

3 Locate the **Position and Orientation of Output** section. Find the **Coordinate system to match** subsection. From the **Take work plane from** list, choose **Washer 3 (pi5)**.

4 From the **Work plane** list, choose **Inner plane (wp1)**.

Rotate 1 (rot1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Select the objects **pi1** and **pi2** only.

3 In the **Settings** window for Rotate, locate the **Point on Axis of Rotation** section.

4 In the **x** text field, type `R_bc`.

5 Locate the **Rotation Angle** section. In the **Rotation** text field, type `30`.

6 Locate the **Axis of Rotation** section. From the **Axis type** list, choose **y-axis**.

Rotate 2 (rot2)

1 Right-click **Rotate 1 (rot1)** and choose **Duplicate**.

2 In the **Settings** window for Rotate, locate the **Input** section.

3 Find the **Input objects** subsection. Select the **Active** toggle button.

4 Select the objects **pi5** and **pi6** only.

5 Locate the **Point on Axis of Rotation** section. In the **x** text field, type `-R_bc`.

Split 1 (spl1)

1 Right-click **Component 1 (comp1)>Geometry 1>Rotate 2 (rot2)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Conversions** and choose **Split**.

3 Select the objects **rot2(2)**, **rot1(2)**, and **pi4** only.

Union 1 (uni1)

1 Right-click **Split 1 (spl1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

3 Select the objects **spl1(18)**, **spl1(20)**, **spl1(30)**, **spl1(52)**, **spl1(38)**, **spl1(6)**, **spl1(48)**, **spl1(16)**, **spl1(27)**, **rot2(1)**, **spl1(2)**, **spl1(25)**, **spl1(1)**, **spl1(15)**, **spl1(43)**, **spl1(39)**, **pi3**, **spl1(17)**, **spl1(54)**, **spl1(37)**, **spl1(12)**, **spl1(24)**, **spl1(34)**, **spl1(33)**, **spl1(53)**, **spl1(21)**, **spl1(36)**, **spl1(9)**, **spl1(3)**, **spl1(51)**, **spl1(10)**, **rot1(1)**, **spl1(35)**, **spl1(42)**, **spl1(19)**, **spl1(46)**, **spl1(7)**, **spl1(45)**, and **spl1(28)** only.

4 In the **Settings** window for Union, locate the **Union** section.

5 Clear the **Keep interior boundaries** check box.

Union 2 (uni2)

1 Right-click **Union 1 (uni1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

3 Select the objects **spl1(14)**, **spl1(40)**, **spl1(8)**, **spl1(11)**, **spl1(4)**, **spl1(22)**, **spl1(23)**, **spl1(41)**, **spl1(32)**, **spl1(50)**, **spl1(5)**, **spl1(26)**, **spl1(49)**, **spl1(29)**, **spl1(13)**, **spl1(31)**, **spl1(47)**, and **spl1(44)** only.

4 In the **Settings** window for Union, locate the **Union** section.

5 Clear the **Keep interior boundaries** check box.

Union 3 (uni3)

1 Right-click **Union 2 (uni2)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

3 Select the objects **uni2** and **uni1** only.

4 Right-click **Union 3 (uni3)** and choose **Build Selected**.

Remove half of the geometry to make use of symmetry.

Work Plane 1 (wp1)

On the **Geometry** toolbar, click **Work Plane**.

Partition Objects 1 (par1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

2 In the **Settings** window for Partition Objects, locate the **Partition Objects** section.

3 Find the **Tool objects** subsection. Select the **Active** toggle button.

4 From the **Partition with** list, choose **Work plane**.

5 Click **Build All Objects**.

Now create the selection for upper half of the geometry to be removed.

Box Selection 1 (boxsel1)

1 On the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

2 In the **Settings** window for Box Selection, type `Upper half` in the **Label** text field.

3 Locate the **Box Limits** section. In the **z minimum** text field, type `-0.01`.

4 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

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 From the **Selection** list, choose **Upper half**.

5 Right-click **Component 1 (comp1)>Geometry 1>Delete Entities 1 (del1)** and choose **Build Selected**.

Create a rigid block to act as symmetry condition for the contact modeling.

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 `0.4`.

4 In the **Depth** text field, type `0.01`.

5 In the **Height** text field, type `0.25`.

6 Locate the **Position** section. In the **x** text field, type `-0.2`.

7 In the **y** text field, type `-0.04`.

8 In the **z** text field, type `-0.2`.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for Form Union/Assembly, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 From the **Pair type** list, choose **Contact pair**.

5 On the **Geometry** toolbar, click **Build All**.

Ignore Edges 1 (ige1)

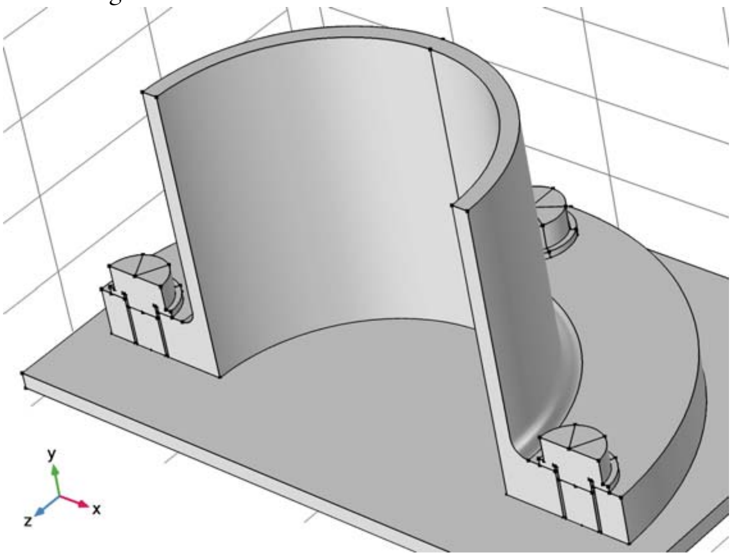
1 On the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Edges**.

2 On the object **fin**, select Edges 27 and 47 only.

This will improve the mesh at the transition to fillet, where large stresses can be expected.

3 On the **Geometry** toolbar, click **Build All**.

This completes the geometry modeling stage. The geometry should now look like that in the figure below.



DEFINITIONS

Define selections to use later in the modeling.

Box 1

1 On the **Definitions** toolbar, click **Box**.

2 In the **Settings** window for Box, type `Symmetry boundaries (Tube)` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **y minimum** text field, type `-0.04`.

5 In the **y maximum** text field, type `0.16`.

6 In the **z minimum** text field, type `-0.005`.

7 In the **z maximum** text field, type `0.01`.

8 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Symmetry boundaries (Bolts)` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 45, 92, and 149 only.

5 Select the **Group by continuous tangent** check box.

Union 1

1 On the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for Union, type `Symmetry boundaries (All)` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. 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 **Symmetry boundaries (Tube)** and **Symmetry boundaries (Bolts)**.

6 Click **OK**.

Union 2

1 On the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for Union, type `Contact boundaries (Washers)` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. 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 **Outer face (Washer 1)**, **Outer face (Washer 2)**, and **Outer face (Washer 3)**.

6 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>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.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1 On the **Physics** toolbar, click **Domains** and choose **Fixed Constraint**.

2 Select Domain 1 only.

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 In the **Settings** window for Symmetry, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Symmetry boundaries (All)**.

Add a constraint suppressing any rigid body translation in the x direction.

Prescribed Displacement 1

1 On the **Physics** toolbar, click **Points** and choose **Prescribed Displacement**.

2 Select Point 22 only.

3 In the **Settings** window for Prescribed Displacement, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in x direction** check box.

Bolt Pre-Tension 1

1 On the **Physics** toolbar, click **Global** and choose **Bolt Pre-Tension**.

2 In the **Settings** window for Bolt Pre-Tension, locate the **Bolt Pre-Tension** section.

3 From the **Pre-tension type** list, choose **Pre-tension stress**.

4 In the σ_p text field, type `800[MPa]*0.8*0.75`.

Bolt Selection 1

1 In the **Model Builder** window, expand the **Bolt Pre-Tension 1** node, then click **Bolt Selection 1**.

2 In the **Settings** window for Bolt Selection, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Pre-tension cut (Bolt 1)**.

Bolt Pre-Tension 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Bolt Pre-Tension 1**.

Bolt Selection 2

1 On the **Physics** toolbar, click **Attributes** and choose **Bolt Selection**.

2 In the **Settings** window for Bolt Selection, locate the **Bolt Selection** section.

3 In the **Bolt label** text field, type `Bolt_2`.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **Pre-tension cut (Bolt 2)**.

Bolt Pre-Tension 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Bolt Pre-Tension 1**.

Bolt Selection 3

1 On the **Physics** toolbar, click **Attributes** and choose **Bolt Selection**.

2 In the **Settings** window for Bolt Selection, locate the **Bolt Selection** section.

3 In the **Bolt label** text field, type `Bolt_3`.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **Pre-tension cut (Bolt 3)**.

Spring Foundation 1

1 On the **Physics** toolbar, click **Domains** and choose **Spring Foundation**.

2 Select Domains 2–8 only.

3 In the **Settings** window for Spring Foundation, locate the **Spring** section.

4 From the **Spring type** list, choose **Total spring constant**.

5 From the list, choose **Diagonal**.

6 In the $\mathbf{k}_{\text{tot}}$ table, enter the following settings:

1e10*(1-ps)	0	0
0	1e12*(1-ps)	0
0	0	1e10*(1-ps)

Before creating the contact conditions, check the automatically created contact pairs. The automatically generated source and destination selections are good in this case, so there is no need to change anything. The pair **ap2** is not needed. You can delete it to clean up the model, but this is not necessary.

DEFINITIONS

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node, then click **Contact Pair 1 (ap1)**.

2 In the **Settings** window for Pair, locate the **Advanced** section.

3 From the **Mapping method** list, choose **Initial configuration**.

4 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Contact Pair 3 (ap3)**.

5 In the **Settings** window for Pair, locate the **Advanced** section.

6 From the **Mapping method** list, choose **Initial configuration**.

SOLID MECHANICS (SOLID)

Contact 1

1 On the **Physics** toolbar, in the Boundary section, click **Pairs** and choose **Contact**.

2 In the **Settings** window for Contact, locate the **Pair Selection** section.

3 In the **Pairs** list, select **Contact Pair 1 (ap1)**.

4 Locate the **Penalty Factor** section. From the **Tuned for** list, choose **Speed**.

Contact 2

1 On the **Physics** toolbar, in the Boundary section, click **Pairs** and choose **Contact**.

2 In the **Settings** window for Contact, locate the **Pair Selection** section.

3 In the **Pairs** list, select **Contact Pair 3 (ap3)**.

4 Locate the **Penalty Factor** section. From the **Tuned for** list, choose **Speed**.

Friction 1

1 On the **Physics** toolbar, click **Attributes** and choose **Friction**.

2 In the **Settings** window for Friction, locate the **Friction** section.

3 In the μ_{stat} text field, type `0.15`.

Boundary Load 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundary 14 only.

3 In the **Settings** window for Boundary Load, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

0	x
lp*M_appl/Wb*X/(Do_pipe/2)	y
0	z

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 **Sequence type** list, choose **User-controlled mesh**.

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**.

Free Tetrahedral 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Tetrahedral 1** and choose **Size**.

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 Select Boundaries 13 and 20 only.

4 Locate the **Element Size** section. From the **Predefined** list, choose **Fine**.

5 Click **Build All**.

Free Tetrahedral 1

Refine the mesh on the destination boundaries of the contact pairs.

Size 2

1 Right-click **Free Tetrahedral 1** and choose **Size**.

2 In the **Settings** window for Size, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Contact boundaries (Washers)**.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Finer**.

6 Click **Build All**.

Since the block is rigid, one brick element is enough to represent it.

Mapped 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Mapped**.

2 Select Boundary 5 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Distribution**.

2 Select Edges 6 and 8 only.

3 In the **Settings** window for Distribution, locate the **Distribution** section.

4 In the **Number of elements** text field, type `1`.

5 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

Free Tetrahedral 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Tetrahedral 1**.

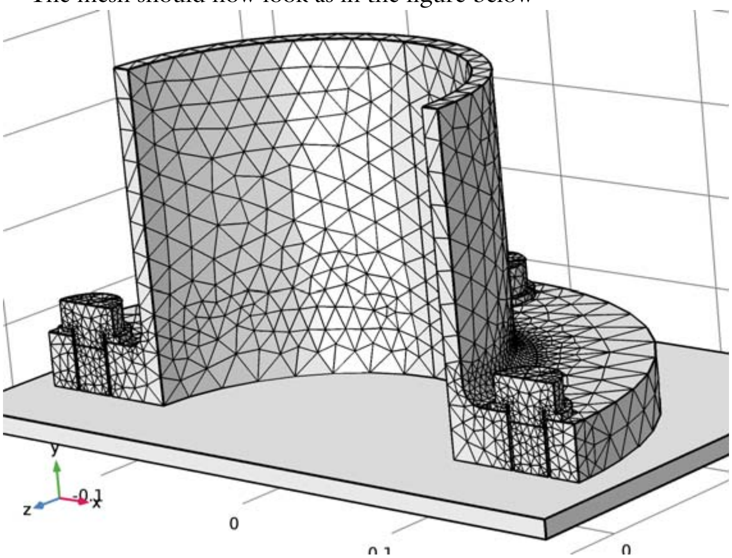
2 In the **Settings** window for Free Tetrahedral, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

Remove Domain 1 from the selection.

4 Click **Build All**.

The mesh should now look as in the figure below



STUDY 1

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for Study, type `Study 1: Prestress` in the **Label** text field.

STUDY 1: PRESTRESS

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1: Prestress** 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 **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
ps	0 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: Prestress>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** node, then click **Displacement field (Material) (comp1.u)**.

4 In the **Settings** window for Field, locate the **Scaling** section.

5 In the **Scale** text field, type `1e-4`.

The default scale for the displacements is 1% of the model size. This is significantly more than can be expected here.

6 On the **Study** toolbar, click **Compute**.

RESULTS

Stress (solid)

Reproduce the plot in Figure 2 with the following steps:

Surface 1

1 In the **Model Builder** window, expand the **Stress (solid)** 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>Stress tensor (Spatial)>solid.sy - Stress tensor, y component**.

3 Click to expand the **Range** section. Select the **Manual color range** check box.

4 In the **Minimum** text field, type `-3e8`.

5 In the **Maximum** text field, type `5e8`.

Deformation

1 In the **Model Builder** window, expand the **Surface 1** node, then click **Deformation**.

2 In the **Settings** window for Deformation, locate the **Scale** section.

3 In the **Scale factor** text field, type `50`.

Surface 1

In the **Model Builder** window, under **Results>Stress (solid)** right-click **Surface 1** and choose **Filter**.

Filter 1

1 In the **Settings** window for Filter, locate the **Element Selection** section.
2 In the **Logical expression for inclusion** text field, type `dom>6`.

   This removes the rigid block from the plot, since it contains the boundaries 1 to 6.

Deformation

1 In the **Model Builder** window, under **Results>Stress (solid)>Surface 1** click **Deformation**.
2 On the **Stress (solid)** toolbar, click **Plot**.

ROOT

Add a new study for the loading of the tube.

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.

STUDY 2

Step 1: Stationary

1 In the **Model Builder** window, click **Study 2**.
2 In the **Settings** window for Study, type `Study 2: External load` in the **Label** text field.

STUDY 2: EXTERNAL LOAD

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2: External load** click **Step 1: Stationary**.
2 In the **Settings** window for Stationary, locate the **Study Extensions** section.
3 Select the **Auxiliary sweep** check box.
4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
lp	range(0.2,0.2,1)	

6 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**.

7 From the **Method** list, choose **Solution**.

8 From the **Study** list, choose **Study 1: Prestress, Stationary**.

9 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

10 From the **Method** list, choose **Solution**.

11 From the **Study** list, choose **Study 1: Prestress, Stationary**.

The pre-deformation in the bolts is to be kept constant during the application of the external load, so it should not be solved for.

Solution 2 (sol2)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, under **Study 2: External load>Solver Configurations>Solution 2 (sol2)** click **Dependent Variables 1**.

4 In the **Settings** window for Dependent Variables, locate the **General** section.

5 From the **Defined by study step** list, choose **User defined**.

6 In the **Model Builder** window, under **Study 2: External load>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** click **Pre-deformation (comp1.solid.pblt1.sblt1.d_pre)**.

7 In the **Settings** window for State, locate the **General** section.

8 Clear the **Solve for this state** check box.

Repeat the same steps for the variables `comp1.solid.pblt1.sblt2.d_pre` and `comp1.solid.pblt1.sblt3.d_pre`."

9 In the **Model Builder** window, under **Study 2: External load>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** click **Displacement field (Material) (comp1.u)**.

10 In the **Settings** window for Field, locate the **Scaling** section.

11 In the **Scale** text field, type `1e-4`.

12 On the **Study** toolbar, click **Compute**.

RESULTS

Stress (solid) 1

The following steps reproduce the plot in Figure 3:

Surface 1

1 In the **Model Builder** window, expand the **Stress (solid) 1** node, then click **Surface 1**.

2 In the **Settings** window for Surface, locate the **Range** section.

3 Select the **Manual color range** check box.

4 In the **Maximum** text field, type `4e8`.

Deformation

1 In the **Model Builder** window, expand the **Surface 1** node, then click **Deformation**.

2 In the **Settings** window for Deformation, locate the **Scale** section.

3 In the **Scale factor** text field, type `50`.

Surface 1

In the **Model Builder** window, under **Results>Stress (solid) 1** right-click **Surface 1** and choose **Filter**.

Filter 1

1 In the **Settings** window for Filter, locate the **Element Selection** section.

2 In the **Logical expression for inclusion** text field, type `dom>6`.

Deformation

Proceed to plot the bolt forces as a function of the applied moment as in Figure 4.

1D Plot Group 3

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 **Study 2: External load/Solution 2 (sol2)**.

Global 1

1 On the **1D Plot Group 3** toolbar, click **Global**.

2 In the **Settings** window for Global, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Bolts> Bolt_1>solid.pblt1.sblt1.F_bolt - Bolt force**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
solid.pblt1.sblt1.F_bolt*2	kN	Force in Bolt 1
solid.pblt1.sblt2.F_bolt	kN	Force in Bolt 2
solid.pblt1.sblt3.F_bolt*2	kN	Force in Bolt 3

The bolts in the symmetry plane need the factor 2 in order to give the full bolt force.

4 Click to expand the **Legends** section. Clear the **Show legends** check box.

5 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type `2`.

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** text field, type `M_appl*lp/1000`.

1D Plot Group 3

1 In the **Model Builder** window, under **Results** click **1D Plot Group 3**.

2 In the **Settings** window for 1D Plot Group, locate the **Plot Settings** section.

3 Select the **x-axis label** check box.

4 In the associated text field, type `Bending Moment on Tube (kNm)`.

5 Select the **y-axis label** check box.

6 In the associated text field, type `Axial Force in bolts (kN)`.

Global 1

1 In the **Model Builder** window, under **Results>1D Plot Group 3** click **Global 1**.

2 In the **Settings** window for Global, locate the **Legends** section.

3 Select the **Show legends** check box.

4 Click to collapse the **Coloring and style** section.

1D Plot Group 3

1 In the **Model Builder** window, under **Results** click **1D Plot Group 3**.

2 In the **Settings** window for 1D Plot Group, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

4 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

The following steps create the plot in Figure 5:

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, locate the **Data** section.

3 From the **Data set** list, choose **Study 2: External load/Solution 2 (sol2)**.

Point Graph 1

1 On the **1D Plot Group 4** toolbar, click **Point Graph**.

2 Select Points 143 and 163 only.

3 In the **Settings** window for Point Graph, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Stress>Stress tensor (Spatial)>solid.sy - Stress tensor, y component**.

4 Locate the **y-Axis Data** section. From the **Unit** list, choose **MPa**.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `M_appl*lp/1000`.

7 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.

8 In the **Width** text field, type `2`.

9 Click to expand the **Legends** section. Select the **Show legends** check box.

10 From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
`Inside`
`Outside`

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, locate the **Plot Settings** section.

3 Select the **x-axis label** check box.

4 In the associated text field, type `Bending Moment on Tube (kNm)`.

5 Select the **y-axis label** check box.

6 In the associated text field, type `Bolt stress (MPa)`.

7 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Finally, reproduce the contact pressure plots shown in Figure 6 and Figure 7:

3D Plot Group 5

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 On the **3D Plot Group 5** toolbar, click **Surface**.

Surface 1

1 In the **Model Builder** window, under **Results>3D Plot Group 5** click **Surface 1**.

2 In the **Settings** window for Surface, click to expand the **Range** section.

3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics>Contact>Contact 1>solid.cnt1.Tn - Contact pressure**.

4 Locate the **Range** section. Select the **Manual color range** check box.

5 In the **Maximum** text field, type `1E8`.

6 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

7 On the **3D Plot Group 5** toolbar, click **Plot**.

3D Plot Group 5

1 In the **Model Builder** window, under **Results** click **3D Plot Group 5**.

2 In the **Settings** window for 3D Plot Group, locate the **Data** section.

3 From the **Data set** list, choose **Study 2: External load/Solution 2 (sol2)**.

4 From the **Parameter value (lp)** list, choose **1**.

5 On the **3D Plot Group 5** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.2a

Thermal Stress Analysis of a Turbine Stator Blade

Introduction

The conditions within gas turbines are extreme. The pressure can be as high as 40 bars, and the temperature far above 1000 K. Any new component must therefore be carefully designed to be able to withstand thermal stress, vibrations due to the rotating machinery and aerodynamic loads exerted by the fluid rushing through the turbine. If a component fails, the high rotational speeds can result in a complete rupture of the whole turbine.

The most extreme conditions are found in the high pressure part downstream of the combustion chamber where hot combustion gas flows through a cascade of rotors and stators. To prevent the parts from melting, relatively "cold" air is taken from bleeding vanes located in the high-pressure compressor casing, then led past the combustion chamber into the turbine casing in order to be used as a coolant. Directly behind the combustion chamber, both internal cooling within ducts and film cooling over the blade side surfaces are applied. Further downstream, where the temperature is somewhat lower, it may suffice with internal cooling. For more details on gas turbines, see Ref. 1.

Since the physics within a gas turbine is very complex, simplified approaches are often used at initial stages of the development of the new components. In this tutorial, the thermal stresses in a stator blade with internal cooling are analyzed.

Note: This application requires the Structural Mechanics Module and the CFD Module or Heat Transfer Module. It also uses the Material Library.

Model Definition

The model geometry is shown in Figure 1. The stator blade profile is a modified version of a design shown in Ref. 2. The model includes some generic mounting details as well as a generic internal cooling duct.

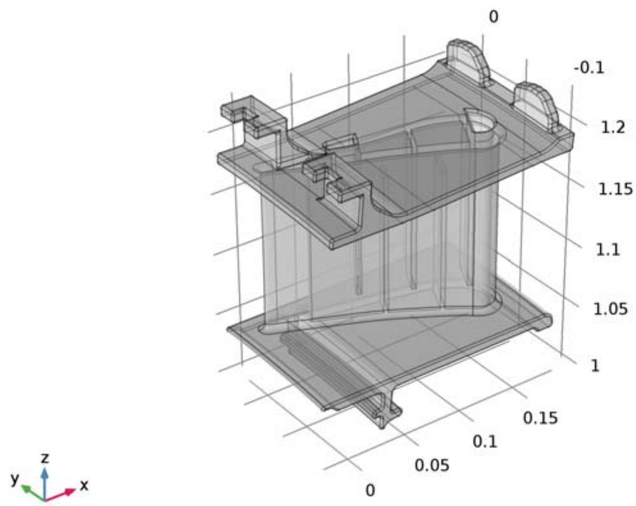


Figure 1: A stator blade with mounting details.

Use the Thermal Stress interface from the Structural Mechanics Module to set up the model. The blade and the mounting details are assumed to be made of the M-152 alloy which is a 12 Cr alloy with high tensile strength (Ref. 1). M-152 is available in the COMSOL Material Library. In addition to the data covered by the Material Library, the linear elastic model requires a reference temperature that is set to 300 K and a Poisson's Ratio that is set to 0.33, a number comparable to that for other stainless steels. Any coating has been neglected.

Figure 2 shows the cooling duct. The duct geometry is simplified and does not include details such as the ribs (Ref. 3) typical for cooling ducts. Instead of simulating the complicated flow in the duct, an average Nusselt number correlation from Ref. 3 is used to calculate a heat transfer coefficient. Assume the cooling fluid to be air at 30 bar and 600 K.

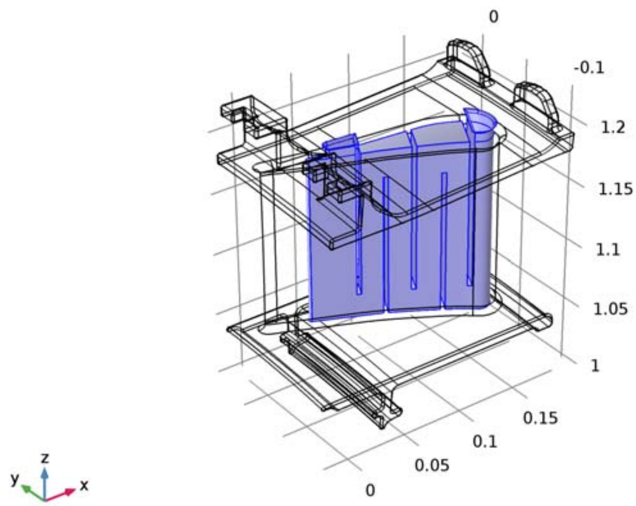


Figure 2: The internal cooling duct.

The heat flux on the stator blade surfaces is calculated using the heat transfer coefficient. The pressure and suction sides are approximated as two flat plates using the local heat transfer coefficient for external forced convection. The combustion gases are approximated as air at 30 bar and 1100 K. The corresponding speed of sound is approximately 650 m/s.

Ref. 4 contains a Mach number plot of stators without film cooling. A typical Mach number is 0.7 on the pressure side (the concave side) and 0.45 on the suction side (the convex side). This corresponds to approximately 450 m/s on the pressure side and 300 m/s on the suction side.

The platform walls adjacent to the stator blades are treated in the same way as the stator itself but with the free stream velocity set to 350 m/s.

The stator blade exchanges heat with the cooling air through the boundaries highlighted in Figure 3. It is assumed that the turbine has a local working temperature of 900 K, and that the heat transfer coefficient to the stator is 25 W/(m^2·K).

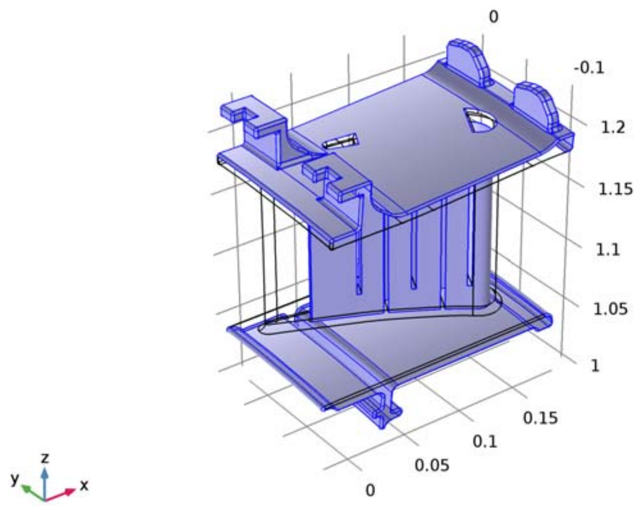


Figure 3: Boundaries through which heat is exchanged with the cooling air.

The attachment of the stator element to a ring support is simulated via roller and spring foundation boundary conditions on few boundaries. All other boundaries are free to deform as a result of thermal expansion.

Results and Discussion

Figure 4 shows a temperature surface plot. The internal cooling creates significant temperature gradients within the blade. However, the trailing edge reaches a temperature close to that of the combustion gases, which indicates that the cooling might be insufficient. The side walls also become very hot, and some additional cooling can be beneficial.

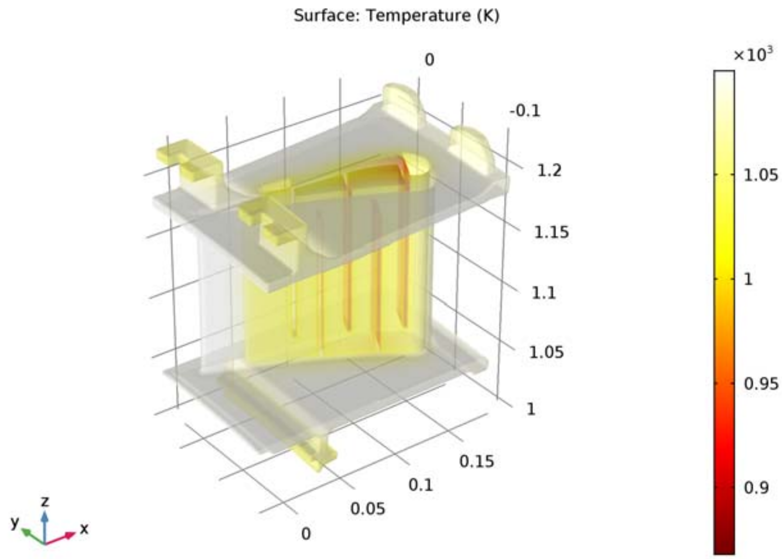


Figure 4: Surface temperature plot.

Figure 5 shows a surface plot of the von Mises stress. The maximum stress with a value close to the yield stress for the material (Ref. 5) occurs in the internal cooling duct. No definite assessment can however be made without conducting a more advanced analysis that includes details of the flow inside the duct.

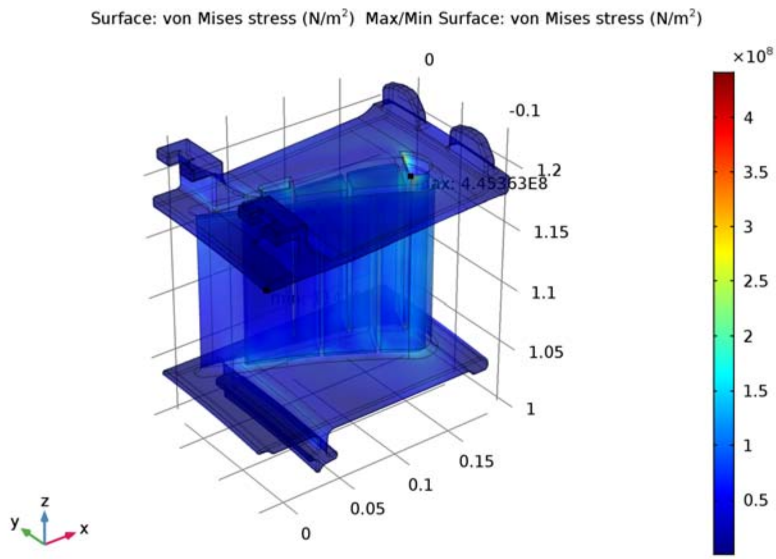


Figure 5: Surface plot of the von Mises stress.

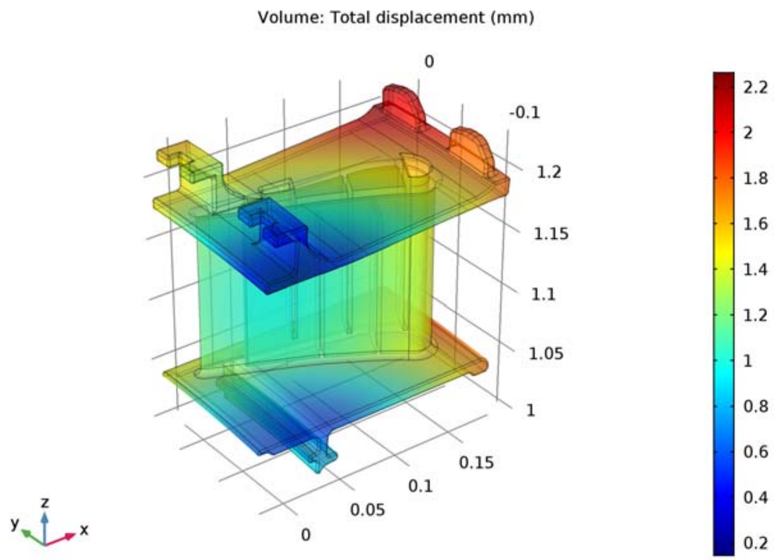


Figure 6: Surface plot of the displacement.

References

1. M.P. Boyce, *Gas Turbine Engineering Handbook*, 2nd ed., Gulf Professional Publishing, 2001.

2. NASA, "Power Turbine," Glenn Research Center, www.grc.nasa.gov/WWW/K-12/airplane/powturb.html.

3. J. Bredberg, "Turbulence Modelling for Internal Cooling of Gas-Turbine Blades," Thesis for the degree of doctor of philosophy, *Chalmers University of Technology*, 2002.

4. P. Dahlander, "Source Term Model Approaches to Film Cooling Simulations," Thesis for the degree of doctor of philosophy, *Chalmers University of Technology*, 2001.

5. http://www.cnalloys.co.uk/stainless-jethete-m152

Application Library path: `Structural_Mechanics_Module/Thermal-Structure_Interaction/turbine_stator`

Note: Instructions below require to select entities corresponding to a particular numbers list. For example:
Select Boundaries 113 and 139 only.
In most cases the easiest way to select them is to click the **Paste Selection** button and paste the numbers as they are printed in the document (for example paste "`113 and 139`" for the example above).

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>Thermal Stress**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, 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
Pr_cool	0.72	0.72	Cooling Prandtl number
U_suction_side	450[m/s]	450 m/s	Gas velocity on stator suction side
U_pressure_side	300[m/s]	300 m/s	Gas velocity on stator pressure side

Name	Expression	Value	Description
U_platform	350[m/s]	350 m/s	Gas velocity along platform walls
T_gas	1100[K]	1100 K	Gas temperature
p_high	30[bar]	3E6 Pa	High pressure level
mu_cool	3.1e-5[Pa*s]	3.1E-5 Pa·s	Viscosity of the cooling air
Cp_cool	770[J/kg/K]	770 J/(kg·K)	Heat capacity of the cooling air
T_cool	800[K]	800 K	Cooling air temperature
H_cool	0.01[m]	0.01 m	Characteristic length scale of cooling channels
T_work	900[K]	900 K	Working temperature
Nu_cool	400	400	Average Nusselt number in cooling channel

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 application's Application Libraries folder and double-click the file `turbine_stator.mphbin`.
5. Click **Import**.

   To see the interior:
6. Click the **Transparency** button on the **Graphics** toolbar.

   The imported geometry should look as shown in the Figure 1.
7. On the **Home** toolbar, click **Build All**.
8. Click the **Zoom Extents** button on the **Graphics** toolbar.

DEFINITIONS

Define a number of sections to simplify the model setup. First define the internal cooling duct boundaries.

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Cooling_Duct` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `222-225, 236, 261-266, 270-271, 278-279, 283-285, 292-305, 308-311, 313-314, 316, 321` in the **Selection** text field.

If you are reading an electronic version of this document, you can copy the geometric entity numbers from the text.

6 Click **OK**.

7 Click the **Transparency** button on the **Graphics** toolbar.

8 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

The selection is shown in Figure 2.

Proceed to select the boundaries through which heat exchanger with the rest of turbine occur Figure 3.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Exchange_Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `3-4, 9, 12, 14, 16, 19-20, 22-24, 28-96, 98-117, 122-135, 138-162, 166-221, 223, 226-231, 239-260, 267-269, 272-277, 280-282, 322-444` in the **Selection** text field.

6 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 **Material Library>Iron Alloys>JETHETE M-152 or Moly Ascoloy>JETHETE M-152 or Moly Ascoloy [solid]**.

4 Click **Add to Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

JETHETE M-152 or Moly Ascoloy [solid] (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **JETHETE M-152 or Moly Ascoloy [solid] (mat1)**.

2 In the **Settings** window for Material, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Poisson's ratio	nu	0.33	1	Young's modulus and Poisson's ratio

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Discretization** in the menu.

2 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

3 In the **Settings** window for Solid Mechanics, click to expand the **Discretization** section.

4 From the **Displacement field** list, choose **Linear**.

MULTIPHYSICS

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Thermal Expansion 1 (te1)**.

2 In the **Settings** window for Thermal Expansion, locate the **Thermal Expansion Properties** section.

3 In the T_{ref} text field, type `300[K]`.

HEAT TRANSFER IN SOLIDS (HT)

On the **Physics** toolbar, click **Solid Mechanics (solid)** and choose **Heat Transfer in Solids (ht)**.

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 `T_gas` in the T text field.

3 In the **Model Builder** window, 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 **Exchange_Boundaries**.

4 Locate the **Heat Flux** section. Click the **Convective heat flux** button.

5 In the h text field, type 25.

6 In the T_{ext} text field, type T_work.

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 **Cooling_Duct**.

4 Locate the **Heat Flux** section. Click the **Convective heat flux** button.

5 In the h text field, type Nu_cool*mu_cool*Cp_cool/2/Pr_cool/H_cool.

6 In the T_{ext} text field, type T_cool.

Heat Flux 3

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 Select Boundaries 137 and 163 only.

3 In the **Settings** window for Heat Flux, locate the **Heat Flux** section.

4 Click the **Convective heat flux** button.

5 From the **Heat transfer coefficient** list, choose **External forced convection**.

6 From the list, choose **Plate, local transfer coefficient**.

7 In the x_{pl} text field, type 0.1675-x.

8 In the U text field, type U_suction_side.

9 In the p_A text field, type p_high.

10 In the T_{ext} text field, type T_gas.

Heat Flux 4

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 Select Boundaries 136 and 312 only.

3 In the **Settings** window for Heat Flux, locate the **Heat Flux** section.

4 Click the **Convective heat flux** button.

5 From the **Heat transfer coefficient** list, choose **External forced convection**.

6 From the list, choose **Plate, local transfer coefficient**.

7 In the x_{pl} text field, type 0.1675-x.

8 In the U text field, type U_pressure_side.

9 In the p_A text field, type `p_high`.

10 In the T_{ext} text field, type `T_gas`.

Heat Flux 5

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 Select Boundaries 15, 21, 118–121, 164, 165, 232, 234, 235, 237, 286, 289–291, 306, 307, 315, 317, and 320 only.

3 In the **Settings** window for Heat Flux, locate the **Heat Flux** section.

4 Click the **Convective heat flux** button.

5 From the **Heat transfer coefficient** list, choose **External forced convection**.

6 From the list, choose **Plate, local transfer coefficient**.

7 In the x_{pl} text field, type `0.19-x`.

8 In the U text field, type `U_platform`.

9 In the p_A text field, type `p_high`.

10 In the T_{ext} text field, type `T_gas`.

SOLID MECHANICS (SOLID)

In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

Roller 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Roller**.

2 Select Boundaries 140, 146, and 213 only.

Spring Foundation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Spring Foundation**.

2 Select Boundaries 426 and 428 only.

3 In the **Settings** window for Spring Foundation, locate the **Spring** section.

4 From the list, choose **Diagonal**.

5 In the $\mathbf{k}_A$ table, enter the following settings:

1e9	0	0
0	0	0
0	0	0

Spring Foundation 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Spring Foundation**.

2 Select Boundaries 82, 90, 192, and 255 only.

3 In the **Settings** window for Spring Foundation, locate the **Spring** section.

4 From the list, choose **Diagonal**.

5 In the $\mathbf{k}_A$ table, enter the following settings:

0	0	0
0	0	0
0	0	1e10

Spring Foundation 3

1 On the **Physics** toolbar, click **Boundaries** and choose **Spring Foundation**.

2 Select Boundary 17 only.

3 In the **Settings** window for Spring Foundation, locate the **Spring** section.

4 From the list, choose **Diagonal**.

5 In the $\mathbf{k}_A$ table, enter the following settings:

0	0	0
0	1e9	0
0	0	0

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Size**.

Size

1 In the **Settings** window for Size, locate the **Element Size** section.

2 From the **Predefined** list, choose **Fine**.

Size 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size 1**.

2 In the **Settings** window for Size, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Cooling_Duct**.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Fine**.

6 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Tetrahedral**.

Free Tetrahedral 1

In the **Settings** window for Free Tetrahedral, click **Build All**.

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Stress (solid)

The first default plot show the von Mises stress. Disable the deformation and create a max/min plot to display the critical point in the stator.

Surface 1

In the **Model Builder** window, expand the **Stress (solid)** node.

Deformation

1 In the **Model Builder** window, expand the **Surface 1** node.

2 Right-click **Deformation** and choose **Disable**.

3 On the **Stress (solid)** toolbar, click **More Plots** and choose **Max/Min Surface**.

Max/Min Surface 1

1 In the **Settings** window for Max/Min Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics>Stress>solid.mises - von Mises stress**.

2 On the **Stress (solid)** toolbar, click **Plot**.

3 Click the **Zoom Extents** button on the **Graphics** toolbar.

4 Click the **Transparency** button on the **Graphics** toolbar.

The second default plot shows the temperature distribution Figure 4.

Temperature (ht)

1 In the **Model Builder** window, under **Results** click **Temperature (ht)**.

2 In the **Settings** window for 3D Plot Group, locate the **Plot Settings** section.

3 Clear the **Plot data set edges** check box.

4 On the **Temperature (ht)** toolbar, click **Plot**.

5 Click the **Zoom Extents** button on the **Graphics** toolbar.

Finally, plot the total displacement (Figure 6).

3D Plot Group 4

On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

Volume 1

1 In the **Model Builder** window, right-click **3D Plot Group 4** and choose **Volume**.
2 In the **Settings** window for Volume, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics> Displacement>solid.disp - Total displacement**.
3 Locate the **Expression** section. From the **Unit** list, choose **mm**.
4 On the **3D Plot Group 4** toolbar, click **Plot**.
5 Click the **Zoom Extents** button on the **Graphics** toolbar.

Created in COMSOL Multiphysics 5.2a

Viscoelastic Structural Damper

Introduction

The example studies a forced response of a typical viscoelastic damper. Damping elements involving layers of viscoelastic materials are often used for reduction of seismic and wind induced vibrations in buildings and other tall structures. The common feature is that the frequency of the forced vibrations is low.

Model Definition

The geometry of the viscoelastic damper is shown in Figure 1, from Ref. 1. The damper consists of two layers of viscoelastic material confined between mounting elements made of steel.

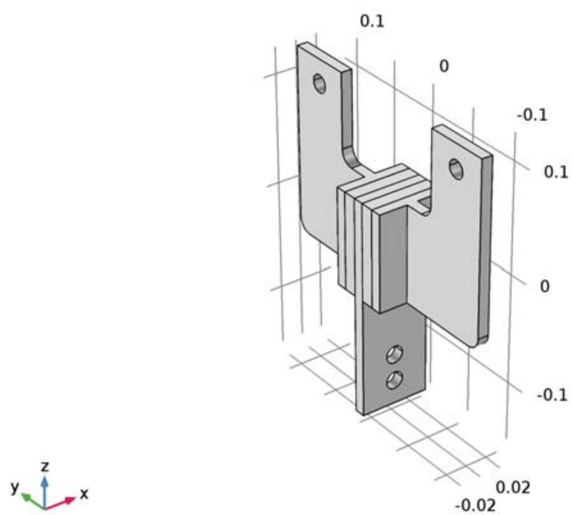


Figure 1: Viscoelastic damping element.

You model the viscoelastic layers by the generalized Maxwell model available in COMSOL Multiphysics. The generalized Maxwell model represents the viscoelastic material as a series of branches, each with a spring-dashpot pair. You can find more details about this material model in the section "About Viscoelastic Materials" in the *Structural Mechanics Module User's Guide*.

Eighteen viscoelastic branches guarantee accurate representation of the material behavior for a wide range of excitation frequencies, when the damper is subjected to forced vibration. The values of the shear moduli and relaxation times for each branch are available in Ref. 1. They are summarized in the following table:

TABLE 1: MODEL DATA FOR THE VISCOELASTIC DAMPER MODEL

PROPERTY	VALUE	DESCRIPTION
G	$5.86 \cdot 10^{-2}$ MPa	Long time shear modulus
ρ	1.06 g/cm^3	Density
G_1	13,3 MPa	Shear modulus branch 1
τ_1	10^{-7} s	Relaxation time branch 1
G_2	286 MPa	Shear modulus branch 2
τ_2	10^{-6} s	Relaxation time branch 2
G_3	$2.91 \cdot 10^{2}$ MPa	Shear modulus branch 3
τ_3	$3.16 \cdot 10^{-6}$ s	Relaxation time branch 3
G_4	$2.12 \cdot 10^{2}$ MPa	Shear modulus branch 4
τ_4	10^{-5} s	Relaxation time branch 4
G_5	$1.12 \cdot 10^{2}$ MPa	Shear modulus branch 5
τ_5	$3.16 \cdot 10^{-5}$ s	Relaxation time branch 5
G_6	61.6 MPa	Shear modulus branch 6
τ_6	10^{-4} s	Relaxation time branch 6
G_7	29.8 MPa	Shear modulus branch 7
τ_7	$3.16 \cdot 10^{-4}$ s	Relaxation time branch 7
G_8	16.1 MPa	Shear modulus branch 8
τ_8	10^{-3} s	Relaxation time branch 8
G_9	7.83 MPa	Shear modulus branch 9
τ_9	$3.16 \cdot 10^{-3}$ s	Relaxation time branch 9
G_{10}	4.15 MPa	Shear modulus branch 10
τ_{10}	10^{-2} s	Relaxation time branch 10
G_{11}	2.03 MPa	Shear modulus branch 11
τ_{11}	$3.16 \cdot 10^{-2}$ s	Relaxation time branch 11
G_{12}	1.11 MPa	Shear modulus branch 12
τ_{12}	0.1 s	Relaxation time branch 12
G_{13}	0.491 MPa	Shear modulus branch 13
τ_{13}	0.316 s	Relaxation time branch 13

TABLE 1: MODEL DATA FOR THE VISCOELASTIC DAMPER MODEL

PROPERTY	VALUE	DESCRIPTION
G_{14}	0.326 MPa	Shear modulus branch 14
τ_{14}	1 s	Relaxation time branch 14
G_{15}	$8.25\cdot10^{-2}$ MPa	Shear modulus branch 15
τ_{15}	3.16 s	Relaxation time branch 15
G_{16}	0.126 MPa	Shear modulus branch 16
τ_{16}	10 s	Relaxation time branch 16
G_{17}	$3.73\cdot10^{-2}$ MPa	Shear modulus branch 17
τ_{17}	100 s	Relaxation time branch 17
G_{18}	$1.18\cdot10^{-2}$ MPa	Shear modulus branch 18
τ_{18}	1000 s	Relaxation time branch 18

One of the mounting elements is fixed; the other two are loaded with periodic forces with frequencies in the range 0–5 Hz.

The time-domain representation of the forced solution is computed using the fast Fourier transform (FFT).

Results and Discussion

The harmonic response at 3 Hz is shown in Figure 2.

In the frequency domain, the viscoelastic properties of the material appear as the storage modulus and loss modulus. The computed variation of the viscoelastic moduli with frequency is shown in Figure 3. The result is in very good agreement with the experimental data (Figure 7 in Ref. 2)

.

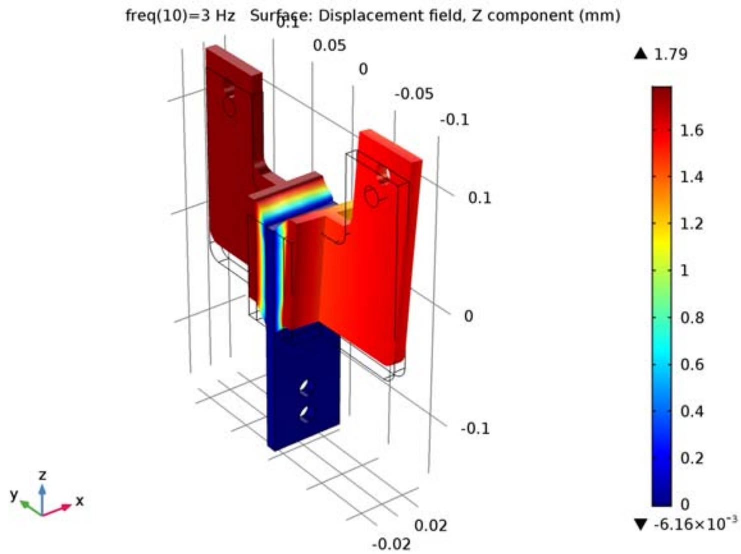


Figure 2: Vertical displacement of the damper, harmonic response.

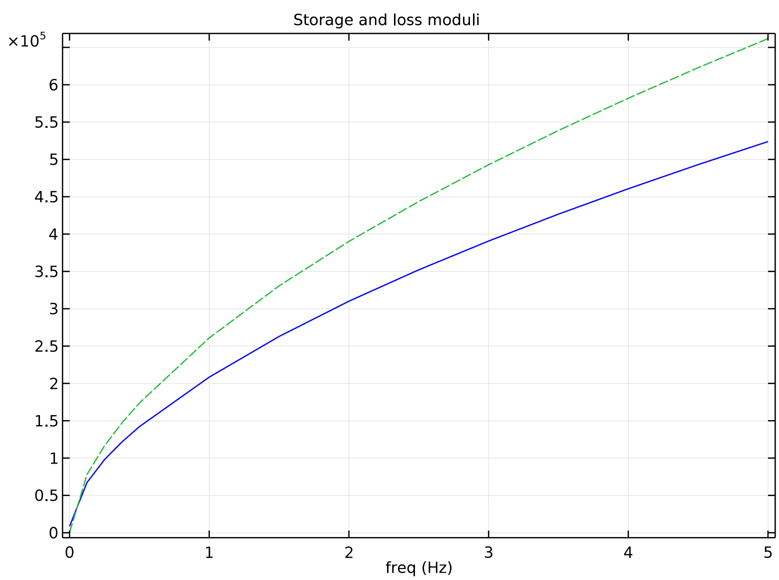


Figure 3: Viscoelastic storage modulus (solid line) and loss modulus (dashed line). Both quantities are normalized by 6.895 to simplify the comparison with Ref. 2.

The time domain solution obtained via FFT for the case of excitation frequency of 3 Hz is shown in Figure 4.

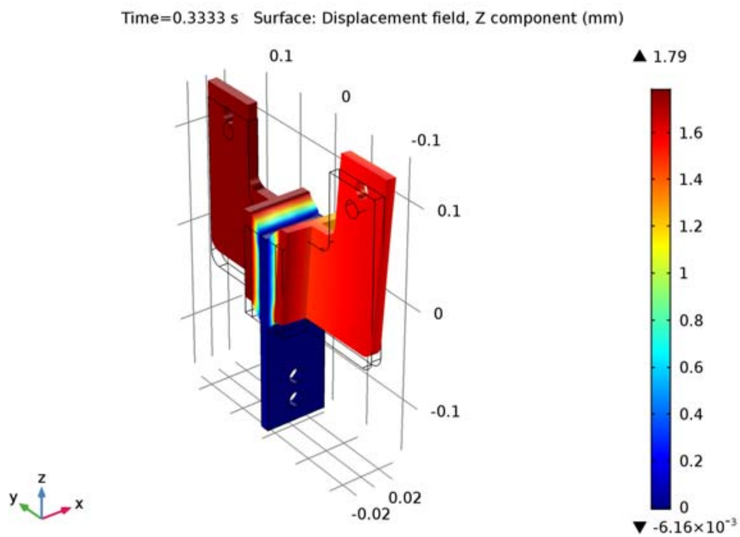


Figure 4: Displacement of the damper after 1/3 second of forced vibrations.

Finally, the total vertical force vs vertical displacement for one of the mounting holes is shown in Figure 5.

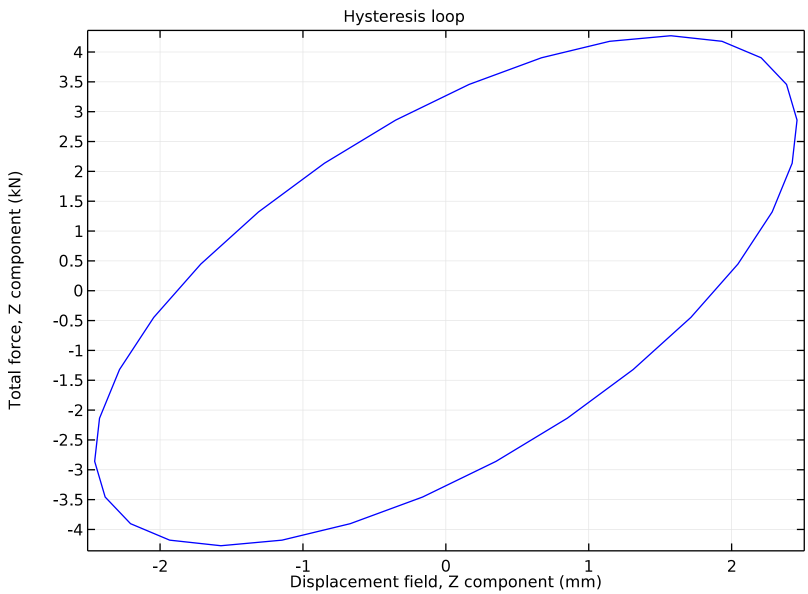


Figure 5: Hysteresis loop for excitation frequency of 3 [Hz] over the time interval of 1/3 second.

Notes About the COMSOL Implementation

You model in 3D and use the Solid Mechanics interface with Linear Elastic Material, add the Viscoelasticity node to the domains representing the viscoelastic layers.

References

1. S.W. Park “Analytical Modeling of Viscoelastic Dampers for Structural and Vibration Control,” *Int. J. Solids and Structures*, vol. 38, pp. 694–701, 2001.

2. K.L. Shen and T.T. Soong, “Modeling of Viscoelastic Dampers for Structural Applications,” *J. Eng. Mech.*, vol. 121, pp. 694–701, 1995.

Application Library path: `Structural_Mechanics_Module/Dynamics_and_Vibration/viscoelastic_damper_frequency`

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>Frequency Domain**.

6 Click **Done**.

GEOMETRY 1

You import the damper geometry from a file.

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 application's Application Libraries folder and double-click the file `viscoelastic_damper.mphbin`.

5 Click **Import**.

6 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

The imported geometry should look similar to that shown in Figure 1.

SOLID MECHANICS (SOLID)

Linear Elastic Material 2

1 On the **Physics** toolbar, click **Domains** and choose **Linear Elastic Material**.

2 In the **Settings** window for Linear Elastic Material, locate the **Linear Elastic Material** section.

3 From the **Specify** list, choose **Bulk modulus and shear modulus**.

4 Select the **Nearly incompressible material** check box.

5 Select Domains 2 and 4 only.

Viscoelasticity 1

1 On the **Physics** toolbar, click **Attributes** and choose **Viscoelasticity**.

Since there are 18 branches in this material model, the data has been collected in a text file which you can load.

2 In the **Settings** window for Viscoelasticity, locate the **Viscoelasticity Model** section.

3 Click **Load from File**.

4 Browse to the application's Application Libraries folder and double-click the file `viscoelastic_damper_viscoelastic_data.txt`.

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.

MATERIALS

Steel AISI 4340 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Steel AISI 4340 (mat1)**.

2 Select Domains 1, 3, and 5 only.

Material 2 (mat2)

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 `Viscoelastic` in the **Label** text field.

3 Select Domains 2 and 4 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Bulk modulus	K	4e8	N/m²	Bulk modulus and shear modulus
Shear modulus	G	5.86e4	N/m²	Bulk modulus and shear modulus
Density	rho	1060	kg/m³	Basic

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Select Boundaries 24–27 only.

Prescribed Displacement 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

2 Select Boundaries 40 and 41 only.

3 In the **Settings** window for Prescribed Displacement, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in x direction** check box.

5 Select the **Prescribed in y direction** check box.

Boundary Load 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundaries 40 and 41 only.

3 In the **Settings** window for Boundary Load, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

0	x
0	y
8.5e6[Pa]	z

Phase 1

1 On the **Physics** toolbar, click **Attributes** and choose **Phase**.

2 In the **Settings** window for Phase, locate the **Phase** section.

3 Specify the ϕ vector as

0	x
0	y
pi/2	z

Prescribed Displacement 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

2 Select Boundaries 32 and 33 only.

3 In the **Settings** window for Prescribed Displacement, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in y direction** check box.

Boundary Load 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundaries 32 and 33 only.

3 In the **Settings** window for Boundary Load, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

5e5[Pa]	x
0	y
8.5e6[Pa]	z

MESH 1

Mesh the side surfaces of the viscoelastic layers and then sweep the resulting mesh into the layers.

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 Boundaries 6 and 20 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 2 and 4 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 3.

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**.

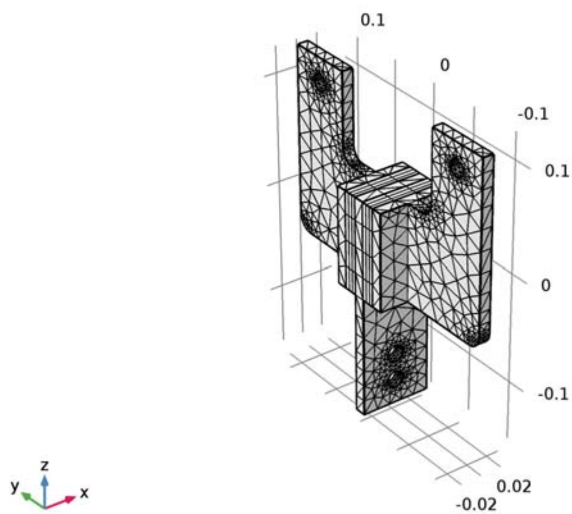
Mesh the rest of the geometry using a free tetrahedral mesh.

Free Tetrahedral 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Tetrahedral**.

2 In the **Settings** window for Free Tetrahedral, click **Build Selected**.

The complete mesh should look similar to that shown in the figure below.



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(0,0.125,0.5) range(1,0.5,5)`.

Solution 1 (sol1)

On the **Study** toolbar, click **Show Default Solver**.

RESULTS

Before computing the solution, set up a displacement plot that will be displayed and updated after every frequency response computation.

3D Plot Group 1

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for 3D Plot Group, type `Displacement, Frequency Domain` in the **Label** text field.

Surface 1

1 Right-click **Displacement, Frequency Domain** 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 (comp1)>Solid Mechanics>Displacement>Displacement field (Material)>w - Displacement field, Z component**.

3 Locate the **Expression** section. From the **Unit** list, choose **mm**.

4 Right-click **Results>Displacement, Frequency Domain>Surface 1** and choose **Deformation**.

STUDY 1

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.

2 In the **Settings** window for Frequency Domain, click to expand the **Results while solving** section.

3 Locate the **Results While Solving** section. Select the **Plot** check box.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node, then click **Solution 1 (sol1)**.

2 In the **Settings** window for Solution, click **Compute**.

RESULTS

Displacement, Frequency Domain

1 In the **Model Builder** window, under **Results** click **Displacement, Frequency Domain**.

2 In the **Settings** window for 3D Plot Group, locate the **Data** section.

3 From the **Parameter value (freq (Hz))** list, choose **3**.

4 Click to expand the **Color legend** section. Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

5 On the **Displacement, Frequency Domain** toolbar, click **Plot**.

6 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

The computed solution should closely resemble that shown in Figure 2.

To plot the storage and loss moduli, follow these steps:

Cut Point 3D 1

1 On the **Results** toolbar, click **Cut Point 3D**.

2 In the **Settings** window for Cut Point 3D, locate the **Point Data** section.

3 In the **X** text field, type `0`.

4 In the **Z** text field, type `0`.

5 In the **Y** text field, type `-0.01`.

1D Plot Group 2

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 **Cut Point 3D 1**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Storage and loss moduli`.

Point Graph 1

1 On the **1D Plot Group 2** toolbar, click **Point Graph**.

2 In the **Settings** window for Point Graph, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `solid.Gstor/6.895`.

1D Plot Group 2

In the **Model Builder** window, under **Results** click **1D Plot Group 2**.

Point Graph 2

1 On the **1D Plot Group 2** toolbar, click **Point Graph**.

2 In the **Settings** window for Point Graph, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `solid.Gloss/6.895`.

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 On the **1D Plot Group 2** toolbar, click **Plot**.

Add a new study to compute by using FFT the solution representation in time domain for the exitation frequency of 3 [Hz].

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 **Empty Study**.

4 Click **Add Study**.

STUDY 2

Frequency to Time FFT

On the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Frequency to Time FFT**.

Step 1: Frequency to Time FFT

1 In the **Settings** window for Frequency to Time FFT, locate the **Study Settings** section.

2 From the **Input study** list, choose **Study 1, Frequency Domain**.

3 In the **Output times** text field, type `range(0,1/(3*30), 1/3)`.

4 Select the **Use window function** check box.

5 From the **Window function** list, choose **Rectangular**.

6 In the **Window start** text field, type `2.9`.

7 In the **Window end** text field, type `3.1`.

8 From the **Scaling** list, choose **Discrete Fourier transform**.

DEFINITIONS

Set up a variable to compute the time domain equivalent of the total force applied to one of the mounting holes.

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**
`Fz1`	`8.5e6[Pa]*cos(2*pi*t*3[1/s])*pi*0.016[m]*0.01[m]`	N	`Total force, Z component`

4 On the **Home** toolbar, click **Compute**.

The default plot will show the stress at the last time moment. Change it to visualize the vertical displacement as shown in Figure 4.

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, Time Domain` 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>Displacement, Time Domain** 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)>w - Displacement field, Z component**.
3 Locate the **Expression** section. From the **Unit** list, choose **mm**.
4 On the **Displacement, Time Domain** toolbar, click **Plot**.

1D Plot Group 4

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

   Plot the total vertical force vs vertical displacement for one of the mounting holes, Figure 5.

2 In the **Settings** window for 1D Plot Group, locate the **Data** section.
3 From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

Point Graph 1

1 On the **1D Plot Group 4** toolbar, click **Point Graph**.

2 In the **Settings** window for Point Graph, locate the **Selection** section.

3 Select the **Active** toggle button.

4 Select Point 23 only.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `Fz1`.

6 From the **Unit** list, choose **kN**.

7 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

8 In the **Expression** text field, type `w`.

9 From the **Unit** list, choose **mm**.

10 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

11 In the **Title** text area, type `Hysteresis loop`.

12 On the **1D Plot Group 4** toolbar, click **Plot**.

Viscoelastic Structural Damper—Transient Analysis

Introduction

The model studies the forced response of a typical viscoelastic damper. Damping elements involving layers of viscoelastic materials are often used for reduction of seismic and wind induced vibrations in buildings and other tall structures. The common feature is that the frequency of the forced vibrations is low.

Model Definition

The geometry of the viscoelastic damper is shown in Figure 1, from Ref. 1. The damper consists of two layers of viscoelastic material confined between mounting elements made of steel.

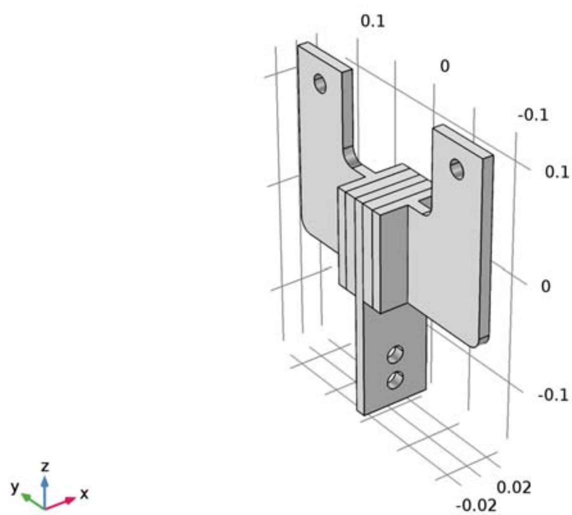


Figure 1: Viscoelastic damping element.

You model the viscoelastic layers by the generalized Maxwell model available in COMSOL Multiphysics. The generalized Maxwell model represents the viscoelastic material as a series of branches, each with a spring-dashpot pair. You can find more details about this material model in the section “About Viscoelastic Materials” in the *Structural Mechanics Module User’s Guide*.

Eighteen viscoelastic branches guarantee accurate representation of the material behavior for different excitation frequencies, when the damper is subjected to forced vibration. The values of the shear moduli and relaxation times for each branch are available in Ref. 1. They are summarized in the following table:

TABLE 1: MODEL DATA FOR THE VISCOELASTIC DAMPER MODEL

PROPERTY	VALUE	DESCRIPTION
K	$4 \cdot 10^4$ MPa	Bulk modulus
G	$5.86 \cdot 10^{-2}$ MPa	Long time shear modulus
ρ	1.06 g/cm^3	Density
G_1	13,3 MPa	Shear modulus branch 1
τ_1	10^{-7} s	Relaxation time branch 1
G_2	286 MPa	Shear modulus branch 2
τ_2	10^{-6} s	Relaxation time branch 2
G_3	$2.91 \cdot 10^2$ MPa	Shear modulus branch 3
τ_3	$3.16 \cdot 10^{-6}$ s	Relaxation time branch 3
G_4	$2.12 \cdot 10^2$ MPa	Shear modulus branch 4
τ_4	10^{-5} s	Relaxation time branch 4
G_5	$1.12 \cdot 10^2$ MPa	Shear modulus branch 5
τ_5	$3.16 \cdot 10^{-5}$ s	Relaxation time branch 5
G_6	61.6 MPa	Shear modulus branch 6
τ_6	10^{-4} s	Relaxation time branch 6
G_7	29.8 MPa	Shear modulus branch 7
τ_7	$3.16 \cdot 10^{-4}$ s	Relaxation time branch 7
G_8	16.1 MPa	Shear modulus branch 8
τ_8	10^{-3} s	Relaxation time branch 8
G_9	7.83 MPa	Shear modulus branch 9
τ_9	$3.16 \cdot 10^{-3}$ s	Relaxation time branch 9
G_{10}	4.15 MPa	Shear modulus branch 10
τ_{10}	10^{-2} s	Relaxation time branch 10
G_{11}	2.03 MPa	Shear modulus branch 11
τ_{11}	$3.16 \cdot 10^{-2}$ s	Relaxation time branch 11
G_{12}	1.11 MPa	Shear modulus branch 12
τ_{12}	0.1 s	Relaxation time branch 12
G_{13}	0.491 MPa	Shear modulus branch 13

TABLE 1: MODEL DATA FOR THE VISCOELASTIC DAMPER MODEL

PROPERTY	VALUE	DESCRIPTION
τ_{13}	0.316 s	Relaxation time branch 13
G_{14}	0.326 MPa	Shear modulus branch 14
τ_{14}	1 s	Relaxation time branch 14
G_{15}	$8.25\cdot10^{-2}$ MPa	Shear modulus branch 15
τ_{15}	3.16 s	Relaxation time branch 15
G_{16}	0.126 MPa	Shear modulus branch 16
τ_{16}	10 s	Relaxation time branch 16
G_{17}	$3.73\cdot10^{-2}$ MPa	Shear modulus branch 17
τ_{17}	100 s	Relaxation time branch 17
G_{18}	$1.18\cdot10^{-2}$ MPa	Shear modulus branch 18
τ_{18}	1000 s	Relaxation time branch 18

One of the mounting elements is fixed; the other two are loaded with periodic forces with a frequency of 3 Hz.

Results and Discussion

Figure 2 shows the results of the transient computations after two seconds of forced vibrations.

The typical viscoelastic hysteresis loops for a point within one of the mounting elements are shown in Figure 3.

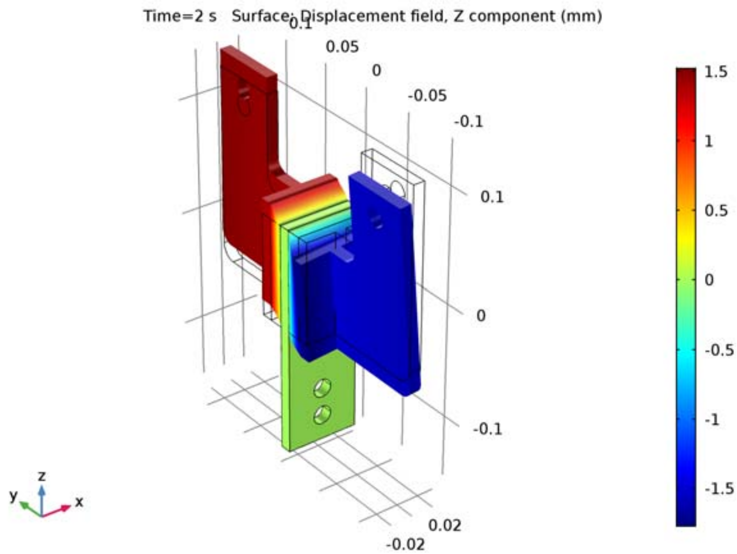


Figure 2: Displacement field.

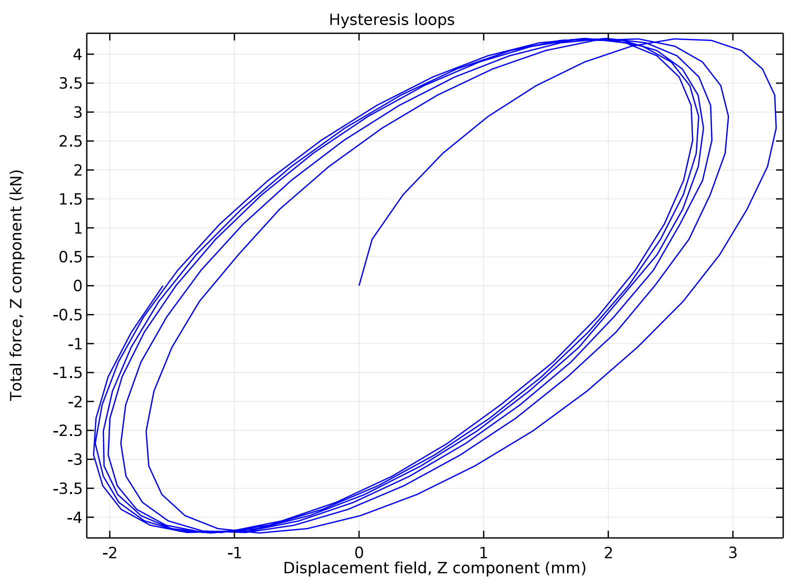


Figure 3: Displacement vs. applied force.

References

1. S.W. Park "Analytical Modeling of Viscoelastic Dampers for Structural and Vibration Control," *Int. J. Solids and Structures*, vol. 38, pp. 694–701, 2001.

2. K.L. Shen and T.T. Soong, "Modeling of Viscoelastic Dampers for Structural Applications," *J. Eng. Mech.*, vol. 121, pp. 694–701, 1995.

Application Library path: `Structural_Mechanics_Module/Dynamics_and_Vibration/viscoelastic_damper_transient`

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>Time Dependent**.

6 Click **Done**.

GEOMETRY 1

You import the predefined geometry from a file.

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 application's Application Libraries folder and double-click the file `viscoelastic_damper.mphbin`.

5 Click **Import**.

6 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

The imported geometry should look similar to that shown in Figure 1.

SOLID MECHANICS (SOLID)

Linear Elastic Material 2

1 On the **Physics** toolbar, click **Domains** and choose **Linear Elastic Material**.

2 In the **Settings** window for Linear Elastic Material, locate the **Linear Elastic Material** section.

3 From the **Specify** list, choose **Bulk modulus and shear modulus**.

4 Select the **Nearly incompressible material** check box.

5 Select Domains 2 and 4 only.

Viscoelasticity 1

1 On the **Physics** toolbar, click **Attributes** and choose **Viscoelasticity**.

Since there are 18 branches in this material model, the data has been collected in a text file which you can load.

2 In the **Settings** window for Viscoelasticity, locate the **Viscoelasticity Model** section.

3 Click **Load from File**.

4 Browse to the application's Application Libraries folder and double-click the file `viscoelastic_damper_viscoelastic_data.txt`.

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.

MATERIALS

Steel AISI 4340 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Steel AISI 4340 (mat1)**.

2 Select Domains 1, 3, and 5 only.

Material 2 (mat2)

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 `Viscoelastic` in the **Label** text field.

3 Select Domains 2 and 4 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Bulk modulus	K	`4e8`	N/m²	Bulk modulus and shear modulus
Shear modulus	G	`5.86e4`	N/m²	Bulk modulus and shear modulus
Density	rho	`1060`	kg/m³	Basic

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Select Boundaries 24–27 only.

Prescribed Displacement 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

2 Select Boundaries 40 and 41 only.

3 In the **Settings** window for Prescribed Displacement, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in x direction** check box.

5 Select the **Prescribed in y direction** check box.

Boundary Load 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundaries 40 and 41 only.

3 In the **Settings** window for Boundary Load, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

`0`	x
`0`	y
`8.5e6[Pa]*sin(pi/2+2*pi*t*3[1/s])`	z

Prescribed Displacement 2

1. On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.
2. Select Boundaries 32 and 33 only.
3. In the **Settings** window for Prescribed Displacement, locate the **Prescribed Displacement** section.
4. Select the **Prescribed in y direction** check box.

Boundary Load 2

1. On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
2. Select Boundaries 32 and 33 only.
3. In the **Settings** window for Boundary Load, locate the **Force** section.
4. Specify the $\mathbf{F}_A$ vector as

`5e5[Pa]*sin(2*pi*t*3[1/s])`	x
`0`	y
`8.5e6[Pa]*sin(2*pi*t*3[1/s])`	z

DEFINITIONS

Set up an integration operator to compute the total force applied to one of the mounting holes. You configure the integration to be performed on the Material frame, because this is the frame used within the Solid Mechanics interface.

Integration 1 (intop1)

1. On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.
2. In the **Settings** window for Integration, locate the **Advanced** section.
3. From the **Frame** list, choose **Material (X, Y, Z)**.
4. Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
5. Select Boundaries 32 and 33 only.

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
`Fz1`	`intop1(solid.FperAreaz)`	N	`Total force, Z component`

MESH 1

Mesh the side surfaces of the viscoelastic layers and then sweep the resulting mesh into the layers.

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 Boundaries 6 and 20 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 2 and 4 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 `3`.

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**.

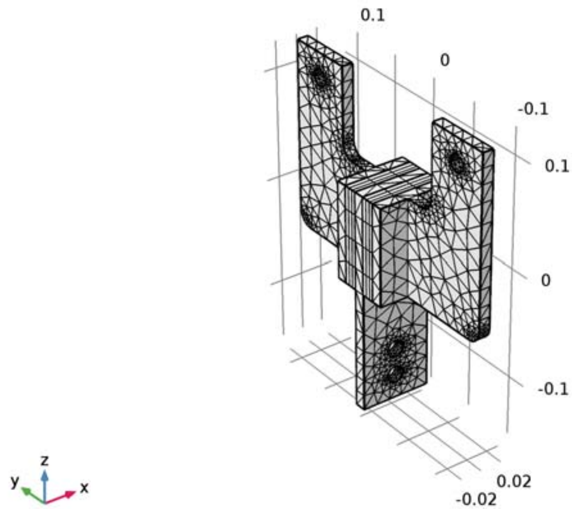
   Mesh the rest of the geometry using a free tetrahedral mesh.

Free Tetrahedral 1

1. In the **Model Builder** window, right-click **Mesh 1** and choose **Free Tetrahedral**.

2 In the **Settings** window for Free Tetrahedral, click **Build Selected**.

The complete mesh should look similar to that shown in the figure below.



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(0,0.01,2)`.

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 **Dependent Variables 1**.

4 In the **Settings** window for Dependent Variables, locate the **Scaling** section.

5 From the **Method** list, choose **Manual**.

6 In the **Scale** text field, type `0.1`.

This scale is applied to the visoelastic strain variables. The scales for the displacement components are choosen according to the size of the geometry. Use the following scale for the pressure help variable.

7 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** node, then click **Auxiliary pressure (comp1.solid.pw)**.

8 In the **Settings** window for Field, locate the **Scaling** section.

9 In the **Scale** text field, type `1e6`.

10 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** click **Time-Dependent Solver 1**.

11 In the **Settings** window for Time-Dependent Solver, click to expand the **Output** section.

12 Clear the **Store time derivatives** check box.

The values of derivatives are not needed for the analysis. By not storing them, you reduce significantly the size of the output data.

RESULTS

Before computing the solution, set up a displacement plot that will be displayed and updated after every time step of the transient analysis.

3D Plot Group 1

On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

Surface 1

1 In the **Model Builder** window, right-click **3D Plot Group 1** 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 (comp1)>Solid Mechanics> Displacement>Displacement field (Material)>w - Displacement field, Z component**.

3 Locate the **Expression** section. From the **Unit** list, choose **mm**.

4 Right-click **Results>3D Plot Group 1>Surface 1** and choose **Deformation**.

STUDY 1

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, click to expand the **Results while solving** section.

3 Locate the **Results While Solving** section. Select the **Plot** check box.

Solution 1 (sol1)

1 In the **Model Builder** window, under **Study 1>Solver Configurations** click **Solution 1 (sol1)**.

2 In the **Settings** window for Solution, click **Compute**.

RESULTS

3D Plot Group 1

1 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

The computed solution should closely resemble that shown in Figure 2.

To plot the displacement vs. applied force, follow these steps:

1D Plot Group 2

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

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 `Hysteresis loops` .

Point Graph 1

1 On the **1D Plot Group 2** toolbar, click **Point Graph**.

2 Select Point 23 only.

3 In the **Settings** window for Point Graph, locate the **x-Axis Data** section.

4 From the **Parameter** list, choose **Expression**.

5 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Displacement>Displacement field (Material)>w - Displacement field, Z component**.

6 Locate the **x-Axis Data** section. From the **Unit** list, choose **mm**.

7 Locate the **y-Axis Data** section. In the **Expression** text field, type `Fz1`.

8 From the **Unit** list, choose **kN**.

9 On the **1D Plot Group 2** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.2a

Stress Relaxation of a Viscoelastic Tube

Introduction

This example studies the temperature effects on the viscoelastic stress relaxation in a generalized Maxwell material with four branches.

Assume that the viscous part of the deformation is incompressible, so that the volume change is purely elastic. The relaxation shear modulus function is approximated by a Prony series as

$$\Gamma(t) = G + \sum_{m=1}^{N} G_m \exp\left(-\frac{t}{\tau_m}\right) \tag{1}$$

where G_m represents the stiffness of the spring in the m^{th} Maxwell branch, and τ_m is the relaxation time of the spring-dashpot pair in the same branch.

The instantaneous shear modulus is defined as

$$G_0 = G + \sum_{m=1}^{N} G_m$$

The alternative form of Equation 1 is

$$\Gamma(t) = G_0\left[\mu_0 + \sum_{m=1}^{N} \mu_m \exp\left(-\frac{t}{\tau_m}\right)\right]$$

where the constants $\mu_m = G_m/G_0$ are such that

$$\sum_{m=0}^{N} \mu_m = 1$$

For many materials the viscoelastic properties have a strong dependence on the temperature. A common assumption is that the material is *thermorheologically simple* (TRS). In a material of this class, a change in the temperature can be transformed directly into a change in the time scale. Thus, the relaxation time for a TRS material is modified to $a_T(T)\tau_m$, where $a_T(T)$ is a shift function. One of the most commonly used shift functions is defined by the WLF (Williams-Landel-Ferry) equation:

$$\log(a_T) = \frac{-C_1(T-T_0)}{C_2 + (T-T_0)}$$

where a base-10 logarithm is assumed. Usually T_0 is the *glass transition temperature* of the material. Note that $a_T(T_0) = 1$. If the temperature drops below $T_0 - C_2$, the WLF equation is no longer valid. The constants C_1 and C_2 are material dependent.

Model Definition

A long thick-walled cylinder has an inner radius of 5 mm and an outer radius of 10 mm. The inner surface is subjected to a prescribed radial displacement of 0.001 mm. A four-branch generalized Maxwell model represents the viscoelastic material.

In this example, you study the decay of the stresses during a period of two hours under the influence of the temperature field. You model a quarter of the cylinder and use the Solid Mechanics interface to compute the displacements in the cylinder cross section. The geometry is shown in Figure 1.

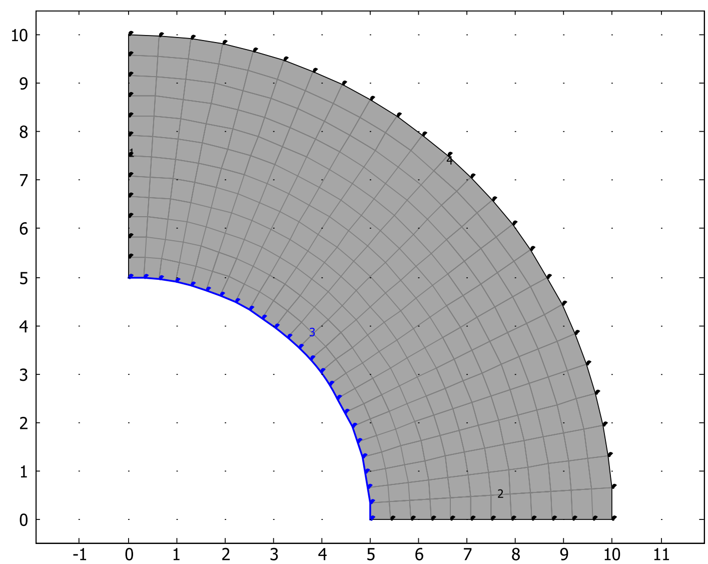


Figure 1: Geometry and mesh.

Study the following two cases:

- Apply a stationary temperature field, causing the problem to lose its axisymmetry.
- Make the temperature field transient.

MATERIAL PARAMETERS

- Elastic data: instantaneous shear modulus $G_0 = 2.746 \cdot 10^4$ MPa and long-term shear modulus of $G = \mu_0 G_0$, bulk modulus $K = 3.988 \cdot 10^4$ MPa.
- Viscoelastic data: four-branch Generalized Maxwell material with the hear modulus per branch defined from $G_m = \mu_m G_0$
  - $\mu_1 = 0.04$, $\tau_1 = 30$ s
  - $\mu_2 = 0.08$, $\tau_2 = 300$ s
  - $\mu_3 = 0.09$, $\tau_3 = 3000$ s
  - $\mu_4 = 0.25$, $\tau_4 = 12000$ s
- Thermal properties: A WLF shift function is used with $C_1 = 17.44$ and $C_2 = 51.6$. These values are reasonable approximations for many polymers.
- The reference temperature is 500 K.
- Heat transfer properties: $\rho = 7850$ kg/m^3 = $7.85 \cdot 10^{-9}$ t/mm^3, $C_p = 2100$ J/(kg·K) = $2100 \cdot 10^6$ N·mm/(t·K), $k = 6 \cdot 10^{-2}$ W/(m·K) = $6 \cdot 10^{-2}$ N/(s·K).

CONSTRAINTS

- The circumferential displacements are constrained on the radial edges.
- The inner edge of the hole is constrained to a radial displacement of 0.001 mm.

HEAT TRANSFER BOUNDARY CONDITIONS

- Stationary analysis: The inner and outer circular edges both have a temperature distribution varying linearly with the y-coordinate from 500 K at the $y = 0$ symmetry section to 506 K at the $x = 0$ symmetry section.
- Transient analysis: The temperature distribution obtained from the stationary analysis is used as initial condition, while all edges are insulated.

Results and Discussion

In the first case, the temperature field is stationary and is shown in Figure 2.

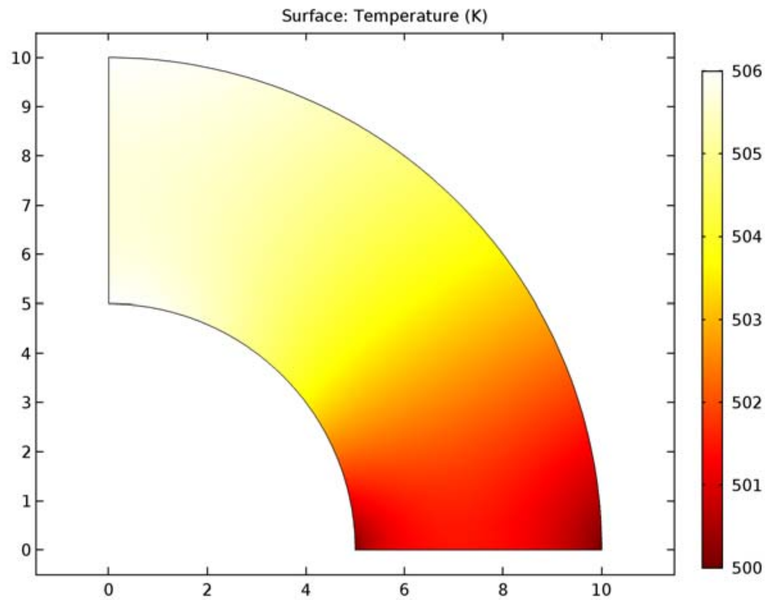


Figure 2: Stationary temperature distribution.

The stress relaxation is faster where the temperature is higher. In Figure 3, the plots of radial stresses at radius 7.5 mm display this effect. The solid line is taken at $y = 0$ (cold) and the dashed line at $x = 0$ (warm).

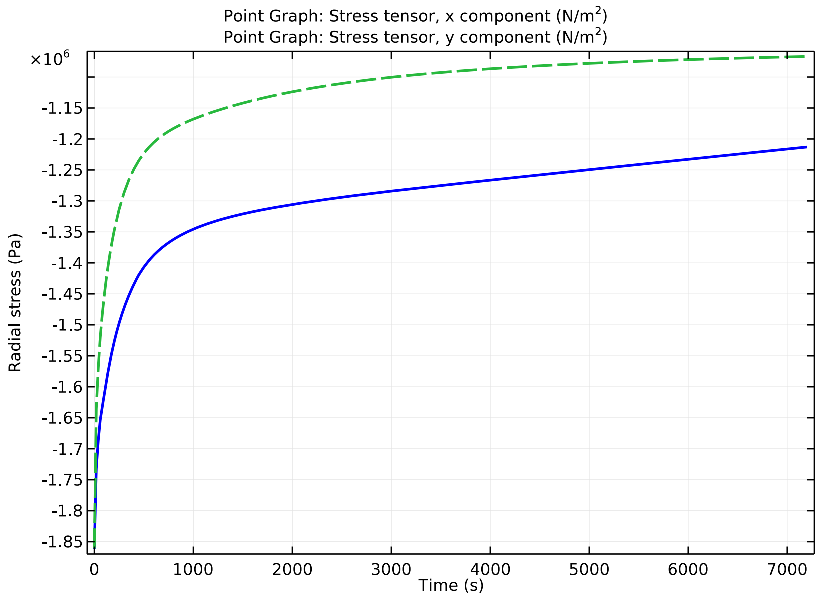


Figure 3: Stress relaxation under a stationary temperature distribution.

In the second case, the temperature initially has the same distribution as shown in Figure 2, but it is allowed to settle in time to a final homogeneous value. Again, compare the radial stresses at radius 7.5 mm. The initial behavior is similar to the previous case, but as the difference in material properties decreases, the curves approach each other; see Figure 4. The strain rate in the initially warm point decreases, while it increases in the initially cold point.

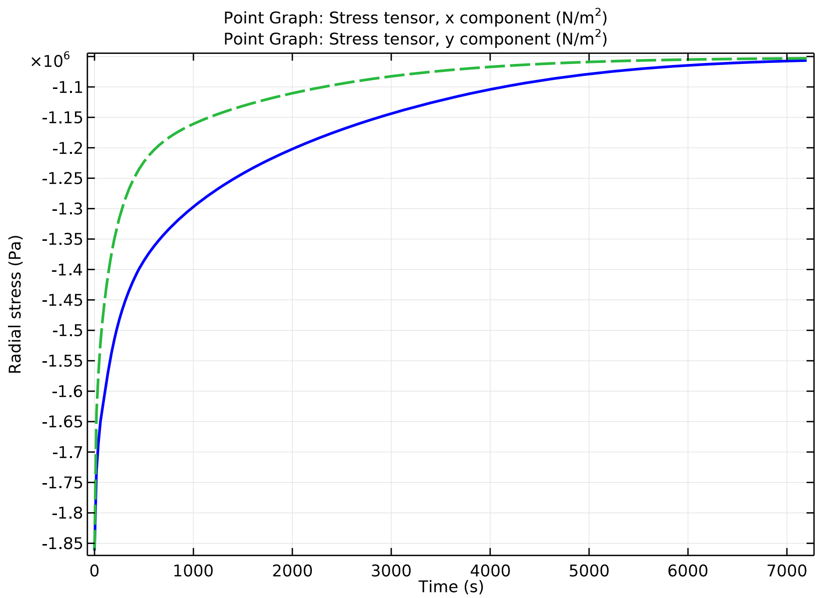


Figure 4: Stress relaxation under temperature settling conditions.

Notes About the COMSOL Implementation

When you solve a problem where a load is applied instantaneously at the beginning of a transient analysis, you can choose between two approaches to compute the initial stresses: either apply the load over a short period of time at the beginning of the time-dependent study. An alternative approach is to set the viscoelastic material to use an instantaneous static stiffness, and to add a separate stationary step before the time-dependent step. In the present example, you use the later method.

The results computed during the stationary step are stored and used as the initial value for the consequent transient analysis. COMSOL Multiphysics handles this automatically if you use a single study with one solver sequence. In this example, however, you study two different relaxation histories with different thermal boundary conditions. In the second case, the thermal boundary conditions differ between the computation of the initial state and the transient analysis. For this reason you use two separate studies for the stationary and the time-dependent studies.

Application Library path: `Structural_Mechanics_Module/Viscoelasticity/viscoelastic_tube`

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 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.

5 Click **Add**.

6 Click **Done**.

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**.

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 `10`.

Circle 2 (c2)

1 Right-click **Circle 1 (c1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

3 In the **Settings** window for Circle, locate the **Size and Shape** section.

4 In the **Radius** text field, type `5`.

Difference 1 (dif1)

1 Right-click **Circle 2 (c2)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

3 Select the object **c1** only.

4 In the **Settings** window for Difference, locate the **Difference** section.

5 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

6 Select the object **c2** only.

Rectangle 1 (r1)

1 Right-click **Difference 1 (dif1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

3 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

4 In the **Width** text field, type 11.

5 In the **Height** text field, type 11.

Intersection 1 (int1)

1 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Intersection**.

3 Click in the **Graphics** window and then press Ctrl+A to select both objects.

4 Right-click **Intersection 1 (int1)** and choose **Build Selected**.

5 Click the **Zoom Extents** button on the **Graphics** toolbar.

Form Union (fin)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

The equations only need to include first-order time derivatives since the inertia effects can be neglected for this class of problems.

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 **Structural Transient Behavior** section.

3 From the list, choose **Quasi-static**.

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

2 In the **Settings** window for Linear Elastic Material, locate the **Linear Elastic Material** section.

3 From the **Specify** list, choose **Bulk modulus and shear modulus**.

Viscoelasticity 1

1 On the **Physics** toolbar, click **Attributes** and choose **Viscoelasticity**.

2 In the **Settings** window for Viscoelasticity, locate the **Viscoelasticity Model** section.

3 In the table, enter the following settings:

Branch	Shear modulus (Pa)	Relaxation time (s)
1	0.04*G_inst	30

4 Click **Add**.

5 In the table, enter the following settings:

Branch	Shear modulus (Pa)	Relaxation time (s)
1	0.08*G_inst	300

6 Click **Add**.

7 In the table, enter the following settings:

Branch	Shear modulus (Pa)	Relaxation time (s)
1	0.09*G_inst	3000

8 Click **Add**.

9 In the table, enter the following settings:

Branch	Shear modulus (Pa)	Relaxation time (s)
1	0.25*G_inst	12000

10 Locate the **Model Inputs** section. From the T list, choose **Temperature (ht)**.

11 Locate the **Thermal Effects** section. From the **Shift function** list, choose **Williams-Landel-Ferry**.

12 In the T_{WLF} text field, type `500[K]`.

13 Locate the **Viscoelasticity Model** section. From the **Static stiffness** list, choose **Instantaneous**.

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 Select Boundaries 1 and 2 only.

Prescribed Displacement 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

2 Select Boundary 3 only.

3 In the **Settings** window for Prescribed Displacement, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.

5 Locate the **Prescribed Displacement** section. Select the **Prescribed in n direction** check box.

6 In the u_{0n} text field, type `-1e-6`.

HEAT TRANSFER IN SOLIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundaries 3 and 4 only.

3 In the **Settings** window for Temperature, locate the **Temperature** section.

4 In the T_0 text field, type `(500+6*y/sqrt(x^2+y^2))[K]`.

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

1 Select Domain 1 only.

2 In the **Settings** window for Material, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Bulk modulus	K	`3.988e4[MPa]`	N/m²	Bulk modulus and shear modulus
Shear modulus	G	`0.54*G_inst`	N/m²	Bulk modulus and shear modulus
Density	rho	`7.85E-9[t/mm^3]`	kg/m³	Basic
Thermal conductivity	k	`0.06[N/(s*K)]`	W/(m·K)	Basic
Heat capacity at constant pressure	Cp	`2.1E9[N*mm/(t*K)]`	J/(kg·K)	Basic

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**
`G_inst`	`2.746e4[MPa]`	Pa	`Instantaneous shear modulus`

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.

Mapped 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Mapped 1** and choose **Distribution**.

Distribution 1

1 Select Boundary 1 only.

2 In the **Settings** window for Distribution, locate the **Distribution** section.

3 In the **Number of elements** text field, type `12`.

Mapped 1

Right-click **Mapped 1** and choose **Distribution**.

Distribution 2

1 Select Boundary 3 only.

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 Click the **Zoom Extents** button on the **Graphics** toolbar.

ROOT

Next, add a stationary study to model the instantaneous preloading step.

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.

STUDY 1

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for Study, type `Study: Stationary` in the **Label** text field.

3 On the **Home** toolbar, click **Compute**.

RESULTS

Stress (solid)

Display the stationary temperature.

Temperature (ht)

1 In the **Model Builder** window, under **Results** click **Temperature (ht)**.

2 On the **Temperature (ht)** toolbar, click **Plot**.

3 Click the **Zoom Extents** button on the **Graphics** toolbar.

Proceed to add a transient study for analyzing the stress relaxation process.

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.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for Study, type `Study: Transient constant temperature` in the **Label** text field.

STUDY: TRANSIENT CONSTANT TEMPERATURE

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study: Transient constant temperature** 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(0,20,200) range(250,50,1000) range(1100,100,2000) range(2200,200,7200)`.

4 Select the **Relative tolerance** check box.

5 In the associated text field, type `1e-3`.

Solution 2 (sol2)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Study: Transient constant temperature>Solver Configurations** node.

Prescribe solution of the stationary study as the initial step in the transient analysis.

3 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Dependent Variables 1**.

4 In the **Settings** window for Dependent Variables, locate the **General** section.

5 From the **Defined by study step** list, choose **User defined**.

6 Locate the **Initial Values of Variables Solved For** section. From the **Method** list, choose **Solution**.

7 From the **Solution** list, choose **Solution 1 (sol1)**.

8 Locate the **Scaling** section. From the **Method** list, choose **Initial value based**.

Force solver to evaluate results in between the specified time steps.

9 In the **Model Builder** window, under **Study: Transient constant temperature>Solver Configurations>Solution 2 (sol2)** click **Time-Dependent Solver 1**.

10 In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.

11 Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Intermediate**.

12 On the **Study** toolbar, click **Compute**.

RESULTS

Temperature (ht) 1

Click the **Zoom Extents** button on the **Graphics** toolbar.

Display stress relaxation.

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 `7.5`.

4 In the **Y** text field, type `0`.

5 Locate the **Data** section. From the **Data set** list, choose **Study: Transient constant temperature/Solution 2 (sol2)**.

Cut Point 2D 2

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`.

4 In the **Y** text field, type `7.5`.

5 Locate the **Data** section. From the **Data set** list, choose **Study: Transient constant temperature/Solution 2 (sol2)**.

1D Plot Group 7

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 **Study: Transient constant temperature/Solution 2 (sol2)**.

Point Graph 1

1 On the **1D Plot Group 7** toolbar, click **Point Graph**.

2 In the **Settings** window for Point Graph, locate the **Data** section.

3 From the **Data set** list, choose **Cut Point 2D 1**.

4 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Stress>Stress tensor (Spatial)>solid.sx - Stress tensor, x component**.

5 On the **1D Plot Group 7** toolbar, click **Plot**.

1D Plot Group 7

In the **Model Builder** window, under **Results** click **1D Plot Group 7**.

Point Graph 2

1 On the **1D Plot Group 7** toolbar, click **Point Graph**.

2 In the **Settings** window for Point Graph, locate the **Data** section.

3 From the **Data set** list, choose **Cut Point 2D 2**.

4 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Stress>Stress tensor (Spatial)>solid.sy - Stress tensor, y component**.

5 On the **1D Plot Group 7** toolbar, click **Plot**.

6 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type `2`.

7 Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

Point Graph 1

1 In the **Model Builder** window, under **Results>1D Plot Group 7** click **Point Graph 1**.

2 In the **Settings** window for Point Graph, locate the **Coloring and Style** section.

3 In the **Width** text field, type `2`.

1D Plot Group 7

1 In the **Model Builder** window, under **Results** click **1D Plot Group 7**.

2 In the **Settings** window for 1D Plot Group, type `Stress relaxation at constant temperature` 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 `Time (s)`.

5 Select the **y-axis label** check box.

6 In the associated text field, type `Radial stress (Pa)`.

Now, simulate stress relaxation at constant temperature.

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 3

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 3** 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(0,20,200) range(250,50,1000) range(1100,100,2000) range(2200,200,7200)`.

4 Select the **Relative tolerance** check box.

5 In the associated text field, type `1e-3`.

6 Locate the **Physics and Variables Selection** section. Select the **Modify physics tree and variables for study step** check box.

7 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Heat Transfer in Solids (ht)**.

8 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Heat Transfer in Solids (ht)>Temperature 1**.

9 Click **Disable**.

10 In the **Model Builder** window, click **Study 3**.

11 In the **Settings** window for Study, type `Study: Transient variable temperature` in the **Label** text field.

Solution 3 (sol3)

On the **Study** toolbar, click **Show Default Solver**.

STUDY: TRANSIENT VARIABLE TEMPERATURE

Solution 3 (sol3)

1 In the **Model Builder** window, expand the **Study: Transient variable temperature>Solver Configurations** node.

Prescribe solution of the stationary study as the initial step in the transient analysis.

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node, then click **Dependent Variables 1**.

3 In the **Settings** window for Dependent Variables, locate the **General** section.

4 From the **Defined by study step** list, choose **User defined**.

5 Locate the **Initial Values of Variables Solved For** section. From the **Method** list, choose **Solution**.

6 From the **Solution** list, choose **Solution 1 (sol1)**.

7 Locate the **Scaling** section. From the **Method** list, choose **Initial value based**.

Force solver to evaluate results in between the specified time steps.

8 In the **Model Builder** window, under **Study: Transient variable temperature>Solver Configurations>Solution 3 (sol3)** click **Time-Dependent Solver 1**.

9 In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.

10 Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Intermediate**.

11 On the **Study** toolbar, click **Compute**.

Display the results of stress relaxation.

RESULTS

Cut Point 2D 1

In the **Model Builder** window, expand the **Results>Data Sets** node.

Cut Point 2D 3

1 Right-click **Cut Point 2D 1** and choose **Duplicate**.

2 In the **Settings** window for Cut Point 2D, locate the **Data** section.

3 From the **Data set** list, choose **Study: Transient variable temperature/Solution 3 (sol3)**.

Cut Point 2D 2

In the **Model Builder** window, under **Results>Data Sets** right-click **Cut Point 2D 2** and choose **Duplicate**.

Cut Point 2D 4

1 In the **Settings** window for Cut Point 2D, locate the **Data** section.

2 From the **Data set** list, choose **Study: Transient variable temperature/Solution 3 (sol3)**.

Stress relaxation at constant temperature

In the **Model Builder** window, under **Results** right-click **Stress relaxation at constant temperature** and choose **Duplicate**.

Stress relaxation at constant temperature 1

In the **Settings** window for 1D Plot Group, type `Stress relaxation at variable temperature` in the **Label** text field.

Point Graph 1

1 In the **Model Builder** window, expand the **Results>Stress relaxation at variable temperature** node, then click **Point Graph 1**.

2 In the **Settings** window for Point Graph, locate the **Data** section.

3 From the **Data set** list, choose **Cut Point 2D 3**.

Point Graph 2

1 In the **Model Builder** window, under **Results>Stress relaxation at variable temperature** click **Point Graph 2**.

2 In the **Settings** window for Point Graph, locate the **Data** section.

3 From the **Data set** list, choose **Cut Point 2D 4**.

4 On the **Stress relaxation at variable temperature** toolbar, click **Plot**.