# A Mechanical Component

The following detailed example consists of a plane stress analysis of a mechanical component. The component is cut from a 4-mm steel plate, and the thickness is small compared to its extent in the xy-plane. This aspect along with the fact that the applied loads lie in the xy-plane mean that you can assume that the out-of-plane stress is negligible. The Plane Stress application mode works under this assumption.
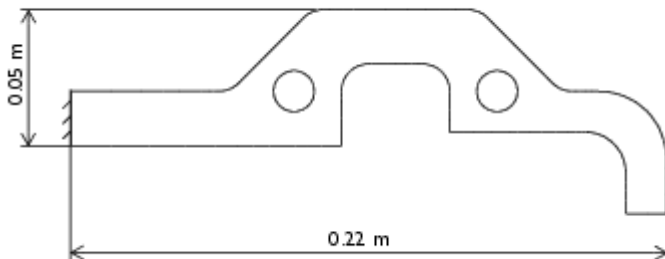
This example also introduces the seven basic analysis types available in the Structural Mechanics Module:

- Static analysis
- Eigenfrequency analysis
- Damped eigenfrequency analysis
- Time-dependent analysis
- Frequency response analysis
- Parametric analysis
- Quasi-static transient analysis

In the static, time-dependent, frequency response, parametric, and quasi-static analyses, the model contains a load to the lower-right end of the component. In all analyses the component is clamped on the left-hand side.

Static Analysis

A static analysis has no explicit or implicit time dependencies. This situation corresponds to the steady state of a transient analysis with constant boundary conditions and material properties.

The purpose of this analysis is to:

- Find the maximum stress level and compare it with the material's yield strength
- Find the static deflection at the point where the load is applied and compare it with the time-dependent analysis.

The Model Library note immediately below appears in the discussion of every model. The path indicates the location of the model file on the **Model Library** page in the **Model Navigator**.
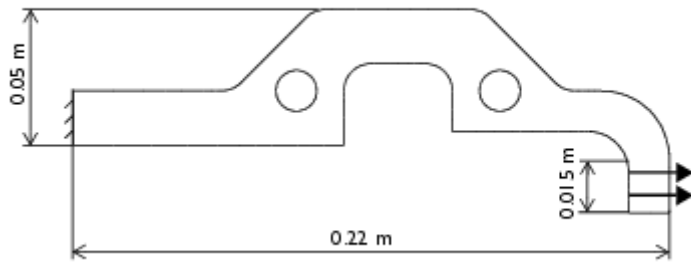
Model Library Path: `Structural_Mechanics_Module/Tutorial_Models/component_static`

MODEL DEFINITION

The model starts with a mechanical component whose shape and overall dimensions are shown in the following figure. It is possible to create the geometry in a CAD package and import it into COMSOL Multiphysics as a DXF file.

Some key parameters for the model:

Material

- Structural steel as taken from the material library

- Thickness of 4 mm

Load

A 900 N force in the x direction on the inside of the right end

Constraints

The left edge is fixed.

RESULTS

After the analysis you find that the von Mises effective stress has a maximum value of $2.8 \cdot 10^8$ N/m^2 = 280 MPa, which, compared with the material's yield strength of 350 MPa, results in a utilization factor of 80%.

The analysis also gives the static displacements at the bottom end of the edge where the load is applied:

| RESULT	x DIRECTION	y DIRECTION
Displacement	6.89e-4 m	1.14e-3 m

Now take a step-by-step look at how to achieve these results using the Structural Mechanics Module.

MODELING USING THE GRAPHICAL USER INTERFACE

Model Navigator

1   In the **Model Navigator** go to the **New** page, then select **2D** from the **Space dimension** list.

2   On that same page go to the list of application modes and select **Structural Mechanics Module>Plane Stress>Static analysis**.
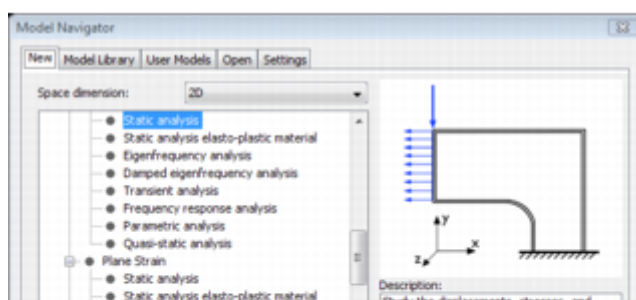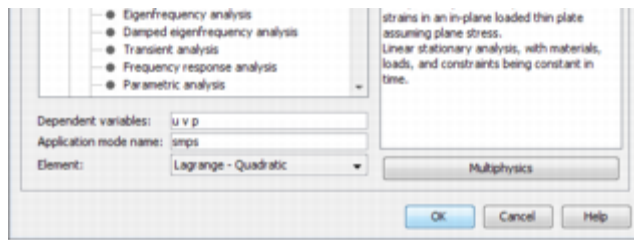
3 Click **OK** to close the **Model Navigator**.

## Geometry Modeling

Instead of drawing the geometry directly in the user interface, you can import a DXF file.

1 From the **File** menu select **Import>CAD Data From File**.

2 The **Import CAD Data from File** dialog box appears. Open the DXF file
   `component_geometry.dxf`, which is located in
   `/models/Structural_Mechanics_Module/Tutorial_Models` in the COMSOL installation
   directory.

3 Click **Import** to close the dialog box, then click the **Zoom Extents** button on the Main toolbar to
   view the entire geometry.
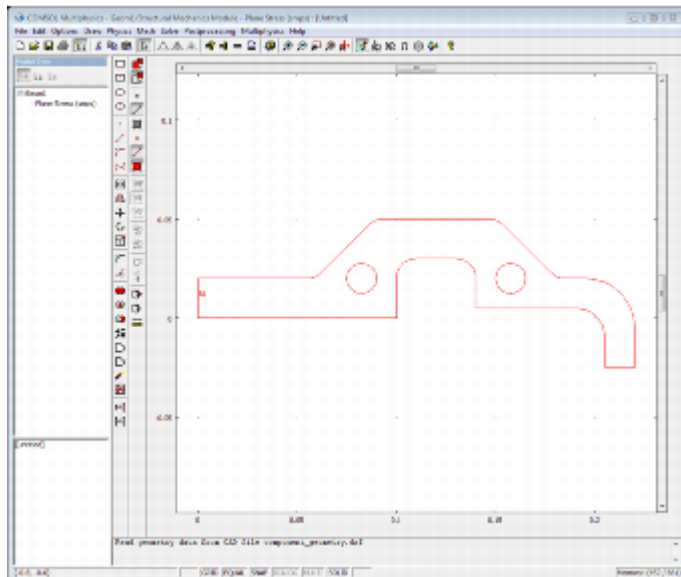
4 Select all the lines in the drawing by clicking the left mouse button and dragging a rubber-band box
   that encloses all the edges (or use the shortcut key Ctrl+A).

5 Construct a solid from the edges imported from the DXF file. To coerce the edges to a solid, click the
   Coerce to Solid button in the Draw toolbar.

6 Split the solid by clicking the **Split Object** button in the Draw toolbar to create independent objects
   for the holes.

7 Click the **Difference** button on the Draw toolbar to cut out the hole in the geometry.
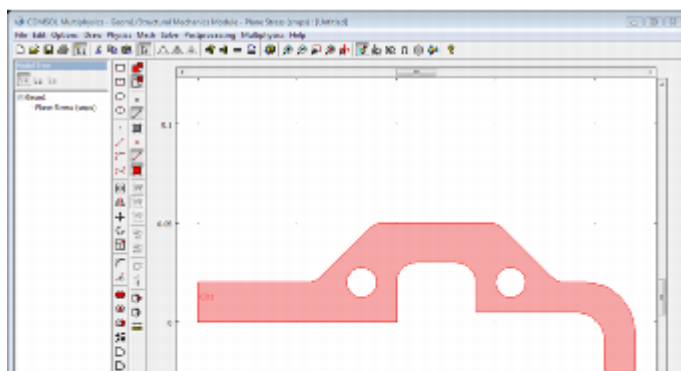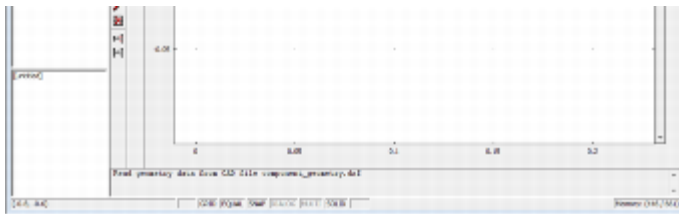
Physics Settings

Application Mode Properties

Set the analysis type to static (the analysis type is already static in this model, so this step is not necessary).

1 From the **Physics** menu select **Properties** to open the **Application Mode Properties** dialog box.

2 The **Analysis type** list defines which analysis to perform and some other properties for the analysis. Because you selected a static analysis from the **Model Navigator**, the analysis type is already defined as being **Static**.
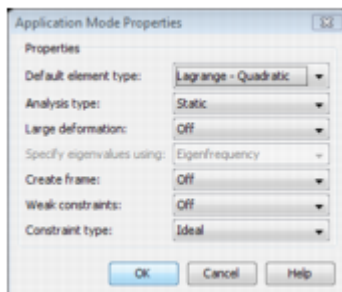
Boundary Settings

In boundary mode you specify loads and constraints. By default all edges are free, which means that there are no loads or constraints. Loads can be defined as force per area (using the thickness) or as force per length. The default is force per length, which this example uses.

The total static force on Edge 12 on the lower right end is 900 N. This results in a distributed force along the edge, which is 15 millimeter long, of 60 kN/meter. The left edge is fixed in both directions.

1 Open the **Boundary Settings** dialog box by going to the **Physics** menu and selecting **Boundary Settings**.

2 Select Boundary 1. Click the **Constraint** tab, then select **Fixed** from the **Constraint condition** list.

3 Select Boundary 12. Click the **Load** tab and enter `60e3` in the $\mathbf{F_X}$ edit field.
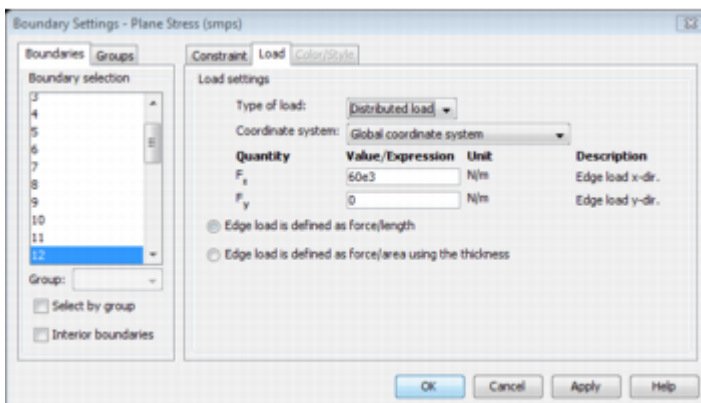
You have set the boundary conditions according to the following table:

	BOUNDARY 1	BOUNDARY 12

Page	Constraint		Load	
	Constraint condition	Fixed	F_x	60e3

You frequently encounter tables such as this one both in the remainder of this chapter and throughout the Model Library. The row marked "Page" indicates on which page in the dialog box you find the setting.
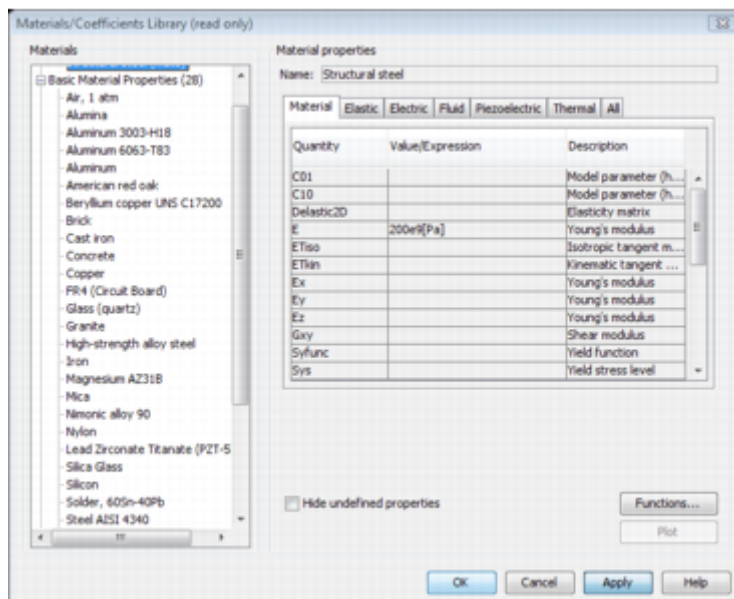
4   Click **OK** to close the **Boundary Settings** dialog box.
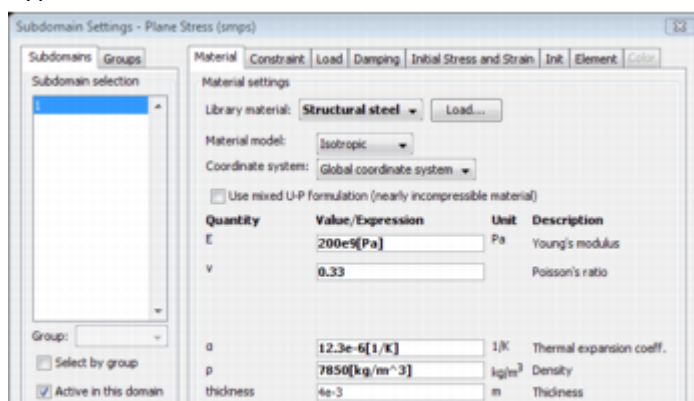
Subdomain Settings

In subdomain mode you specify material properties and element order. The material properties are selected from the material library or entered explicitly by typing the corresponding value or expression. This example uses the material library. Before you can use a material from the library, you must add it to the geometry. Do so from the **Options** menu or directly from the **Subdomain Settings** dialog box.

1   Select **Subdomain Settings** from the **Physics** menu.

2   Select Subdomain 1.

3   Click the **Load** button on the **Material** page to open the **Materials/Coefficients Library** dialog box.

4   Select **Structural steel** from the **Basic Material Properties** folder in the **Materials** list and click **Apply**.

You have now added the structural steel material to the geometry, an entry that you can see in the **Model** part of the **Materials** list.

5   Click **OK** to close the **Material/Coefficients Library** dialog box. **Structural steel** is now selected in the **Library material** list in the **Subdomain Settings** dialog box.

6   Type 4e-3 in the **thickness** edit field.
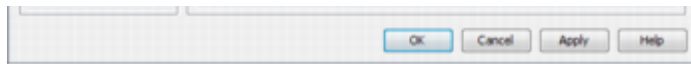
In tabular form, the material settings are:

	SUBDOMAIN 1	
Page	Material	
	Library material	Structural steel
	thickness	4e-3

The **Element** page shows the shape functions currently in use in the selected subdomain. The shape functions define polynomials used in interpolating the dependent variables. Typically, you do not need to change this setting.
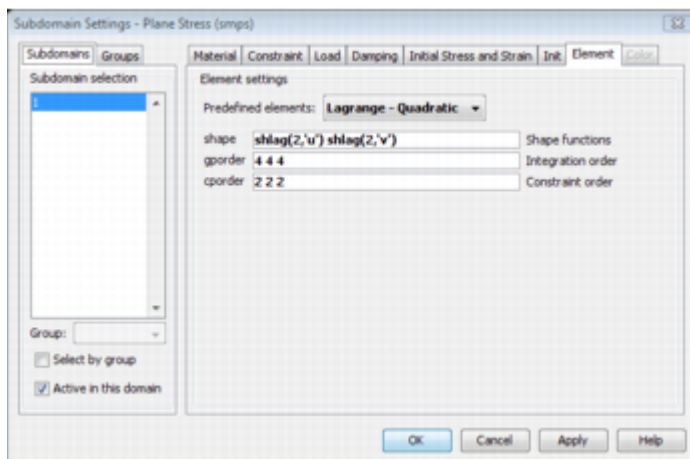
The default element type is quadratic Lagrange elements. They use 2nd-order polynomials, which is often a good trade-off between memory usage and accuracy. You can use linear Lagrange elements to reduce memory consumption when accurate stress or strain results are not required.

Mesh Generation

The **Free Mesh Parameters** dialog box, found on the **Mesh** menu, gives access to a number of parameters that control the mesh's density and gradation. Use the default mesh parameters.

1   Select **Free Mesh Parameters** from the **Mesh** menu to look at the **Free Mesh Parameters** dialog box.
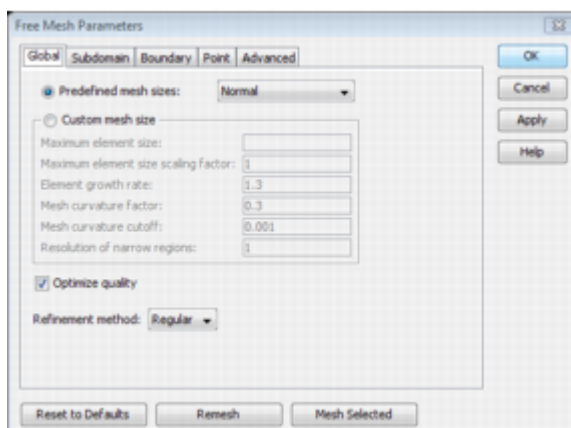
2   Click **OK** to use the default mesh settings.

3   Initialize the mesh by clicking the **Initialize Mesh** button on the Main toolbar.
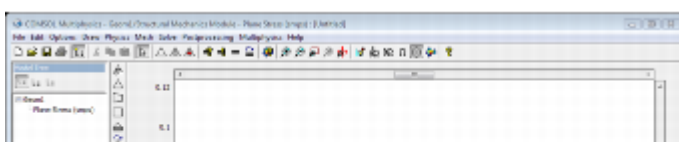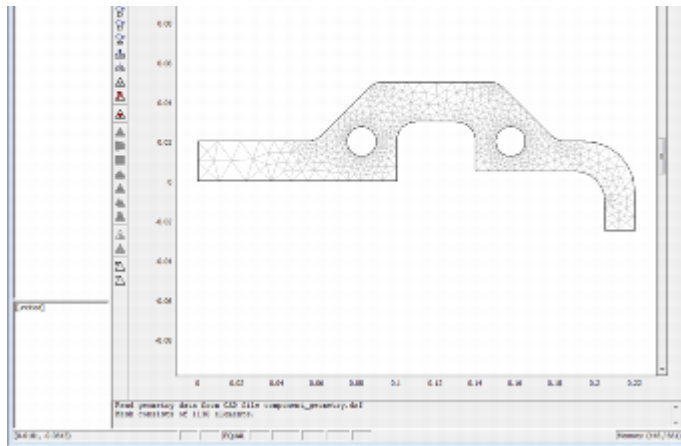
Computing the Solution

The analysis type controls which solver to use through the **Auto select solver** option in the **Solver Parameters** dialog box. This option is enabled as the default, so there is no need to change the solver settings because stationary is the solver associated with the static analysis type.

To compute the solution, either click the **Solve** button (**=**) on the Main toolbar or select **Solve Problem** from the **Solve** menu.

Postprocessing and Visualization

In *Postprocessing mode* you can, for example, add additional plot types and set parameters for plots. The postprocessing utilities can visualize any expression containing, for example, the solution variables, their derivatives, and the space coordinates. Many frequently used expressions are predefined as *postprocessing variables*, and they are directly available from lists in the **Plot Parameters** dialog box.

As soon as the solution is ready, a default plot appears. In the Plane Stress application mode the default visualizes the *von Mises effective stress*. Plot the *von Mises* stress together with the deformed shape of the component:

1   Select **Plot Parameters** from the **Postprocessing** menu.

2   On the **General** page select the **Deformed shape** and **Surface** check boxes.
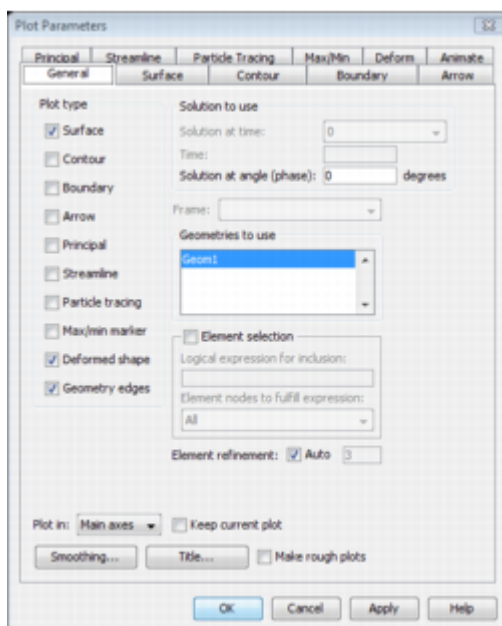
3   Click **OK** to close the **Plot Parameters** dialog box.

4   Click the **Zoom Extents** button on the Main toolbar.
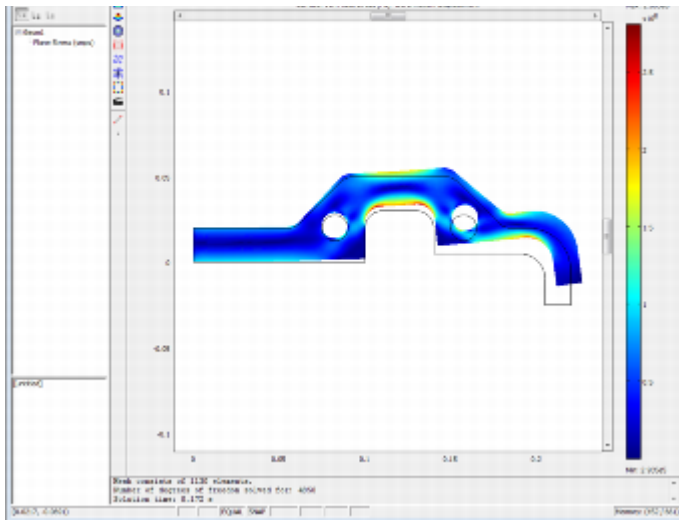
The deformation is exaggerated using automatic scaling. You can control the scaling of the deformation from the **Plot Parameters** dialog box on the **Deform** page.

Eigenfrequency Analysis

An eigenfrequency analysis finds the eigenfrequencies and modes of deformation of a component. The eigenfrequencies f in the structural mechanics field are related to the eigenvalues λ returned by the solvers through

$$f = -\frac{\text{Im}(\lambda)}{2\pi}$$

In COMSOL Multiphysics you can choose between working with eigenfrequencies and working with eigenvalues according to your preferences. Eigenfrequencies are the default option for all application modes in the Structural Mechanics Module.

The purpose of the following eigenfrequency analysis is to find the six lowest eigenfrequencies and corresponding mode shapes.

Model Library Path:
Structural_Mechanics_Module/Tutorial_Models/component_eigen

MODEL DEFINITION

The geometry, material, loads, and constraints are the same as for the static analysis; see the description on page 17 for details.

RESULTS

The first six eigenfrequencies are:

EIGENFREQUENCY NUMBER	FREQUENCY
f_1	300.67 Hz
f_2	1346.42 Hz
f_3	3456.12 Hz
f_4	4405.38 Hz
f_5	8410.73 Hz
f_6	11 kHz

MODELING USING THE GRAPHICAL USER INTERFACE

Model Navigator

The eigenfrequency analysis is described as if it were done after the static analysis described on page 17, so the Plane Stress application mode is already selected.

Geometry Modeling

This was already done in the static analysis.

Physics Settings

This model uses the same material, loads, and constraints as the static analysis.

Application Mode Parameters

Change the analysis type to eigenfrequency analysis:

1 From the **Physics** menu, choose **Properties** to open the **Application Mode Properties** dialog box.

   The **Analysis type** list defines which analysis to perform and which equation to solve.

2 Select **Eigenfrequency** from the **Analysis type** list; then click **OK**.

Mesh Generation

This model uses the same mesh, so there is no difference from the static analysis.

Computing the Solution

The analysis type controls which solver to use through the **Auto select solver** option in the **Solver Parameters** dialog box. This option is enabled as the default, so there is no need to change the solver settings. The eigenfrequency solver is the solver associated with the eigenfrequency analysis type.

Examine the eigenfrequencies solver parameters

1 From the **Solve** menu, choose **Solver Parameters** to open the **Solver Parameters** dialog box.

   The **Eigenfrequency** solver is already selected through the **Auto select solver** option. The number of eigenfrequencies to compute is controlled from the **General** page. Use the default settings to solve for the six lowest eigenfrequencies.
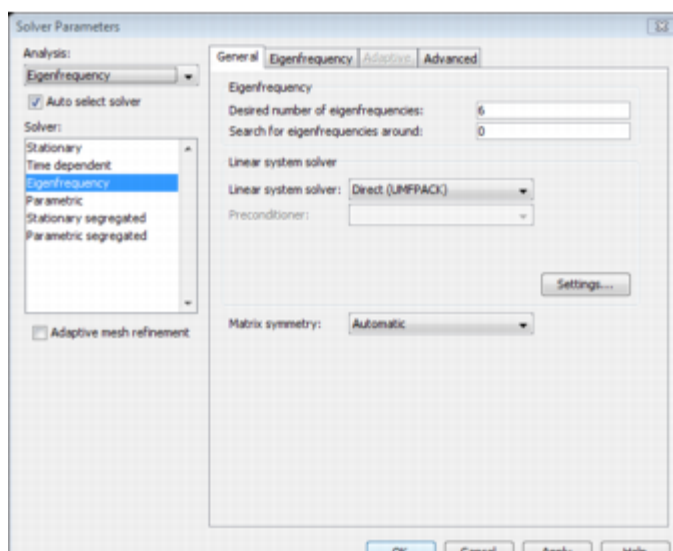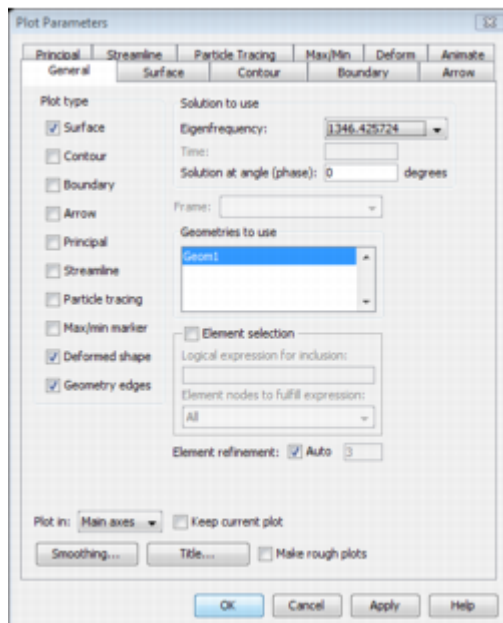
2   Click **OK** to close the **Solver Parameters** dialog box.

3   To compute the solution, either click the **Solve** button (**=**) on the Main toolbar or select **Solve Problem** from the **Solve** menu.

Postprocessing and Visualization

You can select which eigenfrequency to work with from a list on the **General** page in the **Plot Parameters** dialog box. The default plot shows the eigenmode corresponding to the lowest eigenfrequency. The eigenmodes are scaled (normalized) so the size of deformations should be compared only within an eigenmode and not among modes.

1   Choose **Plot Parameters** from the **Postprocessing** menu.

2   Select the **Deformed shape** check box on the **General** page.

3   Select the second eigenfrequency from the **Eigenfrequency** list.

4   Click the **Surface** tab and select **Total displacement** from the **Predefined quantities** list.

5   Click **OK** to close the **Plot Parameters** dialog box to look at the mode shape of the second eiegenfrequency.
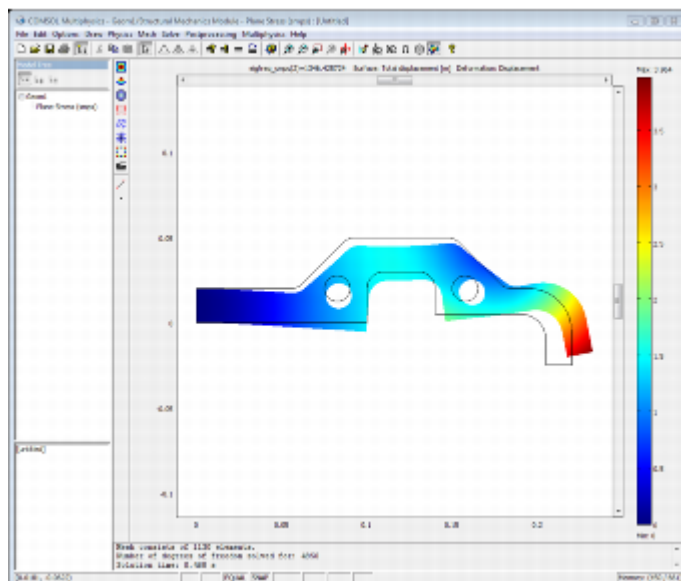
Time-Dependent Analysis

A time-dependent analysis solves for the transient solution of the displacements and velocities as functions of time. The material properties, forces, and boundary conditions can vary in time.

The purpose of this analysis is to find the transient response from a harmonic load with the same amplitude as the static load during the first two periods. The excitation frequency is 500 Hz, which is between the first and second eigenfrequency found in the eigenfrequency analysis.

Model Library Path: `Structural_Mechanics_Module/Tutorial_Models/component_transient`

MODEL DEFINITION

The geometry, material, and constraints are the same as for the static analysis (see the static description on for details).

Load

This model uses a harmonic load with an excitation frequency of 500 Hz on the same edge and with the same amplitude as the static problem. The expression for the load can be written

$$F_x = 900 \cdot \sin(2\pi \cdot 500 \cdot t)$$

where t denotes the time.

Damping

Damping is important in a transient analysis but can be difficult to model. For transient analysis, the Structural Mechanics Module supports Rayleigh damping and loss factor damping. It is also possible to use no damping.

This model uses Rayleigh damping, where you specify damping parameters that are proportional to the mass (α_{dM}) and stiffness (β_{dK}) in the following way:

$$C = \alpha_{dM}M + \beta_{dK}K$$

where C is the damping matrix, M is the mass matrix, and K is the stiffness matrix. The damping is specified locally; you can specify different damping parameters in different parts of the model.

You leave the damping parameters at their default values from the previous analyses because they are used only for transient and frequency response analyses.

To find good values for the Rayleigh damping, you can use the relations between the critical damping ratio and the Rayleigh damping parameters. It is often easier to interpret the critical damping ratios, which are given by

$$\xi_i = \frac{\left(\frac{\alpha_{dM}}{\omega_i} + \beta_{dK} \cdot \omega_i\right)}{2}$$

where ξ_i is the critical damping ratio at a specific angular frequency ω_i. Knowing two pairs of corresponding ξ_i and ω_i results in a system of equations

$$\begin{bmatrix} \frac{1}{(2 \cdot \omega_1)} & \frac{\omega_1}{2} \\ \frac{1}{(2 \cdot \omega_2)} & \frac{\omega_2}{2} \end{bmatrix} \begin{bmatrix} \alpha_{dM} \\ \beta_{dK} \end{bmatrix} = \begin{bmatrix} \xi_1 \\ \xi_2 \end{bmatrix}$$

with the damping parameters as the unknown variables.

The structure has a constant damping ratio of 0.1. Select two frequencies near the excitation frequency, 200 Hz and 600 Hz, to calculate the damping parameters. You can do this in COMSOL Script or MATLAB with the following commands:
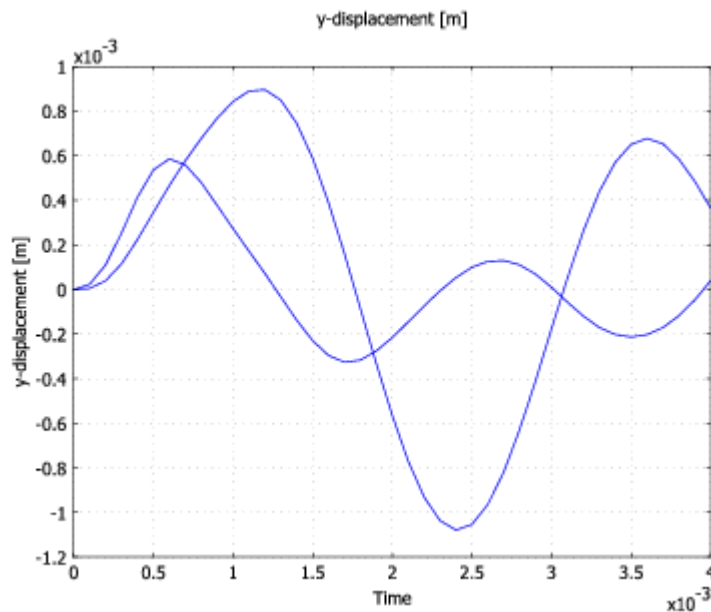
```
b=[0.1;0.1];
```

```
A=[1/(2*200*2*pi) 2*pi*200/2; 1/(2*600*2*pi) 2*pi*600/2];
% A*damp=b
damp=A\b;
alphadM=damp(1)
betadK=damp(2)
```

The result is $\alpha_{dM} = 1.88e2$, $\beta_{dK} = 3.98e-5$.

For more information see the section "Damping" on page 121.

RESULTS

The following plot shows the x- and y-displacements at the bottom end of the edge where the load is applied as a function of time:

MODELING USING THE GRAPHICAL USER INTERFACE

Model Navigator

The time-dependent analysis is described as if it were done after the static analysis described on page 17, so the Plane Stress application mode is already selected.

Geometry Modeling

This was already done in the static analysis (see the description on page 17 for details).

Physics Settings

Application Mode Parameters

Change the analysis type to time-dependent analysis:

1   Select **Properties** from the **Physics** menu to open the **Application Mode Properties** dialog box.

    The **Analysis type** list defines which analysis to perform and which equation to solve.

2   Select **Time dependent** from the **Analysis type** list.

3   Click **OK**.

Boundary Settings

The force at the right end is harmonic with the same amplitude as the force in the static case and with a frequency of 500 Hz,

$$F_x = 60 \cdot 10^3 \sin(2\pi 500t)$$

Use the unit syntax in COMSOL Multiphysics to specify the force in kN/m and the frequency in Hz. The variable t is used by COMSOL Multiphysics to denote the time in second. You specify the time steps in the **Solver Parameters** dialog box, which will be explained later in this model.

The left edge is still fixed in both directions.

1   Open the **Boundary Settings** dialog box by selecting **Boundary Settings** from the **Physics** menu.

2   Set boundary conditions according to the following table and then click **OK**:

	BOUNDARY 1		BOUNDARY 12	
Page	Constraint		Load	
	Constraint condition	Fixed	Fx	60[kN]*sin(2*pi*500[Hz]*t)
			Fy	0
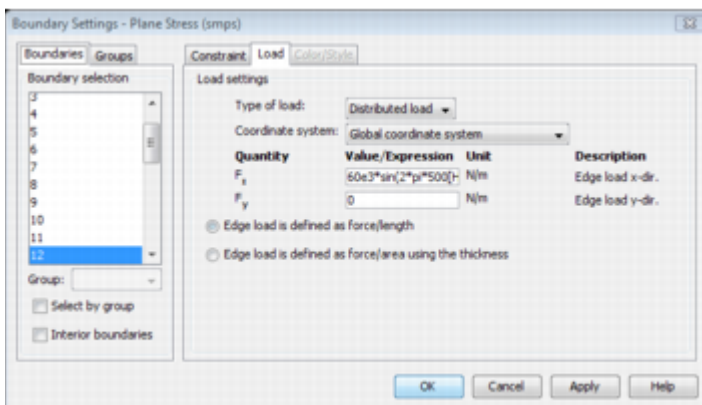
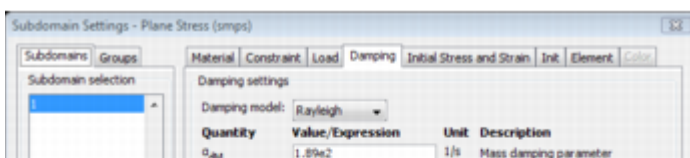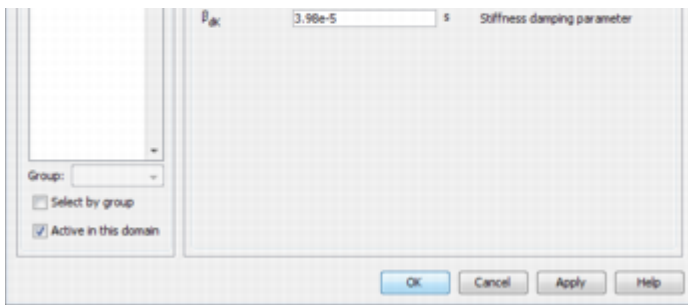## Subdomain Settings

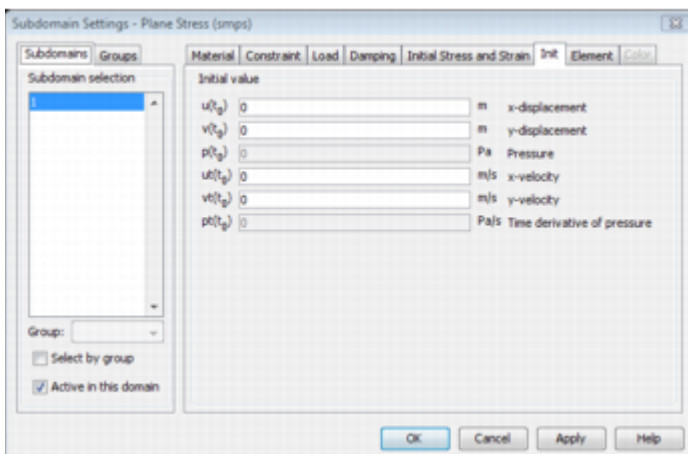The material properties are the same as in the static and eigenfrequency analyses.

1   Select **Subdomain Settings** from the **Physics** menu.

2   Click the **Damping** tab and enter the damping properties according to the following table; when done, click **OK**.

SETTINGS	SUBDOMAIN 1
Page	Damping
αdM	1.89e2
βdK	3.98e-5

To specify the initial values, click the **Init** tab.

In this case the initial deformation and velocity are zero, which are the default values.

Computing the Solution

The analysis type controls which solver to use through the **Auto select solver** option in the **Solver Parameters** dialog box. The option is enabled as the default, so there is no need to change the solver settings. The time-dependent solver is the one associated with the time-dependent analysis type. Solving for two periods with an excitation frequency of 500 Hz means solving for 4 ms.

Specify the time-dependent solver parameters:

1   Choose **Solver Parameters** from the **Solve** menu.

2   Type `0:1e-4:4e-3` in the **Times** edit field. This means that the solution is saved every 0.1 ms during the total solution time of 4 ms.

3   Type `1e-9` in the **Absolute tolerance** edit field. This is important for an accurate analysis (the absolute tolerance must be smaller than the displacements).
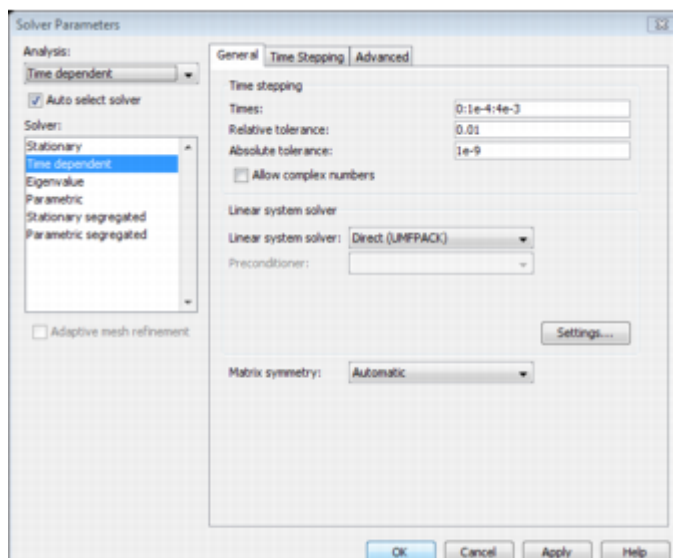
4   Click **OK**.

5   Compute the solution by clicking the **Solve** button (**=**) on the Main toolbar or by selecting **Solve Problem** from the **Solve** menu.

## Postprocessing and Visualization

You have access to the solution at all the computed time steps by going to the **Plot Parameters** dialog box, then the **General** page, and selecting them from the **Output time** list. The default plot shows the last time step.
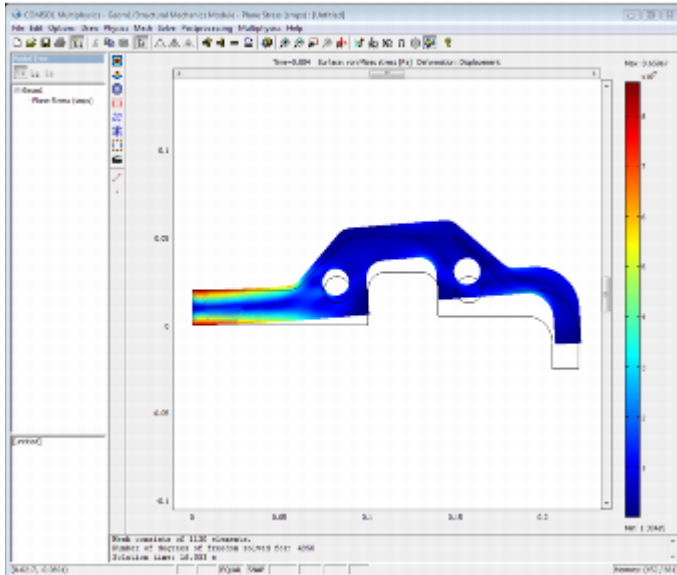
For a more quantitative view of the time evolution of the displacement, plot a graph of the displacements in the x and y directions at the lower-left corner as a function of time.

1   Select **Domain Plot Parameters** from the **Postprocessing** menu.

2   Go to the **General** page and select all the time steps from the **Solutions to use** list.

3   Click the **Point** tab (this selects a point plot automatically).

4   Click at the lower-left corner or select Point 27 from the **Point selection** list.

5   Select **x displacement** from the **Predefined quantities** list.

6   Click **Apply** to plot the x-displacement.

7   Click the **General** tab and then select the **Keep current plot** check box.

8   Click the **Point** tab and then select **y displacement** from the **Predefined quantities** list.

9   Click **Apply** to plot the y-displacement.
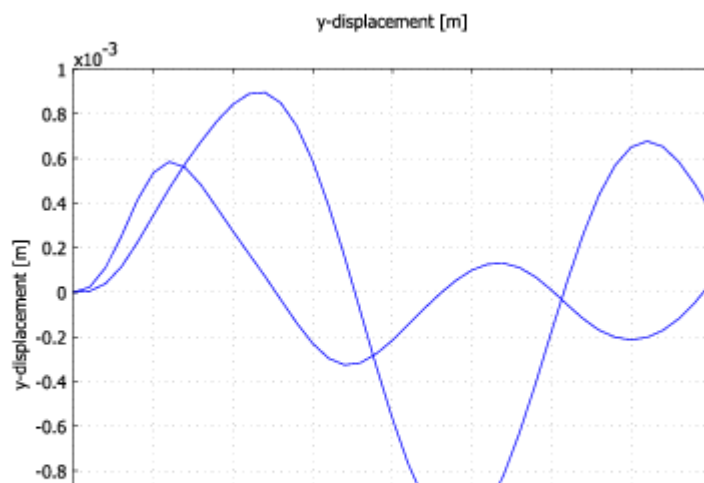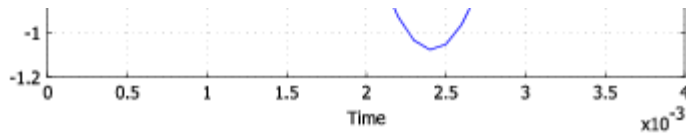
10 Click **OK** to close the dialog box.

Frequency Response Analysis

INTRODUCTION

A frequency response analysis solves for the steady-state response from harmonic excitation loads. The loads can have amplitudes and phase shifts that depend on the excitation frequency, f:

$$F_{freq} = F(f) \cdot \cos\left(2\pi f \cdot t + F_{Ph}(f) \cdot \frac{\pi}{180}\right)$$

where $F(f)$ is the amplitude and $F_{Ph}(f)$ is the phase shift of the load.

The purpose of this analysis is to find the transient response from a harmonic load with an excitation frequency in the range 200–600 Hz, which is near the first eigenfrequency found in the eigenfrequency analysis.

Model Library Path: `Structural_Mechanics_Module/Tutorial_Models/component_frequency`

MODEL DEFINITION

The geometry, material, and constraints are the same as for the static analysis (see the static description on for details).

Loads

This model uses a harmonic load with an excitation frequency between 200 and 600 Hz on the same edge and with the same amplitude as the static problem.
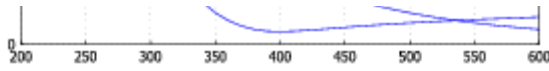
Damping

Damping is modeled using Rayleigh damping in the same way as for transient analysis (see for details): $\alpha_{dM} = 189$, $\beta_{dK} = 3.98 \cdot 10^{-5}$

RESULTS

The x- and y-displacements at the bottom end of the edge where the load is applied as a function of excitation frequency appear in the following figure:
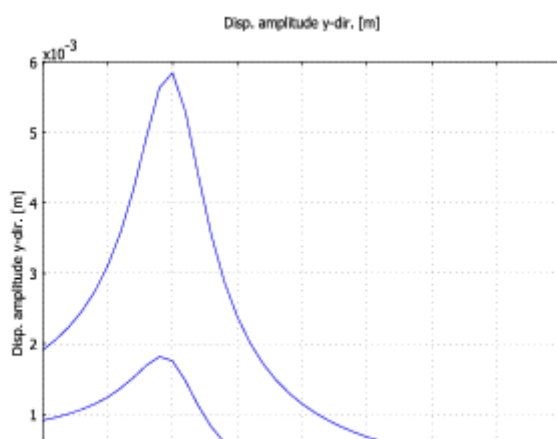
There is a peak near the first eigenfrequency where the steady-state response is as much as eight times higher than the static displacement. The amplitude for 500 Hz is below the transient response, which is natural because the damping reduces the transient effects.

MODELING USING THE GRAPHICAL USER INTERFACE

Model Navigator

The frequency response analysis is described as if it were done after the static analysis described on page 17, so the Plane Stress application mode is already selected.

Geometry Modeling

This was already done in the static analysis (see the static description on page 17 for details).

Physics Settings

Application Mode Parameters

Change the analysis type to frequency response analysis.

1 Choose **Properties** from the **Physics** menu to open the **Application Mode Properties** dialog box.

2 Select **Frequency response** from the **Analysis type** list.

   The **Analysis type** list defines which analysis to perform and which equation to solve.

3 Click **OK**.

Boundary Settings

Use the same amplitude as for the time-dependent analysis without any phase shift, which is the default setting. The left edge is fixed in both directions. In the transient analysis you entered the harmonic excitation load explicitly. The frequency response analysis uses a harmonic assumption and therefore you only need to specify the amplitude of the load, F.

1 Open the **Boundary Settings** dialog box by selecting **Boundary Settings** from the **Physics** menu.

2 Set boundary conditions according to the following table; click **OK** when done.

|	**BOUNDARY 1**		**BOUNDARY 12**	
Page	Constraint		Load	
	Constraint condition	Fixed	F_x	60e3
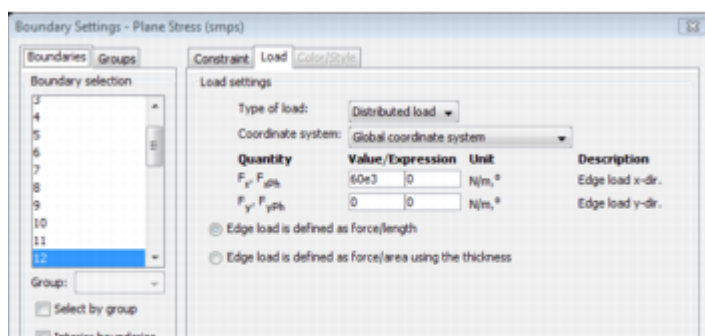
Subdomain Settings

The material properties and damping parameters are the same as in the time-dependent analysis on page 22.

Computing the Solution

Usually when performing a frequency response analysis you want to sweep over a frequency range. This can be done using the parametric solver. The analysis type controls which solver to use through the **Auto select solver** option in the **Solver Parameters** dialog box. This option is enabled as the default, and there is no need to change the solver settings. The parametric solver is the one associated with the frequency response analysis type. Solving for two periods with an excitation frequency of 500 Hz means solving for 0.004 s.
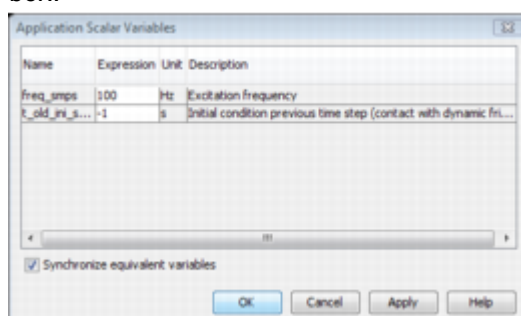
Specify the parametric solver parameters.

1   Select **Solver Parameters** from the **Solve** menu to open the **Solver Parameters** dialog box.

2   Go to the **General** page and enter `freq_smps` in the **Parameter name** edit field.

3   Enter `200:10:600` in the **Parameter values** edit field to specify an excitation frequency from 200 Hz to 600 Hz in steps of 10 Hz.
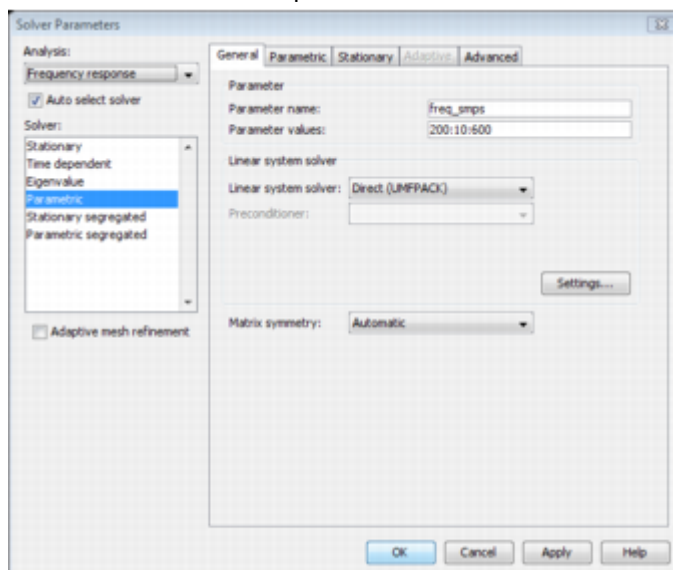
4   Click **OK** to close the **Solver Parameters** dialog box.

If you only want to use the parametric solver to sweep another parameter than the excitation frequency, specify the same **Analysis type** and enter the other parameter in the **Parameter name** edit field. You specify the excitation frequency in the **Application Scalar Variables** dialog box.

5   Select **Scalar Variables** from the **Physics** menu to open the **Application Scalar Variables** dialog box.

You specified a sweep using `freq` as the **Name of the parameter** in the **Solver Parameters** dialog box, which replaces the value of the **Excitation frequency** specified in the **Application Scalar Variables** dialog box.

6   Click **OK** to close the **Application Scalar Variables** dialog box.

7   To compute the solution, either click the **Solve** button (**=**) on the Main toolbar or select **Solve Problem** from the **Solve** menu.

Postprocessing and Visualization

To view the solution at all the different excitation frequencies, open the **Plot Parameters** dialog box, go to the **General** page and select among those frequencies. The default plot shows the von Mises stress for the last excitation frequency in the list.

1   Select **Plot Parameters** from the **Postprocessing** menu.

2   Check **Deformed shape** plot.

3   On the **Surface** page, in the **Surface Data** tab, select **von Mises stress** from the **Predefined quantities** list.
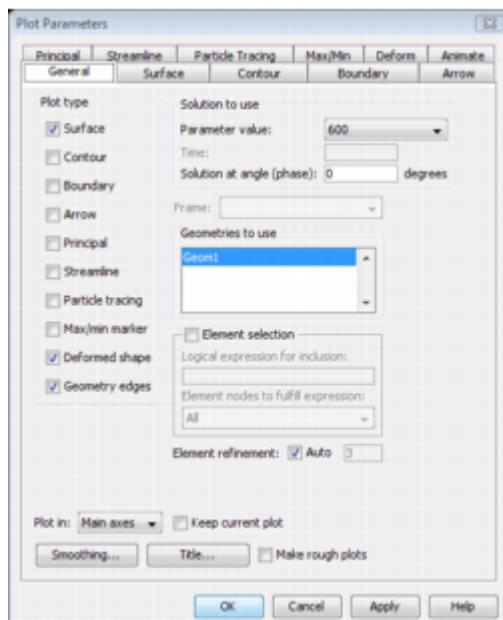
The result of a frequency response analysis is a complex time-dependent displacement field, which can be interpreted as an amplitude, u_{amp}, and a phase angle, u_{phase}. The actual displacement at any point in time is the real part of the solution

$$u = u_{amp}\cos(2\pi f \cdot t + u_{phase})$$

The software can visualize the amplitudes and phases as well as the solution at a specific angle (time). The **Solution at angle** parameter, which you can find on the **General** page, makes this easy. When plotting, COMSOL Multiphysics multiplies the solution by $e^{i\phi}$, where ϕ is the angle in radians that corresponds to the angle specified in degrees in the **Solution at angle** edit field. The plot shows the real part of the evaluated expression

$$u = u_{amp}\cos(\phi + u_{phase})$$

The angle ϕ is available as the variable `phase` (radians) and can be used in plot expressions.

4   Click **OK** to close the dialog box.
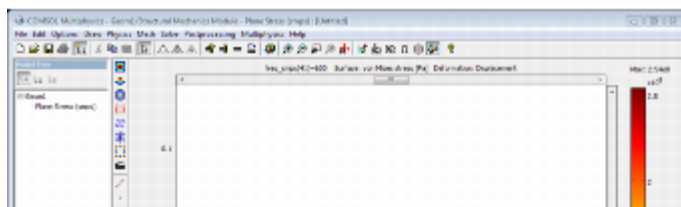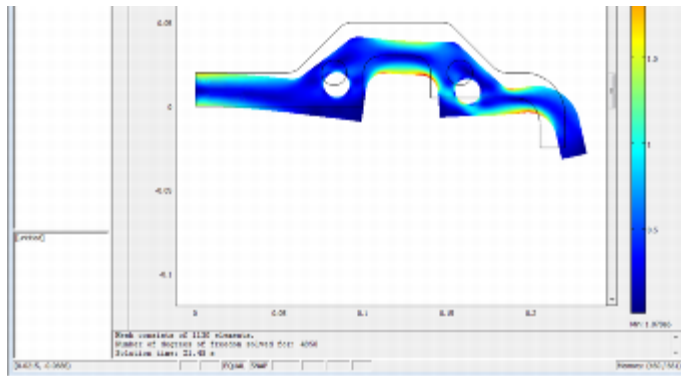
For a more quantitative view of the frequency evolution of the displacement, plot the x- and y-displacement amplitude at the lower-left corner as a function of the frequency.

1 Select **Domain Plot Parameters** from the **Postprocessing** menu.

2 Go to the **General** page and select all the excitation frequencies in the **Solutions to use** list.

3 Click the **Point** tab (this selects a point plot automatically).

4 Click at the lower-left corner or select Point 27 from the **Point selection** list.

5 Select **Disp. amplitude x-dir.** from the **Predefined quantities** list.

6 Click **Apply** to plot the x-displacement amplitude.

7 Click the **General** tab and then select the **Keep current plot** check box.

8 Click the **Point** tab and then select **Disp. amplitude y-dir.** from the **Predefined quantities** list.

9 Click **Apply** to plot the y-displacement amplitude.
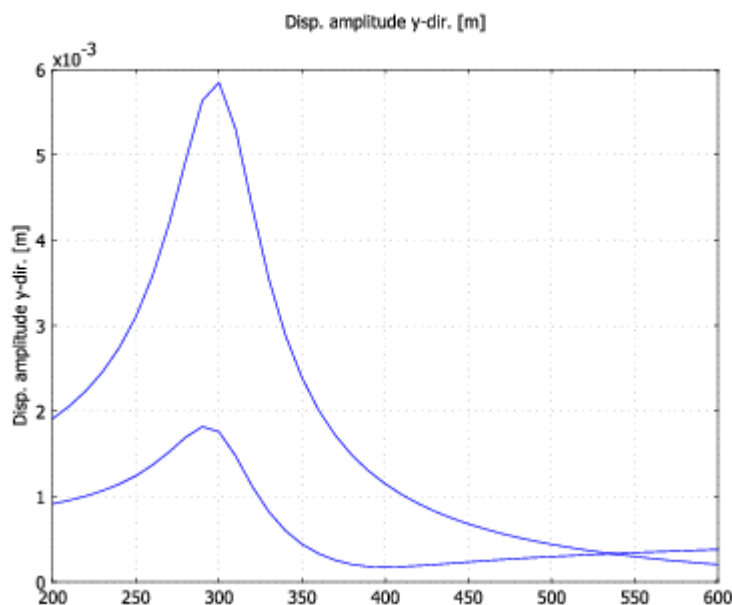
10 Click **OK**.

## Parametric Analysis

A parametric analysis solves for the static response as a function of a parameter. You freely define the parameter name and what it affects; it can be a material property, a load parameter, or something else.

The purpose of this example is to find the static response as a function of the direction of the force. The force is applied at the same edge as in the static analysis.

Model Library Path: `Structural_Mechanics_Module/Tutorial_Models/component_parametric`

MODEL DEFINITION

The geometry, material, and constraints are the same as for the static analysis (see the description on page 17 for details).
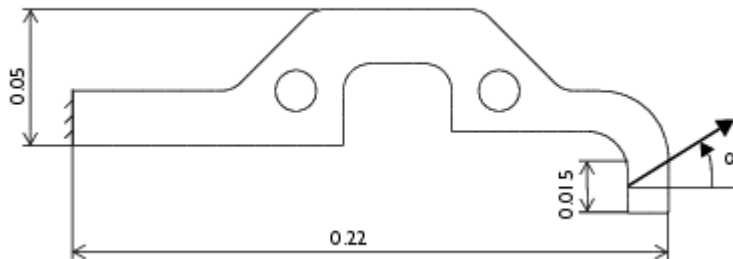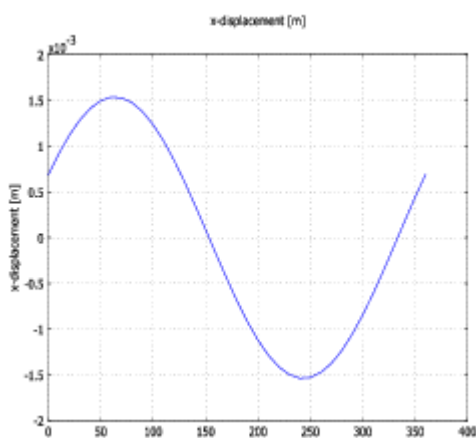

Load

This model uses a static load on the same edge and with the same magnitude as in the static model. The force is free to act in any direction.

RESULTS

The x-displacement at the bottom end of the edge where the load is applied as a function of the direction of the force (α) is shown in the following plot:

MODELING USING THE GRAPHICAL USER INTERFACE

Model Navigator

The parametric analysis is described as if it were done after the static analysis on page 17, so the Plane Stress application mode is already selected.

Geometry Modeling

This was already done in the static analysis (see the description on page 17 for details).

Physics Settings

Application Mode Parameters

Change the analysis type to parametric analysis.

1   Select **Properties** from the **Physics** menu to open the **Application Mode Properties** dialog box.

The **Analysis type** list defines which analysis to perform and which equation to solve.

2   Select **Parametric** from the **Analysis type** list; then click **OK**.

Boundary Settings

This model uses the same magnitude as for the static analysis, but the direction α is added as the parameter to vary. The force is divided into its x- and y-components as a function of the force direction, α:

$$F_x = 60 \cdot 10^3 \cos(\alpha)$$
$$F_y = 60 \cdot 10^3 \sin(\alpha)$$

1   Open the **Boundary Settings** dialog box by selecting **Boundary Settings** from the **Physics** menu.

2   Set boundary conditions according to the following table:

		BOUNDARY 1		**BOUNDARY 12**	
Page		Constraint		Load	
	Constraint condition		Fixed	F_x	60e3*cos (alpha*pi/180)
				F_y	60e3*sin (alpha*pi/180)

Because the cosine and sine functions take radians as input, a transformation from degrees to radians is done to be able to vary α from 0 to $360°$.
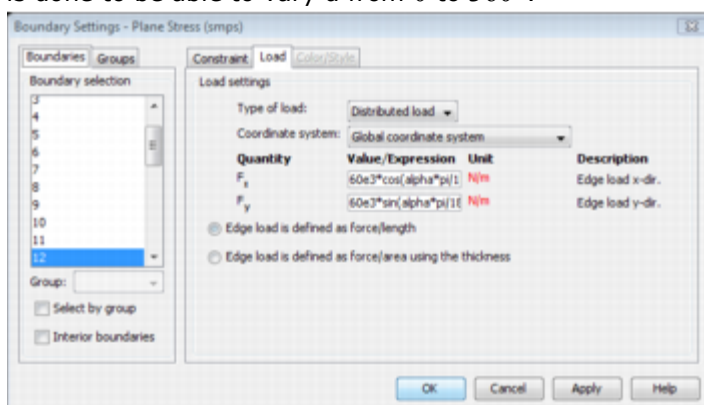
Subdomain Settings

The material properties are the same as in the static analysis on .
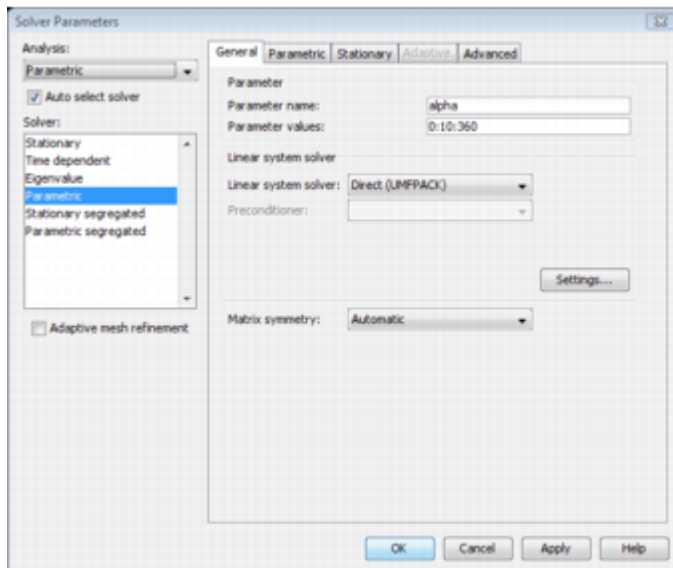
Computing the Solution

The analysis type controls which solver to use through the **Auto select solver** option in the **Solver Parameters** dialog box. This option is enabled as the default so there is no need to change the solver settings. The parametric solver is the one associated with the parametric analysis type.

Specify the force direction α as the parameter to vary between 0 and $360°$.

1   Choose **Solver Parameters** from the **Solve** menu to open the **Solver Parameters** dialog box.

2   In **Parameter**, enter `alpha` in the **Parameter name** edit field.

3   Enter `0:10:360` in the **Parameter values** edit field to specify the direction of the force between 0°
   to 360° in steps of 10°.

4   Click **OK** to close the **Solver Parameters** dialog box.

5   To compute the solution either click the **Solve** button (**=**) on the Main toolbar or choose **Solve
   Problem** from the **Solve** menu.

Postprocessing and Visualization

To view the solution for all different directions of the force, open the **Plot Parameters** dialog box, go to
the **General** page, then select from the **Parameter value** list. The default plot shows the von Mises
stress for the last direction angle `alpha` in the list.

For a more quantitative view of the angle evolution of the displacement, plot the x-displacement at the
lower-left corner as a function of the force direction.

1   Select **Domain Plot Parameters** from the **Postprocessing** menu.

2   Select all load direction angles in the **Solutions to use** list on the **General** page.

3   Click the **Point** tab (this selects a point plot automatically).

4   Click at the lower-left corner or select Point 27 in the **Point selection** list.

5   Select **x-displacement** from the **Predefined quantities** list.

6   Click **Apply** to plot the x-displacement.
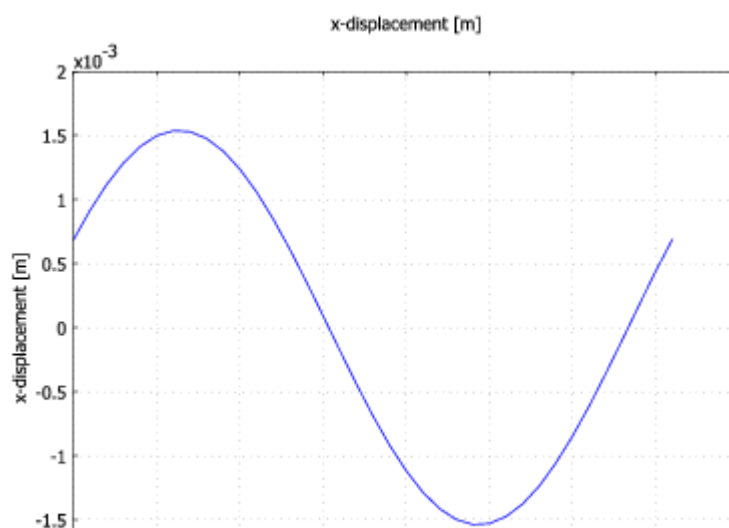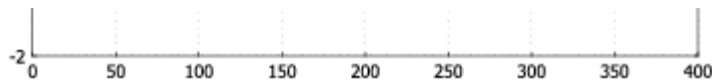
7 Click **OK**.

Quasi-Static Transient Analysis

A quasi-static transient analysis solves for the transient response where the dynamics of the structure are static compared to some other much longer time scale. In this example, a transient temperature problem is coupled to the structure, and the temperature problem has a much longer time scale than the dynamics of the structure.

The goal of this analysis is to find out for how long time the component can be exposed to a temperature of 500 °C before the x-displacement of the loaded edge increases by 30% compared to the static displacement without thermal expansion.

Model Library Path: `Structural_Mechanics_Module/Tutorial_Models/component_quasi_static`

MODEL DEFINITION

The geometry, material, and constraints are the same as for the static analysis (see the description on page 17 for details).

The boundary conditions for the temperature problem are:

• The left edge has fixed temperature of 20 °C

• The loaded edge has fixed temperature of 500 °C

• All other edges and both sides of the plate are cooled convectively with:

    - External temperature, $T_{\text{inf}} = 20$ °C

    - Heat transfer coefficient, $h = 5$ W/(m^2·°C)
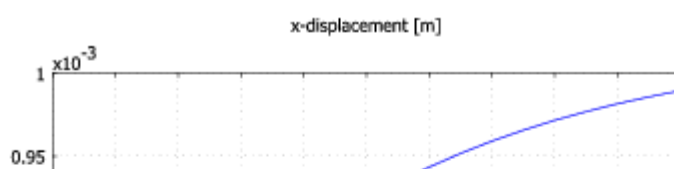
The initial conditions is:

• $T = 0$ °C
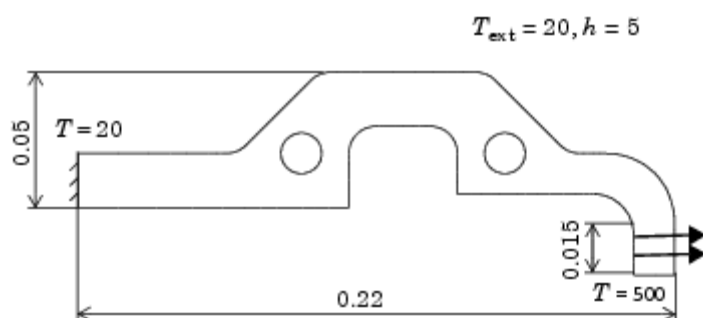
RESULTS

In the following figure you can see a plot of the x-displacement versus time at the bottom end of the edge where the load is applied.
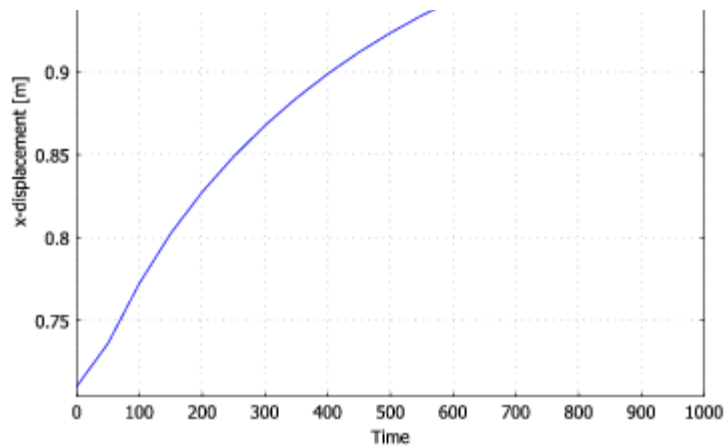
*Figure 3-1: The x-displacement at Point 27 as a function of time*

From the results you can conclude that it takes 500 s for the displacement to increase by 30%.

MODELING USING THE GRAPHICAL USER INTERFACE

Model Navigator

This time the modeling procedure is described as if starting from scratch.

1   Go to the **New** page in the **Model Navigator**, then select **2D** from the **Space dimension** list.

2   Select **Structural Mechanics Module>Thermal-Structural Interaction> Plane Stress with Thermal Expansion>Quasi-static analysis**.
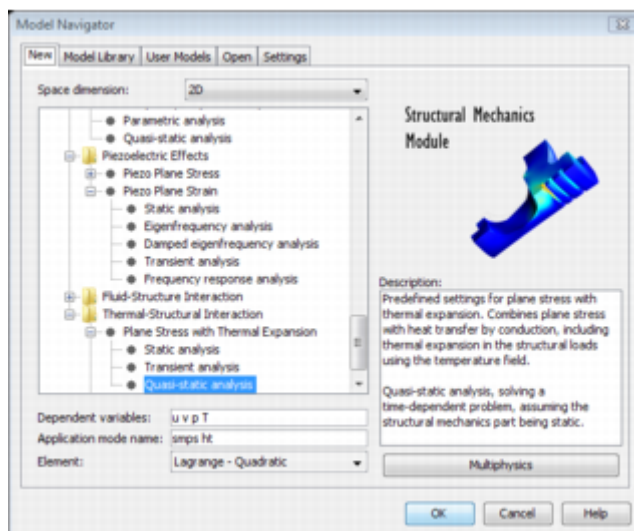
This is a predefined multiphysics coupling that adds both a Plane Stress and a heat transfer application mode (Heat Transfer by Conduction or General Heat Transfer if the license includes the Heat Transfer Module). The predefined multiphysics coupling also sets up the thermal expansion on the **Load** page in the **Subdomain Settings** dialog box for the Plane Stress application mode.

3   Click **OK** to close the **Model Navigator**.

Geometry Modeling

This was described in the static analysis (see the description on for details).

Physics Settings

Application Mode Properties General Heat Transfer

Skip to the next section if you are using the Heat Transfer by Conduction application mode.

1   Select **Properties** from the **Physics** menu to open the **Application Mode Properties** dialog box.

2   From the **Out-of-plane heat transfer** list box select **Enabled**, then click **OK**.

Boundary Settings Plane Stress

This model uses the same boundary conditions for the Plane Stress application mode as for the static analysis (see page 22 for details).

1   Select **Plane Stress** from the **Multiphysics** menu.

2   Apply boundary settings according to "Boundary Settings" on page 22.

Boundary Settings Heat Transfer

1   Select **Heat Transfer by Conduction** from the **Multiphysics** menu, or **General Heat Transfer** if you are using the Heat Transfer Module.

2   Set boundary conditions for the heat transfer application mode according to the following table. Click **OK** when done.

An easy way to apply the settings is if you first select all boundaries, then apply the **Heat flux** boundary condition according to the last two columns of the table. Finish by changing the settings for boundary 1 and 12 according to the table.

|	BOUNDARY 1		BOUNDARY 12		BOUNDARIES 2 −11, 13−30	
Boundary condition	Temperature		Temperature		Heat flux	
	T0	`20[degC]`	T0	`500 [degC]`	Tinf	`20[degC]`
					h	`5[W/ (m^2*K)]`
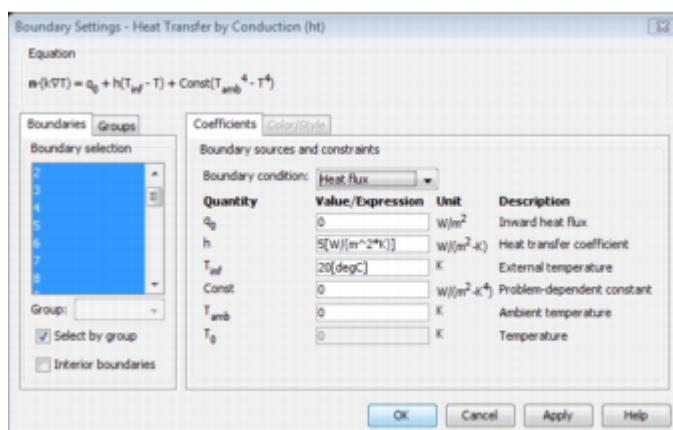
Subdomain Settings Plane Stress

The material properties for the Plane Stress application mode are the same as in the static analysis on page 22.

Specify the reference temperature in the Plane Stress application mode. The temperature coupling is already defined because you used the predefined multiphysics coupling node in the Model Navigator.

1   Select **Plane Stress** from the **Multiphysics** menu.

2   Set the reference temperature for the Plane Stress application mode according to the following table:

SUBDOMAIN	1	
Page	Load	
	Tempref	20 [degC]

You can see that the variable T, which is the temperature from the heat transfer application mode, is entered by default in the **Temp** edit field.
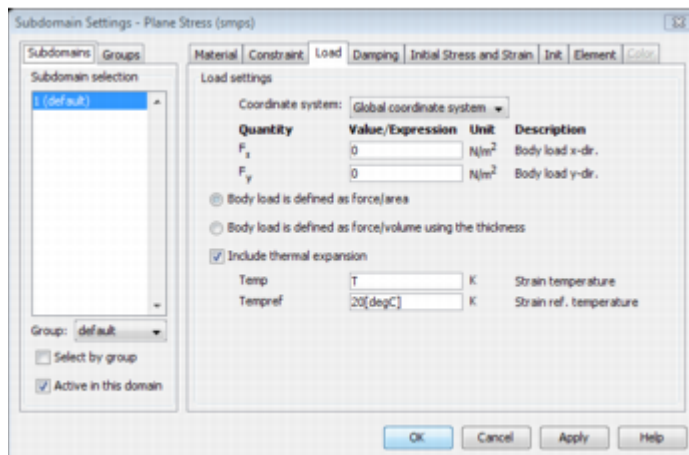
Subdomain Settings Heat Transfer by Conduction

Skip to the next section if you are using the General Heat Transfer application mode from the Heat Transfer Module.

Specify the material properties for the heat transfer application mode. The cooling of the surfaces is the same as for the boundaries, but you must transform it into cooling per volume, taking into account the thickness of the plate and that it has two sides. Instead of 5, h_{trans} is 2500 coming from division by the thickness and multiplication by 2.

1   Select **Heat Transfer by Conduction** from the **Multiphysics** menu.

2   Set the subdomain settings for the heat transfer application mode according to the following table. Click **OK** when finished.

	SUBDOMAIN 1	
Page	Physics	
	Library material	Structural steel
	T_{ext}	20[degC]
	h_{trans}	2500[W/ (m^3*K)]

Subdomain Settings General Heat Transfer

Skip to the next section if you are using the Heat Transfer by Conduction application mode.

1   Select **General Heat Transfer** from the **Multiphysics** menu.

2   Set the subdomain settings according to the following table, then click **OK**.

	SUBDOMAIN 1	
Page	Conduction	
	Library material	Structural steel
Page	Out-of-Plane	
	h_u	5[W/(m^2*K)]
	h_d	5[W/(m^2*K)]
	$T_{ext,u}$	20[degC]
	$T_{ext,d}$	20[degC]
	d_z	4[mm]

Computing the Solution

The analysis type controls which solver to use through the **Auto select solver** option in the **Solver Parameters** dialog box. The option is enabled as the default so there is no need to change the solver settings. The time-dependent solver is the one associated with the quasi-static transient analysis type.

Specify the time-dependent solver parameters:

1   Select **Solver Parameters** from the **Solve** menu to open the **Solver Parameters** dialog box.

2   Enter 0:50:1000 in the **Times** edit field. This means that the software saves the solution every 50 s during the total solution time of 1000 s. This does not influence the time-stepping length, only the times when the solution is saved.

3   Click **OK**.

4   To compute the solution either click the **Solve** button (**=**) on the Main toolbar or select **Solve Problem** from the **Solve** menu.

Postprocessing and Visualization

Look at the final temperature distribution together with the deformed geometry.

1   Select **Plot Parameters** from the **Postprocessing** menu.

2   Select the **Deformed shape** check box in the **Plot type** area.

3   Go to the **Surface** page and select **Heat Transfer by Conduction (ht)>Temperature** (or **General Heat Transfer (htgh)>Temperature**) from the **Predefined quantities** list.

4   Click **OK** to close the **Plot Parameters** dialog box.
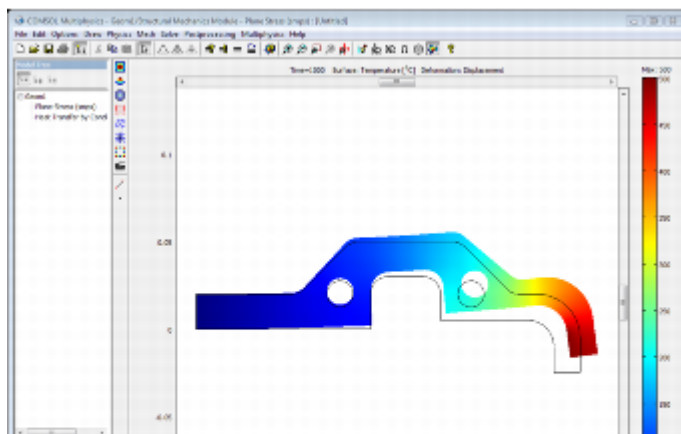
For a quantitative view of the time evolution of the displacement, plot the x-displacement at the lower-left corner as a function of time.

1   Select **Domain Plot Parameters** from the **Postprocessing** menu.

2   Go to the **General** page and select all the time steps in the **Solutions to use** list.

3   Click the **Point** tab (this selects a point plot automatically).

4   Click at the lower-left corner or select Point 27 from the **Point selection** list.

5   Select **x-displacement** from the **Predefined quantities** list.

6   Click **Apply** to plot the x-displacement.
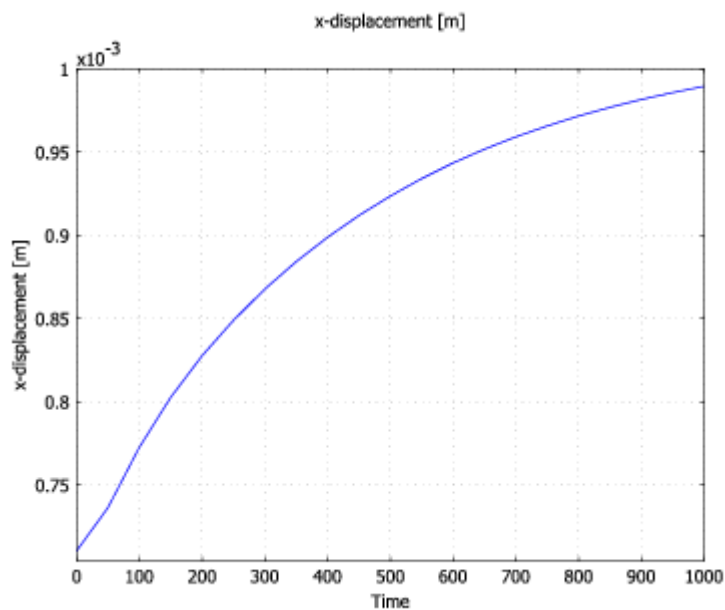
7   Click **OK** to close the **Domain Plot Parameters** dialog box.