



Model created in COMSOL Multiphysics 6.4

Bracket – Transient Analysis

Introduction

Transient analyses provide the time-domain response of a structure subjected to time-dependent loads. A transient analysis can be important when the time scale of the load is such that inertial or damping effects have a significant influence on the behavior of the structure.

In this example, you learn how to add damping properties to the material, define external loads varying with time, set up time-stepping data for the study, and generate animations.

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

Model Definition

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

The model geometry is represented in [Figure 1](#).

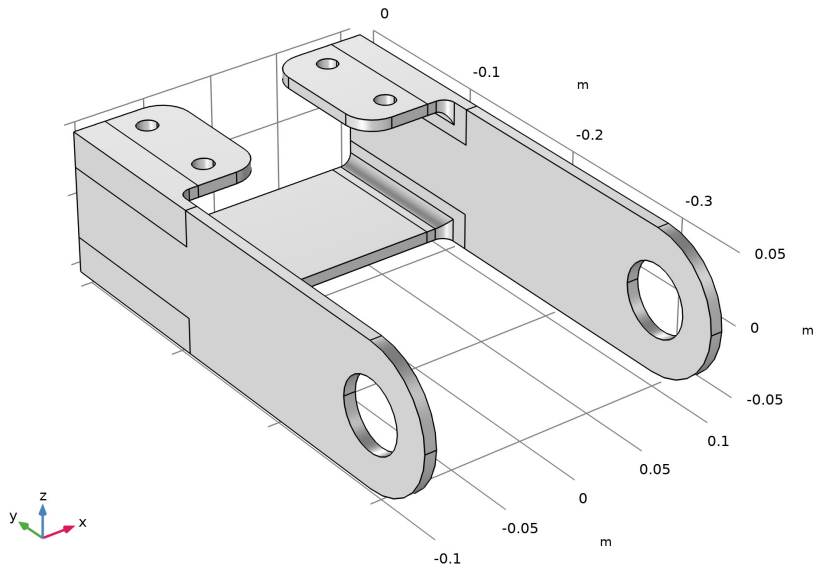


Figure 1: Bracket geometry.

A rigid body, on which the time-varying load is applied, is assumed to be connected to the arms of the bracket. This body is modeled using a rigid connector. The following loads are applied:

- In the x direction, a 400 N load applied between 4 and 4.5 ms
- In the y direction, a 500 N force with 300 Hz sinusoidal time dependence
- In the z direction, a constant load -100 N

Rayleigh damping is chosen here with relative damping set to 0.01 and 0.02 at the frequencies 216 Hz and 1515 Hz, respectively. The resulting damping ratio curve is shown in [Figure 2](#).

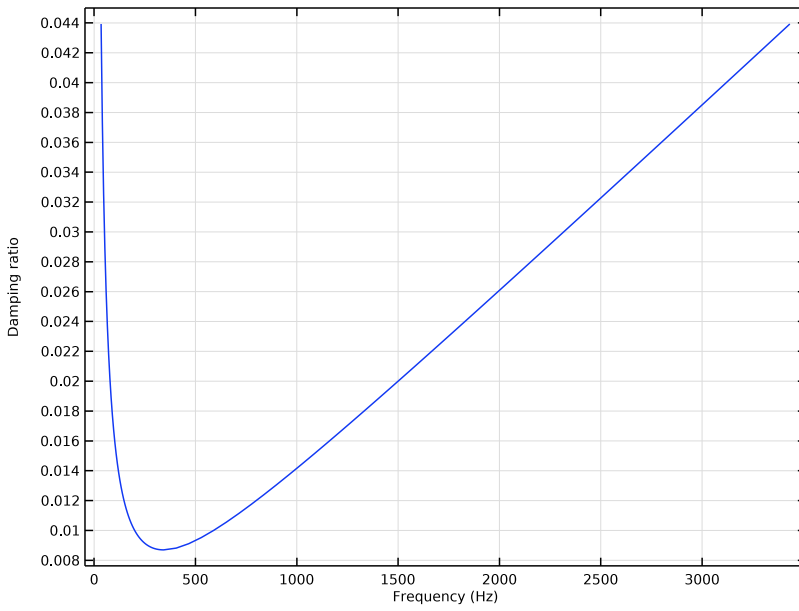


Figure 2: The damping ratio curve.

Results and Discussion

Figure 3 shows the rigid connector's displacements at the center of rotation versus time.

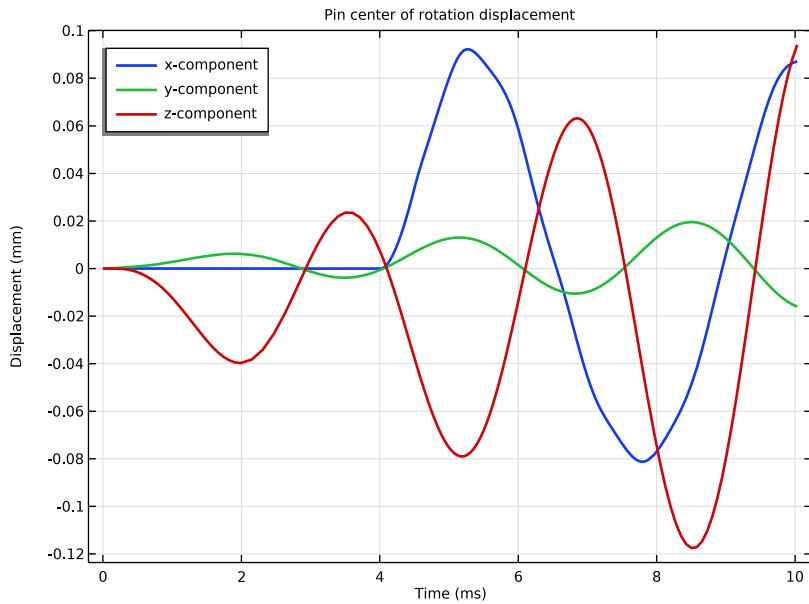
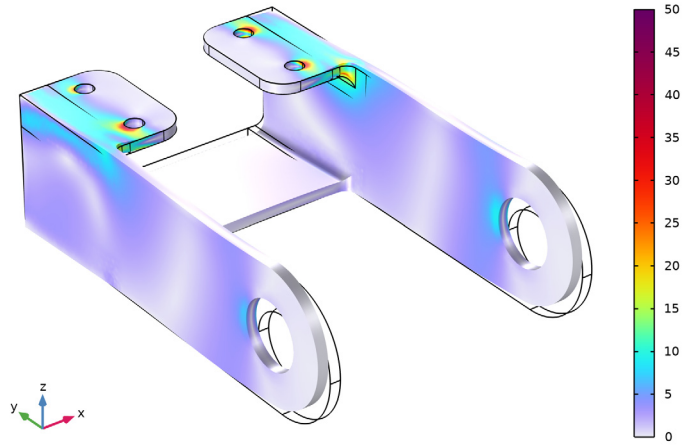


Figure 3: Displacement of the pin center of rotation versus time.

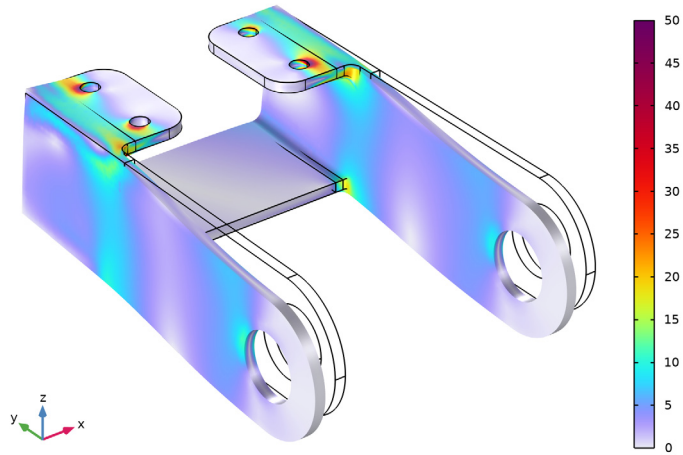
Time=7 ms

von Mises stress (MPa)



Time=8 ms

von Mises stress (MPa)



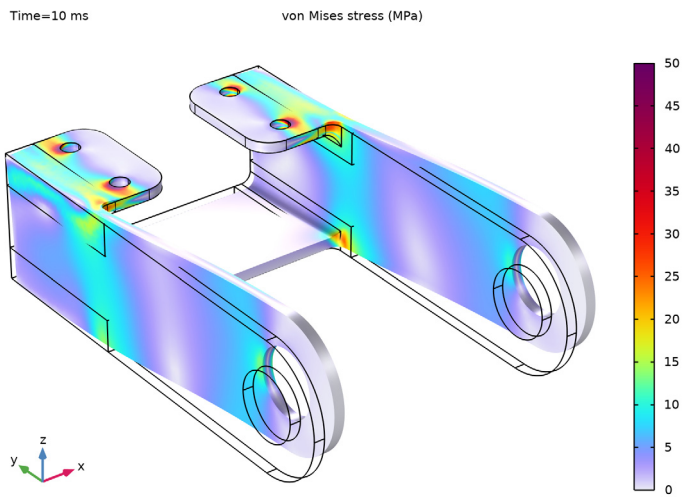
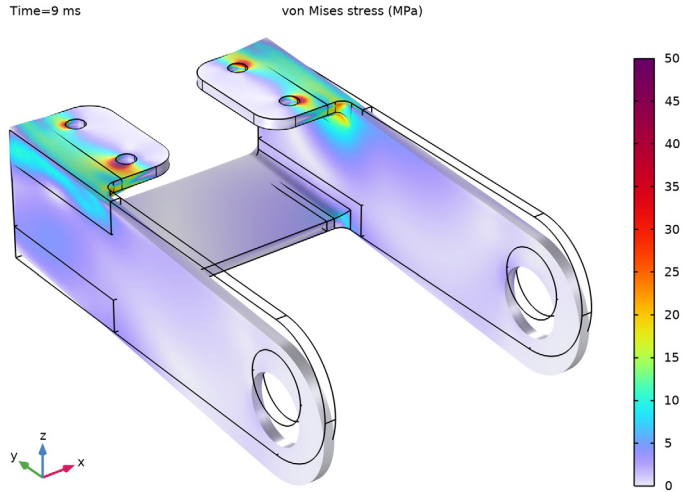


Figure 4: von Mises stress distribution in the bracket (from top to bottom, $t = 7$ ms, 8 ms, 9 ms and 10 ms).

Notes About the COMSOL Implementation

To accurately model the physical problem, you need to apply damping in a dynamic analysis. In COMSOL you have the possibility to add damping of several types, for example loss factor damping, viscous damping, or Rayleigh damping. Damping is also inherent in some material models, like viscoelasticity.

To describe time-dependent boundary conditions you can enter expressions using the built-in time variable, t .

To improve accuracy, the scaling of the displacement variables can be changed to correspond to the expected deformations. Manual scaling is the default for the displacement variables. The default manual scaling used for the structural displacement is 1% of the geometry size. In this example, the size of the geometry is about 200 mm, so the assumed displacement scale is 2 mm. In this case, the expected deformations are in the order of 0.1 mm. The manual scaling factor should thus be changed to 10^{-4} .


Because of the constant load in the z direction, the initial zero displacement assumption is not consistent with the boundary conditions. This singularity at initial time would induce high frequency dynamics requiring a small time step. To avoid issues at the initial time, it is recommended to use a smooth step function to ramp up the load. Another alternative would be to solve a stationary study first with only the static load, and then use the results from that study as initial conditions for the time-dependent analysis.

Two action buttons are provided in the **Damping Settings** section in order to visualize the damping ratio with respect to frequency. The first button shows a dynamic preview plot of the damping ratio, while the second button generates a plot in the **Results** node.

Application Library path: Structural_Mechanics_Module/Tutorials/
bracket_transient

Modeling Instructions

APPLICATION LIBRARIES

- 1 From the **File** menu, choose **Application Libraries**.
- 2 In the **Application Libraries** window, select **Structural Mechanics Module > Tutorials > bracket_basic** in the tree.
- 3 Click  **Open**.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

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

Damping 1

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

In this example, you will use Rayleigh damping defined using damping ratios.

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

3 From the **Input parameters** list, choose **Damping ratios**.

4 In the f_1 text field, type 200.

5 In the ζ_1 text field, type 1e-2.

6 In the f_2 text field, type 1500.

7 In the ζ_2 text field, type 2e-2.

In order to visualize the damping ratio curve, create the **Damping Ratio** plot through an action button from the **Damping Settings** section.

8 Click **Damping Ratio Preview** in the upper-right corner of the **Damping Settings** section.

From the menu, choose **Create Damping Ratio Plot**.

RESULTS

Damping Ratio Plot

1 In the **Model Builder** window, under **Results** click **Damping Ratio Plot**.

2 In the **Damping Ratio Plot** toolbar, click  **Plot**.

GLOBAL DEFINITIONS

Step 1 (step1)

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

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

3 In the **Location** text field, type 1 [ms].

4 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type 2 [ms].

Rectangle 1 (rect1)

1 In the **Home** toolbar, click  **Functions** and choose **Global > Rectangle**.

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

- 3 In the **Lower limit** text field, type 4[ms].
- 4 In the **Upper limit** text field, type 4.5[ms].
- 5 Click to expand the **Smoothing** section. Clear the **Size of transition zone** checkbox.

Parameters 1

- 1 In the **Model Builder** window, click **Parameters 1**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 In the table, enter the following settings:


| Name | Expression | Value | Description |
|------|----------------------------|-------|----------------------------|
| t | 0[s] | 0 s | Time |
| F_x | 400[N]*rect1(t) | 0 N | Applied force, x-component |
| F_y | 500[N]*sin(2*pi*300[Hz]*t) | 0 N | Applied force, y-component |
| F_z | -100[N]*step1(t) | -0 N | Applied force, z-component |

This way of adding time dependent functions as parameters is possible, since they can be evaluated when **t** is also a parameter. By doing so, it is easy to verify that the loads in all directions are zero at the initial time. This is consistent with the initial conditions.

SOLID MECHANICS (SOLID)

To represent the pin between the arms of the bracket use a rigid connector to which a load is applied.

Rigid Connector 1



- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Rigid Connector**.
- 2 In the **Settings** window for **Rigid Connector**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Pin Holes**.

Applied Force 1

- 1 In the **Physics** toolbar, click  **Attributes** and choose **Applied Force**.
- 2 In the **Settings** window for **Applied Force**, locate the **Applied Force** section.
- 3 Specify the **F** vector as

| | |
|-----|---|
| F_x | x |
| F_y | y |
| F_z | z |

ADD STUDY


- 1 In the **Home** toolbar, click  **Add Study** to open the **Add Study** window.
- 2 Go to the **Add Study** window.
- 3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies > Time Dependent**.
- 4 Click the **Add Study** button in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Study** to close the **Add Study** window.

STUDY 1

Step 1: Time Dependent


- 1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
- 2 From the **Time unit** list, choose **ms**.
- 3 In the **Output times** text field, type range (0,0.2,10).
With this time stepping, the solver automatically stores the solution every 0.2 ms from 0 to 10 ms.
The time-dependent solver adapts its time stepping based on a tolerance criterion. This ensures that the solver takes sufficiently small time steps if large variations occur between the specified output times.
- 4 Click to expand the **Results While Solving** section. Select the **Plot** checkbox.

Solution 1 (sol1)

- 1 In the **Study** toolbar, click  **Show Default Solver**.
Adjust the manual scaling in the solver settings to match the expected deformations.
- 2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.
- 3 In the **Model Builder** window, expand the **Study 1 > Solver Configurations > Solution 1 (sol1) > Dependent Variables 1** node, then click **Displacement Field (compl.u)**.
- 4 In the **Settings** window for **Field**, locate the **Scaling** section.
- 5 In the **Scale** text field, type $1e-4$.
You can speed up the computation for a small model by disabling the convergence plot. To do this, go to the **Options** menu and select **Preferences**. Under **Results**, click to clear the **Generate convergence plots** checkbox.
Create probes to visualize the pin displacement while computing.

DEFINITIONS

Pin displacement, x-component

- 1 In the **Definitions** toolbar, click  **Probes** and choose **Global Variable Probe**.
- 2 In the **Settings** window for **Global Variable Probe**, type Pin displacement, x-component in the **Label** text field.
- 3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1) > Solid Mechanics > Rigid connectors > Rigid Connector 1 > Rigid body displacement (spatial frame) - m > solid.rig1.u - Rigid body displacement, x-component**.
- 4 Locate the **Expression** section. From the **Table and plot unit** list, choose **mm**.


Pin displacement, y-component

- 1 Right-click **Pin displacement, x-component** and choose **Duplicate**.
- 2 In the **Settings** window for **Global Variable Probe**, type Pin displacement, y-component in the **Label** text field.
- 3 Locate the **Expression** section. In the **Expression** text field, type `solid.rig1.v`.

Pin displacement, z-component

- 1 Right-click **Pin displacement, y-component** and choose **Duplicate**.
- 2 In the **Settings** window for **Global Variable Probe**, type Pin displacement, z-component in the **Label** text field.
- 3 Locate the **Expression** section. In the **Expression** text field, type `solid.rig1.w`.

STUDY 1


In the **Study** toolbar, click  **Compute**.

RESULTS

Stress (solid)



The default plot shows the stress distribution at the final time. You can change the time for the plot display in the **Time list** of the plot group settings.

Volume 1

- 1 Click the  **Show Grid** button in the **Graphics** toolbar.
- 2 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Volume 1**.
- 3 In the **Settings** window for **Volume**, click to expand the **Range** section.
- 4 Select the **Manual color range** checkbox.

- 5 In the **Maximum** text field, type 50.
- 6 Locate the **Coloring and Style** section. From the **Color table transformation** list, choose **Nonlinear**.
- 7 In the **Color calibration parameter** text field, type -1.
To visualize the results in an animation, create a player.

Animation 1

- 1 In the **Results** toolbar, click  **Animation** and choose **Player**.
- 2 In the **Settings** window for **Animation**, locate the **Scene** section.
- 3 From the **Subject** list, choose **Stress (solid)**.
- 4 Locate the **Frames** section. From the **Frame selection** list, choose **All**.
- 5 Click  **Show Frame**.

The movie is generated, and then played. To replay the movie, click the **Play** button in the **Graphics** toolbar.

If you want to export a movie, right-click **Export** and create an **Animation** node.

Displacement at Center of Pin

- 1 In the **Model Builder** window, under **Results** click **Probe Plot Group 2**.
- 2 In the **Settings** window for **ID Plot Group**, type Displacement at Center of Pin in the **Label** text field.
- 3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
- 4 In the **Title** text area, type Pin center of rotation displacement.
- 5 Locate the **Plot Settings** section.
- 6 Select the **y-axis label** checkbox. In the associated text field, type Displacement (mm).
- 7 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Probe Table Graph 1

- 1 In the **Model Builder** window, expand the **Displacement at Center of Pin** node, then click **Probe Table Graph 1**.
- 2 In the **Settings** window for **Table Graph**, locate the **Coloring and Style** section.
- 3 From the **Width** list, choose **2**.
- 4 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends

x - component

y - component

z - component

6 In the **Displacement at Center of Pin** toolbar, click  **Plot**.