Created in COMSOL Multiphysics 5.6

Bracket — Transient Analysis

Introduction

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

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

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

Model Definition

This model is an extension 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.

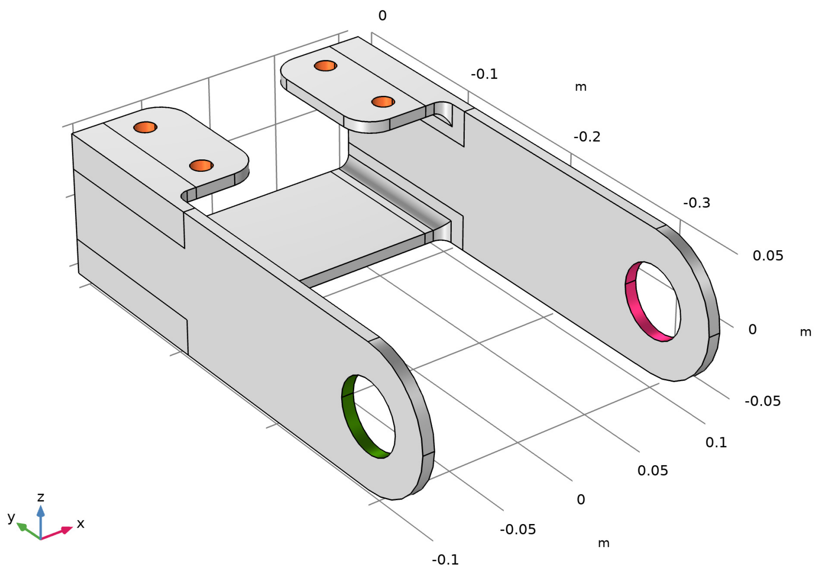

Figure 1: Bracket geometry.

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

Results and Discussion

Figure 2 shows the rigid connector's displacements at the center of rotation versus the time.

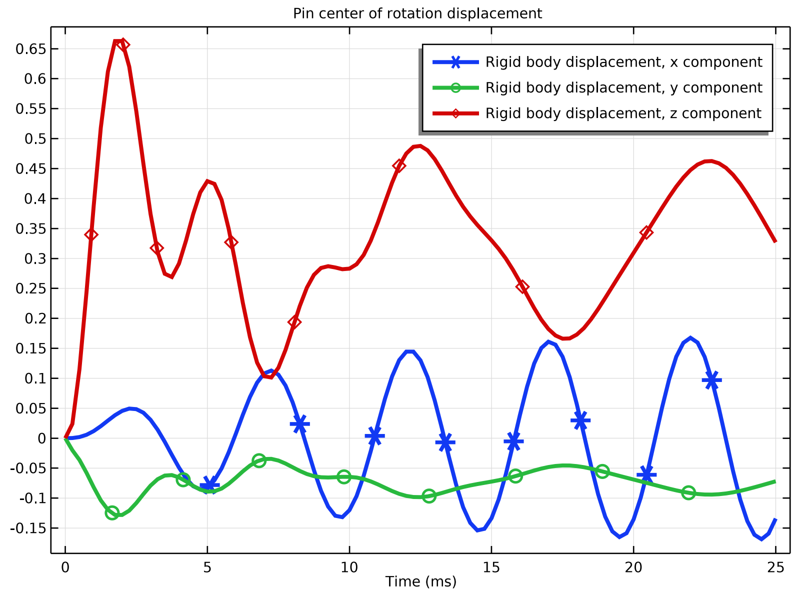

Figure 2: Displacement of the pin center of rotation vs time.

Notes About the COMSOL Implementation

To accurately model the physical problem, you need to apply damping in a dynamic analysis. In COMSOL you have the possibility to add damping of several types: isotropic loss factor, anisotropic loss factor, viscous damping, or Rayleigh damping.

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

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

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_transient`

Modeling Instructions

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

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

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

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

Damping 1

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

In this example, you will use Rayleigh damping.

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

3 In the α_{dM} text field, type `50`.

4 In the β_{dK} text field, type `1e-4`.

To represent the pin between the arms of the bracket you can use a rigid connector, to which a load 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

`100*sin(2*pi*t*200[1/s])`	x
`-11000`	y
`700*sin(2*pi*t*100[1/s])`	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 **Add Study** 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(0,0.25,25)`.

With this time stepping, the solver automatically stores the solution every 0.25 ms from 0 to 25 ms.

The time-dependent solver adapts its time stepping based on a tolerance criterion. This ensures that the solver takes small enough time steps if large variations occur between the specified output times.

4 From the **Tolerance** list, choose **User controlled**.

5 In the **Relative tolerance** text field, type `0.001`.

6 Click to expand the **Results While Solving** section. Select the **Plot** check box.

Solution 1 (sol1)

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

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

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

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

RESULTS

Stress (solid)

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

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

Plot the displacement of the center of rotation of the rigid body versus the time.

1D Plot Group 3

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

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

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Pin center of rotation displacement`.

5 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Global 1

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

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

3 In the table, enter the following settings:

Expression	Unit	Description
solid.rig1.u	mm	Rigid body displacement, x component
solid.rig1.v	mm	Rigid body displacement, y component
solid.rig1.w	mm	Rigid body displacement, z component

4 Click to expand the **Legends** section. Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

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

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

1D Plot Group 3

1 In the **Model Builder** window, click **1D Plot Group 3**.

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

3 From the **Position** list, choose **Upper right**.

4 In the **1D Plot Group 3** toolbar, click **Plot**.

To visualize the results in an animation, create a player.

Animation 1

1 In the **Results** toolbar, click **Animation** and choose **File**.

2 In the **Settings** window for **Animation**, locate the **Target** section.

3 From the **Target** list, choose **Player**.

4 Locate the **Frames** section. From the **Frame selection** list, choose **All**.

5 Click **Show Frame**.

COMSOL Multiphysics generates the movie and then plays it. To replay the movie, click the **Play** button in the **Graphics** toolbar.

If you want to export a movie in GIF, Flash, or AVI format, right-click **Export** and create an **Animation** node.