

# INTRODUCTION TO Structural Mechanics Module

# Introduction to the Structural Mechanics Module

© 1998–2019 COMSOL

Protected by patents listed on [www.comsol.com/patents](http://www.comsol.com/patents), and U.S. Patents 7,519,518; 7,596,474; 7,623,991; 8,457,932; 8,954,302; 9,098,106; 9,146,652; 9,323,503; 9,372,673; 9,454,625; and 10,019,544. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement ([www.comsol.com/comsol-license-agreement](http://www.comsol.com/comsol-license-agreement)) and may be used or copied only under the terms of the license agreement.

COMSOL, the COMSOL logo, COMSOL Multiphysics, COMSOL Desktop, COMSOL Compiler, COMSOL Server, and LiveLink are either registered trademarks or trademarks of COMSOL AB. All other trademarks are the property of their respective owners, and COMSOL AB and its subsidiaries and products are not affiliated with, endorsed by, sponsored by, or supported by those trademark owners. For a list of such trademark owners, see [www.comsol.com/trademarks](http://www.comsol.com/trademarks).

Version: COMSOL 5.5

## Contact Information

Visit the Contact COMSOL page at [www.comsol.com/contact](http://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](http://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](http://www.comsol.com/support/case). Other useful links include:

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

Part number: CM021105

# Contents

---

Introduction .....	5
Structural Mechanics Simulations .....	5
The Structural Mechanics Module Physics Interfaces. ....	9
Physics Interface Guide by Space Dimension and Study Type ...	14
The Fundamentals: A Static Linear Analysis .....	21
Parametric Study .....	48
Modeling Techniques for Structural Mechanics. ....	58
Including Initial Strain .....	59
Modeling Thermal Expansion.....	66
Examples in the Application Library.....	72



# Introduction

---

The Structural Mechanics Module is tailor-made for modeling and simulating applications and designs in the fields of structural and solid mechanics. Engineers and scientists use it to design new structures and devices and to study the performance of existing structures.

With this module you can perform static and dynamic analyses in for solids (2D, 2D axisymmetry, 3D), shells (2D axisymmetry, 3D), plates (2D), membranes (2D axisymmetry, 3D), trusses (2D, 3D), beams (2D, 3D), and pipes (2D, 3D). Other capabilities are for thermal stress, geometric nonlinearities (large deformations), and structural contact.

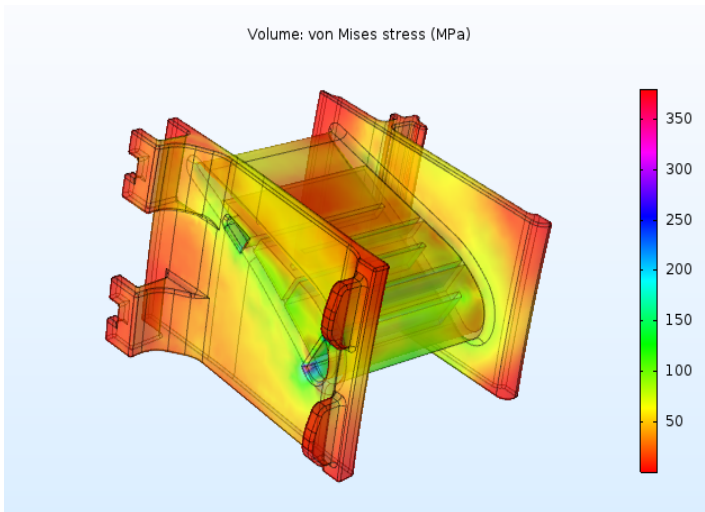


Figure 1: Von Mises stresses caused by thermal expansion in a turbine stator model. From the Heat Transfer Module application library: Thermal Stress Analysis of a Turbine Stator Blade (*turbine\_stator*). This model uses both the Structural Mechanics and Heat Transfer Modules.

The Structural Mechanics physics interfaces are the backbone of the module. These have predefined formulations for the capabilities described above. This guide gives an overview of these physics interfaces as well as examples of the modeling procedures used in them.

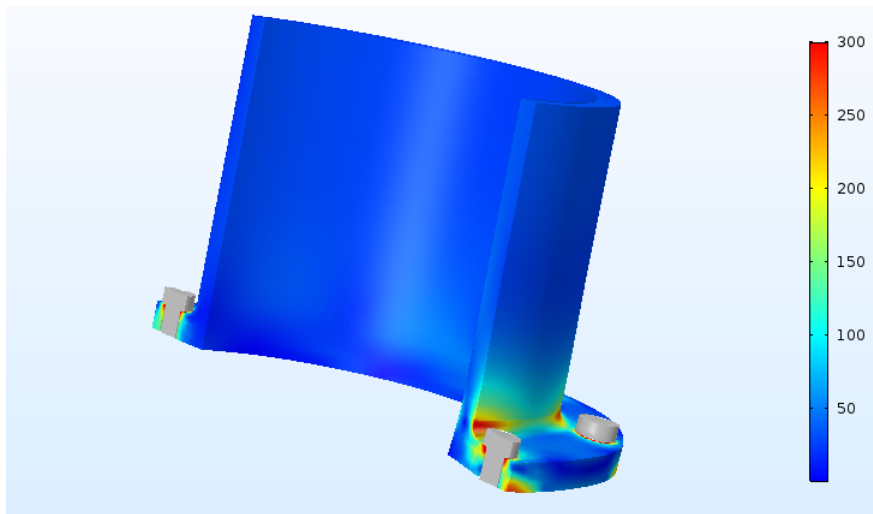
## Structural Mechanics Simulations

Simulations in structural mechanics are used in a wide range of applications — from the microscale of MEMS components to the geomechanics scale of civil

engineering. These types of simulations are also frequently used to study the behavior of existing structures, from microscopic biostructures to glaciers.

Structural mechanics was the first engineering field to use the concept of finite elements as a standard tool. Over time, these verifiable and validated formulations have been developed and are applicable to a wide range of materials. Simulations can often replace experimental measurements. For example, finite element simulations are used extensively in safety-critical applications within the aerospace and nuclear industries.

A traditional use of structural analysis tools is depicted below. The device being studied is a pipe with a bolted flange, and the purpose of the study is two-fold: to estimate the stress in the pipe and to evaluate the performance of the bolted joint. [Figure 2](#) shows the deformation (exaggerated) and the von Mises stresses in the pipe.



*Figure 2: The deformation (exaggerated) and the von Mises stresses in the pipe. From the Structural Mechanics Module application library: Prestressed Bolts in a Tube Connection (tube\_connection).*

A less traditional application of mechanical design for a MEMS device is shown in [Figure 3](#). The microactuator is subjected to controlled thermal expansion by the application of a current through parts of the structure, which induces Joule heating. The thermal expansion then causes the actuator to deflect and the simulation predicts the deflection as a function of the operating conditions. The

simulation also reveals the limitations of the design, because the device will not work properly if the legs of the actuator make contact along the free faces.

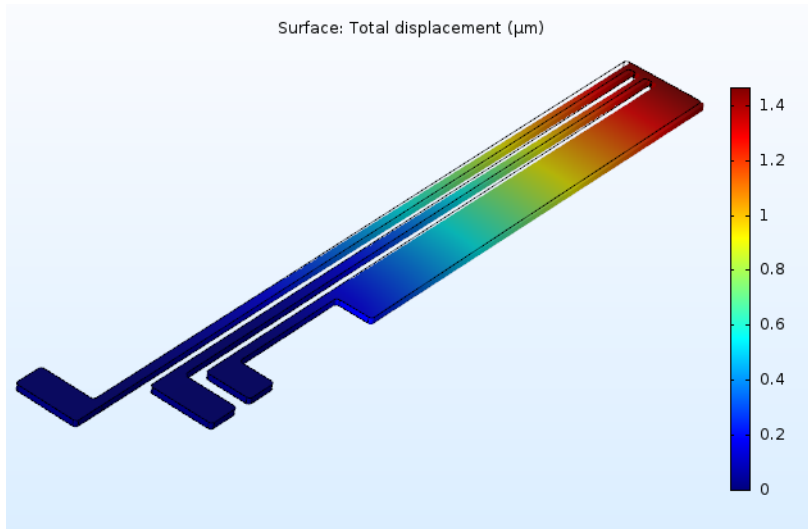


Figure 3: The total displacement in a MEMS device. From the MEMS Module application library: Thermal Actuator (*thermal\_actuator\_tem*).

Structural analysis is also important outside of traditional structural engineering, for example in the field of bioscience. Figure 4 shows the results of a simulation of part of the vascular system in a young child. The purpose of the simulation is to study what happens when surgery is performed on a child with a malformed aorta. The aorta and its ramified blood vessels are embedded in biological tissue. Pressure from the moving fluid is applied as a face load in the structural analysis.

The Structural Mechanics Module includes several types of couplings for fluid-structure interaction.

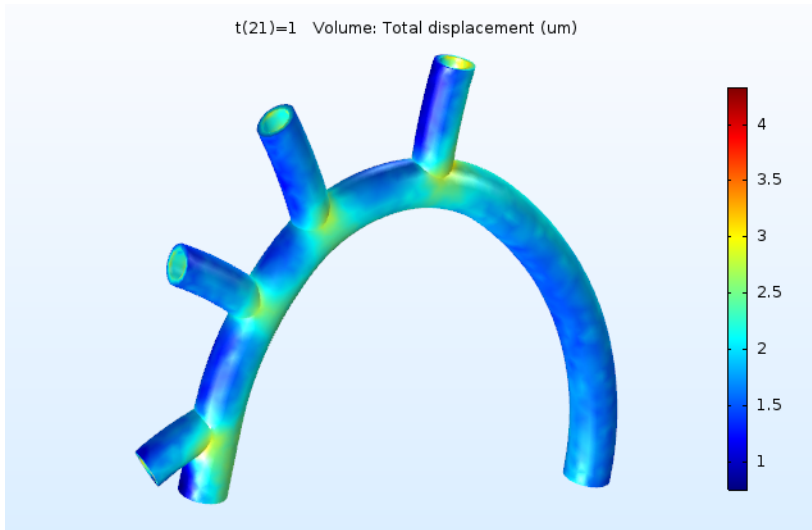


Figure 4: Displacements in the blood vessel. From the Structural Mechanics Module application library: Fluid-Structure Interaction in a Network of Blood Vessels (blood\_vessel).



# The Structural Mechanics Module Physics Interfaces

The figure below shows the physics interfaces that are in the COMSOL Model Wizard and included with the Structural Mechanics Module. In addition to the structural mechanics capabilities, the module also has substantial multiphysics capabilities, including AC/DC, acoustics, CFD, heat transfer, Joule heating, piezoelectricity, and magnetostriction.

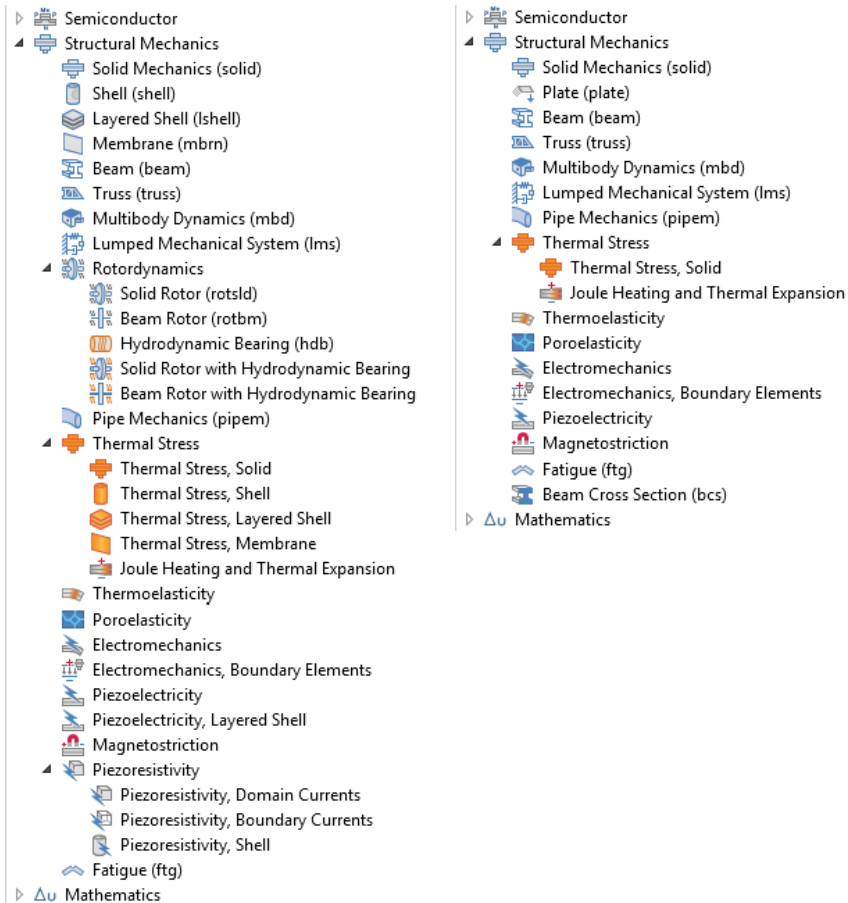























Figure 5: The 3D and 2D component physics interfaces available with the Structural Mechanics Module. The Layered Shell; Multibody Dynamics; Lumped Mechanical System; Rotordynamics; Thermal Stress, Shell; Thermal Stress, Layered Shell; Thermal Stress, Membrane; Poroelasticity; Thermoelasticity; Magnetostriction; Piezoresistivity; Electromechanics; Piezoelectricity, Layered Shell; and Fatigue interfaces require additional licenses..

A short description of the Structural Mechanics interfaces follows.

## SOLID MECHANICS

The Solid Mechanics interface (  ) defines the quantities and features for stress analysis and general linear and nonlinear solid mechanics. In 2D, plane stress, plane strain, and generalized plane strain formulations are available. The Linear Elastic Material node is the default material model. The elastic material model can be expanded with features for thermal expansion, hygroscopic swelling, viscoelasticity, damping, and initial stress and strain. The description of elastic materials in the module includes isotropic, orthotropic, and fully anisotropic materials. The material can be activated or deactivated based on for example time or temperature. A number of preset study types are available for solid mechanics (shown below). Also see [Physics Interface Guide by Space Dimension and Study Type](#).

- ▲  General Studies
  -  Eigenfrequency
  -  Frequency Domain
  -  Stationary
  -  Time Dependent
- ▲  Preset Studies for Selected Physics Interfaces
  -  Bolt Pretension
  -  Eigenfrequency, Prestressed
  -  Frequency Domain, Prestressed
  -  Frequency Domain, Modal
  -  Linear Buckling
  -  Random Vibration (PSD)
  -  Response Spectrum
  -  Time Dependent, Modal
- ▲  More Studies
  -  Frequency Domain, Prestressed, Modal
  -  Frequency Domain, AWE Reduced-Order Model
  -  Frequency Domain, Modal Reduced-Order Model
  -  Time Dependent, Prestressed, Modal
  -  Time Dependent, Modal Reduced-Order Model

## *Geomechanics, Nonlinear Structural Materials, Composite Materials, Rotordynamics, and Fatigue*

There are five additional modules available to enhance the Structural Mechanics Module: the Geomechanics Module, the Nonlinear Structural Materials Module, the Composite Materials Module, the Rotordynamics Module, and the Fatigue Module

- With the addition of the Geomechanics Module, you can add soil plasticity, nonlinear elasticity, creep, concrete, rocks, brittle damage, and specialized soil models like ‘hardening soil’ and ‘modified Cam clay’ to the physics interface.


- With the Nonlinear Structural Materials Module, hyperelasticity, nonlinear elasticity, plasticity, porous plasticity, creep, viscoplasticity, brittle damage, and shape memory alloys can be modeled.
- With the Composite Material Module, you get the possibility to model layered shells, either using the Layered Shell interface, or through a special material model in the Shell and Membrane interfaces.
- With the Rotordynamics module you can study the dynamics of rotors, including advanced bearing models and gears. The rotor can be modeled either as beams, or using 3D solids. The hydrodynamic bearings can also be used in solid mechanics and multibody dynamics models.
- Using the Fatigue Module, you can augment your stress analysis with low or high cycle fatigue evaluation.


The Multibody Dynamics Module is a related product, which does not require the Structural Mechanics Module. Here, you can study rigid and flexible bodies connected by different types of mechanical joints, gears, cam-follower mechanisms, springs, and dampers. You can also model lumped mechanical systems, defined by components like mass, spring, damper, or impedance.

You will also find a subset of the structural mechanics capabilities in the MEMS Module and the Acoustics Module.

The exact contents of the menus and windows shown in this document may vary depending on the license or licenses you have. All illustrations assume the presence of these additional modules, even though the examples given only require the Structural Mechanics Module.


## SHELL AND PLATE

The Shell interface (  ) is intended for the structural analysis of thin-walled structures. The formulation used in the Shell interface is a Mindlin-Reissner type, which means that transverse shear deformations are accounted for. It can be used for rather thick shells as well as thin ones. It is possible to prescribe an offset in a direction normal to a selected surface, which for example can be used when meshing imported geometries. The Shell interface also includes other features such as damping, thermal expansion, and initial stresses and strains. With the Composite Materials Module, it is also possible to analyze layered shells. Through the addition of the Nonlinear Structural Materials Module, it is possible to model material behavior like plasticity, creep, and viscoplasticity. The preset studies available are the same as for the Solid Mechanics interface, except Mode Analysis and Bolt Pre-Tension.


The Plate interface (  ) is a 2D analogy to the 3D Shell interface. Plates are similar to shells, but act in a single plane and usually act only with out-of-plane

loads. The formulation and features for this physics interface are similar to those for the Shell interface.


## MEMBRANE

The Membrane interface () is used for thin membranes, which can be considered as curved plane stress elements in 3D having in-plane as well as out-of-plane deformation. The difference between a shell and a membrane is that a membrane does not have any bending stiffness. Thin films and fabric are structures suited for modeling with the Membrane interface. With the Nonlinear Structural Materials Module, hyperelasticity, nonlinear elasticity, plasticity, creep, and viscoplasticity can be modeled in addition to the default linear elastic material. With the Composite Materials Module, it is also possible to analyze layered membranes. The preset studies for this physics interface are the same as for the Solid Mechanics interface, except Mode Analysis, Linear Buckling, and Bolt Pre-Tension.


## BEAM

The Beam interface () is intended for the modeling of slender structures (beams) that can be fully described by their cross-section properties, such as area and moment of inertia. The Beam interface defines stresses and strains using Hermitian elements and Euler-Bernoulli or Timoshenko theory. Beam elements are used to model both planar and three-dimensional frame structures. This physics interface is also suitable for modeling reinforcements of solid and shell structures. It includes a library for rectangular, box, circular, pipe, hat, H-profile, U-profile, C-profile, and T-profile beam sections. Additional features include damping, thermal expansion, and initial stresses and strains. The preset studies for this physics interface are the same as for the Solid Mechanics interface, except Mode Analysis, Bolt Pre-Tension, and the Prestressed study types.

## BEAM CROSS SECTION


A separate 2D physics interface called the Beam Cross Section interface () can be used to compute the properties of general cross sections, to be used as inputs in beam analysis. The interface can also be used for detailed stress evaluation in beams, given the forces and moments acting of the section.

## TRUSS


The Truss interface () may be used to model slender structures that can only sustain axial forces. Trusses are modeled using Lagrange shape functions, which allow the specification of small strains as well as Green-Lagrange strains for large deformations. In addition to classical truss structures, the interface can be used for


modeling cables exposed to gravity forces (sagging cables). Additional features include damping, thermal expansion, and initial stresses and strains. The default material model is linear elastic, and together with the Nonlinear Structural Materials Module, it is possible to also model plasticity. The preset studies for this physics interface are the same as for the Solid Mechanics interface, except Mode Analysis and Bolt Pre-Tension.



## PIPE MECHANICS


The Pipe Mechanics () interface is intended for stress and deformation analysis in pipe system, either standalone, or together with the Pipe Flow interface. Pipes are similar to beams, but stresses from internal pressure and temperature gradients through the pipe wall are part of the formulation.

## OTHER STRUCTURAL MECHANICS INTERFACES


The Thermal Stress, Solid interface () combines the Solid Mechanics interface with the Heat Transfer in Solids interface. The temperature field is automatically coupled to a structure's thermal expansion, and also to any temperature dependent material properties. Similar interface are available also for shells and membranes.

The Joule Heating and Thermal Expansion interface () combines three physics interfaces: Joule Heating, Heat Transfer in Solids, and Solid Mechanics. It describes the conduction of electric current in a structure, the subsequent electric heating caused by the ohmic losses, and the thermal stresses induced by the temperature field.


The Piezoelectricity interface () combines the Solid Mechanics and Electrostatics interfaces to model piezoelectric materials. The piezoelectric coupling can be in stress-charge or strain-charge form. All solid mechanics and electrostatics functionalities are also accessible through this physics interface, for example, for modeling the surrounding linear elastic solids or air domains. For the Layered Shell interface the Piezoelectricity, Layered Shell interface () offers similar capabilities.

The Magnetostriction interface () combines the Solid Mechanics and Magnetic Fields interfaces to model magnetostrictive materials. Linear as well as nonlinear materials models are available. This interface requires the AC/DC module.

## FLUID FLOW

There are several Fluid-Structure Interaction (FSI) interfaces, found under the Fluid Flow branch in the Model Wizard. These interfaces combine fluid flow with structural mechanics to capture the interactions between fluids and structures. In the Fluid-Solid Interaction interface (), a Solid Mechanics interface and a







Single Phase Flow interface model the solid and the fluid, respectively. Together with CFD Module, also turbulent flow and two-phase flow can be studied. The FSI couplings appear on the boundaries between the fluid and the solid. The Fluid-Structure Interaction interface uses an arbitrary Lagrangian-Eulerian (ALE) method; it combines the fluid flow formulated by an Eulerian description and a spatial frame with solid mechanics formulated by a Lagrangian description and a material (reference) frame.









The Fluid-Solid Interaction, Fixed Geometry () interface can be used to model phenomena where a fluid and a deformable solid structure affect each other. Both the fluid loading on the structure and the structural velocity transmission to the fluid can be taken into account. The interface models situations where the displacements of the solid are assumed to be small enough for the geometry of the fluid domain to be considered as fixed during the simulation.






There are also similar interfaces for coupling shells and membranes to fluid flow.

### Physics Interface Guide by Space Dimension and Study Type




The table below lists the physics interfaces available specifically with this module in addition to those included in the basic COMSOL Multiphysics license.

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
 <b>Elastic Waves</b>				
Elastic Waves, Time Explicit		elte	3D, 2D	time dependent
 <b>Fluid Flow</b>				
Fluid-Solid Interaction <sup>2</sup>		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Fluid-Solid Interaction, Fixed Geometry <sup>2</sup>		—	3D, 2D, 2D axisymmetric	stationary; stationary, one-way coupled; time dependent; time dependent, one-way coupled
Fluid-Shell Interaction <sup>2</sup>		—	3D, 2D axisymmetric	stationary; time dependent





PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Fluid-Shell Interaction, Fixed Geometry <sup>2</sup>		—	3D, 2D axisymmetric	stationary; stationary, one-way coupled; time dependent; time dependent, one-way coupled
Fluid-Membrane Interaction <sup>2</sup>		—	3D, 2D axisymmetric	stationary; time dependent
Fluid-Membrane Interaction, Fixed Geometry <sup>2</sup>		—	3D, 2D axisymmetric	stationary; stationary, one-way coupled; time dependent; time dependent, one-way coupled
Conjugate Heat Transfer, Fluid-Solid Interaction <sup>2</sup>		—	3D, 2D, 2D axisymmetric	stationary; time dependent;
Fluid-Solid Interaction, Two-Phase Flow, Phase Field <sup>2, 6</sup>		—	3D, 2D, 2D axisymmetric	time dependent; time dependent with phase initialization
Fluid-Solid Interaction, Two-Phase Flow, Phase Field, Fixed Geometry <sup>2, 6</sup>		—	3D, 2D, 2D axisymmetric	time dependent; time dependent with phase initialization
 <b>Structural Mechanics</b>				
Solid Mechanics <sup>1</sup>		solid	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; eigenfrequency, prestressed; mode analysis; time dependent; time dependent, modal; time dependent, prestressed, modal; time dependent, modal reduced-order model; frequency domain; frequency domain, modal; frequency domain, prestressed; frequency domain, prestressed, modal; frequency domain, modal reduced-order model; frequency domain, AWE reduced-order model; response spectrum; random vibration (PSD); linear buckling; bolt pre-tension

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Thermal Stress, Solid <sup>2</sup>		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Thermal Stress, Shell <sup>2, 4</sup>		—	3D, 2D axisymmetric	stationary; time dependent
Thermal Stress, Membrane <sup>2, 4</sup>		—	3D, 2D axisymmetric	stationary; time dependent
Joule Heating and Thermal Expansion <sup>2</sup>		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Shell		shell	3D	stationary; eigenfrequency; eigenfrequency, prestressed; time dependent; time dependent, modal; time dependent, prestressed, modal; time dependent, modal reduced-order model; frequency domain; frequency domain, modal; frequency domain, prestressed; frequency domain, prestressed, modal; frequency domain, modal reduced-order model; response spectrum; random vibration (PSD); linear buckling



PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Plate		plate	2D	stationary; eigenfrequency; eigenfrequency, prestressed; time dependent; time dependent, modal; time dependent, prestressed, modal; time dependent, modal reduced-order model; frequency domain; frequency domain, modal; frequency domain, prestressed; frequency domain, prestressed, modal; frequency domain, modal reduced-order model; response spectrum; random vibration (PSD); linear buckling
Beam		beam	3D, 2D	stationary; eigenfrequency; time dependent, modal; time dependent, modal reduced-order model; frequency domain; frequency domain, modal; frequency domain, modal reduced-order model; time dependent; response spectrum; random vibration (PSD); linear buckling
Beam Cross Section		bcs	2D	stationary

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Truss		truss	3D, 2D	stationary; eigenfrequency; eigenfrequency, prestressed; time dependent; time dependent, modal; time dependent, prestressed, modal; time dependent, modal reduced-order model; frequency domain; frequency domain, modal; frequency domain, prestressed; frequency domain, prestressed, modal; frequency domain, modal reduced-order model; response spectrum; random vibration (PSD); linear buckling
Membrane		mbm	3D, 2D axisymmetric	stationary; eigenfrequency; eigenfrequency, prestressed; time dependent; time dependent, modal; time dependent, prestressed, modal; time dependent, modal reduced-order model; frequency domain; frequency domain, modal; frequency domain, prestressed; frequency domain, prestressed, modal; frequency domain, modal reduced-order model; response spectrum; random vibration (PSD)

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Pipe Mechanics		pipem	3D, 2D	stationary; eigenfrequency; time dependent, modal; time dependent, modal reduced-order model; frequency domain; frequency domain, modal; frequency domain, modal reduced-order model; time dependent; response spectrum; random vibration (PSD)
Piezoelectricity <sup>2</sup>		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; eigenfrequency, prestressed; time dependent; time dependent, modal; time dependent, prestressed, modal; frequency domain; frequency domain, modal; frequency domain, prestressed; frequency domain, prestressed, modal; small-signal analysis, frequency domain; linear buckling
Piezoelectricity, Layered Shell <sup>2, 3, 5</sup>		—	3D	stationary; eigenfrequency; time dependent; frequency domain
Magnetostriction <sup>2,3</sup>		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; time dependent; frequency domain; small-signal analysis, frequency domain; eigenfrequency, prestressed; frequency domain, prestressed
<sup>1</sup> This physics interface is included with the core COMSOL package but has added functionality for this module.				
<sup>2</sup> This physics interface is a predefined multiphysics coupling that automatically adds all the physics interfaces and coupling features required.				
<sup>3</sup> Requires the addition of the AC/DC Module.				

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
<sup>4</sup> Requires the addition of the Heat Transfer Module				
<sup>5</sup> Requires the addition of the Composite Materials Module				
<sup>6</sup> Requires the addition of the CFD Module or the Microfluidics Module				

## The Fundamentals: A Static Linear Analysis

---

This section summarizes the fundamentals for modeling structural mechanics problems and applying them in COMSOL Multiphysics and the Structural Mechanics Module. It includes instructions for creating a geometry, generating a mesh, defining material properties and boundary conditions. After the solution is computed, you will learn how to display and analyze the results.

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 6](#).

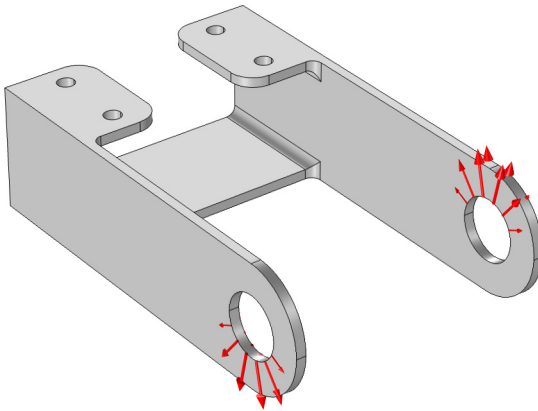


Figure 6: The geometry of the bracket together with the load distribution.

In this initial analysis, the mounting bolts are assumed to be fixed and securely bonded to the bracket, so they will be replaced by boundary conditions at the bolt holes. To model the external load from the pin, specify a surface pressure  $p$  with a trigonometric distribution on the inner surfaces of the two holes, given by

$$p = P_0 \cos(\alpha) \quad -\frac{\pi}{2} < \alpha < \frac{\pi}{2}$$

where  $P_0$  is the peak load intensity, and  $\alpha$  is the angle from the direction of the load resultant. One of the arms is loaded upward, and the other downward. The load distribution is shown in [Figure 6](#).

## 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.






**Note:** These instructions are for Windows users but apply with minor differences to Linux and Mac.

- 1 To start the software, double-click the COMSOL icon on the desktop. When the software opens, you can choose to use the Model Wizard to create a new COMSOL model or Blank Model to create one manually.

For this tutorial, click the Model Wizard button.

If COMSOL is already open, you can start the Model Wizard by selecting New  from the File menu and then click Model Wizard .

The Model Wizard guides you through the first steps of setting up a model. The next window lets you select the dimension of the modeling space.


- 2 On the Select Space Dimension page click 3D .
- 3 In the Select Physics tree under Structural Mechanics, click Solid Mechanics (solid) .
- 4 Click Add then click Study .
- 5 In the Select Study tree under General Studies, click Stationary .
- 6 Click Done .

## Global Definitions - Parameters

---

It is good modeling practice to gather constants and parameters in one place so that you can change them easily. Using parameters will also improve the readability of your input data.

The following parameters will be defined: the peak load intensity  $P_0$  and the  $y$ -coordinate of the center of the pin holes,  $Y_C$ .

- 1 In the Model Builder Window, under Global Definitions, select 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

The third column in the table is filled out automatically.

## Importing the Geometry

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). The file contains the assembly of both the bracket and mounting bolts.

**Note:** The location of the files used in this exercise varies based on your installation. For example, if the installation is on your hard drive, the file path might be similar to C:\Program\COMSOL55\Multiphysics\applications\Structural\_Mechanics\_Module\Tutorials.

1 On the Home toolbar click Import .

**Note:** On Linux and Mac, the Home toolbar refers to the specific set of controls near the top of the Desktop.

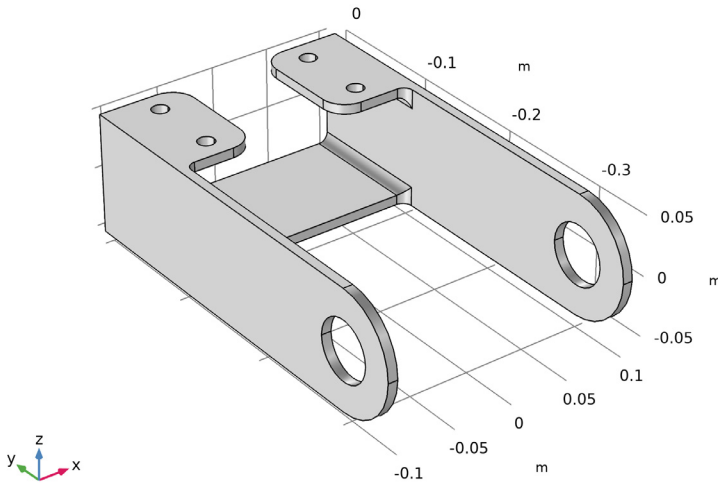
2 In the Settings window for Import, locate the Import section.

3 From the Source list, choose COMSOL Multiphysics file.

4 Click Browse.

5 Browse to the folder Structural\_Mechanics\_Module\Tutorials under the COMSOL installation directory and double-click the file bracket.mphbin.

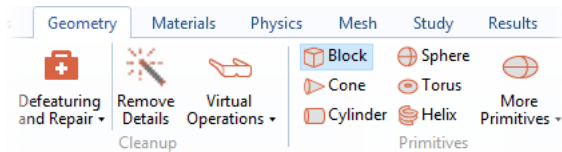
6 Click Import.



It is possible to automatically create a free tetrahedral mesh on the whole component. Such a strategy may however not be optimal for the large flat regions. For this reason, you will partition the geometry so that you can create a better mesh.

For a small linear problem like this, it would still be more efficient just to use a rather fine tetrahedral mesh, but the approach shown here is useful for more general cases.

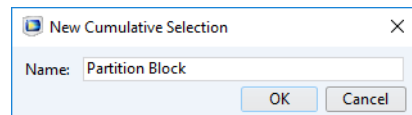
1 On the Geometry toolbar, select Block .



2 In the Settings window for Block, locate the Selections of Resulting Entities section.

3 Click New.

4 In the New Cumulative Selection dialog box, type Partition Block in the Name text field.



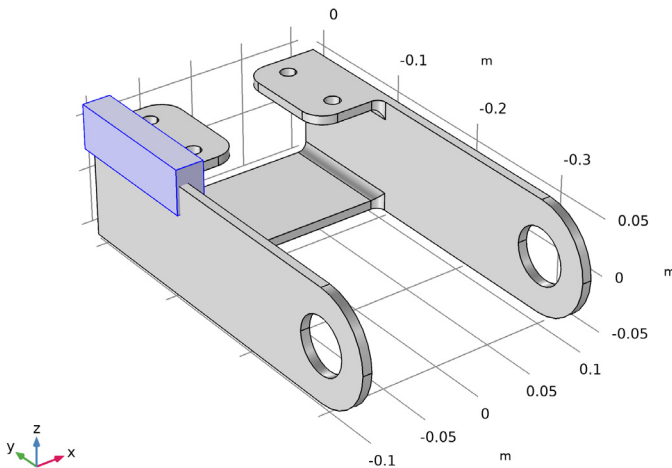
5 Click OK.





- 6 Locate the Size and Shape section, and enter the values shown below.

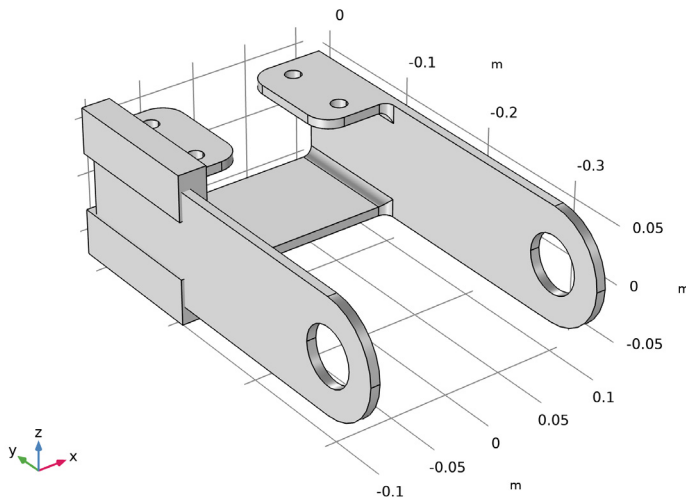
▼ Size and Shape	
Width:	0.025 m
Depth:	0.13 m
Height:	0.04 m
▼ Position	
Base:	Corner ▼
x:	-0.11 m
y:	-0.12 m
z:	0.025 m

You have now created a block which encompasses the part of the geometry which has a more complex geometry, and thus requires free tetrahedral mesh. Now mirror this block twice, so that all four fillets are covered.





- 7 On the Geometry toolbar, click Transforms  and choose Mirror .
- 8 In the Settings window for Mirror, locate the Input section. From the Input objects list, choose Partition Block.
- 9 Select the Keep input objects check box.

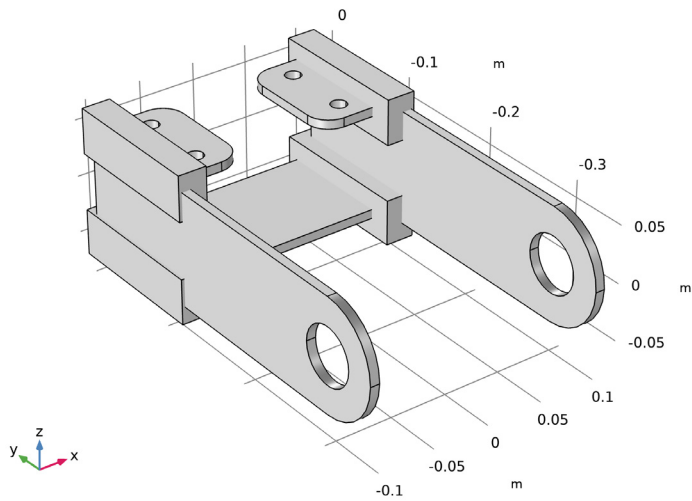
- 10** Locate the Selections of Resulting Entities section. From the Contribute to list, choose Partition Block. Click Build Selected  .





Create the second mirror copy.


- 11** On the Geometry toolbar, click Transforms  and choose Mirror .
- 12** In the Settings window for Mirror, locate the Input section.
- 13** From the Input objects list, choose Partition Block.
- 14** Select the Keep input objects check box.
- 15** Locate the Normal Vector to Plane of Reflection section. In the x text field, type 1. In the z text field, type 0.
- 16** Locate the Selections of Resulting Entities section. From the Contribute to list, choose Partition Block.

17 Click Build Selected  .

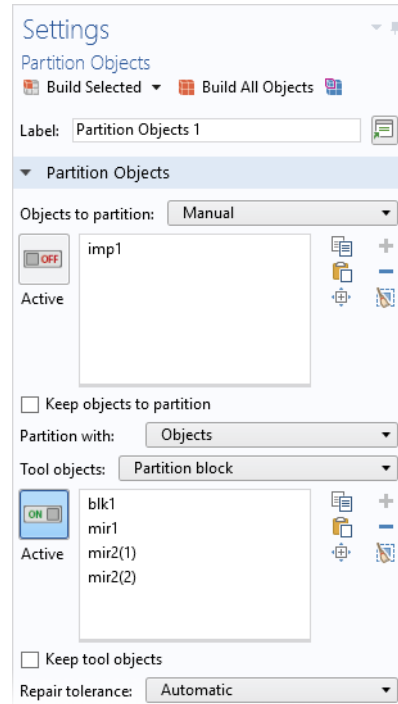



## Partition objects

- 1 On the Geometry toolbar, click Booleans and Partitions  and choose Partition Objects .
- 2 In the Settings for Partition Objects, locate the Partition Objects section. In Objects to partition, select imp1.

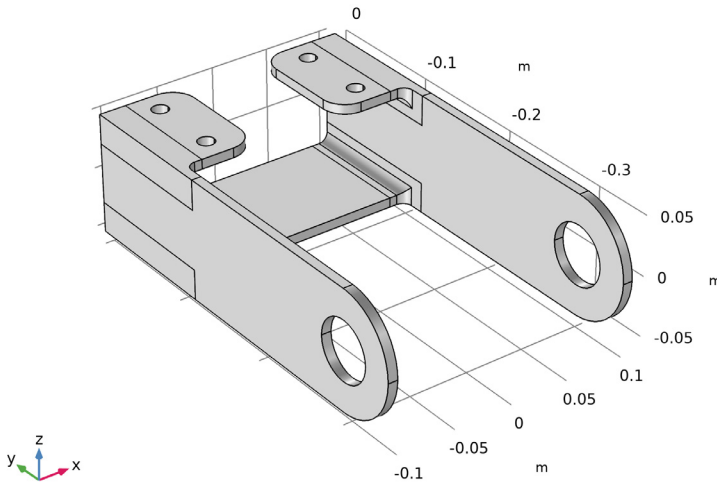
**Note:** There are many ways to select geometric entities. When you know the geometric entity to add, as in these exercises, you can click the Paste Selection button  and enter the information in the Selection text field. In this case, ‘imp1’ is the whole bracket, so quickest way to select it is to click on it in the graphics window.

For more information about selecting geometric entities in the Graphics window, see the *COMSOL Multiphysics Reference Manual*.






- 3 In the Tool objects subsection, select the Active toggle button. Then from the Tool objects list, select Partition Block.
- 4 Click Build Selected .

The solid geometry has now been partitioned into nine separate domains, five of which are thin objects with a constant thickness.



### Form Union (fin)

- 1 In the Model Builder under Component 1 (comp1)>Geometry 1 right-click Form Union (fin)  and choose Build All  (or click the button on the Settings window).
- 2 Click the Zoom Extents button  on the Graphics toolbar.


The Form Union node determines how the parts of an assembly will be considered in the analysis. By using the default setting, Form a union, any adjacent domains will be bonded, and the interior boundaries assume continuity. If Form an assembly were selected instead, different parts would not be connected to each other, and then structural contact could be modeled. In this case, the node would also be renamed to Form Assembly, so that you can see which state the model is in by just looking at the geometry sequence.

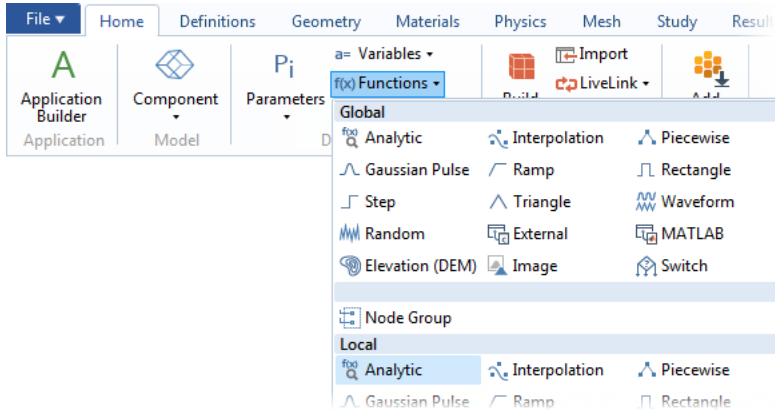
## Definitions - Functions and Selections

---

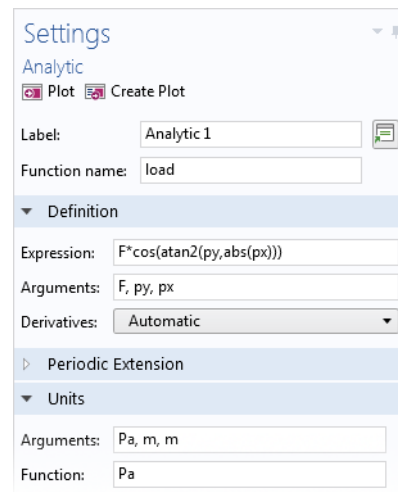
Here you want to define expressions for the load applied to the load-carrying holes. Assume the load distribution to be defined by a trigonometric function.

## Analytic 1 (an1)

1 On the Home toolbar, click Functions and under Local choose 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  $F \cdot \cos(\text{atan2}(py, \text{abs}(px)))$  (or copy and paste this text).
- 4 In the Arguments text field, type  $F$ ,  $py$ ,  $px$ .
- 5 Locate the Units section. In the Arguments text field, type  $\text{Pa}$ ,  $\text{m}$ ,  $\text{m}$ .
- 6 In the Function text field, type  $\text{Pa}$ .

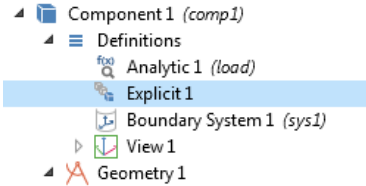



Regardless of whether the model is treated as a single entity or as an assembly, you still may need to access different parts of it for definitions, such as multiple specifications of similar boundary conditions. There are different ways in which you can create selections, for example by explicit selection of objects or by geometrical location. A box selection is independent of the geometry, and allows you to change the geometry topology while still keeping the desired selection.

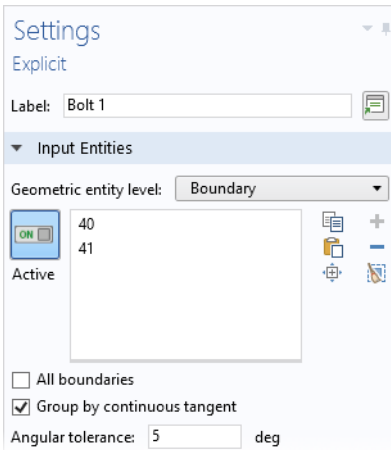
In this example, one selection contains the bolt domains, while other selections contain the load-bearing boundaries of the pin holes.

### *Explicit 1*


- 1 On the Definitions toolbar, click Explicit . An Explicit node is added under Definitions.





- 2 In the Settings window for Explicit, in the Label text field, type **Bolt 1**. You can also click any node in the model builder tree and press F2 to rename, or right-click and select Rename.
- 3 On the Graphics toolbar, click Wireframe Rendering  in order to see also hidden boundaries better.
- 4 Locate the Input Entities section. From the Geometric entity level list, choose Boundary.
- 5 Select Boundary 41 only.
- 6 Select the Group by continuous tangent check box. Boundary 40 is automatically added to the selection list.



### *Explicit 2, Explicit 3, and Explicit 4*

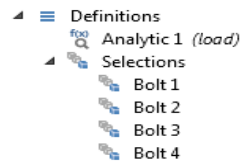
Repeat the steps completed for Explicit 1 to add a total of four Explicit nodes .

- 1 On the Definitions toolbar, click Explicit  three times.
- 2 In the Model Builder click the next Explicit node  to open the Settings window. Edit the Settings window as indicated in the table below.
- 3 Use the table as a guide, and in the Graphics window, select the boundaries as indicated. Click to select the Group by continuous tangent check box for each node.



DEFAULT NODE LABEL	NEW NODE LABEL	SELECT THIS BOUNDARY
Explicit 2	Bolt 2	43
Explicit 3	Bolt 3	55
Explicit 4	Bolt 4	57

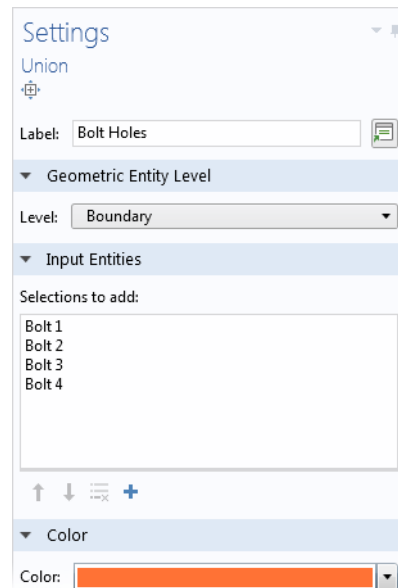
The node sequence under Definitions should look like this at this point:

You will now create a selection containing all the bolt holes.



### Union 1


- 1 On the Definitions toolbar, click Union .
- 2 In the Settings window for Union, in the Label text field enter **Bolt Holes**.
- 3 Locate the Geometric Entity Level section. From the Level list, choose Boundary.
- 4 Locate the Color section. On Windows, select the eighth color in the first row of the palette (orange). On other platforms, choose Color 8 from the Color list.
- 5 Locate the Input Entities section. Under Selections to add, click the Add button .
- 6 In the Add dialog box, in the Selections to add list, choose Bolt 1, Bolt 2, Bolt 3, and Bolt 4.
- 7 Click OK.





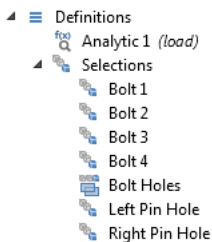
Create selections for the two holes carrying the load.

### *Explicit 5*

- 1 On the Definitions toolbar, click Explicit .
- 2 In the Settings window for Explicit, in the Label text field, type **Left Pin Hole**.
- 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.
- 6 Locate the Color section. On Windows, select the ninth color in the first row of the palette (green). On other platforms, choose Color 9 from the Color list.



### *Explicit 6*


Repeat the steps above, but use the label **Right Pin Hole**, select boundary 75, and use Color 12 (a red color). Now the sequence under Definitions should look like this:



### *Adjacent 1*

Add one more selection to be used during mesh generation.




- 1 On the Definitions toolbar, click Adjacent .
- 2 In the Settings window for Adjacent, in the Label text field, type **Bolt Hole Edges**.
- 3 Locate the Input Entities section. From the Geometric entity level list, choose Boundary.
- 4 Under Input Selections, click the Add button .
- 5 In the Add dialog box, select Bolt Holes in the Input selections list
- 6 Click OK.
- 7 Locate the Output Entities section. From the Geometric entity level list, choose Adjacent Edges.

- 8 On the Graphics toolbar, click Wireframe Rendering  to reset the graphics to a view with hidden surfaces.

## 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.

- 1 On the Home toolbar click Add Material  to open the Add Material window.
- 2 In the Add Material window under Built-In, click Structural steel.
- 3 Click the Add to Component button .
- 4 On the Home toolbar click Add Material  again to close the window.

## Solid Mechanics


---

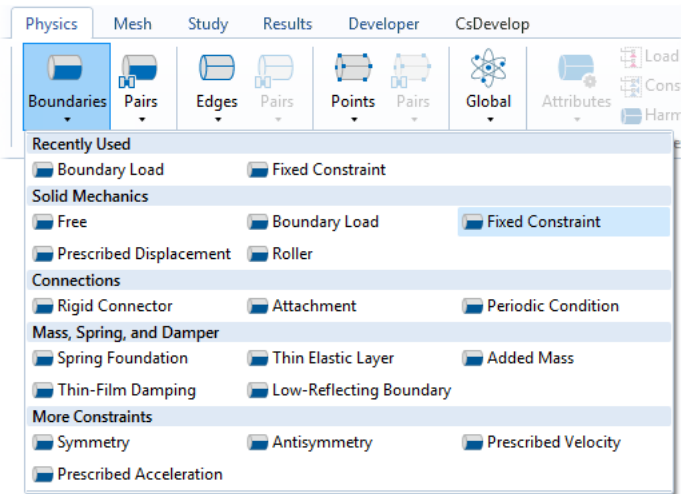
Now it is time to define the physics settings, such as material models, loads and constraints. Initially, the analysis was specified to be stationary using the classical equations associated with solid mechanics.

By default, the Solid Mechanics interface assumes that the materials are linear elastic, which is appropriate for this example. All that is left to do is to define the constraints and loads.

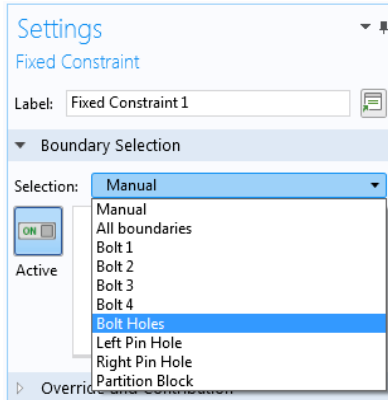
### *Fixed Constraint 1*

Assume that the boundaries of the bolt holes are perfectly constrained.

- 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.



### *Boundary Load 1*

Apply a boundary load to the bracket holes in the normal direction. This is done by using the built-in boundary coordinate system.

- 1 On the Physics toolbar click Boundaries  and choose Boundary Load .
- 2 In the Settings window for Boundary Load, in the Label text field, type Boundary Load, Left.

- 3 Locate the Boundary Selection section. From the Selection list, choose Left Pin Hole.
- 4 Locate the Coordinate System Selection section. From the Coordinate system list, choose Boundary System 1 (sys1). The input of the load is now made in this coordinate system, and you can directly access the normal to the boundaries.
- 5 Locate the Force section. Specify the  $\mathbf{F}_A$  vector as:

0	t1
0	t2
$\text{load}(-P0, Y-YC, Z) * (Z < 0)$	n

Note that the coordinate names in the second column of the table were updated to reflect the chosen coordinate system.

The previously defined function 'load()' is called in order to provide the load distribution. Note how the load is limited to act on the lower half of the hole through multiplication with the boolean expression  $Z < 0$ .

#### *Boundary Load, Left 1*


- 1 Right-click Boundary Load, Left and choose Duplicate.
- 2 In the Settings window for Boundary Load, in the Label text field, modify the text to **Boundary Load, Right**.
- 3 Locate the Boundary Selection section. From the Selection list, choose Right Pin Hole.
- 4 Change the expression for the load to  $\text{load}(-P0, Y-YC, Z) * (Z > 0)$ , since only the upper half of the right hole is loaded.

## **Mesh 1**

---

Start by creating an edge mesh around the bolt holes to make sure that they are properly resolved.

#### *Edge 1*

- 1 In the Model Builder window, under Component 1 (comp1), right-click Mesh 1 and choose More Operations -> Edge .
- 2 In the Settings window for Edge, locate the Edge Selection section. From the Selection list, choose Bolt Hole Edges.



#### *Distribution 1*

- 1 In the Model Builder, Right-click Edge 1 and choose Distribution .


- 2 In the Settings window for Distribution, locate the Distribution section. In the Number of elements text field, type 8.

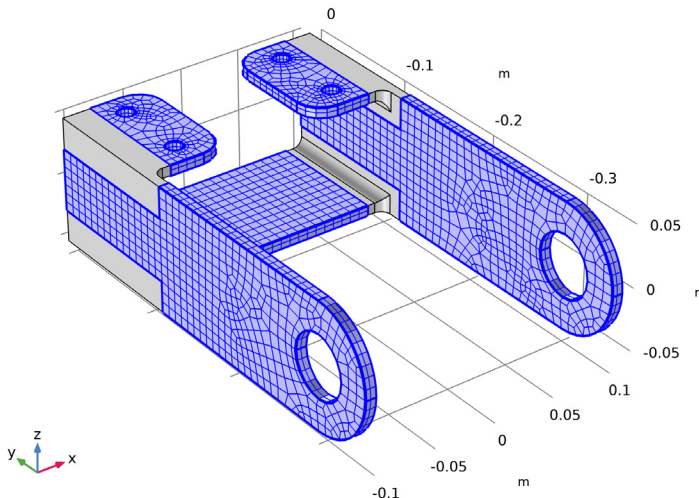
Create a mesh which is swept through the thin flat parts, and then use a free tetrahedral mesh for the remaining more complex parts of the geometry. Note that the transition between the two types of elements is automatic.


#### *Swept 1*

- 1 On the Mesh toolbar, click Swept .
- 2 In the Settings window for Swept, locate the Domain Selection section. From the Geometric entity level list, choose Domain, and select domains 1, 4, 5, 6, and 9.
- 3 Click to expand the Source faces section. Select Boundaries 1, 33, 37, 50, and 72.
- 4 In the Model Builder, right-click Swept 1 and choose Size .

#### *Size 1*



- 1 In the Settings window for Size, locate the Element Size section. Click the Custom button.
- 2 Locate the Element Size Parameters section. Select the Maximum element size check box. In the associated text field, type 8[mm].
- 3 Click Build Selected  to generate the swept mesh in the thin parts..



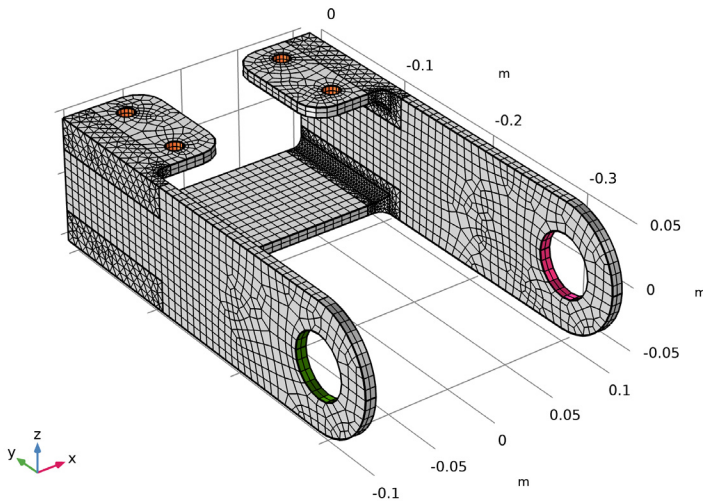
- 4 On the Mesh toolbar, click Free Tetrahedral .

### Free Tetrahedral 1

The default selection is Remaining, that is all not yet meshed domains.

- 1 In the Model Builder, right-click Free Tetrahedral 1 and choose Size .
- 2 In the Settings window for Size, locate the Element Size section. From the Predefined list, choose Finer.
- 3 Click Build All .

You should now have a mesh which looks similar to this:



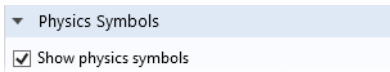
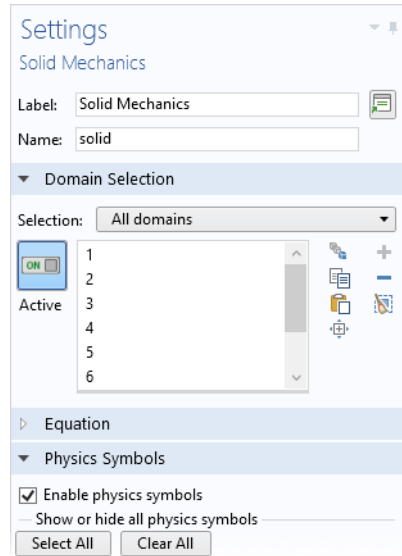
## DISPLAYING PHYSICS SYMBOLS IN THE GRAPHICS WINDOW


When working with the physics features, the current selection is shown in the Graphics window. You can add symbols to describe the type of condition applied to the selection.

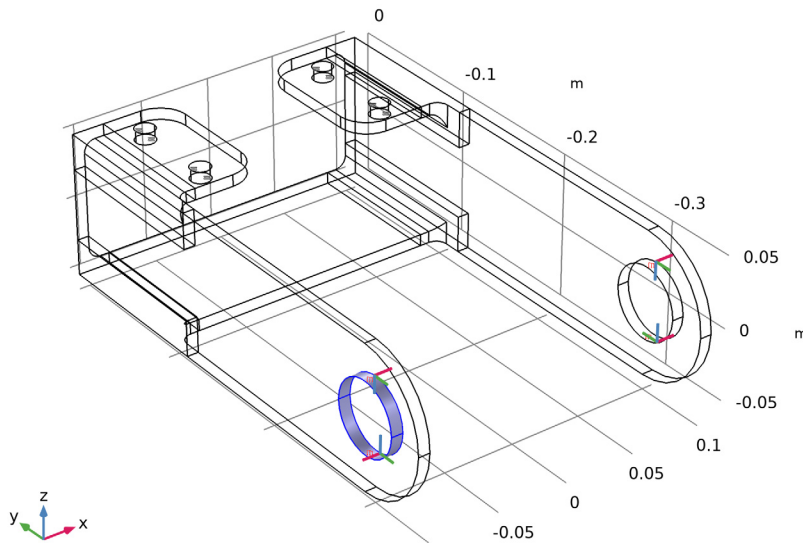
The symbols indicate only the type of settings applied to the model and not the actual magnitude or direction. If you are interested in visualizing an actual applied load, a solution must be computed first.

You turn the physics symbols on by selecting Enable physics symbols in the settings for a physics interface.

When physics symbols are enabled for the interface, a Physics Symbols section will be shown in the settings for each node in the Model Builder tree that has an associated symbol. You can then control the state of the physics symbols for individual boundary conditions.




You can click the Wireframe Rendering button  to see the symbols everywhere.



## Study

To check the actual load distribution, you can compute the initial value. For a large model, this is much faster than computing the actual solution, but the approach will only work as long as the loads are independent of the solution. The steps below show how to visualize the load distribution in the current geometry.

- On the Study toolbar click Get Initial Value .

The Study node automatically defines a solver sequence for the simulation based on the selected physics interface (Solid Mechanics) and study type (Stationary). If a mesh had not been created yet, the Study node automatically generates it at the same time as the solver sequence.

Study1

Step 1: Stationary

Solver Configurations

Solution 1 (sol1)

Compile Equations: Stationary

Dependent Variables 1

Displacement field (comp1.u)

Stationary Solver 1

Direct

Advanced

Fully Coupled 1

Suggested Direct Solver (solid)

Suggested Iterative Solver (solid)




**Note:** In practice, do not rely only on the standard mesh settings. For most real life problems, suitable meshing parameters should be set up from the Mesh toolbar.

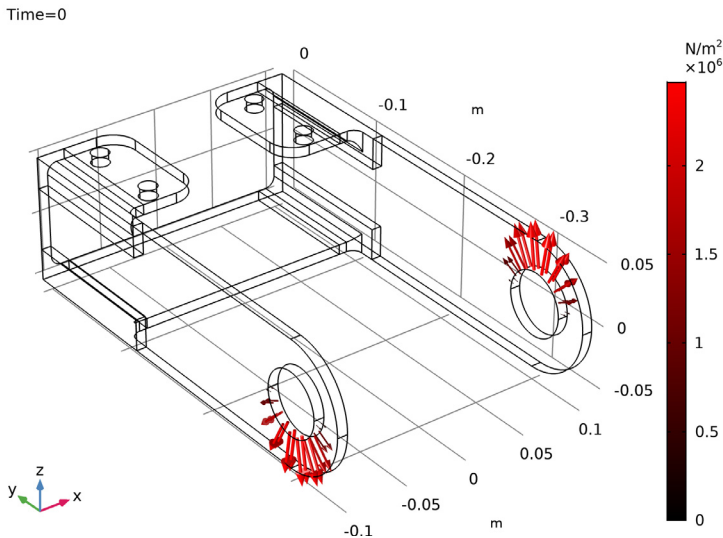
## Results

Among the default plots that are generated, there is a group containing arrow plots of the applied loads.

### *Boundary Loads (solid)*


- 1 In the Model Builder window, expand the Results>Applied Loads (solid) node, then click Boundary Loads (solid).
- 2 In the Boundary Loads (solid) toolbar, click Plot . This step may not be necessary, depending on your settings for automatic update of plots in the Results node.

From the Graphics window you can now check that the applied loads have the intended distributions.



## Study


---

Solve the model. On the Home or Study toolbar, click Compute . Solving the problem will take a few seconds.

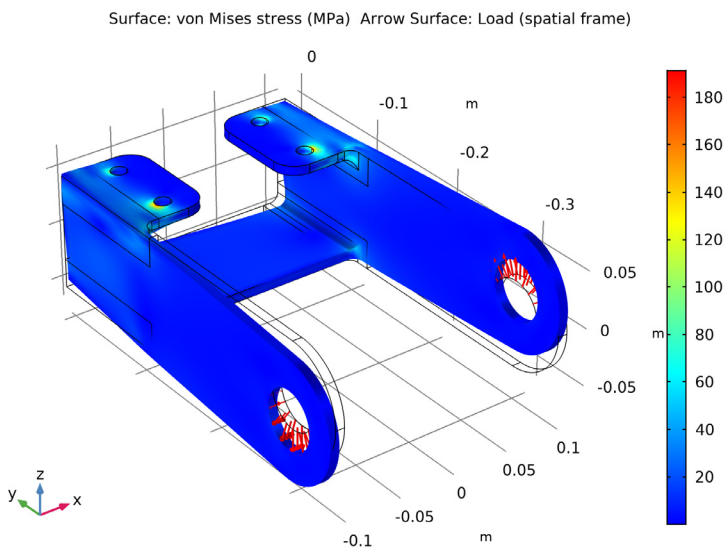
## Results

---

### Stress (solid)



- 1 Expand the Stress (solid) node, and then click Surface 1.
- 2 In the settings window for Surface, locate the Expression setting.
- 3 From the Unit list, choose MPa.
- 4 Click the Zoom Extents button  on the Graphics toolbar.

The default plot displays 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.





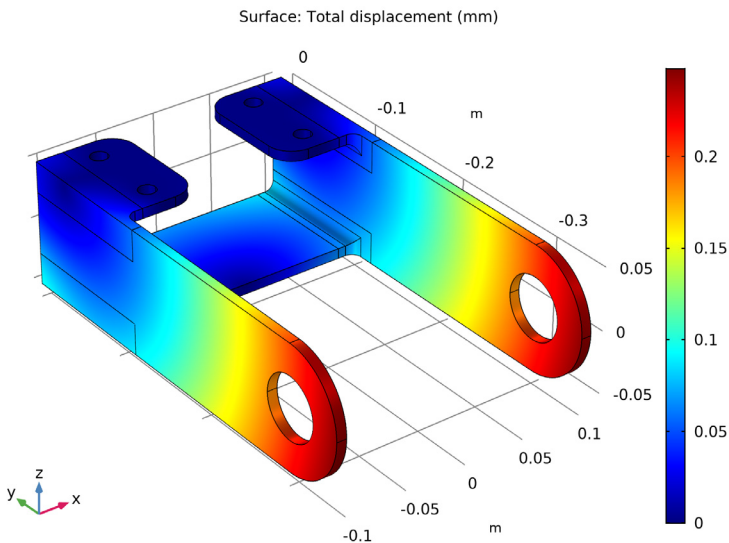
### 3D Plot Group 3

Add a plot group to display the displacement of the bracket.

- 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. Click anywhere on the Desktop for the setting to take effect.



### *Total Displacement*

- 1 On the Total Displacement toolbar, click Surface . Or right-click Total Displacement and choose Surface.
- 2 In the Settings window for Surface, locate the Expression section. Note that the total displacement, `solid.disp`, was selected as the default result.
- 3 From the Unit list, choose mm (or enter mm in the text field).
- 4 On the Total Displacement toolbar, click Plot . You can also click Plot on the Settings window for Surface.






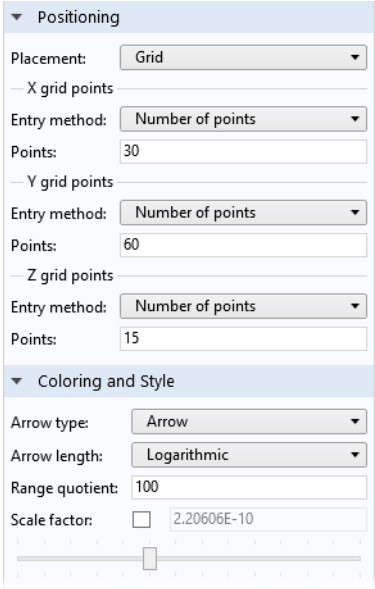
### *3D Plot Group 4*

Create a third plot to display the principal stresses.

- 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

- 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. Enter values as follows and replace the defaults:
  - Find the X grid points subsection. In the Points text field, type 30.
  - Find the Y grid points subsection. In the Points text field, type 60.
  - Find the Z grid points subsection. In the Points text field, type 15.
- 3 Locate the Coloring and Style section. From the Arrow length list, choose Logarithmic.
- 4 On the Principal Stress toolbar, click Plot .



▼ Positioning

Placement:

— X grid points

Entry method:

Points:

— Y grid points

Entry method:

Points:

— Z grid points

Entry method:

Points:

▼ Coloring and Style

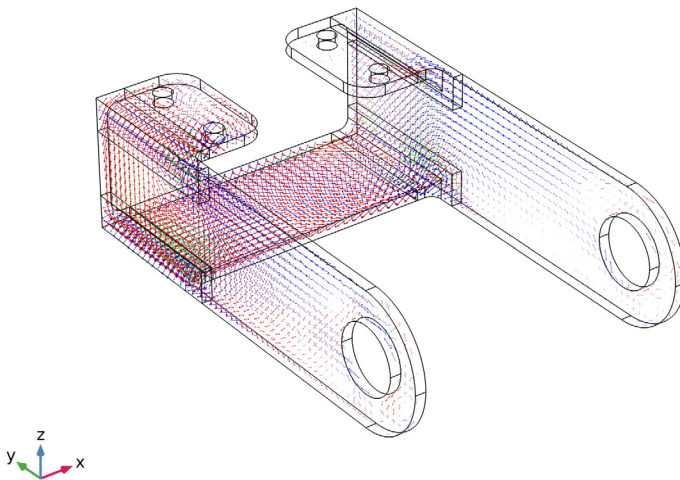
Arrow type:

Arrow length:

Range quotient:

Scale factor: ☐

Principal stress




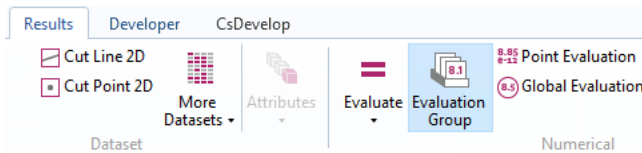
## Derived Values

---

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.





### *Evaluation Group 1*

- 1 In the Results toolbar, click Evaluation Group .



- 2 In the Model Builder window, under Results click Evaluation Group 1.
- 3 In the Settings window for Evaluation Group, type Evaluation Group: Reactions in the Label text field.

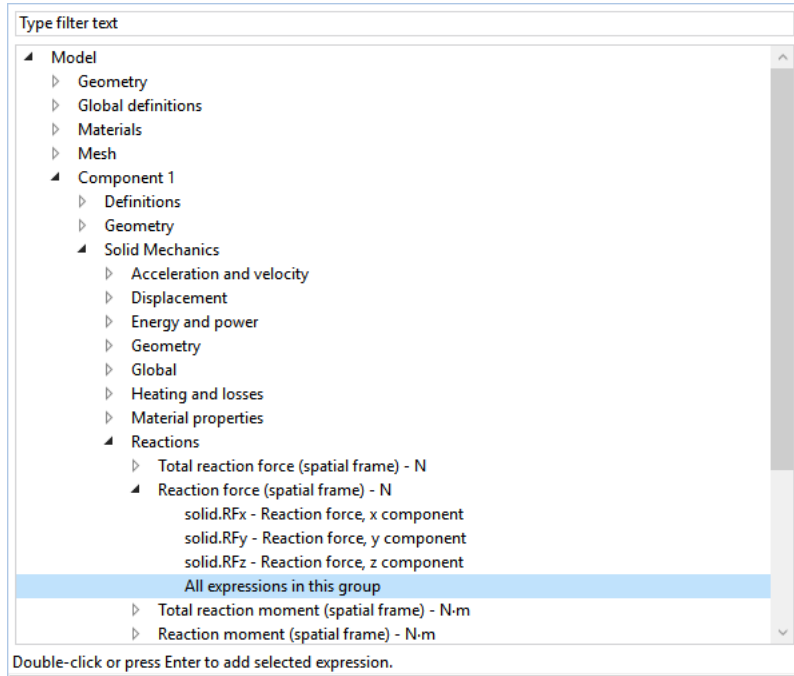
### *Surface Integration - Bolt 1*

- 1 On the Evaluation Group: Reactions toolbar click Integration  and choose Surface Integration . Alternatively, in the Model Builder, right-click Evaluation Group: Reactions and choose Integration>Surface Integration .
- 2 In the Settings window for Surface Integration type Bolt 1.
- 3 Locate the Selection section. From the Selection list, choose Bolt 1.
- 4 Locate the Expressions section. Click Replace Expression  in the upper-right corner of the Expressions section. From the menu, choose

Component 1>Solid Mechanics>Reactions>

Reaction force (spatial frame) - N>All expressions in this group.

This will add all three components of the reaction force vector in the Expression column.



- 5 In the table in the Expressions section, fill in the Descriptions column as shown below.

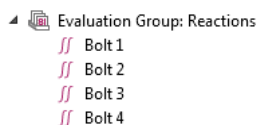
Expressions		
Expression	Unit	Description
solid.RFx	N	Bolt 1, X
solid.RFy	N	Bolt 1, Y
solid.RFz	N	Bolt 1, Z


### Bolt 2, Bolt 3, and Bolt 4

- 1 Right-click Bolt 1 and choose Duplicate. Repeat this two more times, to add a total of four Surface Integration nodes. Adjust the settings according to the table below.

NODE LABEL	SELECTION	DESCRIPTION FIELDS
Bolt 2	Bolt 2	Bolt 2, X etc.
Bolt 3	Bolt 3	Bolt 3, X etc.
Bolt 4	Bolt 4	Bolt 4, X etc.

Now, the evaluation group in the Model Builder should look like this:



- 2 In the Evaluation Group: Reactions toolbar, click Evaluate . All reaction forces in the evaluation group are now displayed.

Messages

Progress

Log

Evaluation Group: Reactions

8.85  
e-12

8.5  
e-1
850  
e-3
0.85

Bolt 1, X (N)	Bolt 1, Y (N)	Bolt 1, Z (N)	Bolt 2, X (N)	Bolt 2, Y (N)	Bolt 2, Z (N)	Bolt 3, X (N)	Bolt 3, Y (N)	Bolt 3, Z (N)
450.78	306.77	2534.7	-451.00	72.831	-1702.7	451.53	-306.63	-2534.7

# Parametric Study

---

In the previous section, a bracket loaded by an actuator was analyzed. This section extends this analysis to study the effect of the actuator’s position. This is equivalent to changing the direction of the applied load. You will use a load-angle parameter to set up a parametric study.

COMSOL Multiphysics has two ways to perform parametric studies — a Parametric Sweep node or an Auxiliary sweep from the Stationary Solver node. In this example, either method can be 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 parameterization.

## Load the Model

---

Either continue working on the existing model or open a saved version of the model from the Application Libraries. Browse to the `Structural_Mechanics_Module\Tutorials` folder. Double-click to open `bracket_static.mph`.

## Definitions — Parameters and Selections

---

- 1 In the Model Builder expand the Global Definitions node, then click Parameters 1  $P_1$  .
- 2 In the Settings window for Parameters, locate the Parameters section.
- 3 In the table, add the following settings:

NAME	EXPRESSION	VALUE	DESCRIPTION
theta0	0[deg]	0 rad	Load direction angle



## Definitions — Local Coordinate System

---

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  $\varphi=0$  subsection. In the table, enter the following settings:

X	Y	Z
0	$\sin(\text{theta0})$	$\cos(\text{theta0})$

The orientation of the coordinate system now depends on the parameter theta0.

Settings

Cylindrical System

Label: 


Name:

Settings

Frame:

Spatial (x, y, z)

Coordinate names

First	Second	Third
r	phi	a

Origin

xw (m)	yw (m)	zw (m)
0	YC	0

Longitudinal axis

xw	yw	zw
1	0	0

Direction of axis  $\varphi=0$ 

xw	yw	zw
0	$\sin(\text{theta0})$	$\cos(\text{theta0})$

## Definitions - Functions

---

Change the expression for the load distribution.

### *Analytic 1 (load)*

- 1 In the Model Builder under Component 1 (comp1)>Definitions click Analytic 1 (load).
- 2 In the Settings window for Analytic, locate the Definition section. In the Expression text field, replace the previous expression by  $F \cdot \cos(p)$ .
- 3 In the Arguments text field, type  $F, p$ .
- 4 Locate the Units section. In the Arguments text field, replace the previous contents with Pa, rad.

▼ Definition

Expression:

Arguments:

Derivatives:

► Periodic Extension

▼ Units

Arguments:

Function:

## Solid Mechanics

---

Change the boundary loads to consider the parameterized direction.

### *Boundary Load, Left*

- 1 In the Model Builder expand the Component 1 (comp1)>Solid Mechanics (solid) node, then click Boundary Load, Left.
- 2 Locate the Coordinate System Selection section. From the Coordinate system list, choose Cylindrical System 2 (sys2).

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

<code>load(-P0,sys2.phi)*(abs(sys2.phi)&gt;pi/2)</code>	r
0	phi
0	a

Note how the boolean expression `abs(sys2.phi)>pi/2` is used to filter the load so that it is applied only to half of the bolt hole. The loaded area does however rotate with the coordinate system, which is controlled by the parameter `theta0`.

#### *Boundary Load, Right*

- 1 In the Model Builder window, click Boundary Load 2.
- 2 Locate the Coordinate System Selection section. From the Coordinate system list, choose Cylindrical System 2 (sys2).
- 3 In the Force section, specify the  $\mathbf{F}_A$  vector as


<code>load(P0,sys2.phi)*(abs(sys2.phi)&lt;pi/2)</code>	r
0	phi
0	a

Now add an auxiliary sweep parameter.

## **Study- Auxiliary Sweep**

Parametric studies can be set up from scratch or, as in this example, added to an existing study. This model computes the stress in the bracket for different load orientations. Start by selecting the sweep parameter, in this case the load direction angle. Then run the analysis.

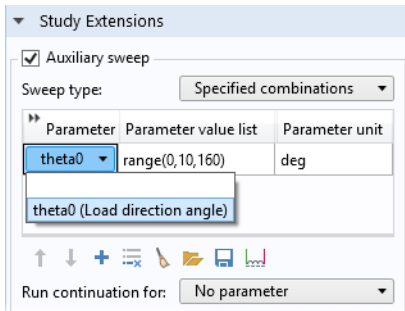
#### *Step 1: Stationary*


- 1 In the Model Builder expand the Study 1 node and click Step 1: Stationary.
- 2 In the Settings window for Stationary, click to expand the Study Extensions section. Select the Auxiliary sweep check box.
- 3 Under the table, click Add .

- 4 Choose **theta0** (Load direction angle) from the list. In the table, enter the additional settings:

PARAMETER NAME	PARAMETER VALUE LIST	PARAMETER UNIT
theta0	range(0,10,160)	deg

- 5 From the Run continuation for list, choose No parameter, since you are analyzing a set of independent loads.




- 6 On the Settings window for Stationary click Compute .


## Results

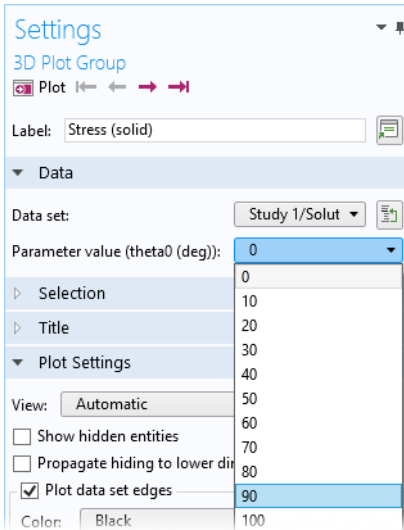
The default plot shows the solution for the last parameter value (160[deg]) You can easily change the parameter value to display the plot and then compare solutions for different load cases.


**Note:** Click the Zoom Extents button  to view the new default plots.

*Stress (solid)*

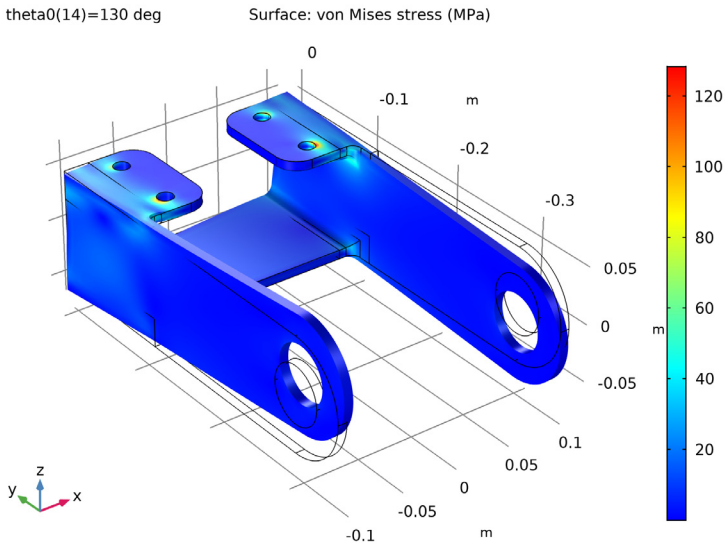
- 1 In the Settings window for 3D Plot Group, click the Plot First button  to plot the first solution at load angle 0.

- 2 Locate the Data section. From the Parameter value (theta0 (deg)) list, choose 90. Click Plot .





- 3 Repeat this and choose 130. Click Plot .

The stress distribution when at the angle 130 is shown below.



You will now create a plot showing how the reaction forces vary with the load angle.


#### *Evaluation Group: Reactions*

- 1 In the Model Builder, click Evaluation Groups: Reactions.
- 2 In the Evaluation Groups: Reactions toolbar, click Evaluate .
- 3 In the Table window, on the toolbar in the Evaluation Group: Reactions tab, click Table Graph .

#### *1D Plot Group 5*


- 1 In the Model Builder window, under Results click 1D Plot Group 5.
- 2 In the Settings window for 1D Plot Group, type Reaction Force, X-component 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 Reaction Force (N).

## Table Graph 1

- 1 In the Model Builder window click Table Graph 1.
- 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, choose Bolt 1, X (N), Bolt 2, X (N), Bolt 3, X (N), and Bolt 4, X (N).
- 5 Locate the Coloring and Style section. In the Width text field, type 2.
- 6 Find the Line markers subsection. From the Marker list, choose Cycle.
- 7 Click to expand the Legends section. Select the Show legends check box.
- 8 In the Reaction Force, X-component toolbar, click Plot  to show the variation of the four X-direction reaction forces as function of the load angle parameter.

### Settings

#### Table Graph

 Plot

Label:

---

#### Data

Source:

Evaluation group:

x-axis data:

Plot columns:

Columns:

theta0 (deg)  
Bolt 1, X (N)  
Bolt 1, Y (N)  
Bolt 1, Z (N)  
Bolt 2, X (N)  
Bolt 2, Y (N)  
Bolt 2, Z (N)  
Bolt 3, X (N)

Transformation:

☐ Plot imaginary part

---

#### Preprocessing

---

#### Coloring and Style

Line style

Line:

Color:

Width:

Line markers

Marker:

Positioning:

Number:

---

#### Legends

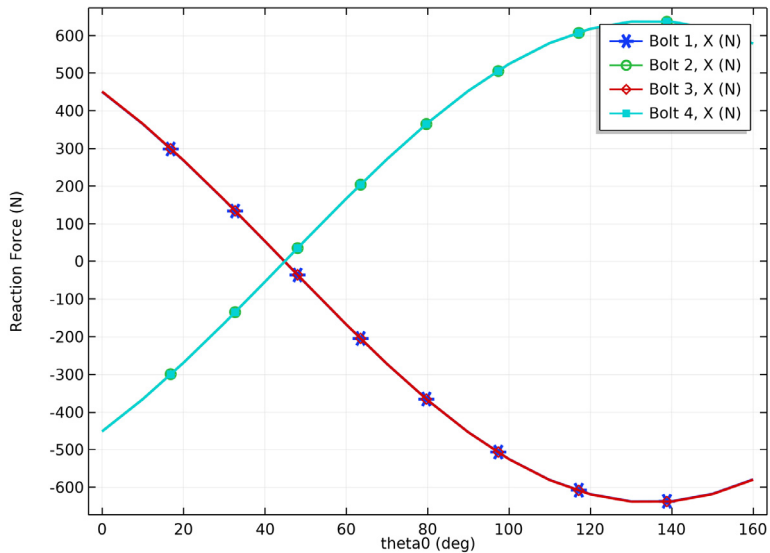
☒ Show legends

Legends:

Prefix:


Suffix:

The graph should look like this:



You will now create similar plots for the reaction forces in the other two directions. When you already have a plot which is similar to the one you are going to create, it is usually more efficient to duplicate an existing plot, and then change its properties, than to start from new plot.


#### Reaction Force, X-component 1

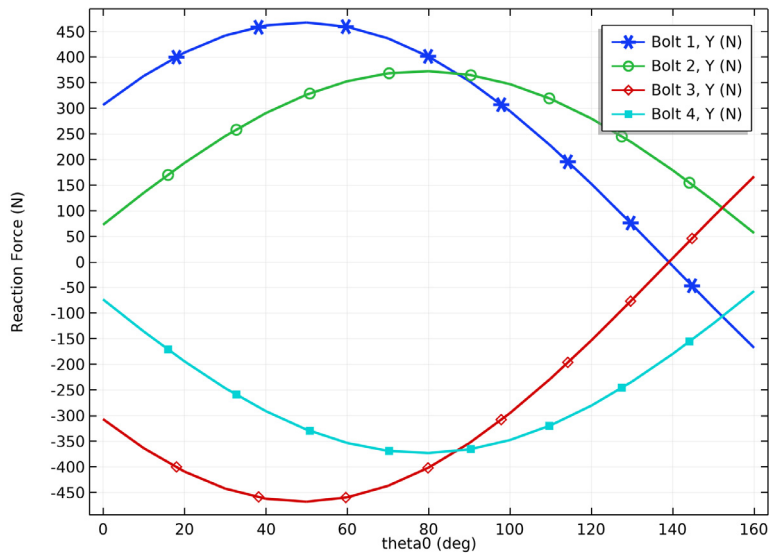
- 1 Right-click Reaction Force, X-component, and choose Duplicate .
- 2 Click Reaction Force, X-component 1. In the Settings window for 1D Plot Group, change to Reaction Force, Y-component in the Label text field.

#### Table Graph 1

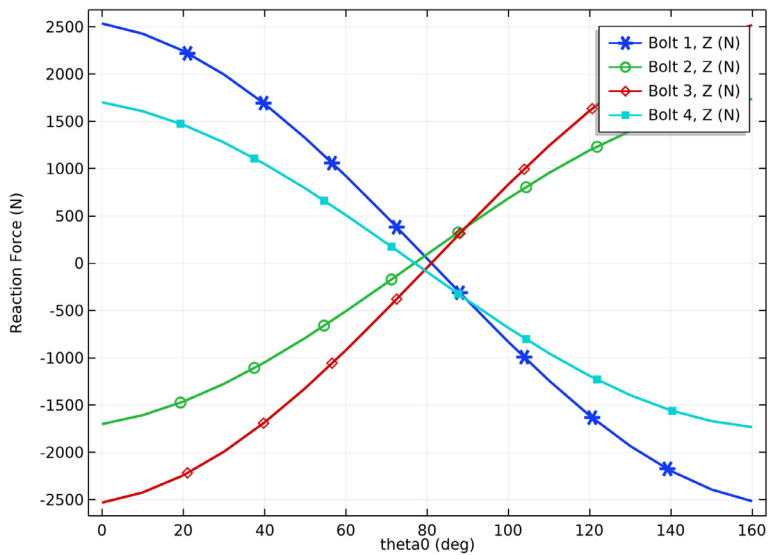
- 1 Under Reaction Force, Y-component click Table Graph 1.
- 2 In the Settings window for Table Graph, locate the Data section.
- 3 In the Columns list, choose Bolt 1, Y (N), Bolt 2, Y (N), Bolt 3, Y (N), and Bolt 4, Y (N).



4 In the Reaction Force, Y-component toolbar, click Plot .



Repeat the steps above to create a similar graph also of the Z-direction reaction forces.



## *Modeling Techniques for Structural Mechanics*

The next set of tutorials build on the previous examples to demonstrate the following structural mechanics modeling techniques: How can I model thermal stress? How can I add an initial strain to the simulation? To get started, go to [Including Initial Strain](#) or [Modeling Thermal Expansion](#).

## Including Initial Strain

---

Initial stresses and strains can be specified in a subnode to a material model. 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 come from another physics interface in the same study, or even be results from another study.




In this example, you add a pin geometry to the bracket assembly. Then you specify an initial strain to simulate that the pin is slightly too short in the axial direction, and then you investigate its effect on the assembly.

### Definitions - Parameters

---

Parameters defining the original length of the pin,  $L_0$ , and the current length,  $L$ , are used to calculate the initial strain. The prestrain is the only load acting on the structure, which is fixed at the bolt holes.

#### *Parameters*

- 1 Browse to the `Structural_Mechanics_Module\Tutorials` folder and double-click to open `bracket_basic.mph`.
- 2 On the Home toolbar click Parameters  and select Parameters  1 .
- In the Parameters table, define a strain value that corresponds to a reduction of the pin length from 215 mm to 214 mm.
- 3 In the Settings window for Parameters, locate the Parameters section.
- 4 In the table, enter the following settings:

NAME	EXPRESSION	VALUE	DESCRIPTION
$L_0$	215[mm]	0.21500 m	Initial pin length
$L$	214[mm]	0.21400 m	Current pin length
InitStrain	$(L-L_0)/L_0$	-0.0046512	Pin strain


### Geometry

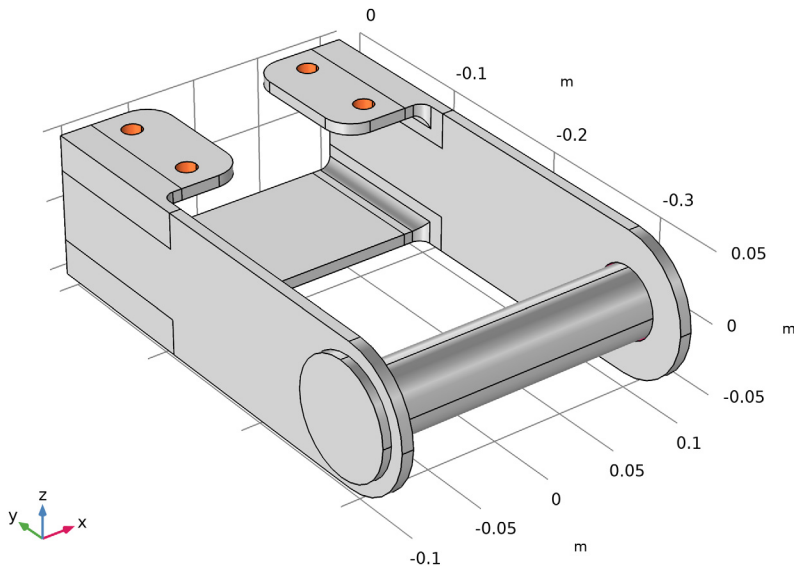
---

Next, the pin geometry is added to the bracket assembly. This is done by importing it into the existing geometry.



**Note:** The location of the files used in this exercise varies based on your installation. For example, if the installation is on your hard drive, the file path might be similar to `C:\Program\COMSOL55\Multiphysics\applications\Structural_Mechanics_Module\Tutorials`.

### Import

- 1 On the Home toolbar click Import . An Import 2 node is added to the Model Builder.
- 2 In the Settings window for Import, locate the Import section. From the Source list, choose COMSOL Multiphysics file.
- 3 Click Browse and go to the model folder `Structural_Mechanics_Module\Tutorials` and double-click the file `bracket_pin.mphbin`.
- 4 Click Import.



### Form Union (fin)

- 1 In the Model Builder under 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 Click the Build Selected button .

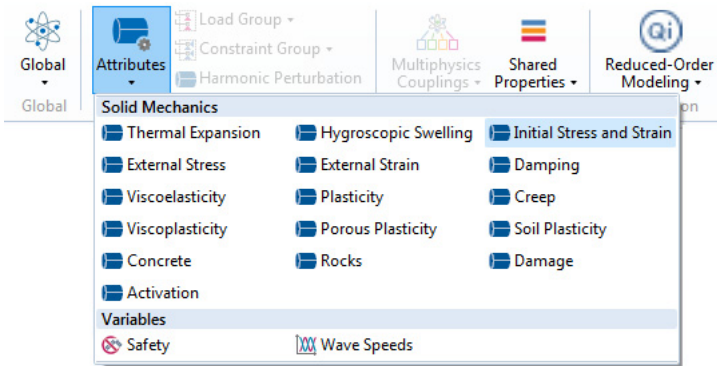
The effect of this operation is that the bracket and pin are now independent objects. The adjacent boundaries are not shared.

## Solid Mechanics (solid)


### Linear Elastic Material 1: Adding Initial Stress and Strain

The initial strain is specified under the Linear Elastic Material node.

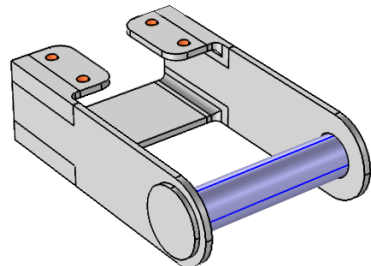
- 1 In the Model Builder window, expand Solid Mechanics and click Linear Elastic Material 1. On the Physics toolbar click Attributes and select Initial Stress and Strain .



As an alternative, you can right-click on the Linear Elastic Material 1 node, and select Initial Stress and Strain from the context menu.

- 2 In the Settings window for Initial Stress and Strain, locate the Domain Selection section. From the Selection list, choose Manual.
- 3 To the right of the Selection list, click the Clear Selection button . Then select only Domain 2 (the pin).

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



- 4 Under the Initial Stress and Strain section, enter `InitStrain` in the first component of the  $\epsilon_0$  table.

▼ Initial Stress and Strain

Initial stress:

$S_0$	0	0	0	$N/m^2$
	0	0	0	
	0	0	0	

Initial strain:

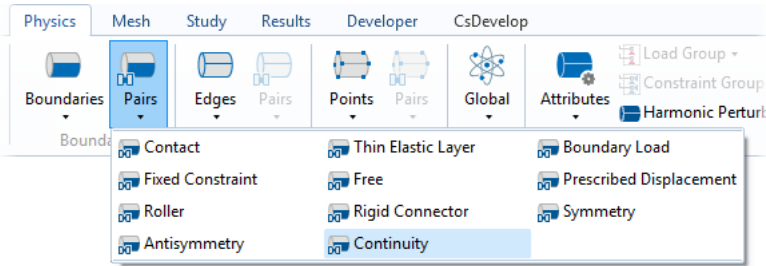
$\epsilon_0$	InitStrain	0	0	1
	0	0	0	
	0	0	0	


### Continuity 1: Connecting the parts

Since you selected Form Assembly in the geometry sequence, the pin and the bracket are separate objects, even though some of their boundaries touch. To connect the objects, you can either use a contact condition, or a continuity condition. Since the pin is compressing the bracket, the boundaries will always be in contact, and a continuity condition is sufficient.

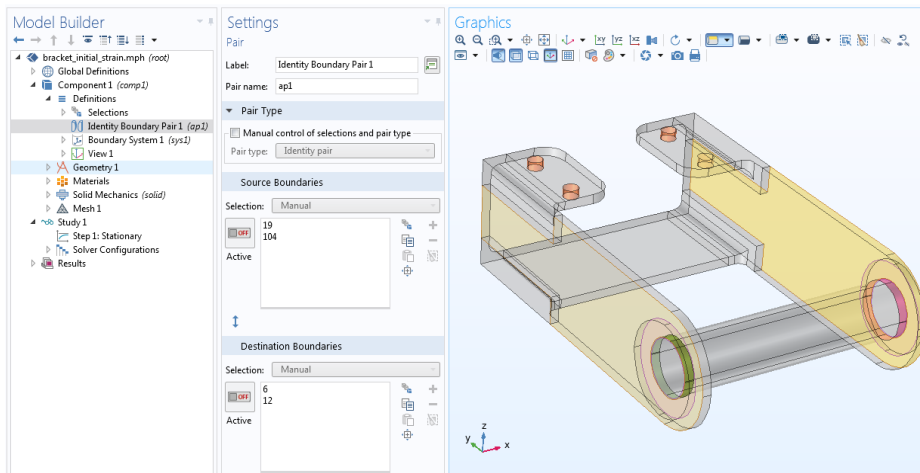
The advantage of using an assembly in this case is that the mesh on the bracket and the mesh on the head of the pin do not have to match.

- 1 On the Physics toolbar click Pairs and select Continuity .







- 2 In the Settings window for Continuity, locate the Pair Selection section. Under Pairs, click Add .
- 3 In the Add dialog box, select Identity Boundary Pair 1 (ap1) and click OK.

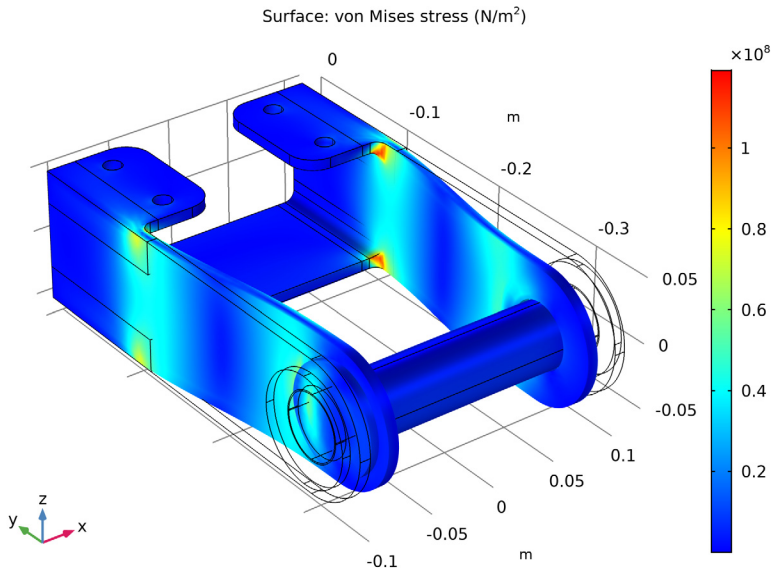
The pair was automatically created by detection of adjacent boundaries when the geometry was built. You can check, and if needed, modify its definition under the Definitions branch in the Model Builder window.



## Add a Study and Compute

- 1 On the Home toolbar click Add Study  to open the Add Study window. Go to the Add Study window.
- 2 Find the Studies subsection. In the Select Study tree under General Studies click Stationary .
- 3 In the Add study window, click Add Study .
- 4 On the Home toolbar, click Add Study again to close the Add Study window.
- 5 On the Home toolbar, click Compute .

The default plot shows the von Mises stress in the bracket.





The results show 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.

## Results


---

You can also plot a principal strain to visualize the resulting 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 as input.


### 3D Plot Group 2

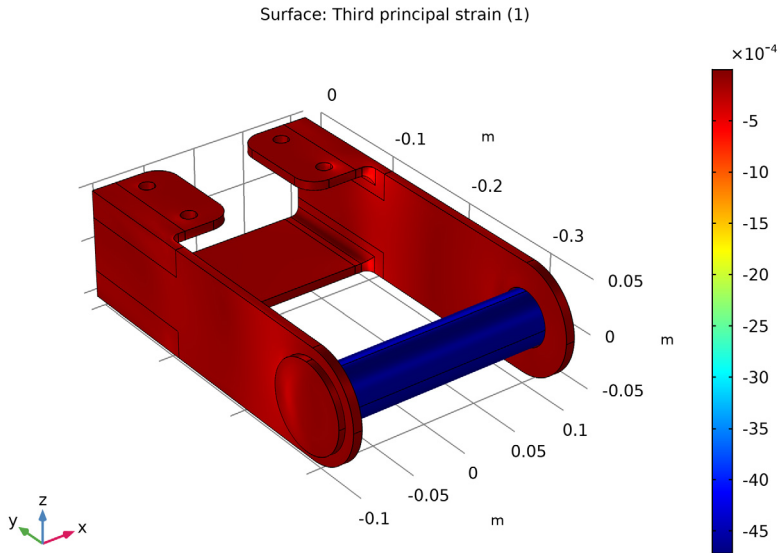
- 1 On the Home toolbar, click Add Plot Group  and choose 3D Plot Group .
- 2 In the Settings window for 3D Plot Group, type Third Principal Strain in the Label text field (replace the default).

### Third Principal Strain

- 1 On the Third Principal Strain toolbar click Surface  (or right-click the Third principal strain node 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 toolbar or Settings window, click Plot  .



## Modeling Thermal Expansion

---

In this example, a temperature field is applied to the bracket and pin assembly and the thermal stresses are calculated.









COMSOL Multiphysics contains physics interfaces for structural analysis as well as thermal analysis. You can define the analyses separately and then simulate the thermal-structure interaction by coupling them using the appropriate variables and terms in the structural analysis equations, or you can add a predefined Thermal Stress interface at the beginning of the setup.

If you were using a Solid Mechanics interface, adding thermal expansion to a material model is as easy as it was to add the initial strain — you would add a Thermal Expansion node to the Linear Elastic Material.



However, an even easier option is to use a predefined multiphysics interface, the Thermal Stress interface. This physics interface contains both the structural and thermal equations along with the coupling, which is included by default.

### Select Physics, Choose the Study, and Load the Geometry

---

- 1 Click the Model Wizard button.  
If COMSOL is already open, you can start the Model Wizard by selecting New  from the File menu and then click Model Wizard .
- 2 On the Select Space Dimension page click 3D .
- 3 In the Select Physics tree under Structural Mechanics, click Thermal Stress .
- 4 Click Add, then click Study .
- 5 In the Select Study tree under General Studies, click Stationary .
- 6 Click Done .
- 7 On the Home toolbar click Import .
- 8 In the Settings window for Import locate the Import section.
- 9 From the Source list, choose COMSOL Multiphysics file.
- 10 Click Browse.
- 11 Browse to the folder `Structural_Mechanics_Module\Tutorials` under the COMSOL installation directory and double-click the file `bracket.mphbin`.
- 12 Click Import.

### *Form Union (fin)*

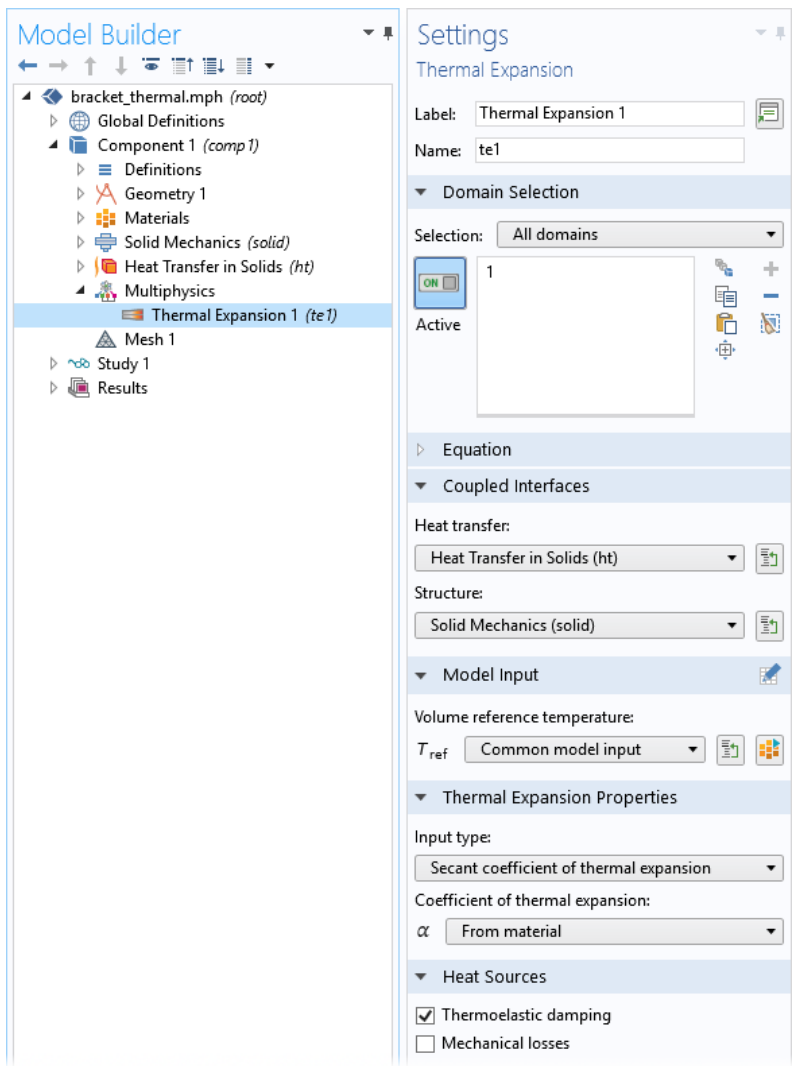
- | In the Model Builder under Component 1 (comp1)>Geometry 1 click Form Union (fin) . On the Settings window, click Build Selected  (or right-click and select the same option).

## **Thermal Stress**


---

When you add a Thermal Stress multiphysics interface, a Solid Mechanics interface, a Heat Transfer in Solids interface, and a Multiphysics node are added to the Model Builder.

Under the Multiphysics node there is one subnode: Thermal Expansion. When you click this node, you see that the coupling is set up automatically after adding the multiphysics interface.






**Note:** The thermal expansion contribution requires both a coefficient of thermal expansion (CTE) and a volume reference temperature, which is the temperature at which there are no thermal strains. The default is that the volume reference temperature is obtained from Default Model Inputs, since this is a quantity which

usually should be shared between different physics interfaces. To see or modify its value, click the Go to Source button . This will move you to the Default Model Inputs node under Global Definitions in the Model Builder. The default value is room temperature, 293.15 K.

## Material and Physics Settings

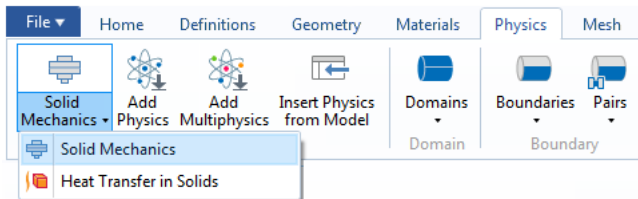
---


Add the material. It contains structural as well as thermal properties.

- 1 On the Home toolbar click Add Material .
- 2 In the Add Material window under Built-In, click Structural steel. Click Add to Component .
- 3 On the Home toolbar click Add Material  again to close the window.


Now set the boundary conditions.

- 1 Make sure that Solid Mechanics is the currently active interface. To do this, either select the physics interface from the list on the Physics toolbar or click any node under Solid Mechanics in the Model Builder.



- 2 On the Physics toolbar click Boundaries  and choose Roller.
- 3 In the Settings window for Roller select Boundaries 17 and 27.

In this analysis, the fixed constraints at the bolt holes are replaced by springs in order to avoid large local stresses caused by the thermal expansion.

- 1 On the Physics toolbar click Boundaries  and choose Spring Foundation.
- 2 In the Settings window for Spring Foundation select Boundaries 18–21 and 31–34.





3 Locate the Spring section. Choose Diagonal from the list below the text field. The text field changes to a matrix input.

4 Enter  $1e7$  in the first two diagonal elements of the  $k_A$  matrix as shown in the figure. This provides a stiffness in the global X and Y directions.


Spring type:			
Spring constant per unit area			
1e7	0	0	N/(m·m³)
0	1e7	0	
0	0	0	

Diagonal

As the Heat Transfer in Solids interface includes a heat balance, the thermal boundary conditions must also be set. A heat flux is prescribed at the two boundaries below the bolt holes. All other boundaries are assumed to be surrounded by air with room temperature, which is modeled by a convection condition.

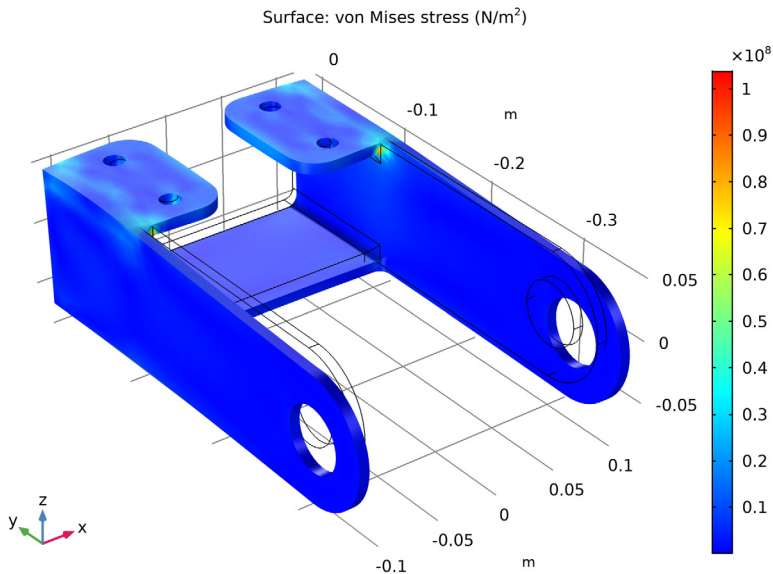
- 1 On the Physics toolbar make sure that Heat Transfer in Solids is the currently active physics interface (or click the Heat Transfer in Solids node in the Model Builder).
- 2 On the Physics toolbar click Boundaries  and choose Heat Flux .
- 3 In the Settings window for Heat Flux, select All boundaries in the Selection list. Then remove boundaries 17 and 27 from the selection.
- 4 In the Heat Flux section, click the Convective heat flux button. Enter 10 in the text field for the Heat transfer coefficient,  $h$ .
- 5 On the Physics toolbar click Boundaries  and choose Heat Flux .
- 6 In the Settings window for Heat Flux, select boundaries 17 and 27.
- 7 In the Heat Flux section, enter  $1e4$  in the General inward heat flux,  $q_0$ , text field.

## Compute and Display Results

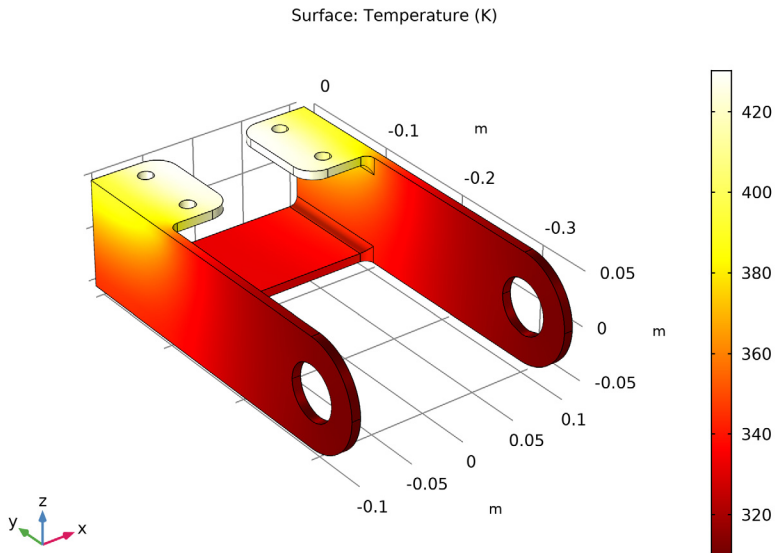
On the Study or Home toolbar click Compute .

Under the Results node, three plot groups are automatically added to show the default results for a structural analysis and a thermal analysis. The first plot group, Stress (solid), shows the von Mises stresses on a scaled deformed geometry. Due

to the boundary conditions and the non-uniform temperature distribution, thermal stresses develop in the structure.



The second default plot group, Temperature (ht), displays the temperature distribution. The temperature is highest where the inward heat flux is prescribed, and decreases as heat is removed by convection from all other boundaries.



As a final step, pick one of the plots to use as a model thumbnail.

- 1 In the Model Builder under Results click any of the plot groups.
- 2 Click the Root node (the first node in the model tree). On the Settings window for Root under Thumbnail, click Set from Graphics Window.

Make adjustments to the image in the Graphics window using the toolbar buttons until the image is one that is suitable to your purposes.

## Examples in the Application Library

This example concludes the introduction.

In the Structural Mechanics Module application library, there are several more detailed examples that use the bracket geometry to extend this tutorial. These show how to:

- Model with special features such as rigid connectors and spring conditions (bracket\_rigid\_connector, bracket\_spring)
- Model thin structures using the Shell interface in combination with the Solid Mechanics interface (bracket\_shell)
- Perform structural dynamics analysis using eigenfrequency extraction, transient analysis, and frequency response analysis (bracket\_eigenfrequency, bracket\_transient, bracket\_frequency)
- Perform a linearized buckling analysis (bracket\_linear\_buckling)
- Perform a contact analysis (bracket\_contact)



- Perform a random vibration analysis (bracket\_random\_vibration)

