

INTRODUCTION TO

Structural Mechanics Module

Introduction to the Structural Mechanics Module

© 1998–2024 COMSOL

Protected by patents listed on www.comsol.com/patents, or see Help > About COMSOL Multiphysics on the File menu in the COMSOL Desktop for less detailed lists of U.S. Patents that may apply. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement (www.comsol.com/sla) 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.

Version: COMSOL 6.3

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries 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 on the COMSOL Access page at www.comsol.com/support/case. Useful links:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/product-update
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/forum
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/videos
- Support Knowledge Base: www.comsol.com/support/knowledgebase
- Learning Center: <https://www.comsol.com/support/learning-center>

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 . . .	16
The Fundamentals: A Static Linear Analysis	25
Parametric Study	65
Modeling Techniques for Structural Mechanics.	76
Including Initial Strain	77
Modeling Thermal Expansion.	84
Examples in the Application Library	94

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 (1D, 1D axisymmetry, 2D, 2D axisymmetry, 3D), shells (2D axisymmetry, 3D), plates (2D), membranes (2D axisymmetry, 3D), trusses (2D, 3D), wires (2D, 3D), beams (2D, 3D), and pipes (2D, 3D). Other examples of capabilities are thermal stress, geometric nonlinearities (large deformations), and structural contact.

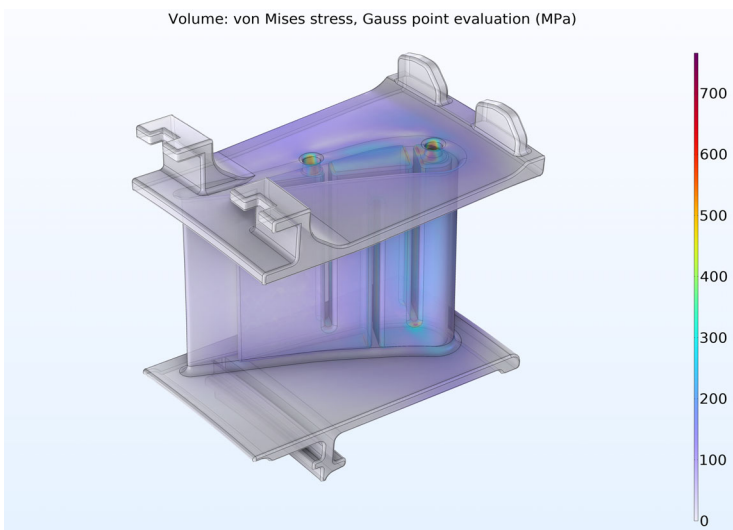


Figure 1: von Mises stresses caused by thermal expansion in a turbine stator model. From the Structural Mechanics Module application library: Thermal Stress Analysis of a Turbine Stator Blade (*turbine_stator*). This model uses both the Structural Mechanics and Heat Transfer Modules.

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: (i) to estimate the stress in the pipe and (ii) to evaluate the performance of the bolted joint. [Figure 2](#) shows the deformation and the von Mises stresses in the pipe.

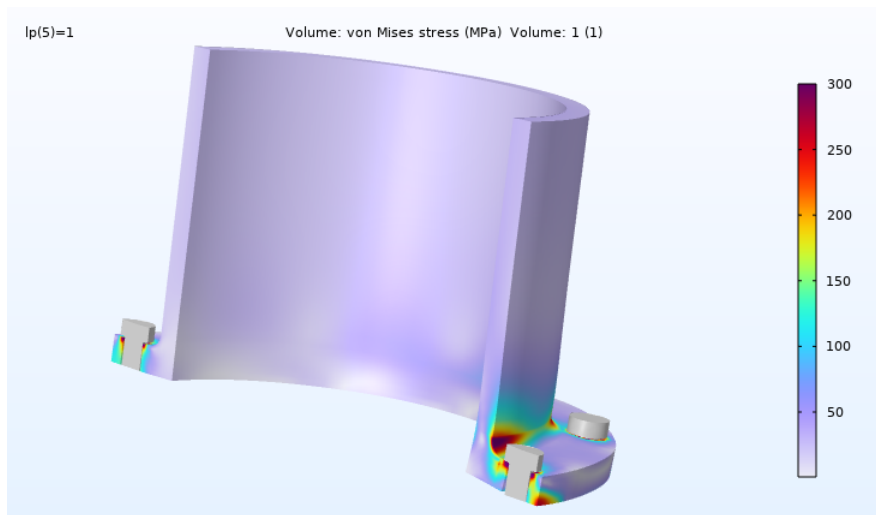


Figure 2: The deformation (exaggerated) and the von Mises stress in the pipe. From the Structural Mechanics Module application library: Prestressed Bolts in a Tube Connection (tube_connection).

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.

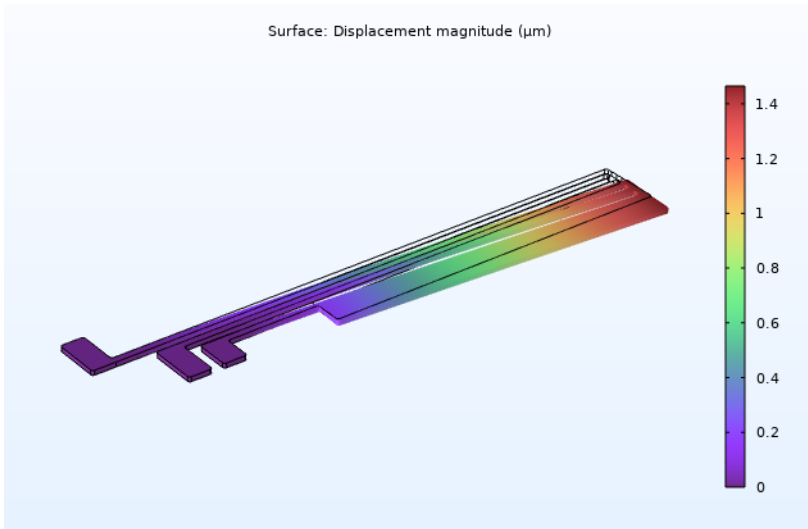


Figure 3: The total displacement in a MEMS device. From the MEMS Module application library: Thermal Actuator (*thermal_actuator_tem*).

Structural analysis is also important outside of traditional structural engineering, for example in the field of biomechanics. 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.

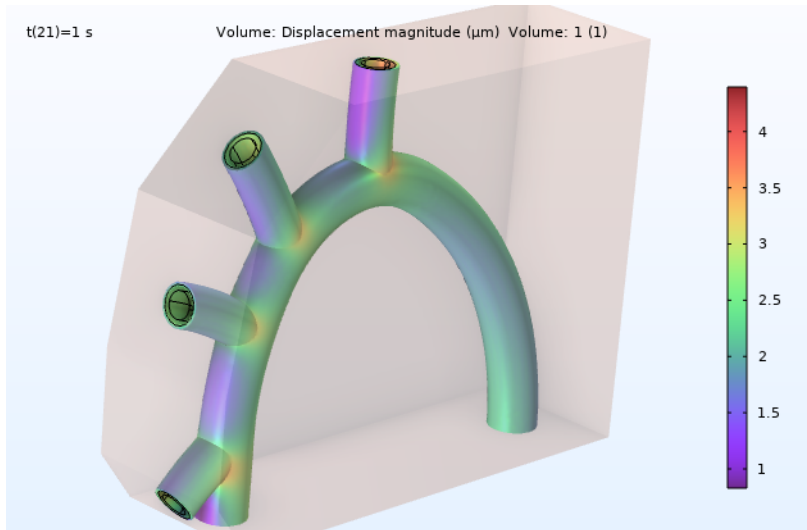


Figure 4: Displacements in the blood vessel. From the Structural Mechanics Module application library: Fluid-Structure Interaction in a Network of Blood Vessels (blood_vessel).

The Structural Mechanics Module Physics Interfaces


The figures below show the physics interfaces that are available in the COMSOL Model Wizard and included with the Structural Mechanics Module and its related products. In addition to the structural mechanics capabilities, the module also has substantial multiphysics capabilities, including electromagnetics, acoustics, fluid flow, heat transfer, poroelasticity, piezoelectricity, and magnetostriction.

- Solid Mechanics (solid)
 - Shell (shell)
 - Layered Shell (lshell)
 - Membrane (mbrn)
 - Beam (beam)
 - Truss (truss)
 - Wire (wire)
 - Multibody Dynamics (mbd)
 - Lumped Mechanical System (lms)
 - Rotordynamics
 - Solid Rotor (rotsld)
 - Solid Rotor, Fixed Frame (srotf)
 - Beam Rotor (rotbrn)
 - Hydrodynamic Bearing (hdb)
 - Solid Rotor with Hydrodynamic Bearing
 - Beam Rotor with Hydrodynamic Bearing
 - Pipe Mechanics (pipem)
 - Fatigue (ftg)
 - Beam Cross Section (bcs)
 - Phase-Field Damage
- Electromagnetics–Structure Interaction
 - Electromechanics
 - Electromechanics, Solid
 - Electromechanics, Shell
 - Electromechanics, Membrane
 - Electromechanics, Boundary Elements
 - Piezoelectricity
 - Piezoelectricity, Solid
 - Piezoelectricity, Layered Shell
 - Piezoelectricity and Pyroelectricity
 - Electrostriction
 - Ferroelectroelasticity
 - Magnetostriction
 - Piezomagnetism
 - Nonlinear Magnetostriction
 - Magnetomechanics
 - Magnetomechanics
 - Magnetomechanics, No Currents
 - Rotating Machinery, Magnetic–Structure Interaction
 - Magnetic–Elastic Interaction in Rotating Machinery
 - Magnetic–Rigid–Body Interaction in Rotating Machinery
 - Piezoresistivity
 - Piezoresistivity, Domain Currents
 - Piezoresistivity, Boundary Currents
 - Piezoresistivity, Shell
 - Piezoresistivity, Layered Shell
- Fluid–Structure Interaction
 - Fluid–Solid Interaction
 - Fluid–Solid Interaction, Fixed Geometry
 - Fluid–Shell Interaction
 - Fluid–Shell Interaction, Fixed Geometry
 - Fluid–Membrane Interaction
 - Fluid–Membrane Interaction, Fixed Geometry
 - Fluid–Multibody Interaction
 - Fluid–Multibody Interaction, Assembly
 - Fluid–Pipe Interaction, Fixed Geometry
 - Thin-Film Damping
 - Solid–Thin-Film Damping
 - Shell–Thin-Film Damping
 - Viscoelastic Flow
 - Fluid–Solid Interaction
 - Fluid–Solid Interaction, Fixed Geometry
 - Conjugate Heat Transfer
 - Fluid–Solid Interaction
 - Two-Phase Flow, Phase Field
 - Fluid–Solid Interaction
 - Fluid–Solid Interaction, Fixed Geometry
- Poroelasticity
 - Poroelasticity, Solid
 - Poroelasticity, Layered Shell
 - Unsaturated Poroelasticity
- Thermal–Structure Interaction
 - Thermal Stress, Solid
 - Thermal Stress, Shell
 - Thermal Stress, Layered Shell
 - Thermal Stress, Membrane
 - Joule Heating and Thermal Expansion
 - Thermoelasticity

Figure 5: The 3D component physics interfaces available with the Structural Mechanics Module. Several of the physics interfaces require additional licenses.

A short description of the main 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 1D and 2D, plane stress, plane strain, and generalized plane strain formulations are available. In 2D axisymmetry, there are formulations with and without twist.

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
 -  Boundary Mode Analysis
 -  Eigenfrequency, Prestressed
 -  Frequency Domain, Prestressed
 -  Frequency Domain, Modal
 -  Linear Buckling
 -  Random Vibration (PSD)
 -  Response Spectrum
 -  Time Dependent, Modal
- ▼  Optimization
 -  Shape Optimization, Frequency Domain
 -  Shape Optimization, Stationary
 -  Topology Optimization, Frequency Domain
 -  Topology Optimization, Stationary
- ▼  More Studies
 -  Frequency Domain, Prestressed, Modal
 -  Frequency Domain, AWE Reduced-Order Model
 -  Frequency Domain, Modal Reduced-Order Model
 -  Time Dependent, Modal Reduced-Order Model

Figure 7: Study types for 3D Solid Mechanics.

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, volume-preserving and pressure-dependent 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 special material models 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, including random vibration fatigue.

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 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 mechanical 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. Stresses in


welds can be evaluated for connected shells. Special features for connecting shells through a large number of spot welds or fasteners are available. 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, hyperelasticity, creep, and viscoplasticity.

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 possess any bending stiffness. Thin films and fabrics are structures suited for modeling with the Membrane interface. In order to be stable, a membrane must be in tension; if not, it will wrinkle. A special formulation in the Membrane interface makes it possible to model also wrinkling. 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.

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.

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 analyses. The interface can also be used for detailed stress evaluation in beams, given the forces and moments acting on the section. With the 3D version

of the same interface, it is also possible to plot the full 3D representation of the stresses in a beam, given the section forces.


TRUSS

The Truss interface () can be used to model slender structures that can only sustain axial forces. Trusses are modeled using Lagrangian shape functions, which allow the specification of small strains as well as Green–Lagrange strains for large deformations. 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 and shape memory alloys.


PIPE MECHANICS


The Pipe Mechanics interface () is intended for stress and deformation analysis in pipe systems, 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. There are also pipe-specific formulations for stress evaluation and stiffness correction in curved pipes.


WIRE


The Wire () interface can be used for modeling cables, both prestressed and exposed to gravity forces (sagging cables). Wires are similar to truss elements, but cannot sustain any compressive forces.



OTHER STRUCTURAL MECHANICS INTERFACES



The Thermal Stress, Solid multiphysics 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 interfaces are available also for shells and membranes.



The Joule Heating and Thermal Expansion multiphysics 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 multiphysics 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 Ferroelectroelasticity () and Electrostriction () multiphysics interfaces combine Solid Mechanics and Electrostatics together with the constitutive relationships required to model linear and nonlinear ferroelectric materials with polarization saturation and possible hysteresis. Many piezoelectric materials exhibit such nonlinear ferroelectroelastic behavior at large applied electric fields. This interfaces requires the AC/DC module or the MEMS module.

The Piezomagnetism () and Nonlinear Magnetostriction () multiphysics interfaces combine the Solid Mechanics and Magnetic Fields interfaces to model linear and nonlinear magnetostrictive materials, respectively. The interfaces require the AC/DC module.


A similar multiphysics interface, Magnetomechanics (), combines Solid Mechanics and Magnetic Fields interfaces together with a moving mesh functionality to model the deformation of magnetically actuated structures. For magnetomechanical interaction where some parts are rotating, there is a dedicated multiphysics interface, Magnetic–Elastic Interaction in Rotating Machinery ()


FLUID FLOW



There are several Fluid–Structure Interaction (FSI) multiphysics interfaces. 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 the 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 in an Eulerian description and a spatial frame with solid mechanics formulated in 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, but 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. Both the fluid loading on the structure and the structural velocity transmission to the fluid can be taken into account.

There are also similar interfaces for coupling shells and membranes to fluid flow, as well as a special interface coupling the Pipe Flow interface to the Pipe Mechanics interface.












The Fluid–Solid Interaction, Conjugate Heat Transfer interface (), combines the FSI functionality with heat transfer in the fluid and solid domains.









There are also multiphysics interfaces targeted at porous flow, such as the Poroelasticity, Solid () interface. It combines Solid Mechanics with Darcy's Law, to provide a bidirectional coupling between the flow through an elastic matrix and the deformation of the matrix.




The interaction between structures and thin liquid or gas layers can be modeled using the Solid Thin-Film Damping () and Shell Thin-Film Damping () multiphysics interfaces.




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
 Acoustics				
 Elastic Waves				
Elastic Waves, Time Explicit		elte	3D, 2D, 2D axisymmetric	time dependent
Piezoelectric Waves, Time Explicit		—	3D, 2D, 2D axisymmetric	time dependent
 Chemical Species Transport				
Transport in Solids		ts	all dimensions	stationary; time dependent
 Fluid Flow				
 Fluid-Structure Interaction				
Fluid–Solid Interaction ²		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Fluid–Shell Interaction ²		—	3D, 2D axisymmetric	stationary; time dependent
Fluid–Shell Interaction, Fixed Geometry		—	3D, 2D axisymmetric	stationary; stationary, one way; time dependent; time dependent, one way








PHYSICS INTERFACE		ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
	Fluid–Membrane Interaction ²		—	3D, 2D axisymmetric	stationary; time dependent
	Fluid–Membrane Interaction, Fixed Geometry ²		—	3D, 2D axisymmetric	stationary; stationary, one way; time dependent; time dependent, one way
	Fluid–Pipe Interaction, Fixed Geometry ^{2,7}		—	3D, 2D	stationary; time dependent
	Conjugate Heat Transfer, Fluid–Solid Interaction ²		—	3D, 2D, 2D axisymmetric	stationary; time dependent;
	Fluid–Solid Interaction, Viscoelastic Flow ^{2,10}		—	3D, 2D, 2D axisymmetric	time dependent; time dependent with phase initialization
	Fluid–Solid Interaction, Two-Phase Flow, Phase Field ^{2,6}		—	3D, 2D, 2D axisymmetric	time dependent; time dependent with phase initialization
	Solid–Thin-Film Damping ²		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; time dependent; 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)
	Shell–Thin-Film Damping ²		—	3D, 2D axisymmetric	stationary; eigenfrequency; time dependent; 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)










PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
 Structural Mechanics				
Solid Mechanics ¹		solid	3D, 2D, 2D axisymmetric, 1D, 1D axisymmetric	stationary; eigenfrequency; eigenfrequency, prestressed; mode analysis; time dependent; time dependent, 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 pretension
Shell		shell	3D, 2D axisymmetric	stationary; eigenfrequency; eigenfrequency, prestressed; time dependent; time dependent, 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, 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	3D, 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, 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
Wire		wire	3D, 2D	stationary; eigenfrequency; eigenfrequency, prestressed; time dependent; time dependent, 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	
Membrane		mbrn	3D, 2D axisymmetric	stationary; eigenfrequency; eigenfrequency, prestressed; time dependent; time dependent, 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)	
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)	
Phase Field Damage ^{2,11}		—	3D, 2D, 2D axisymmetric	stationary; time dependent	
 Thermal–Structure Interaction					
	Thermal Stress, Solid ²		—	3D, 2D, 2D axisymmetric	stationary; time dependent
	Thermal Stress, Shell ^{2,4}		—	3D, 2D axisymmetric	stationary; time dependent
	Thermal Stress, Membrane ^{2,4}		—	3D, 2D axisymmetric	stationary; time dependent
	Joule Heating and Thermal Expansion ²		—	3D, 2D, 2D axisymmetric	stationary; time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
 Piezoelectricity				
Piezoelectricity, Solid ²		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; eigenfrequency, prestressed; time dependent; time dependent, modal; frequency domain; frequency domain, modal; frequency domain, prestressed; frequency domain, prestressed, modal; small-signal analysis, frequency domain; linear buckling
Piezoelectricity, Layered Shell ^{2,5}		—	3D	stationary; eigenfrequency; time dependent; frequency domain
 Magnetomechanics				
Piezomagnetism ^{2,3}		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; time dependent; frequency domain; small-signal analysis, frequency domain; eigenfrequency, prestressed; frequency domain, prestressed
Nonlinear Magnetostriction ^{2,3}		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; time dependent; frequency domain; small-signal analysis, frequency domain; eigenfrequency, prestressed; frequency domain, prestressed
Magnetomechanics ^{2,3}		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; time dependent; frequency domain; small-signal analysis, frequency domain; eigenfrequency, prestressed; frequency domain, prestressed

PHYSICS INTERFACE		ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Magnetomechanics, No Currents ^{2,3}		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; time dependent; frequency domain; small-signal analysis, frequency domain; eigenfrequency, prestressed; frequency domain, prestressed	
Magnetic–Elastic Interaction in Rotating Machinery ^{2,3}		—	3D, 2D	stationary; time dependent; frequency domain;	
 Electrostriction					
Ferroelectroelasticity ^{2,9}		—	3D, 2D, 2D axisymmetric	stationary; time dependent; frequency domain	
Electrostriction ^{2,9}		—	3D, 2D, 2D axisymmetric	stationary; time dependent; frequency domain	
 Piezoresistivity					
Piezoresistivity, Shell ^{2,12}		—	3D	stationary; time dependent; time dependent, modal; frequency domain; frequency domain, modal; small-signal analysis, frequency domain; eigenfrequency, prestressed; frequency domain, prestressed	
Piezoresistivity, Layered Shell ^{2,5,12}		—	3D	stationary; time dependent; time dependent, modal; frequency domain; frequency domain, modal; small-signal analysis, frequency domain; eigenfrequency, prestressed; frequency domain, prestressed	
 Poroelasticity					

PHYSICS INTERFACE		ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
	Unsaturated Poroelasticity ^{2,8}		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Δu Mathematics					
 Moving Interface					
	Phase Field in Solids		pfs	3D, 2D, 2D axisymmetric	stationary, time dependent
¹ This physics interface is included with the core COMSOL Multiphysics software but has added functionality for this module.					
² This physics interface is a predefined multiphysics coupling that automatically adds all the physics interfaces and coupling features required.					
³ Requires the addition of the AC/DC Module.					
⁴ Requires the addition of the Heat Transfer Module.					
⁵ Requires the addition of the Composite Materials Module.					
⁶ Requires the addition of the CFD Module, or the Polymer Flow, or the Microfluidics Module.					
⁷ Requires the addition of the Pipe Flow Module.					
⁸ Requires the addition of the Porous Media Flow Module.					
⁹ Requires the addition of the AC/DC Module or the MEMS Module.					
¹⁰ Requires the addition of the Polymer Flow Module.					
¹¹ Requires the addition of the Nonlinear Structural Materials Module or the Geomechanics Module.					
¹² Requires the addition of the MEMS 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 imposing 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 and the loading are shown in [Figure 8](#).

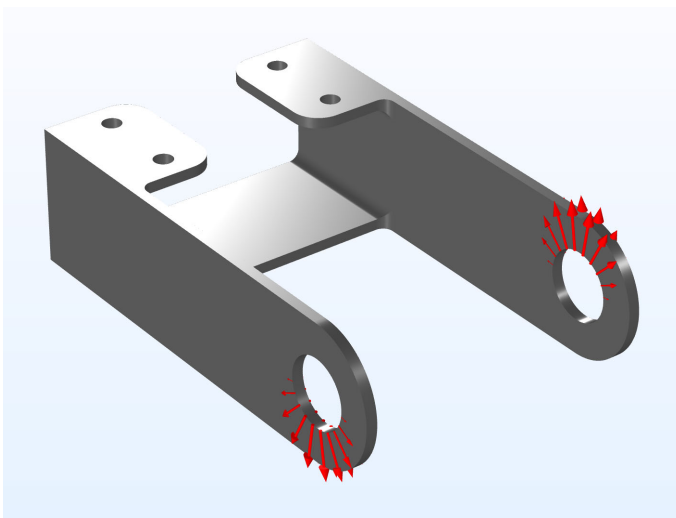


Figure 8: The geometry of the bracket together with the load distribution.

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 α is the angle from the direction of the load resultant. One of the arms is loaded upward, and the other downward.

This example contains several steps that are not strictly necessary for the task. Rather, they are intended to showcase useful functionality. Examples of this are the coloring of various selections, addition of certain plots, and so on.



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 also 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 a 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 the Add button (or right-click on the Solid Mechanics (solid) node), and select + Add Physics.
- 5 Click Study .
- 6 In the Select Study tree under General Studies, click Stationary .
- 7 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. You can also easily add new parameters when needed.

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

- 2 In the Settings window for Parameters, locate the Parameters section.
- 3 In the table, enter the following settings:


NAME	EXPRESSION	VALUE	DESCRIPTION
Fh	800[N]	800 N	Force per hole
dHole	50[mm]	0.05 m	Hole diameter
tArm	8[mm]	0.008 m	Thickness of arm
p0	Fh/(dHole*tArm*pi/4)	2.5465E6 N/m ²	Peak load intensity

The third column in the table is filled out automatically. Note that you can enter data in any unit; it will automatically be converted to the base units of your model. You can choose unit system under the top level node in the model tree.

The value of the peak load intensity, p0, is computed based on the resultant load applied to each hole. Creating parameters using expressions containing other parameters is a powerful tool.

In any node in the Model Builder, you can add comments to explain the settings. Here, you may want to explain the expression for the peak load intensity.

You add and modify comments in the Properties tab for a certain node.

- 1 To open the Properties tab, right-click the Parameters 1 node, and select Properties and Comments .
- 2 In the Settings window for Properties, locate the Node Properties section.
- 3 In the Comments edit field enter the following text:

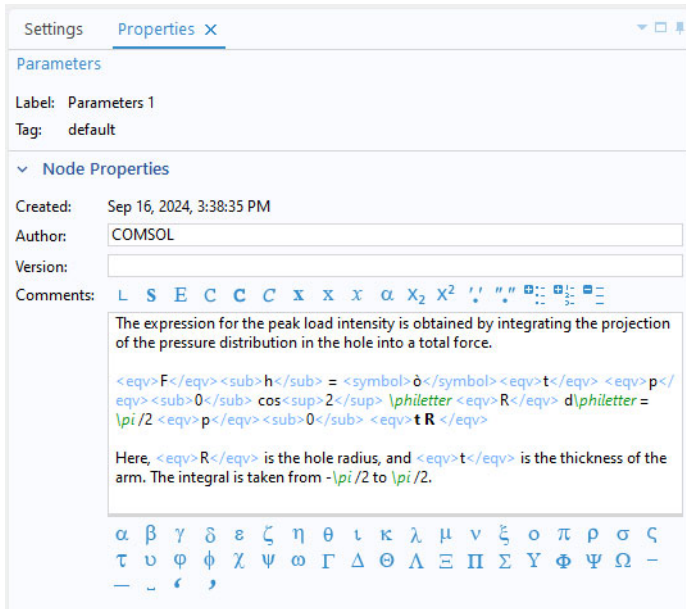
The expression for the peak load intensity is obtained by integrating the projection of the pressure distribution in the hole into a total force.

$$F_h = \int_{-\pi/2}^{\pi/2} p_0 \cos^2 \phi R d\phi = \pi/2 p_0 t R$$

Here, R is the hole radius and t is the thickness of the arm. The integral is taken from $-\pi/2$ to $\pi/2$.

To improve the visualization, enrich the text fonts with subscripts, superscripts, or equation variable formatting. This can be done by selecting one or more

characters and then clicking the associated button above the text field. In the figure you can see how the text looks after formatting.



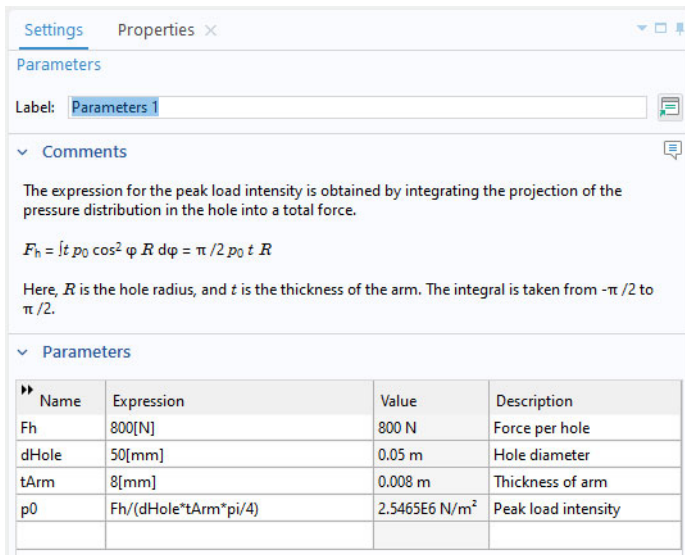
If you want to directly paste the final formatted expression, copy the following text:

The expression for the peak load intensity is obtained by integrating the projection of the pressure distribution in the hole into a total force.

$$F_h = \int_{-\pi/2}^{\pi/2} p \cos^2 \phi \, d\phi = \pi R t \int_{-\pi/2}^{\pi/2} p \cos^2 \phi \, d\phi$$

Here, R is the hole radius, and t is the thickness of the arm. The integral is taken from $-\pi/2$ to $\pi/2$.

Once you have added a comment for a node in the model, it will be visible in the Settings tab.



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).

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 Files\COMSOL\COMSOL63\Multiphysics\applications\Structural_Mechanics_Module\Tutorials.


1 In the Home toolbar click Import .

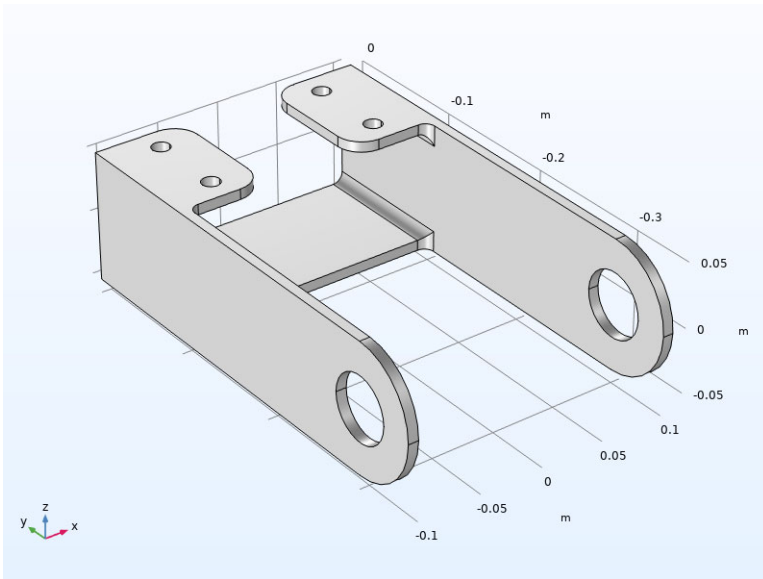
Note: On Linux and Mac, the Home toolbar refers to the specific set of controls near the top of the Desktop.

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

3 From the Source list, choose COMSOL Multiphysics file.

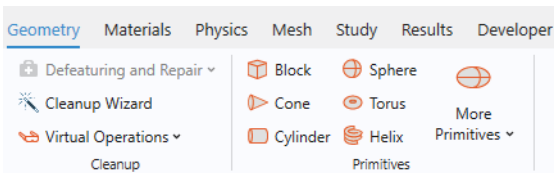
4 Click Browse .

- 5 Browse to the folder `Structural_Mechanics_Module\Tutorials` under the COMSOL installation directory and double-click the file `bracket.mphbin`.
- 6 Below the filename, click Import .



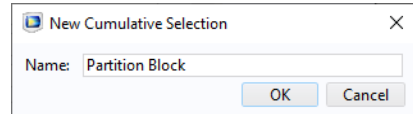
It is possible to automatically create a free tetrahedral mesh covering the whole component. Such a strategy is however not optimal for the large flat regions. For this reason, you will partition the geometry so that you can create a better mesh. For a small linear problem like this, it would still be more efficient just to use a rather fine tetrahedral mesh, but the approach shown here is useful for more general cases.

- 1 In the Geometry toolbar, select Block .



- 2 In the Settings window for Block, locate the Selections of Resulting Entities section.
- 3 Find the Cumulative selection subsection. Click New.

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



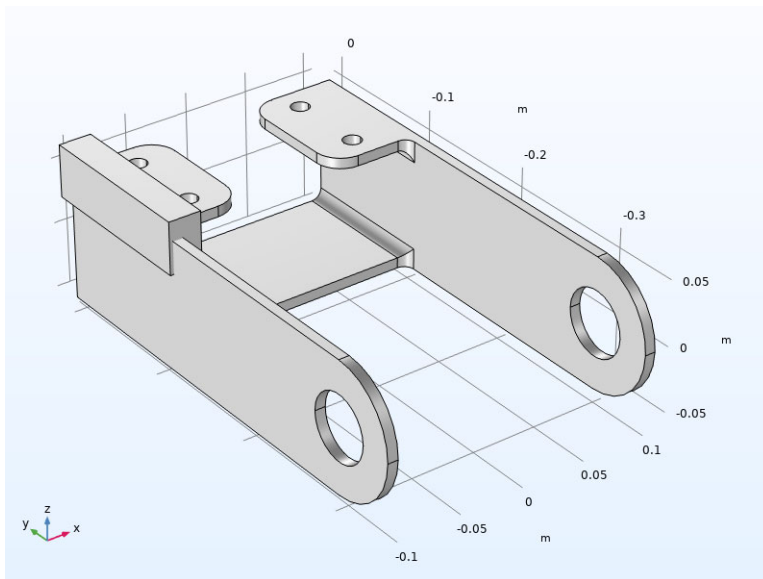
5 Click OK.


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

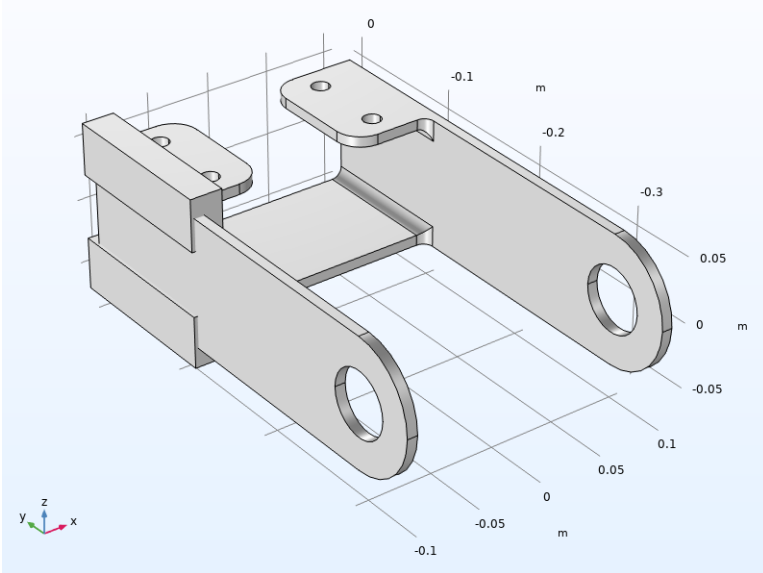
7 Click Build Selected 

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.





8 In the Geometry toolbar, click Transforms  and choose Mirror .


- 9 In the Settings window for Mirror, locate the Input section. From the Input objects list, choose Partition Block.
- 10 Select the Keep input objects checkbox.
- 11 Locate the Selections of Resulting Entities section. Find the Cumulative Selection subsection. From the Contribute to list, choose Partition Block. Click Build Selected .

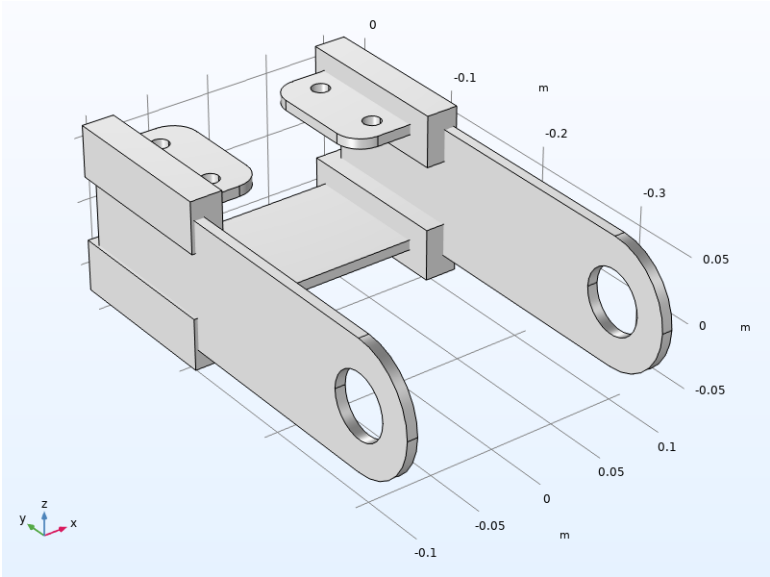


Since the x-y plane happens to be the default mirror plane, you did not have to change any settings to specify it.



Create the second mirror copy.


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

18 Click Build Selected  .

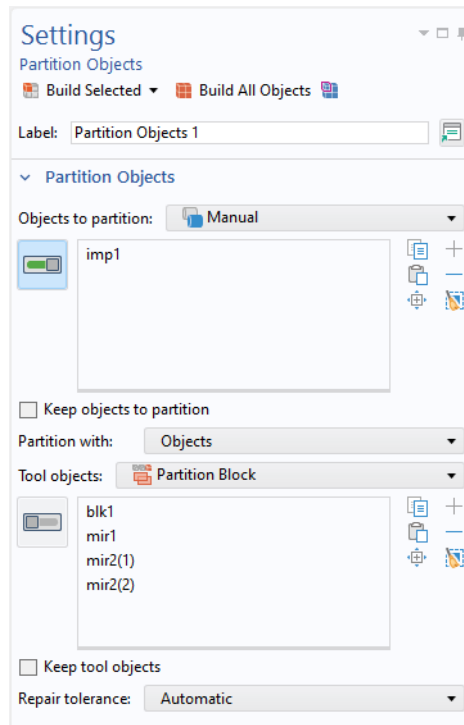



Partition objects

- 1 In 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.

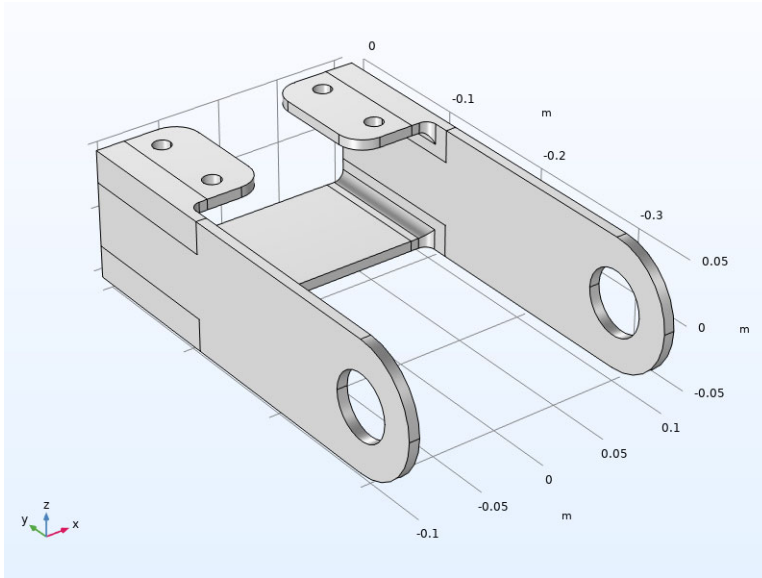
Note: There are many ways to select geometric entities. When you know the geometric entity to add, as in these exercises, you can click the Paste Selection button  and enter the information in the Selection text field. In this case, ‘imp1’ is the whole bracket, so quickest way to select it is to click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

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



- 3 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)


| In the Model Builder under Component 1 (comp1) > Geometry 1, locate Form Union (fin)  and click the Build All  button in the Settings window.

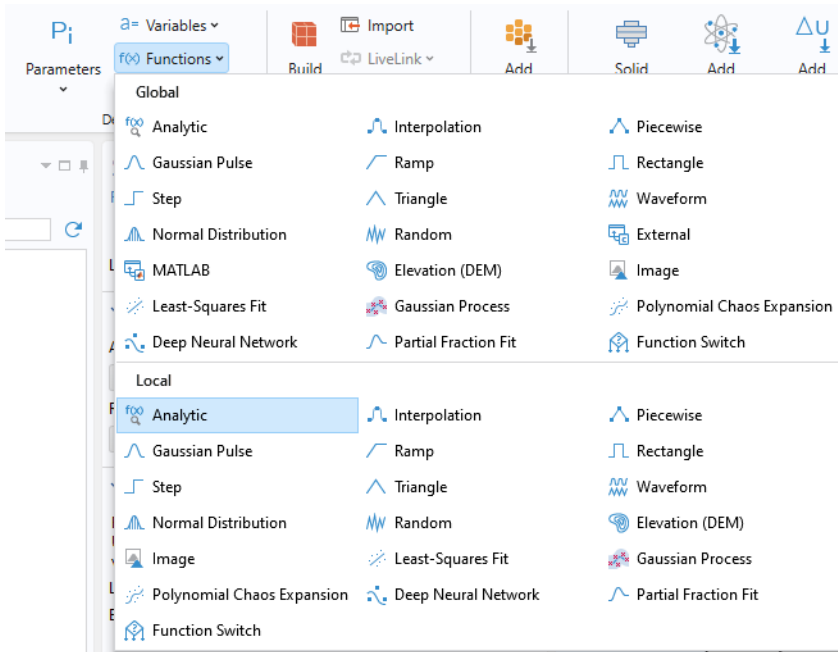
The Form Union node determines how the objects will be considered in the analysis. By using the default setting, Form a union, any adjacent objects will be bonded, and the interior boundaries assume continuity. If Form an assembly were selected instead, each object would be considered as different parts of an assembly and 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

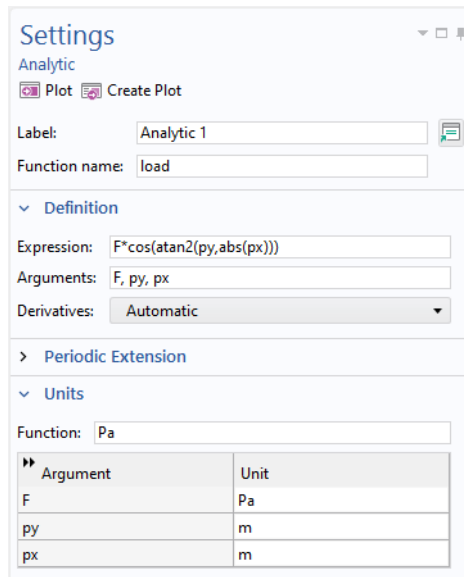
Here you want to define an expression for the load applied to the load-carrying holes. Assume the load distribution to be defined by a trigonometric function.

Analytic 1 (an1)

- In the Home toolbar, click Functions and under Local choose Analytic .




- In the Settings window for Analytic, type 100 in the Function Name text field.
- 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).
- In the Arguments text field, type F, py, px . These values will be used when the function is referenced later, and represent the peak load intensity, and coordinates relative to the center of the hole.
- Locate the Units section. In the Function text field, type Pa.
- In the table, enter Pa, m, and m as shown in the figure.

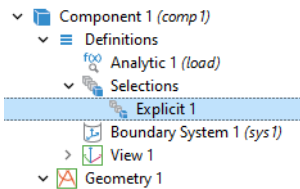



Regardless of whether the geometry is treated as a single entity or as an assembly, you still may need to access different parts of it for definitions, such as multiple specifications of similar boundary conditions. There are different ways in which you can create selections, for example by explicit 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.

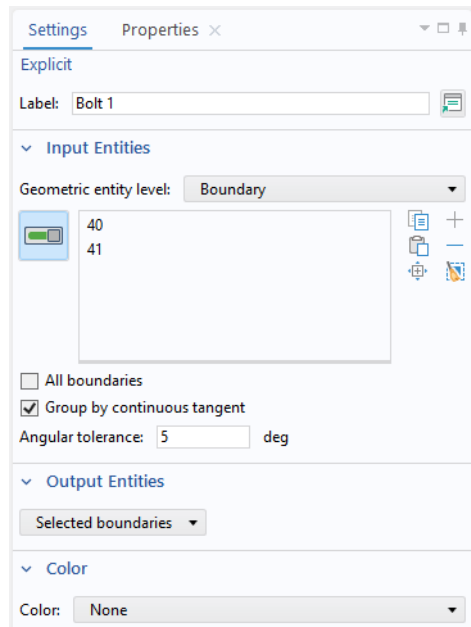
In this example, one selection contains the bolt domains, while other selections contain the boundaries of the pin holes that carry load.

Bolt 1


- 1 In the Definitions toolbar, click Explicit . An Explicit node is added under Definitions > Selections.





- 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 In 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. Note how it is highlighted in the graphics window.
- 6 Select the Group by continuous tangent checkbox. Boundary 40 is automatically added to the selection list.



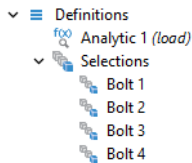
Bolt 2, Bolt 3, and Bolt 4

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

- 1 In the Definitions toolbar, click Explicit  three times.
- 2 In the Model Builder click each Explicit node  to open the Settings window.
- 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 checkbox 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

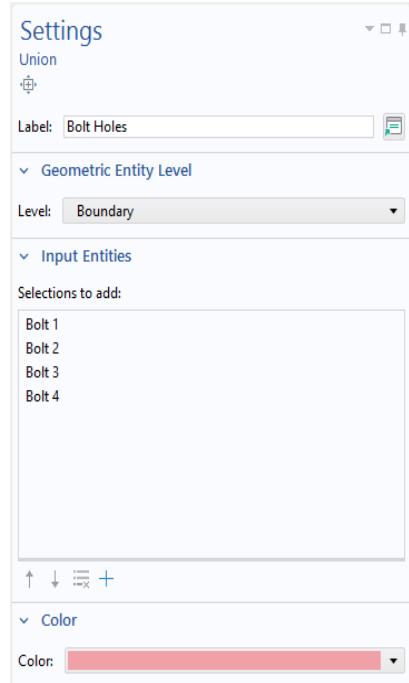
The node sequence under Definitions should look like this at this point:



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


Bolt Holes

- 1 In 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 Input Entities section. Under Selections to add, click the Add button .
- 5 In the Add dialog, in the Selections to add list, choose Bolt 1, Bolt 2, Bolt 3, and Bolt 4.
- 6 Click OK.
- 7 Locate the Color section. On Windows, select the eighth color in the first row of the palette (a pink color). On other platforms, choose Color 8 from the Color list.



Note: This assumes default settings for color themes.
Create selections for the two holes carrying the load.

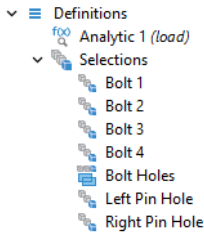
Left Pin Hole

- 1 In 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 checkbox.
- 6 Locate the Color section. On Windows, select the ninth color in the first row of the palette (a light blue-gray color). On other platforms, choose Color 9 from the Color list.

Right Pin Hole


Repeat the steps above, but use the label Right Pin Hole, select Boundary 75, and use the second color in the second row (a light brown color) on Windows

platform, or choose Color 12 on other platforms. Now the sequence under Definitions should look like this:





Pin Hole

Add a selection that combines the pin hole boundaries on both sides.

- 1 In the Definitions toolbar, click Union .
- 2 In the Settings window for Union, type Pin 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, in the Selections to add list, choose Left Pin Hole and Right Pin Hole.
- 6 Click OK.




Bolt Hole Edges

Add one more selection to be used during mesh generation.

- 1 In 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, 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. Note how the edges of the bolt holes are highlighted.
- 8 In 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 In the Materials 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 In the Materials toolbar click Add Material  again to close the window.



Solid Mechanics

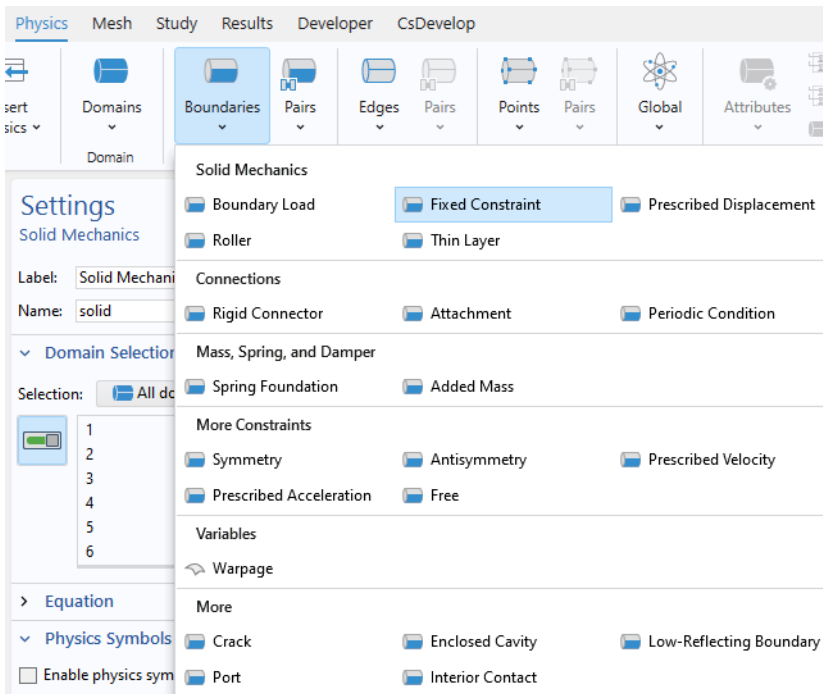
Now it is time to define the physics settings, such as material models, loads, and constraints. Initially, the analysis was specified to be stationary, using the classical equations associated with solid mechanics.

By default, the Solid Mechanics interface assumes that the participating material models are linear elastic, which is appropriate for this example.

Fixed Constraint 1

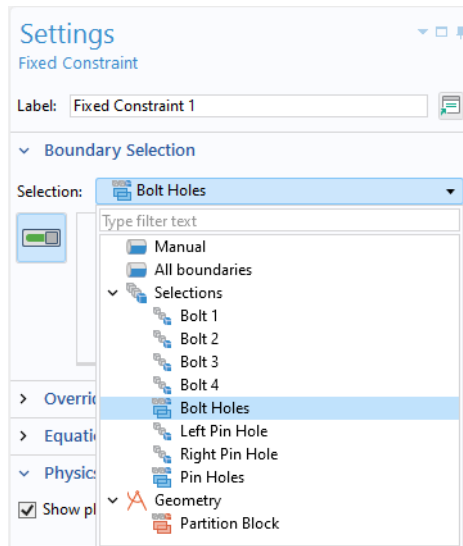
Assume that the boundaries of the bolt holes are perfectly constrained.

1 In the Physics toolbar click Boundaries  and choose Fixed Constraint .






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

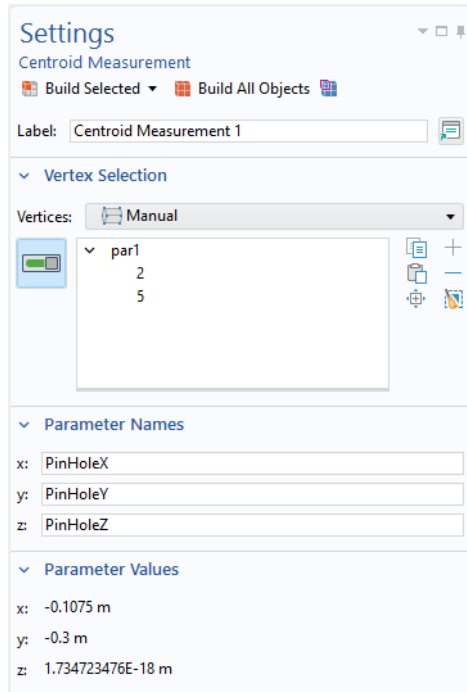
3 From the Selection list, choose Bolt Holes.



Centroid Measurement I



Before adding the load, find the location of the pin hole center line. You can directly create parameters for the coordinates to be used in the load expression.

- 1 In the Geometry toolbar, click Measurements  and choose Centroid Measurement .
- 2 On the object par1, select Point 2 and 5 only.
- 3 Locate the Parameter Names section. In the x text field, type PinHoleX.
- 4 Then type PinHoleY and PinHoleZ, in the y and z text field respectively.
- 5 In the Settings window for Centroid Measurement, click Build Selected .



Boundary Load I

The load is acting in the direction normal to the boundaries. To achieve that, use the built-in boundary coordinate system.

- 1 In the Physics toolbar click Boundaries  and choose Boundary Load .
- 2 Locate the Boundary Selection section. From the Selection list, choose Pin Holes.
- 3 Locate the Coordinate System Selection section. From the Coordinate system list, choose Boundary System 1 (sys1). The input of the load is now interpreted in this coordinate system, and you can directly access the normal to the boundaries.

4 Locate the Force section. Specify the \mathbf{F}_A vector as:

0	t1
0	t2
load(-p0,Y-PinHoleY,Z)*(X*Z>0)	n



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

The previously defined function 'load()' is referenced in order to provide the load distribution. Note how Boolean expressions like $X*Z>0$ can be used to limit the part of the hole where the load is active: upper half of the right pin hole or lower half of the left pin hole. It would also have been possible to include that part of the logic in the function 'load()' by supplying the X coordinate as a fourth argument. Or, you could simply have used one Boundary Load feature for each hole, and selected only the loaded part of the hole in each of them.


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 Mesh toolbar, click More Generators  and choose 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 that is swept through the thin flat parts, and then use a free tetrahedral mesh in the parts with a more complex geometry. Note that the transition between the two types of elements is automatic. A layer of pyramid elements will be generated there.

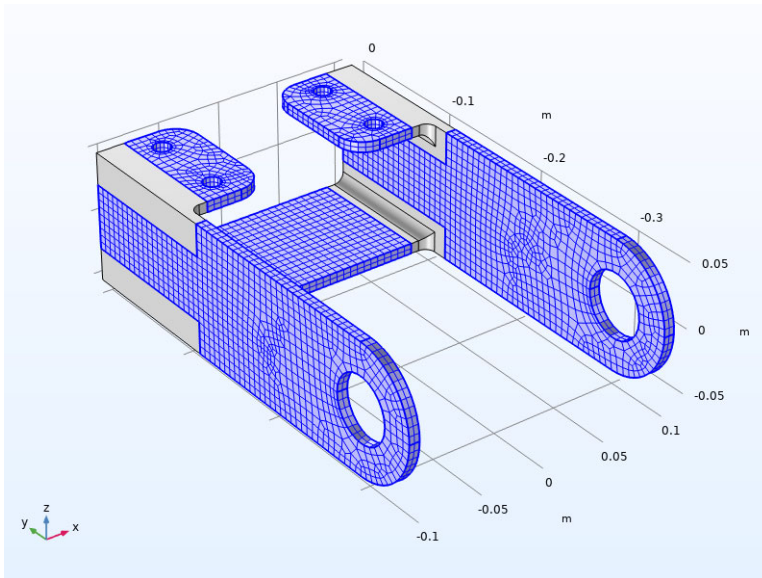
Swept 1

- 1 In 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.

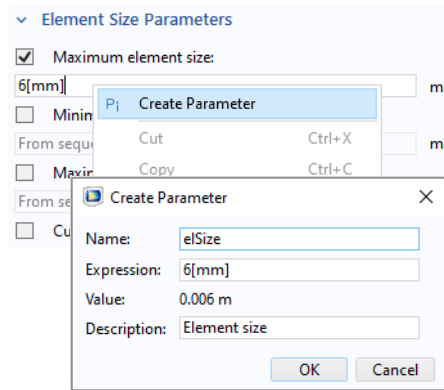
Size 1

- 1 In the Model Builder, right-click Swept 1 and choose Size .
- 2 In the Settings window for Size, locate the Element Size section. Click the Custom button.
- 3 Locate the Element Size Parameters section. Select the Maximum element size checkbox. In the associated text field, type 6[mm].
- 4 Click Build Selected  to generate the swept mesh in the thin parts.



You are going to use the element size later in the modeling. It is then a good idea to convert it into a parameter. It is possible to add new parameters on the fly from any input field that supports parameters. You can right-click in an empty text field to do that, but you can also convert an existing value into a parameter.

- 5 In the Maximum element size text field, select all text. Right-click and choose Create Parameter.
- 6 In the Create Parameter dialog, type e1Size in the Name text field, and Element size in the Description text field.
- 7 Click OK.




The value in the expression for the element size is now automatically substituted by the new parameter name.

The parameter was added to the list in the Parameters 1 node, and can later be modified there if necessary. You can also select any parameter in a text field, right-click it, and immediately modify its value or description.


Size

- 1 In the Model Builder window, under 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


- 1 In the Mesh toolbar, click Free Tetrahedral .
- The default selection is Remaining, that is all not yet meshed domains.


Size 1

- 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.
- 3 Click the Custom button.
- 4 Locate the Element Size Parameters section.
- 5 Select the Maximum element size checkbox. In the associated text field, type e1Size.

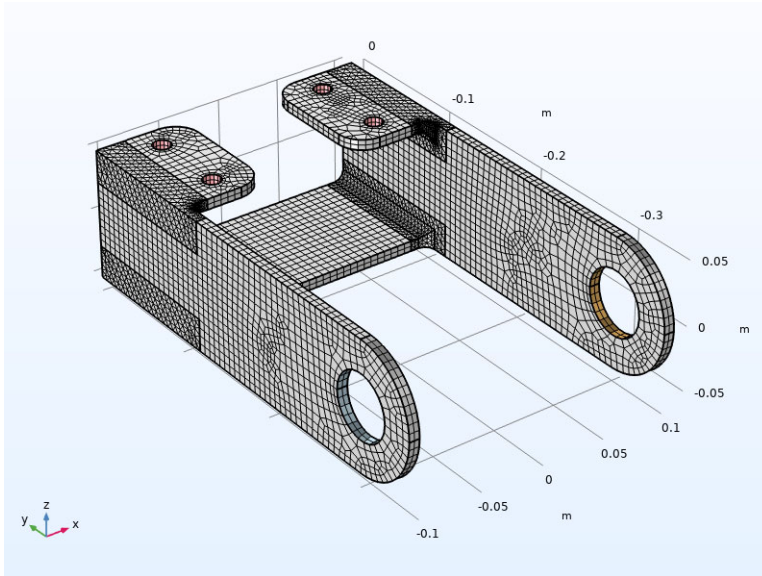
Size 2

Use a finer mesh in the fillets where high stresses can be anticipated.

- 1 In the Model Builder, right-click Free Tetrahedral 1 and choose Size .
- 2 In the Settings window for Size, locate the Geometric Entity Selection section. From the Geometric entity level list, choose Boundary, and select Boundaries 24, 28, 63, and 70.

- 3 In the Settings window for Size, locate the Element Size section. From the Predefined list, choose Extremely Fine.
- 4 Click Build All  .

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

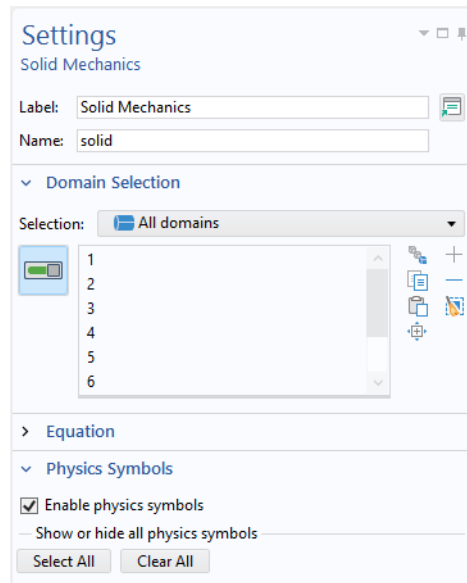


DISPLAYING PHYSICS SYMBOLS IN THE GRAPHICS WINDOW

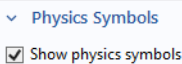
When working with the physics features, the current selection is shown in the Graphics window. You can add symbols to describe the type of condition applied to the selection.


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.

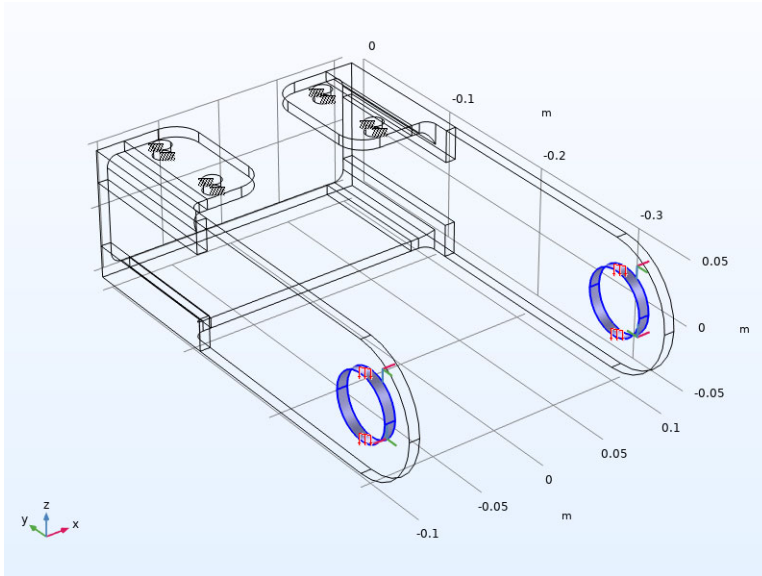
You turn the physics symbols on by selecting Enable physics symbols in the settings for a physics interface.



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.



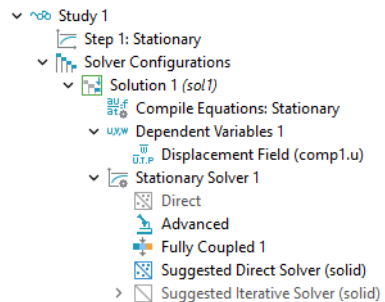
Study

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 is only relevant as long as the loads are independent of the solution. The steps below show how to visualize the load distribution in the current geometry.

▮ In 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.

If you expand the Study 1 node and its subnodes, you can see the details of the solver sequence generated.






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, just as in this example.

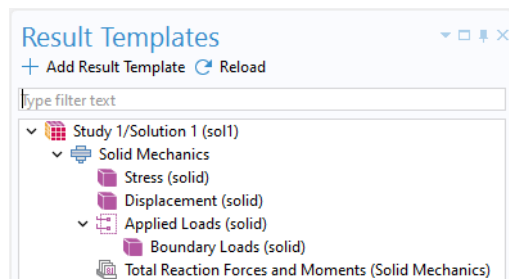
Results

Each physics interface has a set of default plots that are generated when a study is run. In this case, it is a stress plot. All stresses are zero as initial value.


There are also a number of predefined result templates which can be added if needed. The applied loads are available as such a result template.

Add Result Template

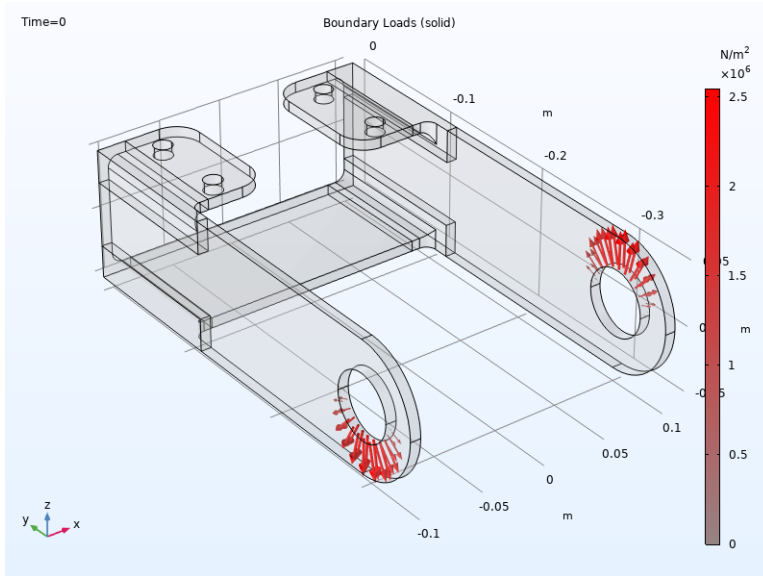
- 1 In the Home toolbar, click Result Templates .
- 2 Go to the Result Templates window.
- 3 In the tree, select Boundary Loads (solid).
- 4 Click Add Result Template  in the window toolbar.
- 5 In the Home toolbar, click Result Templates  again to hide the Result Templates window.




Boundary Loads (solid)

- 1 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 and magnitudes.



Study




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

Results

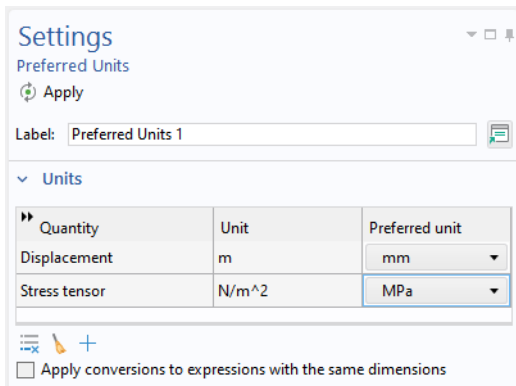
The default plot displays the von Mises stress distribution together with an exaggerated (automatically scaled) picture of the deformation.

Set default units for result presentation.

Preferred Units



- 1 In the Results toolbar, click Configurations  and choose Preferred Units .
- 2 In the Settings window locate the Units section, click Add Physical Quantity .
- 3 In the Physical Quantity dialog, select General > Displacement (m) in the tree.

- 4 Click OK.
- 5 In the Settings window for Preferred Units, locate the Units section.
- 6 In the table, select mm from the Preferred unit list.
- 7 Click Add Physical Quantity + .
- 8 In the Physical Quantity dialog, select Solid Mechanics > Stress tensor (N/m²) in the tree.
- 9 Click OK.
- 10 In the Settings window for Preferred Units, locate the Unit section.
- 11 In the table, select MPa from the Preferred unit list.

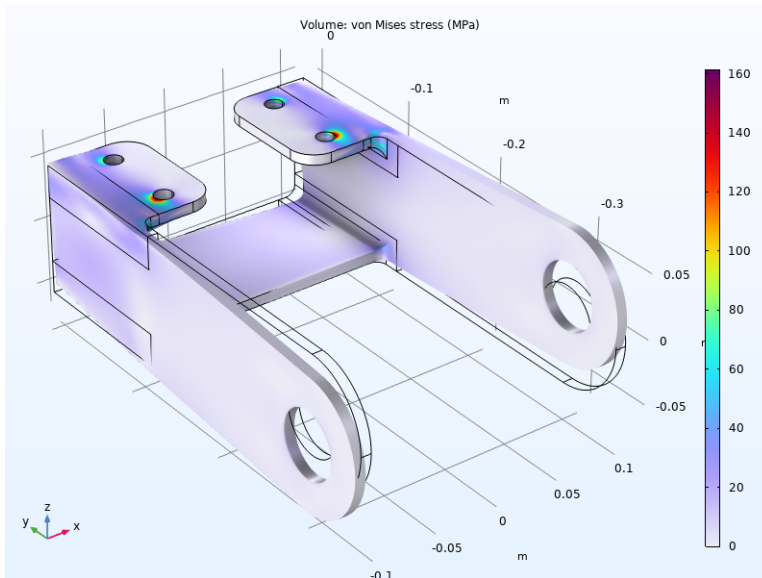


- 12 Click Apply .

Stress (solid)

- 1 In the Model Builder window, expand the Stress (solid) node, then click Volume 1.
- 2 In the Graphics window toolbar, click  next to  Scene Light, then choose Ambient Occlusion. Note how the plot changes when you turn this option on and off.

3 Click the Zoom Extents button  in the Graphics toolbar.



In structural mechanics models, it is common with high stresses at singular boundary conditions. Here, this is the case at the bolt holes. In order to visualize the interesting parts of the stress distribution better, it is often useful to truncate the value range in plots.

4 In the Settings window for Volume, click to expand the Range section. Select the Manual color range checkbox.

5 In the Maximum text field, type 70.

Now, enhance the plot with load arrows. The easiest way of doing that is to copy from the predefined load plot, and then adapt the arrow plot slightly.

1 In the Model Builder window, expand the Boundary Loads (solid) node under Applied Loads (solid).

2 Right-click Boundary Load 1 and choose Copy.

3 Right-click Stress (solid) and choose Paste Arrow Surface.


4 In the Settings window for Arrow Surface, locate the Coloring and Style section. From the Arrow base list, choose Head.

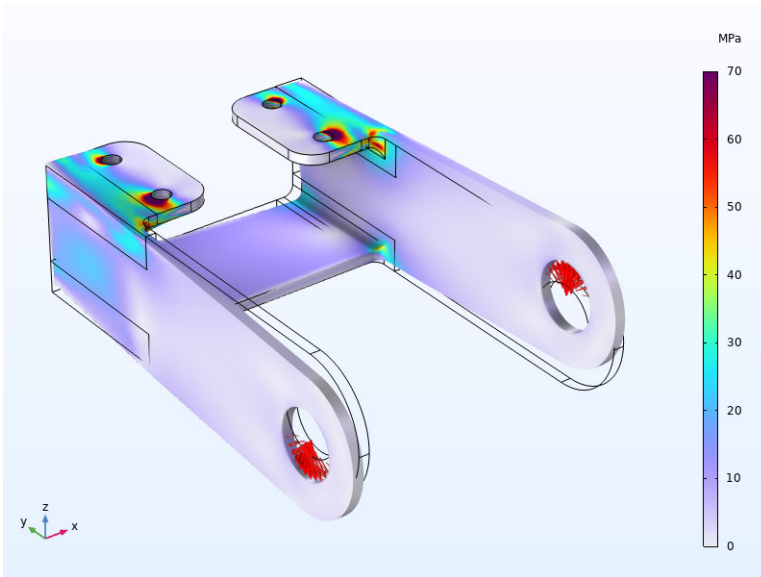
5 Select the Scale factor checkbox. In the associated text field, type $1E-8$.

6 Click to expand the Inherit Style section. From the Plot list, choose Volume 1.

7 Clear the Arrow scale factor, Color, and Color and data range checkboxes.




8 Expand the Boundary Load 1 node, then click Color Expression.

- 9 In the Settings window for Color Expression, locate the Coloring and Style section. Clear the Color legend checkbox.
- 10 In the Model Builder window, under Results click Stress (solid).
- 11 In the Settings window for 3D Plot Group, click to expand the Title section.
- 12 From the Title type list, choose Custom.
- 13 Find the Types and data subsection. Clear the Type, Description, and the Unit checkbox.
- 14 Locate the Color Legend section. Select the Show units checkbox.
- 15 Click the Show Grid  button in the Graphics toolbar to switch off the grid.



Total Displacement

Add a plot showing the displacement of the bracket. This is another one of the plots available under Result Templates.

- 1 In the Home toolbar, click Result Templates .
- 2 Go to the Result Templates window.
- 3 In the tree, select Study 1/Solution 1 > Solid Mechanics > Displacement (solid).
- 4 Click Add Result Template  in the window toolbar.
- 5 In the Home toolbar, click Result Templates  again to hide the Result Templates window.


- 6 In the Model Builder window, under Results, click Displacement (solid).
- 7 In the settings window, type Total Displacement in the Label text field.

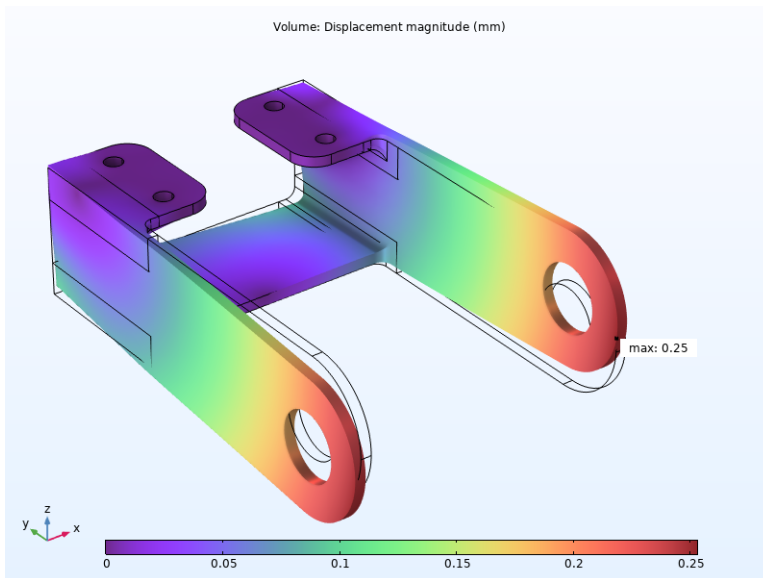
Marker 1

Add a marker indicating the position of the maximum displacement.

- 1 Expand the Total Displacement node, then right-click Volume 1 and choose Marker . . .
- 2 In the Settings window for Marker, locate the Display section.
- 3 From the Display list, choose Max.
- 4 Locate the Text Format section. In the Display precision text field, type 2.
- 5 Locate the Coloring and Style section. From the Background color list, choose From theme.

Total Displacement

- 1 In the Model Builder window, click Total Displacement.
- 2 In the Settings window for 3D Plot Group, locate the Color Legend section.
- 3 From the Position list, choose Bottom.
- 4 On the Total Displacement toolbar, click Plot .





You may have noticed that in the plot group, the stresses and displacements are displayed as volume plots. It would also have been possible to use surface plots.



For a solid geometry in 3D, the visual difference is small. Each of these plots have their merits, though. In a large model, a volume plot can take noticeably longer time to generate, since the expression to be visualized is evaluated not only on the surface, but also in the interior. If you want to plot only parts of the structure, the volume plot is easier to work with, since you can select domains, rather than boundaries. Also, for some expressions, an evaluation in the domain is more accurate than when the same expression is evaluated only on the boundaries.

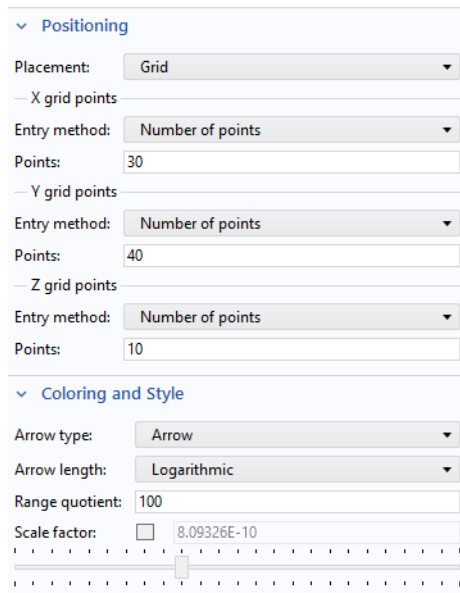
Principal Stress


Create a third plot to display the principal stresses.

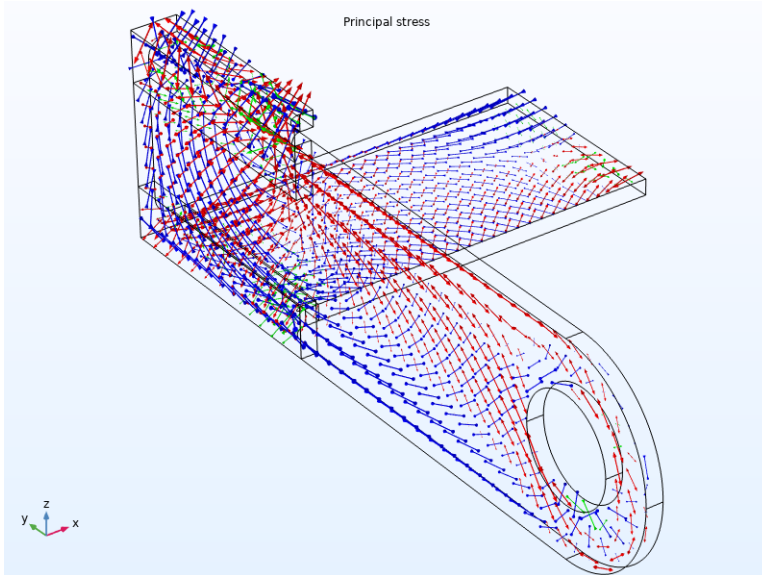
- 1 In the Home toolbar, click Add Plot Group  and choose 3D Plot Group .
- 2 In the Settings window for 3D Plot Group, type Principal Stress in the Label text field.
- 3 Click to expand the Selection section. From the Geometric entity level list, choose Domain.
- 4 Select Domains 1–4 only.
- 5 Select the Apply to dataset edges checkbox.

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. 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 40.
 - Find the Z grid points subsection. In the Points text field, type 10.
- 3 Locate the Coloring and Style section. From the Arrow length list, choose Logarithmic.







4 On the Principal Stress toolbar, click Plot .

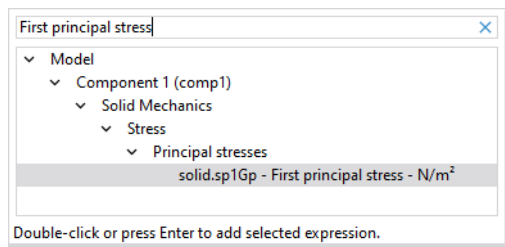





The principal stresses are color coded so that red represents the first principal stress, green the second, and blue the third.

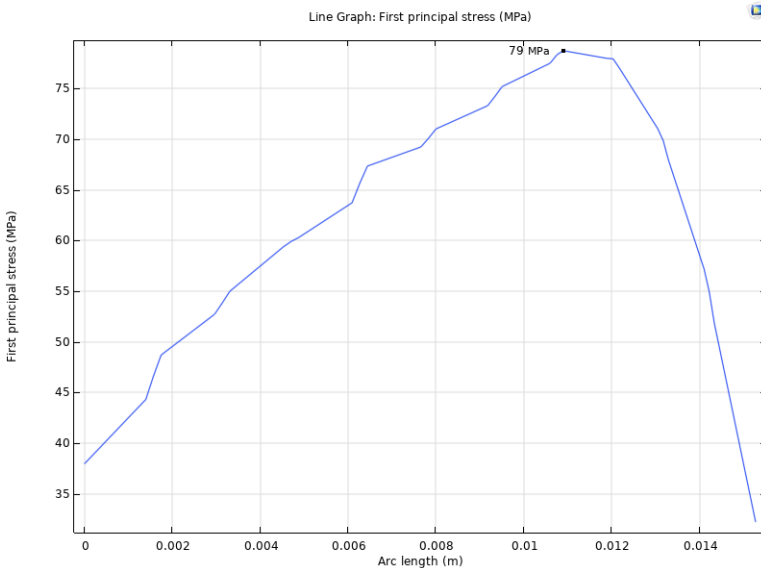
Stress Along Fillet

Plot the stress distribution along the fillet having high stress levels.

- 1 In the Home toolbar, click Add Plot Group  and choose 1D Plot Group .
- 2 In the Settings window for 1D Plot Group, type Stress Along Fillet in the Label text field.
- 3 Right-click Stress Along Fillet and choose Line Graph .
- 4 Select Edge 48 only.
- 5 In the Settings window for Line Graph, click Replace Expression  in the upper-right corner of the y-Axis Data section.
- 6 In the filter text field type First principal stress, then choose solid.sp1Gp - First principal stress - N/m².



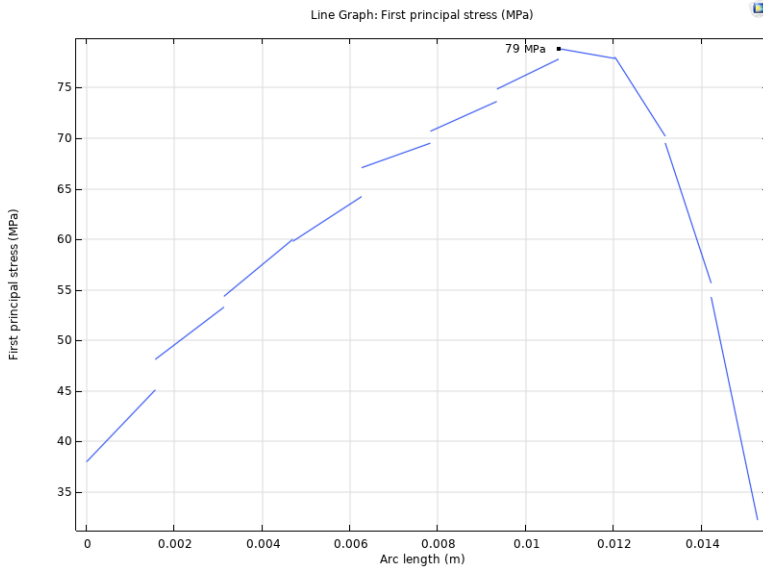
- 7 Right-click Line Graph 1 and choose Graph Marker .
- 8 In the Settings window for Graph Marker, locate the Display section.
- 9 From the Display list, choose Max.
- 10 Select the Include unit checkbox.
- 11 Locate the Text Format section. In the Precision text field, type 2.
- 12 Click to expand the Coloring and Style section. From the Anchor point list, choose Middle right .
- 13 In the Stress Along Fillet toolbar, click Plot .



You can get an indication of the discretization errors by switching off the averaging between adjacent elements.

- 14 In the Model Builder window, click Line Graph 1.
- 15 In the Settings window for Line Graph, click to expand the Quality section.
- 16 From the Smoothing list, choose None.

17 In the Stress Along Fillet toolbar, click Plot  .



Solid Mechanics

Now, assume that the bracket is made from cast iron, a material that is much stronger in compression than in tension. The von Mises criterion, which is isotropic with respect to the stress direction is not suitable in this case. By adding a Safety node, it is possible to check the margins against failure for many different types of materials.

Safety 1

1 In the Model Builder window, under Solid Mechanics (solid) click Linear Elastic Material 1.

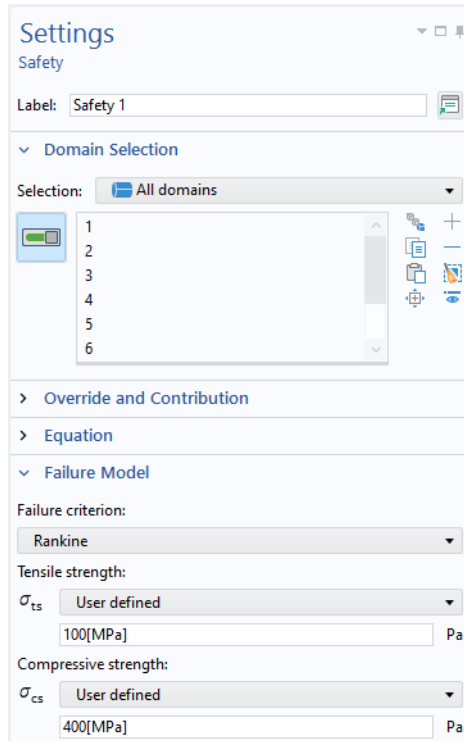
- In the Physics toolbar, click Attributes  and choose Safety.

The Rankine criterion is suitable for materials where the failure is governed by either the most positive or most negative stress.

- In the Settings window for Safety, locate the Failure Model section.
- From the Failure criterion list, choose Rankine.


Here, you can choose to either augment the Structural Steel node under Materials or enter the strength values directly in the Safety node. When a single material is used, it does not matter. When there are several materials in a model, it is a good practice to use the From Material option.

- From the Tensile strength list, choose User defined. In the associated text field, type 100[MPa].
- From the Compressive strength list, choose User defined. In the associated text field, type 400[MPa].










Study

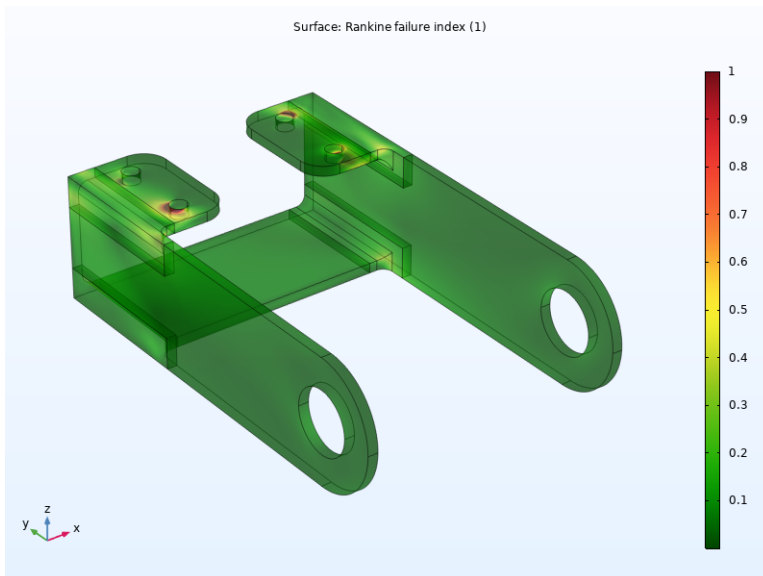
Evaluating safety factors can be done without recomputing the solution. The new variables defined by the recently added Safety node must, however, be made part of the solution. For this, you use Update Solution, which is much faster than actually solving.

- In the Study toolbar, click Update Solution .

Results

After updating the solution, a new plot has been added to Result Templates.


- 1 In the Home toolbar, click Result Templates .
- 2 Go to the Result Templates window.
- 3 In the tree, select Study 1/Solution 1 (sol1) > Solid Mechanics > Failure Indices (solid) > Failure Index (Safety 1).
- 4 Click Add Result Template  in the window toolbar.
- 5 In the Home toolbar, click Result Templates  again to hide the Result Templates window.
- 6 In the Model Builder window, expand the Failure Index (Safety 1) node, then click Surface 1.
- 7 In the Settings window for Surface, click to expand the Range section.
- 8 Select the Manual color range checkbox.
- 9 In the Maximum text field, type 1.
- 10 In the Model Builder window, click Failure Index (Safety 1).
- 11 In the Settings window for 3D Plot Group, locate the Plot Settings section.
- 12 From the View list, choose New View.
- 13 In the Failure Index (Safety 1) toolbar, click Plot .
- 14 In the Graphics toolbar, click the Scene Light , the Transparency , and the Show Grid  buttons.

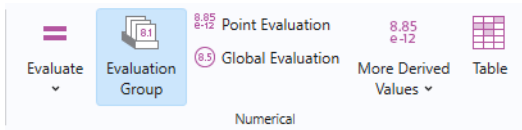


Derived Values

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: Reactions




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.

Bolt 1

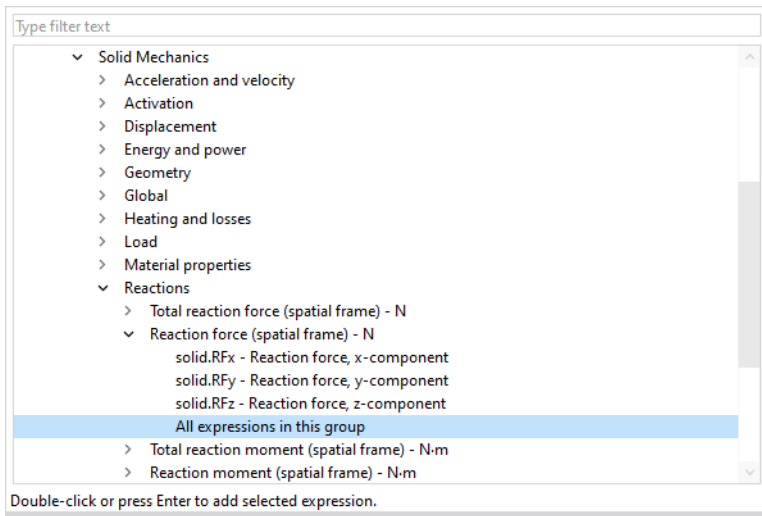
1 On the Evaluation Group: Reactions toolbar click Integration  and choose Surface Integration . Alternatively, in the Model Builder, right-click Evaluation Group: Reactions and choose Integration > Surface Integration .

2 In the Settings window for Surface Integration type Bolt 1 in the Label text field.

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.

- 5 From the menu, choose Component 1 > Solid Mechanics > Reactions > Reaction force (spatial frame) - N > All expressions in this group.



- 6 Press Enter.

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

- 7 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

↑ ↓ ↻ ↷ ⌂ ⌂

Expression:
solid.RFx

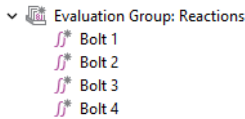
Description:
Bolt 1, X

Bolt 2, Bolt 3, and Bolt 4

- 1 Right-click Bolt 1 and choose Duplicate. Repeat this two more times; adding 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 and so on
Bolt 3	Bolt 3	Bolt 3, X and so on
Bolt 4	Bolt 4	Bolt 4, X and so on

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



- 2 In the Evaluation Group: Reactions toolbar, click Evaluate $\mathbf{=}$. All reaction forces in the evaluation group are now displayed.

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)	B
460.00	312.63	2578.8	-459.96	74.612	-1731.3	460.18	-312.68	-2580.1	-4

You can use the individual reaction forces to check the capacity of the bolts. Verifying that the sum of all reaction forces matches the applied loads is also a good sanity check for an analysis.

Parametric Study

In the previous section, a bracket loaded by an actuator was analyzed. This section extends the 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

In this model, the stress in the bracket is computed for different load orientations. First add a parameter to set the load direction angle.



- 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 In 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	PinHoleY	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: Cylindrical System 2

Name: sys2

Coordinate Names

Frame: Spatial (x, y, z)

Coordinate names

First	Second	Third
r	phi	a

Settings

Origin

x (m)	y (m)	z (m)
0	PinHoleY	0

Longitudinal axis

x	y	z
1	0	0

Direction of axis $\varphi=0$

x	y	z
0	sin(theta0)	cos(theta0)

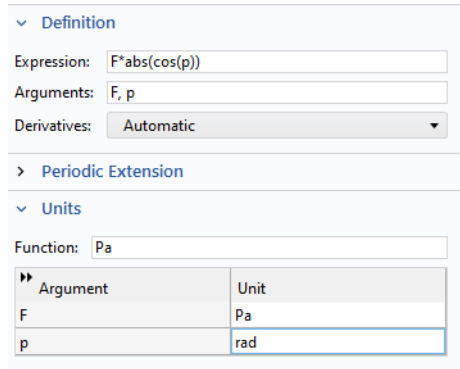
Definitions - Functions

Change the expression for the load distribution.

Analytic 1 (load)

- | 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 $F \cdot \text{abs}(\cos(p))$.
- 3 In the Arguments text field, type F, p .
- 4 Locate the Units section. In the table (which just shrunk to two rows to match the current arguments), replace the previous contents on the second row with rad .



Solid Mechanics

Change the boundary load to consider the parameterized direction.

Boundary Load 1

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

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

$\text{load}(p0, \text{sys2}.phi) * (\text{sign}(X) * (\text{abs}(\text{sys2}.phi) - \pi/2) < 0)$	r
0	phi
0	a

Note how the Boolean expression $\text{sign}(X) * \text{abs}(\text{sys2}.phi) > \pi/2 < 0$ is used to filter the load so that it is applied only to half of the pin hole, as well as in opposite directions in the left and right pin holes. The loaded area does however rotate with the coordinate system, which is controlled by the parameter $\text{theta}0$.

Now, add an auxiliary sweep parameter.

Study - Auxiliary Sweep

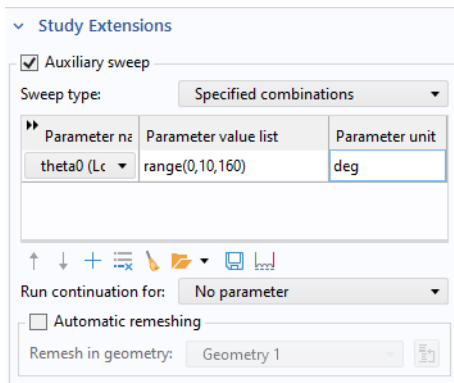
Parametric studies can be set up from scratch or, as in this example, added to an existing study. This model computes the stress in the bracket for different load orientations. Start by selecting the sweep parameter, in this case the load direction angle. Then run the analysis.

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 checkbox.
- 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 (Load direction angle)	range(0,10,160)	deg

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





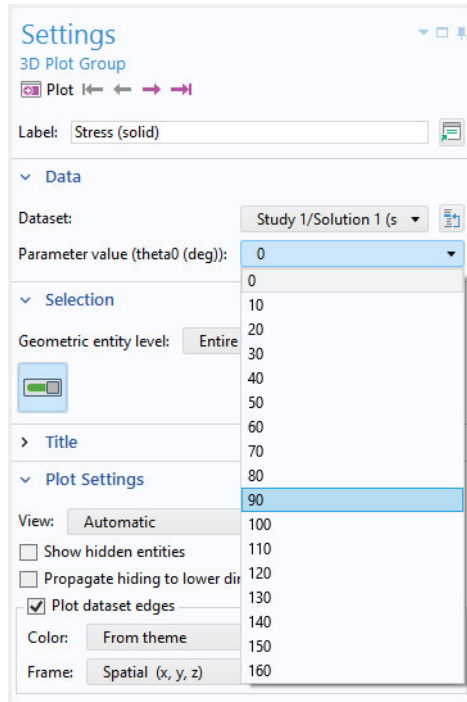
- 6 In the Settings window for Stationary click Compute = .

Results

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

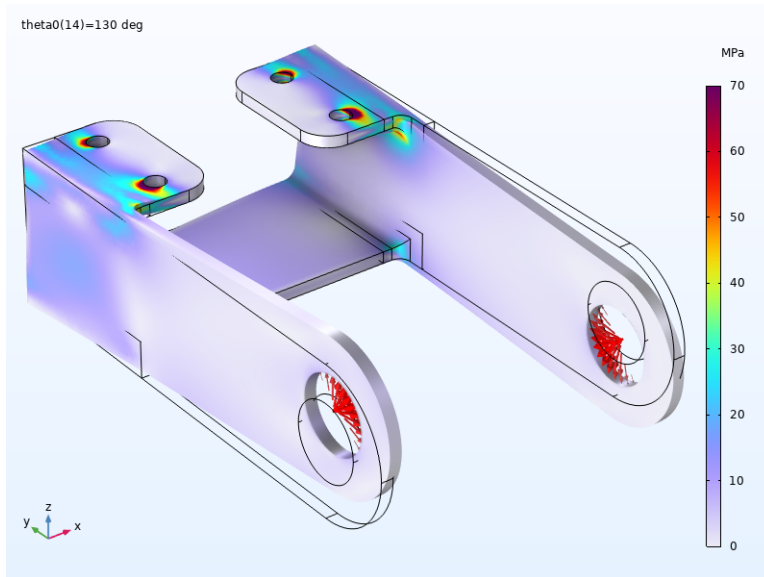
Stress (solid)

- 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 degrees is shown below.



Now create a plot showing how the reaction forces vary with the load angle.


Evaluation Group: Reactions

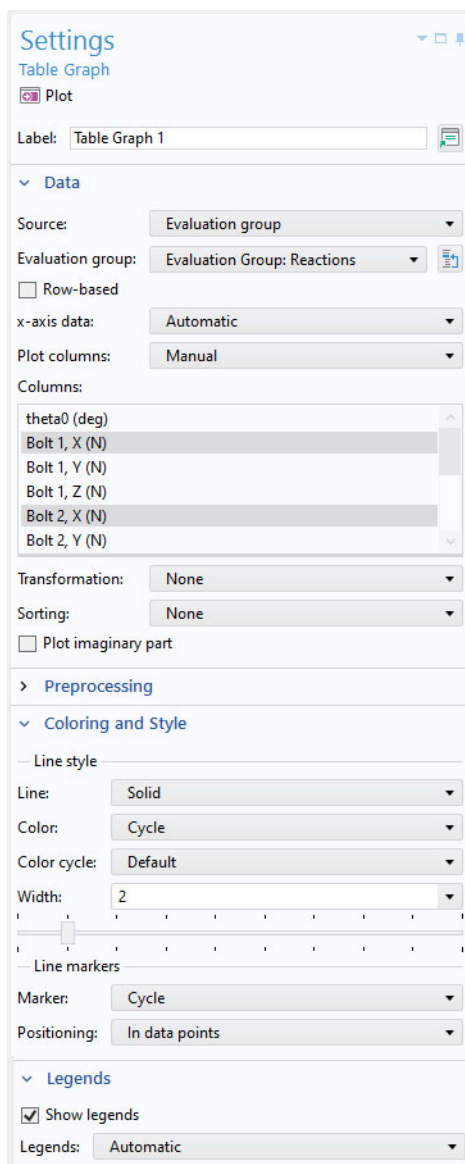
- 1 In the Model Builder, click Evaluation Groups: Reactions.
- 2 In the Evaluation Groups: Reactions toolbar, click Evaluate = .
- 3 In the Evaluation Groups: Reactions toolbar, click Table Graph ⌂ .

Reaction Force, X-component

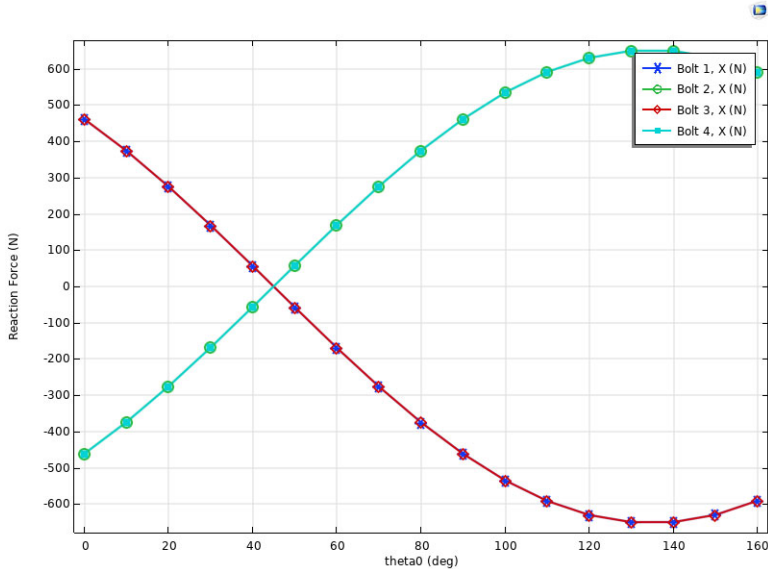
- 1 In the Model Builder window, under Results click 1D Plot Group 7.
- 2 In the Settings window for 1D Plot Group, type Reaction Force, X-component in the Label text field.
- 3 Locate the Plot Settings section. Select the y-axis label checkbox.
- 4 In the associated text field, type Reaction Force (N).

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 checkbox.
- 8 In the Reaction Force, X-component toolbar, click Plot  to show the variation in the four X-direction reaction forces as function of the load angle parameter.



The graph should look like this:



Next, create similar plots for the reaction forces in the other two directions. When you already have a plot that 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



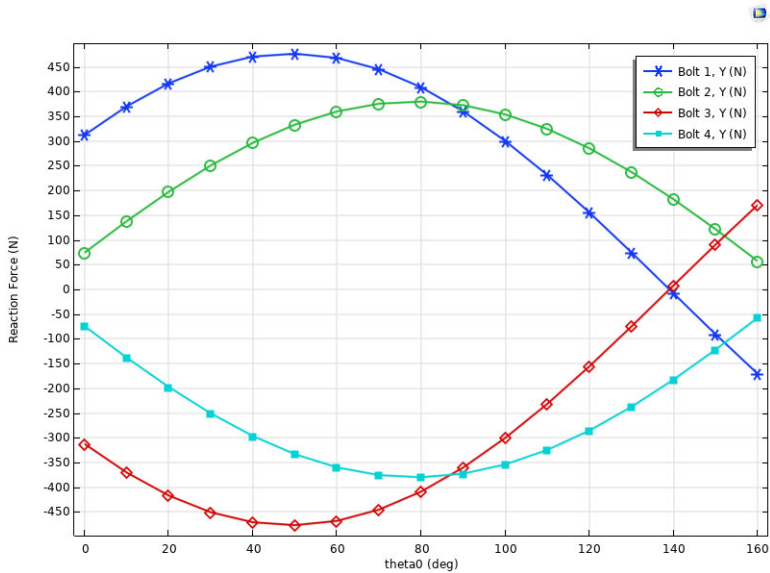
- 1 Right-click Reaction Force, X-component, and choose Duplicate .
- 2 Click Reaction Force, X-component 1. In the Settings window for 1D Plot Group, change to Reaction Force, Y-component in the Label text field.

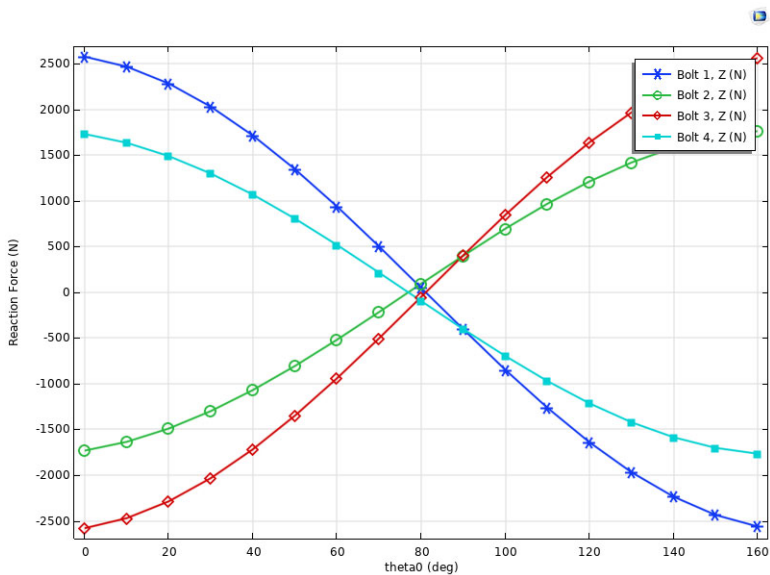
Table Graph 1

- 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  .

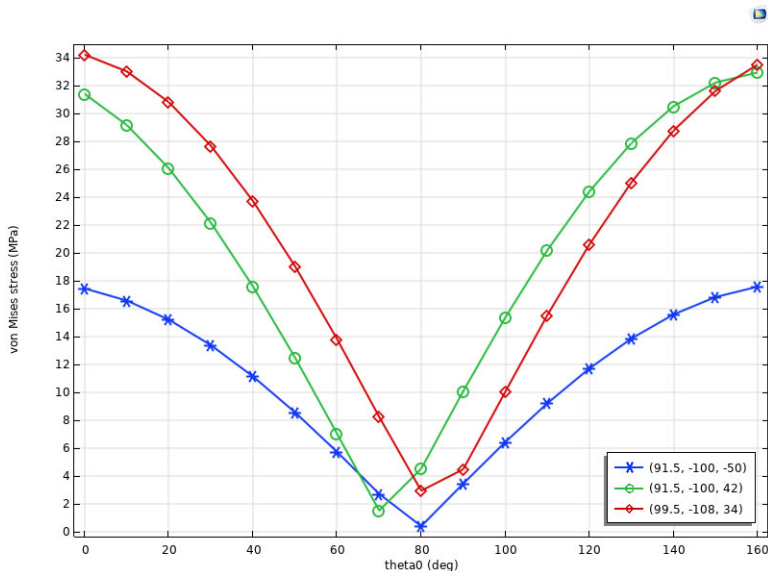


Repeat the steps above to create a similar graph also of the Z-direction reaction forces.



Stress as Function of Load Angle

- 1 In the Home toolbar, click Add Plot Group  and choose 1D Plot Group .
 - 2 In the Settings window for 1D Plot Group, type Stress as Function of Load Angle in the Label text field.
 - 3 Expand the Title section. From the Title type list, choose None.
 - 4 Locate the Legend section. From the Position list, choose Lower right .
 - 5 Right-click Stress as Function of Load Angle and choose Point Graph .
 - 6 Select Points 86, 88, and 103 only.
 - 7 In the Settings window for Point Graph, locate the y-Axis Data section. In the Expression text field, type `solid.mises`.
 - 8 Click to expand the Coloring and Style section. Locate the Coloring and Style section. In the Width text field, type 2.
 - 9 Find the Line markers subsection. From the Marker list, choose Cycle.
- Annotate the graphs by the location of each point.
- 10 Click to expand the Legends section. Select the Show legends checkbox.
 - 11 From the Legends list, choose Evaluated.
 - 12 In the Legend text field, type `(eval(X,mm), eval(Y,mm), eval(Z,mm))`.
 - 13 In the Stress as Function of Load Angle toolbar, click Plot .



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 In the Home toolbar click Parameters P_i and select Parameters 1 P_i .
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

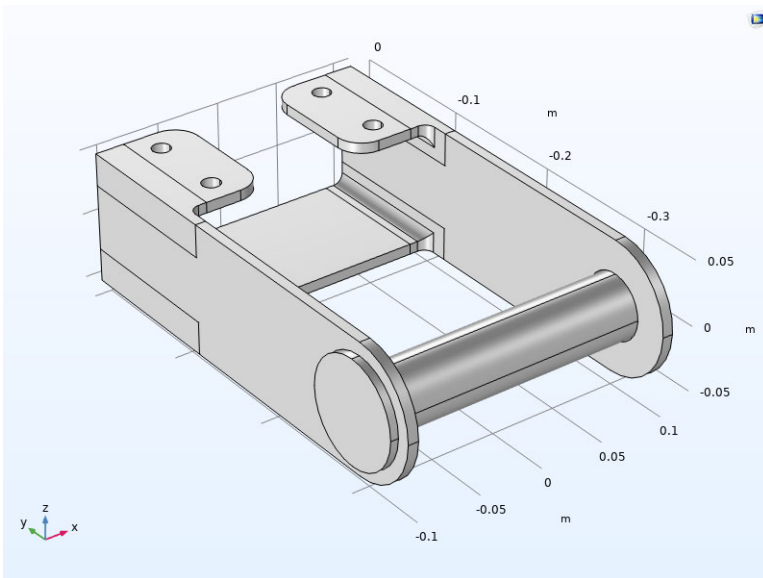
Geometry

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 Files\COMSOL\COMSOL63\Multiphysics\applications\Structural_Mechanics_Module\Tutorials.

Import

- 1 In 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 Below the filename, click Import .



Form Union (fin)

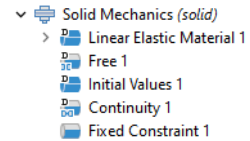
- 1 In the Model Builder under Geometry 1 click Form Union (fin) .
- 2 In the Settings window for Form Union/Assembly, locate the Form Union/Assembly section.
- 3 From the Action list, choose Form an assembly.
- 4 Click the Build Selected button .

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

Since the pins will be bonded to the bracket, it would have been possible to use either an assembly or a union. The advantage of using an assembly in this case is that it is not necessary to have a matching mesh on the two parts.



Solid Mechanics (solid)

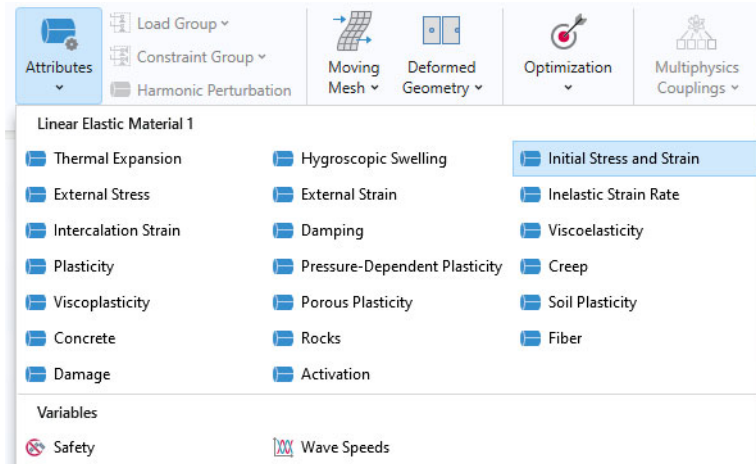
If you expand the Solid Mechanics (solid) node, you can see that a Continuity node has been automatically added. This means that the flanges of the pin will be bonded to the bracket arms. Since the boundaries of the pin are geometrically adjacent to the bracket, this was automatically identified. You can check, and, if needed, modify the definition of a pair under the Definitions branch in the Model Builder window.



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, under Solid Mechanics (solid), click Linear Elastic Material 1. In the Physics toolbar click Attributes  and select Initial Stress and Strain .

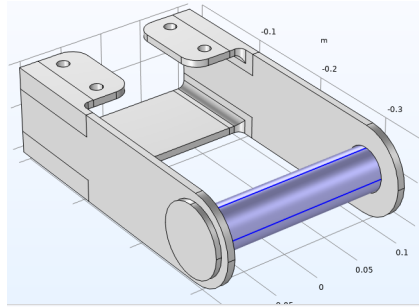


As an alternative, you can right-click on the Linear Elastic Material 1 node, and select Initial Stress and Strain from the context menu.

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 pin, which coincides with the global x direction.



4 Under the Initial Stress and Strain section, enter `InitStrain` in the first component of the ϵ_0 table.

Initial Stress and Strain

Initial stress:

0	0	0	N/m ²
0	0	0	
0	0	0	

Initial strain:

InitStrain	0	0	1
0	0	0	
0	0	0	

Mesh

The current mesh sequence ends with a node that creates a rather fine free tetrahedral mesh on the remaining (not yet meshed) domains. This would now include also the pin. In order to avoid getting a large number of unnecessary degrees of freedom in the pin, modify the mesh sequence.

1 In the Model Builder, expand the Component 1 (comp1) > Mesh 1 node. Then expand the Free Tetrahedral 1 node.

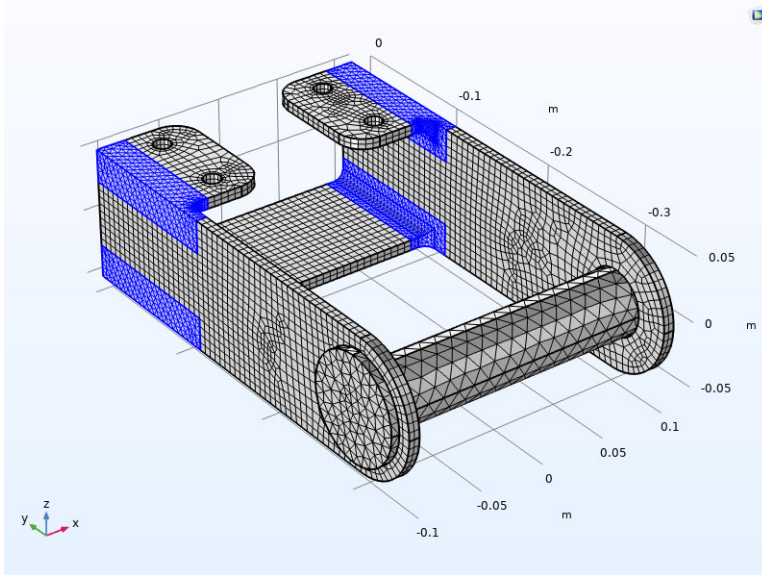
2 Click the Size 1 node.

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





4 From the Geometric entity level list, choose Domain.

5 Select Domains 5, 6, 10, and 11.

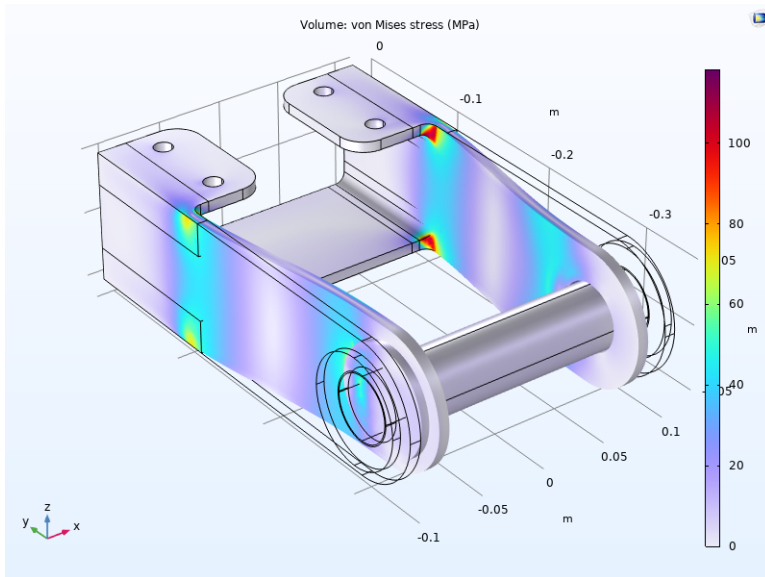
6 Click Build All  .



Add a Study and Compute

- 1 In 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 In the Home toolbar, click Add Study again to close the Add Study window.
- 5 In 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 supporting plates.

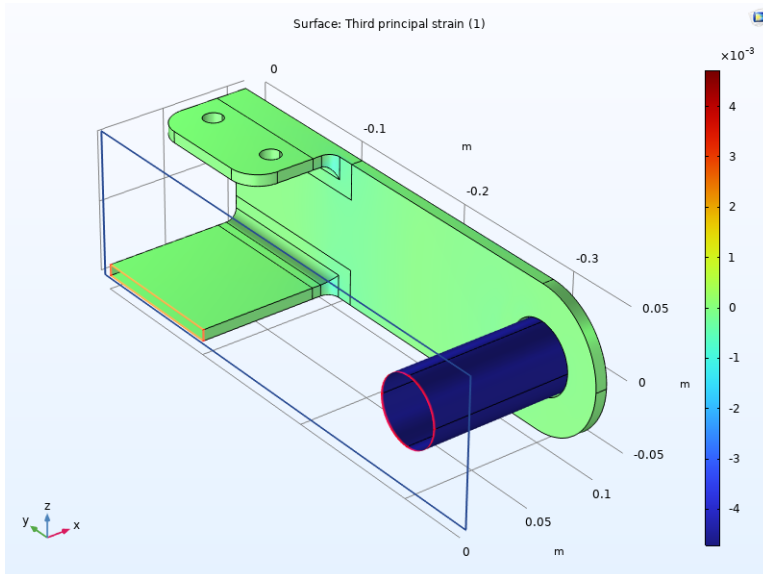
Results

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.

Third Principal Strain

- 1 In 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).
- 3 On the Third Principal Strain toolbar click Surface  (or right-click the Third principal strain node and choose Surface).
- 4 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.

- 5 Locate the Coloring and Style section. From the Scale list, choose Linear symmetric.
- 6 In the Graphics window toolbar, click  next to  Clipping, then choose Add Clip Plane.
- 7 In the Graphics window toolbar, click  next to  Clipping Active, then deselect Show Gizmos.



Modeling Thermal Expansion

In this example, a temperature field is applied to the bracket 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 multiphysics interface at the beginning of the setup. In principle, you have three different options:












- Start the analysis by adding a Thermal Stress, Solid multiphysics interface. This will add a Solid Mechanics interface, a Heat Transfer in Solids interface, and a Thermal Expansion multiphysics coupling.
- Add a Solid Mechanics interface, a Heat Transfer in Solids interface, and then add a Thermal Expansion multiphysics coupling manually.
- Add a Solid Mechanics interface and a Heat Transfer in Solids interface. Under the Linear Elastic Material, add a Thermal Expansion subnode, and make sure that it picks up the computed temperature.

These different types of approaches are relevant for many other multiphysics simulations too.



If you know from the beginning of your simulation project that you are going to perform a coupled analysis, adding the predefined multiphysics interface is the recommended approach. Not only is it more convenient, but doing so will also in general change some settings in the involved physics interfaces to better suit the combined analysis. An example of this is that when you add the Thermal Stress, Solid multiphysics interface, then the Solid Mechanics interface will assume that a time-dependent analysis is quasistatic from the structural point of view, so that inertial terms are ignored. This, in turn, will affect the time-stepping algorithm chosen in a time-dependent solver.

In this example, you start from the pure geometry, and use the automatic meshing. The number of modeling steps is kept at a minimum.

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, expand the folder Thermal Structure Interaction , and click Thermal Stress, Solid .
- 4 Click Add, then click Study .
- 5 In the Select Study tree under General Studies, click Stationary .
- 6 Click Done .
- 7 In 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 Below the filename, click Import .

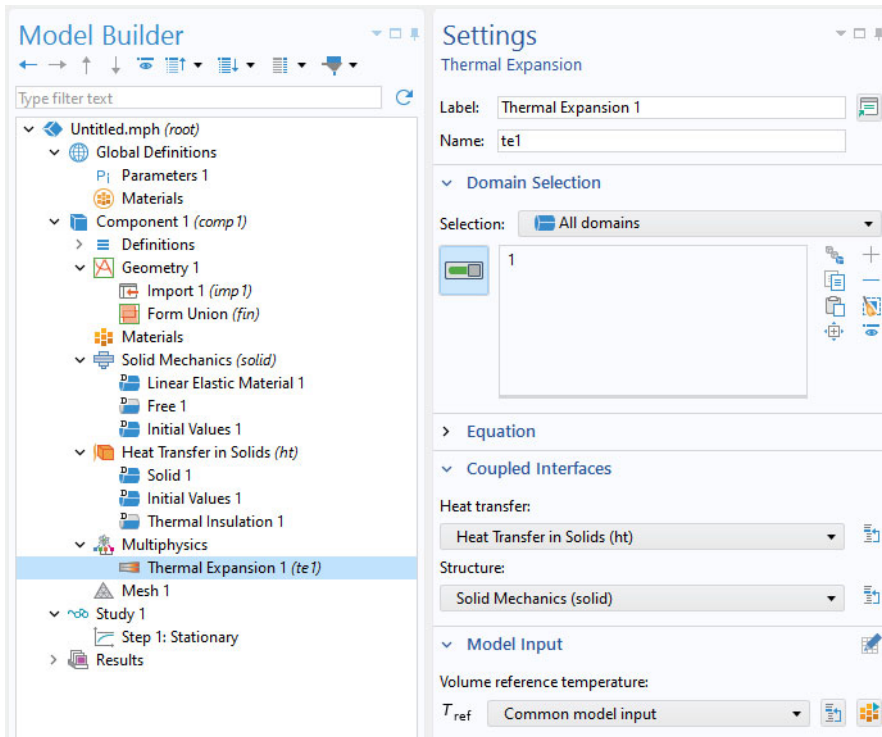
Form Union (fin)


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

Thermal Stress

When you add a Thermal Stress, Solid 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 that 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

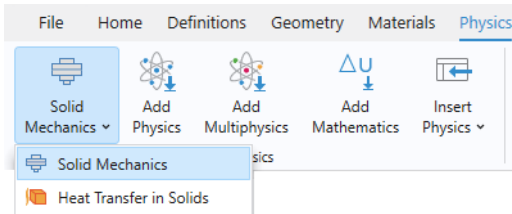
Add the material. It contains structural as well as thermal properties.



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

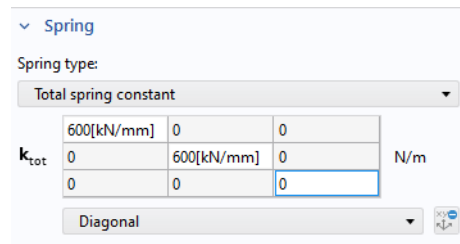
Solid Mechanics (solid)

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





- 2 In the Physics toolbar click Boundaries  and choose Roller.
- 3 In the Settings window for Roller select Boundaries 17 and 27.
- 4 In the Physics toolbar click Boundaries  and choose Spring Foundation.
- 5 In the Settings window for Spring Foundation select Boundaries 18–21 and 31–34 (the bolt holes).
- 6 Locate the Spring section. From the Spring Type list, choose Total spring constant.
- 7 Choose Diagonal from the list below the text field. The text field changes to a matrix input.
- 8 Enter 600[kN/mm] in the first two diagonal elements of the k_{tot} matrix as shown in the figure. This provides a stiffness in the global X and Y directions.

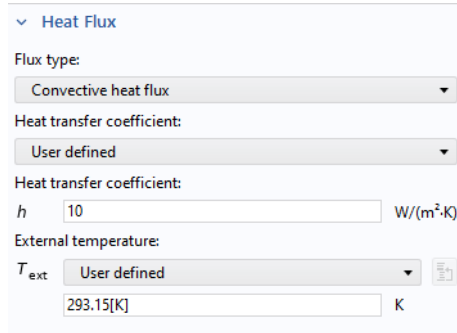




Heat Transfer in Solids (ht)

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 In 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 In 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 Locate the Heat Flux section. From the Flux type list, choose Convective heat flux. Enter 10 in the text field for the Heat transfer coefficient, h .

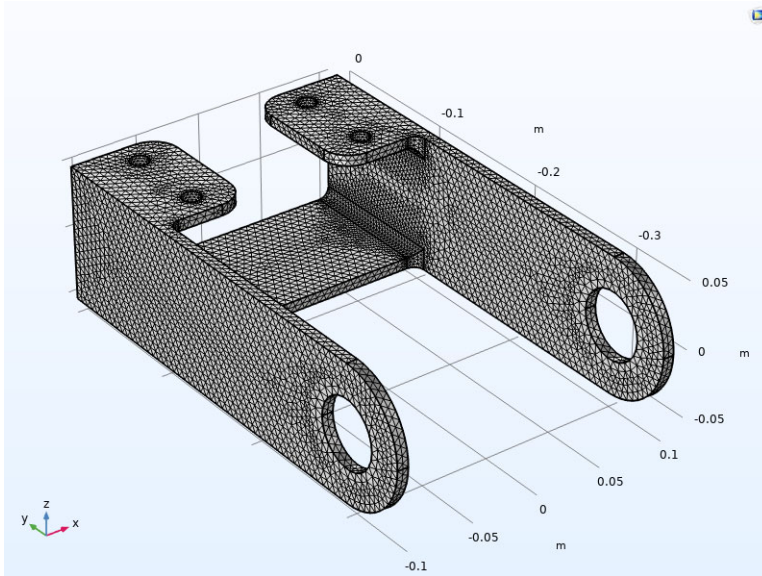


- 5 In the Physics toolbar click Boundaries  and choose Heat Flux .
- 6 In the Settings window for Heat Flux, select Boundaries 17 and 27.
- 7 In the Heat Flux section, enter $1e4$ in the General inward heat flux, q_0 , text field.

Mesh 1

- 1 In the Model Builder Window, click Mesh 1.
- 2 In the settings window for Mesh, locate the Physics-Controlled Mesh section. From the Element size list, choose Finer.
- 3 Locate the Sequence Type section, and select User-controlled mesh from the list.
- 4 Under Mesh 1, click Size.
- 5 In the Settings window for Size, locate the Element Size section, and select Finer from the Predefined list.
- 6 Right-click the Free Tetrahedral 1 node, and choose Size.
- 7 In the Settings window for Size, locate the Element Size section, and click the Custom button.

- 8 Locate the Element Size Parameters section. Select the Maximum Element Size checkbox, and type 8[mm] in the associated text field.
- 9 Click Build All  . The mesh should now look like in the figure below.



Compute and Display Results

In general, the resulting system of equations in a multiphysics problem can be solved using two different strategies:


- In the fully coupled approach, all degrees of freedom (DOF) are solved simultaneously using a single large system of equations.
- In the segregated approach, one smaller system of equations is solved for each type of physics. The couplings between the types of DOF are introduced through the load vectors. Except for some special cases, it is then necessary to repeat the solutions several times, until convergence.

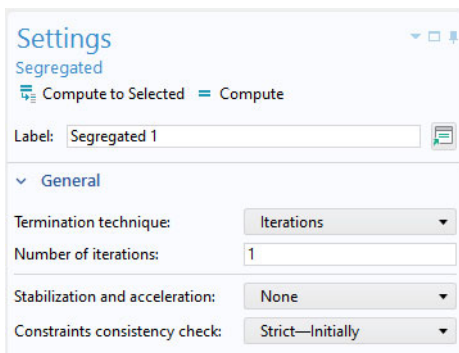
In most cases, the segregated approach will be the default suggestion when the solver sequence is generated for a multiphysics problem. Based on your knowledge of the underlying properties of a problem, it is often possible to reduce the

solution time by making appropriate changes to the solver settings. In this example, there are two important properties that can be used:

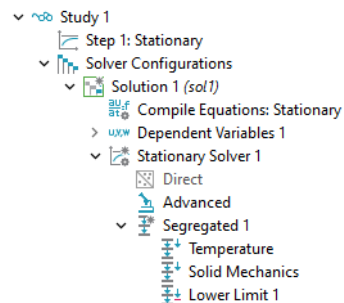
- Both the heat transfer problem and the solid mechanics problem are linear.
- The heat transfer problem is independent of the solution to the solid mechanics problem. This is a very common situation in thermal-stress analysis.


As long as the heat transfer problems is solved first, it is thus sufficient to make one pass through the segregated solver, and no iterations are needed for either physics interface.

- 1 In the Study toolbar, click Show Default Solver .
- 2 In the Model Builder window, expand the Solution 1 node. Then, expand the Stationary Solver 1 node.
- 3 Click Segregated 1. In the Settings window, locate the General section.
- 4 From the Termination technique list, choose Iterations.



- 5 Expand the Segregated 1 node, and inspect the segregated steps. Note that the Temperature node appears above the Solid Mechanics node, which is the optimal order for this problem, since the coupling between the physics interfaces is unidirectional.






- 6 On the Study or Home toolbar, click Compute .

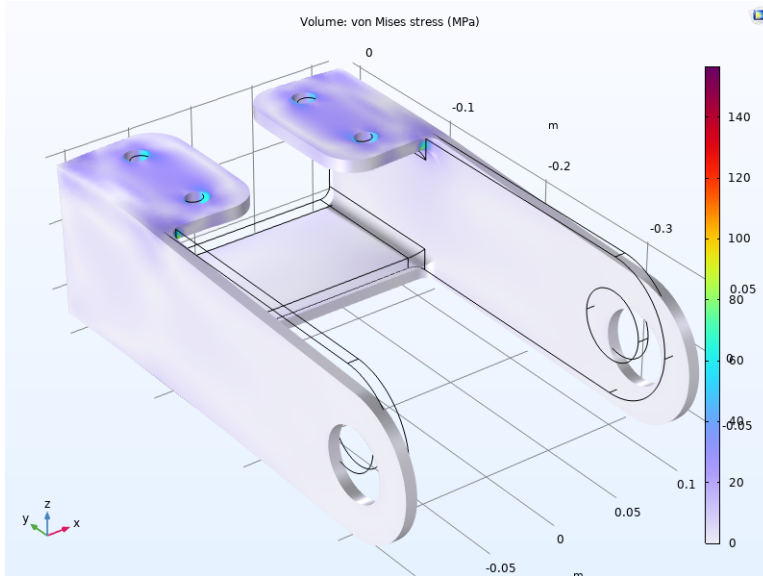
Under the Results node, two plot groups are automatically added to show the default results for a structural analysis and a thermal analysis. The first plot group, Stress (solid), shows the von Mises stresses on a scaled deformed geometry. Due

to the boundary conditions and the nonuniform temperature distribution, thermal stresses develop in the structure.

Improve the plot slightly.

Volume 1

- 1 In the Model Builder window, expand the Stress (solid) node, then click Volume 1.
- 2 In the Settings window for Volume, locate the Expression section. From the Unit list, choose MPa.
- 3 Click the Show Grid  button in the Graphics toolbar to switch off the grid.
- 4 Click the Zoom Extents button  in the Graphics toolbar.
- 5 Click Plot  .




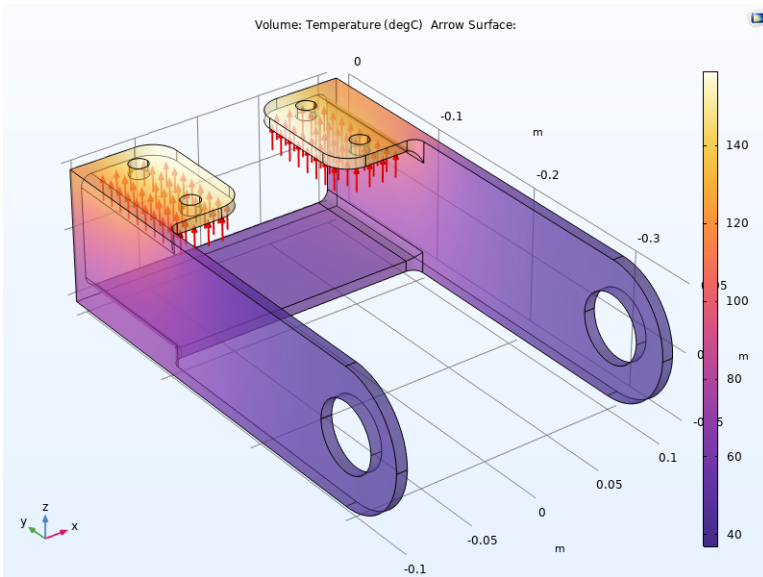
The second default plot group, Temperature (ht), displays the temperature distribution. The temperature is highest where the inward heat flux is prescribed, and decreases as heat is removed by convection from all other boundaries.

Add some arrows indicating the thermal loading.

Temperature (ht)

- 1 In the Model Builder window, right-click Temperature (ht), and choose Arrow Surface.

- 2 In the Settings window for Arrow Surface, locate the Expression section.
- 3 In the X-component and Y-component text fields, type 0.
- 4 In the Z-component text field, type 1.
- 5 Locate the Arrow Positioning section. In the Number of arrows text field, type 100.
- 6 Locate the Coloring and Style section. From the Arrow base list, choose Head.
- 7 Select the Scale factor checkbox. In the associated text field, type 0.02.
- 8 Right-click Arrow Surface 1 and choose Selection.
- 9 Select Boundaries 17 and 27.
- 10 Click Volume and locate the Expression section. From the Unit list, choose degC.
- 11 Right-click Volume and choose Transparency.
- 12 Set the Transparency value to 0.4.
- 13 Set the Fresnel transmittance to 0.4.
- 14 Click Plot  .

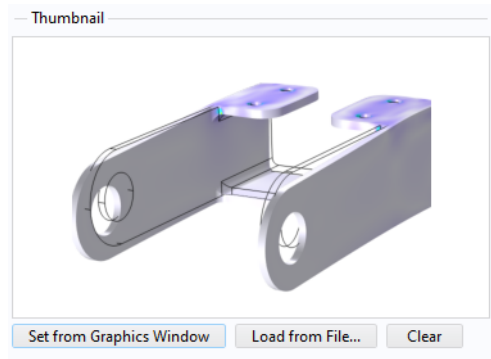


As a final step, pick one of the plots to use as a model thumbnail.

- 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). In the Settings window for Root under Thumbnail, click Set from Graphics Window.

Adjust 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, frequency response analysis, and analysis of nonharmonic periodic excitation (bracket_eigenfrequency, bracket_transient, bracket_frequency, bracket_general_periodic)
- Perform a linearized buckling analysis (bracket_linear_buckling)
- Perform a contact analysis (bracket_contact)
- Perform a random vibration analysis (bracket_random_vibration)
- Perform an inertia relief analysis (bracket_inertia_relief)
- Create a use reduced-order model (bracket_rom)