Model created in COMSOL Multiphysics 6.4

Vibrations of an Impeller

Introduction

This tutorial model demonstrates the use of dynamic cyclic symmetry with postprocessing on the full geometry. A 3D impeller with eight identical blades can be divided into eight sectors of symmetry. The model computes the fundamental frequencies for the full impeller geometry and compares them to the values computed for a single sector with the cyclic symmetry boundary conditions applied on two sector boundaries. It also demonstrates how to set up a frequency response analysis for one sector of symmetry, and how to postprocess the results into the full geometry by using the sector datasets. The results for one sector are in very good agreement with the computations on the full geometry, while both the computational time and memory requirements are significantly reduced.

Model Definition

Figure 1 shows the impeller geometry. The problem is solved using the Cartesian coordinate system in 3D.

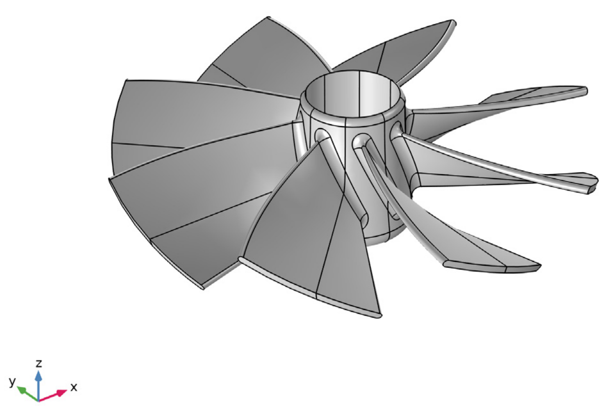

Figure 1: Impeller geometry.

The geometry can be divided into eight identical parts, each represented by a sector with an angle $\theta = \pi/4$ with respect to rotation around the z-axis; see Figure 2.

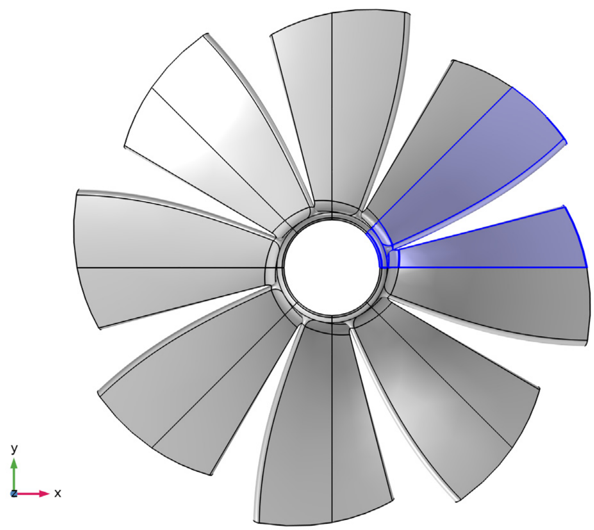

Figure 2: Sector of periodicity.

The impeller is made of aluminum, and is supposed to be mounted on a shaft. The mounting boundary is modeled via a fixed constraint, and all possible effects of the shaft rotation are neglected.

The analysis is based on the Floquet theory which can be applied to the problem of small-amplitude vibrations of spatially periodic structures, Ref. 1. This includes the case of cyclic symmetry studied in this example.

For an eigenfrequency study, one can show that all the eigenmodes of the full problem can be found by performing the analysis on one sector of symmetry only and imposing the cyclic symmetry of the eigenmodes with an angle of periodicity $\varphi = m\theta$, where the cyclic symmetry mode number m can vary from 0 to $N/2$, with N being the total number of sectors so that $\theta = 2\pi/N$.

Results and Discussion

In the first part of the analysis, you perform an eigenfrequency analysis of a single sector of periodicity, and then of the full geometry. A sweep over all required values of the cyclic symmetry parameter recovers all the eigenfrequencies of the full model with decent accuracy. See the Modeling Instructions section for in-detail comparison of the results and discussion of the performance gains.

In the second part, you perform a frequency-response analysis. Again, first of the sector of periodicity, and then of the full impeller geometry. The excitation is a pressure load applied to all free boundaries of the impeller. You enter it as a normal component of the boundary load using the expression

$$F_n = -p_0 \exp[-jm\,\mathrm{atan}(Y/X)]$$

using the magnitude of $p_0 = 10^4$ Pa and cyclic symmetry parameter $m = 3$. The excitation frequency is 200 Hz. Figure 3 and Figure 4 show very good agreement between the results computed on the full and reduced geometry.

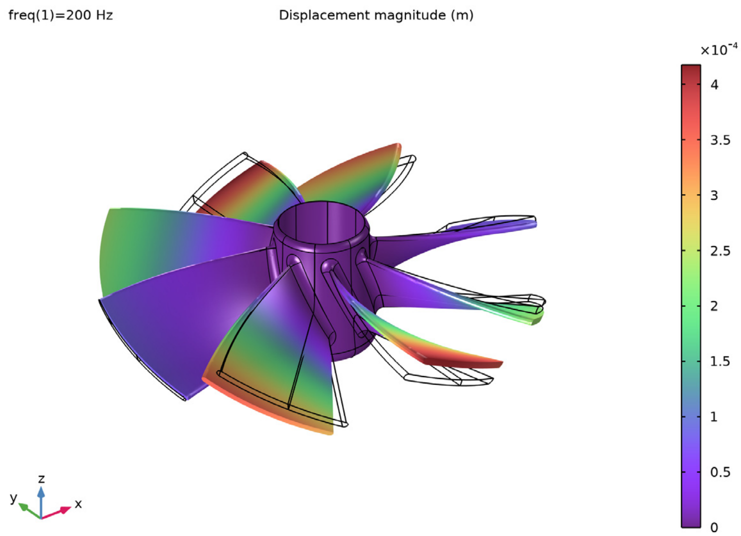

Figure 3: Frequency response computed on the sector of periodicity only, and then visualized over the full geometry.

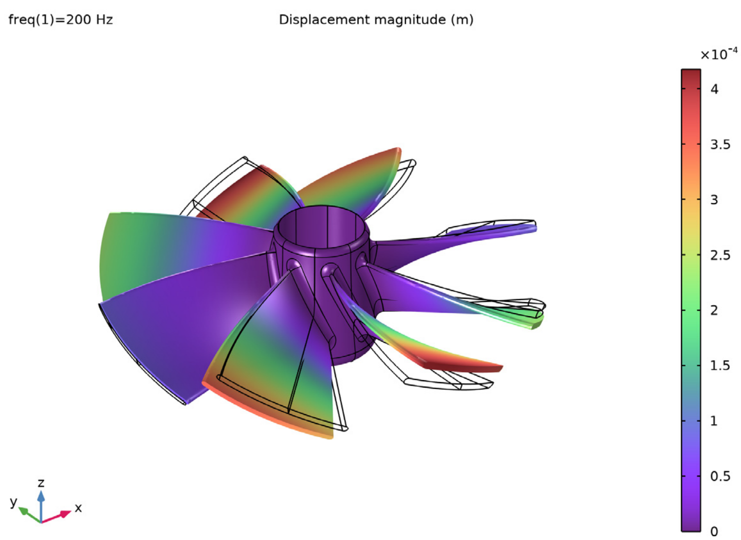

Figure 4: Frequency response computed for the full geometry.

MESHING

You use an unstructured mesh with the same size of the mesh elements for both calculations on one sector of symmetry and on the full geometry, see Figure 5. This helps to compare the results for this tutorial model. In practice, the mesh used for computations on the sector could be much finer, so that the results obtained via such geometry reduction would provide significantly better resolution of the results under the same memory requirements as for the full geometry (with a coarser mesh).

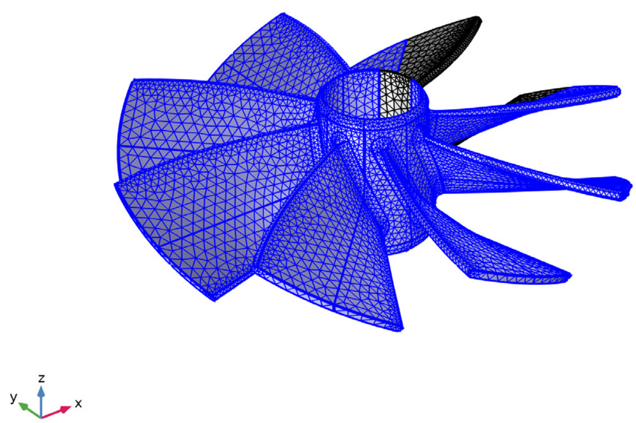

Figure 5: Meshed geometry.

CYCLIC SYMMETRY CONDITIONS AND POSTPROCESSING

To set up the cyclic symmetry conditions, you use the predefined functionality available in COMSOL Multiphysics within the Solid Mechanics interface under the Periodic Condition boundary feature. This imposes the proper boundary coupling condition on the sector boundaries.

You visualize the results computed for one sector over the full geometry by making use of the predefined type of derived dataset called **Sector 3D**, which is available under the **Results** node in the COMSOL Desktop.

Reference

1. B. Lalanne and M. Touratier, "Aeroelastic Vibrations and Stability in Cyclic Symmetric Domains," *Int. J. Rotating Machinery*, vol. 6, no. 6, pp. 445–452, 2000.

Application Library path: `Structural_Mechanics_Module/Dynamics_and_Vibration/impeller`

Modeling Instructions

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

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.
2 In the **Select Physics** tree, select **Structural Mechanics** > **Solid Mechanics (solid)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies** > **Eigenfrequency**.
6 Click **Done**.

GEOMETRY 1

Import the prebuilt geometry for the impeller from a file.

Import 1 (imp1)

1 In the **Geometry** toolbar, click **Import**.
2 In the **Settings** window for **Import**, locate the **Source** section.
3 Click **Browse**.
4 Browse to the model's Application Libraries folder and double-click the file `impeller.mphbin`.
5 Click **Import**.
6 In the **Geometry** toolbar, click **Build All**.
7 Click the **Go to Default View** button in the **Graphics** toolbar.

8 Click the **Go to XY View** button in the **Graphics** toolbar.

The complete geometry should look similar to that shown in Figure 1 and Figure 2.

Now create selections to easily apply loads and boundary conditions.

DEFINITIONS

Constrained Boundaries

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 55 only.

5 Select the **Group by continuous tangent** checkbox.

6 In the **Label** text field, type `Constrained Boundaries`.

Explicit 2

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 102 only.

5 Select the **Group by continuous tangent** checkbox.

Tip

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 In the **Label** text field, type `Tip`.

5 Select Boundaries 1, 3, 10, 14, 18, 21, 29, 44, 77, 95, and 147–152 only.

Load Boundaries

1 In the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, locate the **Geometric Entity Level** section.

3 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 **Explicit 2** and **Tip**.

6 Click **OK**.

7 In the **Settings** window for **Union**, type `Load Boundaries` in the **Label** text field.

8 Click in the **Graphics** window and then press Ctrl+A to select all boundaries.

Free Boundaries

1 In the **Definitions** toolbar, click **Complement**.

2 In the **Settings** window for **Complement**, locate the **Geometric Entity Level** section.

3 From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to invert**, click **Add**.

5 In the **Add** dialog, select **Constrained Boundaries** in the **Selections to invert** list.

6 Click **OK**.

7 In the **Settings** window for **Complement**, type `Free Boundaries` in the **Label** text field.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** 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
N	8	8	Number of sectors
theta	2*pi/N	0.7854	Unit sector angle
mn	3	3	Azimuthal mode number
p0	1e4[Pa]	10000 Pa	Load magnitude

SOLID MECHANICS (FULL)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, type `Solid Mechanics (Full)` in the **Label** text field.

COMPONENT 1 (COMP1)

Add a second Solid Mechanics interface to use for the computations on the reduced geometry only.

ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Structural Mechanics** > **Solid Mechanics (solid)**.

4 Click the **Add to Component I** button in the window toolbar.

5 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

SOLID MECHANICS (SECTOR)

1 Select Domain 8 only.

2 In the **Settings** window for **Solid Mechanics**, type `Solid Mechanics (Sector)` in the **Label** text field.

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in** > **Aluminum**.

4 Click the **Add to Component** button in the window toolbar.

5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

SOLID MECHANICS (FULL) (SOLID)

Fixed Constraint 1

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 **Constrained Boundaries**.

SOLID MECHANICS (SECTOR) (SOLID2)

In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

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

2 From the **Selection** list, choose **Constrained Boundaries**.

For a reduced geometry, you set up the Cyclic symmetry condition on the sector boundaries.

Periodic Condition 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Periodic Condition**.

2 Select Boundaries 112 and 134 only.

3 In the **Settings** window for **Periodic Condition**, locate the **Periodicity Settings** section.

4 From the **Type of periodicity** list, choose **Cyclic symmetry**.

5 In the m text field, type `mn`.

6 Right-click **Periodic Condition 1** and choose **Manual Destination Selection**.

7 Locate the **Destination Selection** section. Click **Clear Selection**.

8 Select Boundary 112 only.

Follow these steps to create a free unstructured mesh that will be identical in all eight sectors.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Fine**.

Identical Mesh 1

1 In the **Mesh** toolbar, click **More Attributes** and choose **Identical Mesh**.

2 Select Boundary 134 only.

3 In the **Settings** window for **Identical Mesh**, locate the **Second Entity Group** section.

4 Click to select the **Activate Selection** toggle button.

5 Select Boundary 112 only.

Free Tetrahedral 1

1 In the **Mesh** toolbar, click **Free Tetrahedral**.

2 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 8 only.

Copy Domain 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Copying Operations** > **Copy Domain**.

2 Select Domain 8 only.

3 In the **Settings** window for **Copy Domain**, locate the **Destination Domains** section.

4 Click to select the **Activate Selection** toggle button.

5 Select Domains 1–7 only.

6 Click **Build All**.

7 Click the **Go to Default View** button in the **Graphics** toolbar.

The resulting mesh should look similar to that shown in Figure 5.

STUDY 1

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 1** click **Step 1: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.

3 Select the **Desired number of eigenfrequencies** checkbox. In the associated text field, type `32`.

4 Locate the **Physics and Variables Selection** section. In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Solid Mechanics (Sector) (solid2)**.

5 In the **Model Builder** window, click **Study 1**.

6 In the **Settings** window for **Study**, type `Eigenfrequency Study (Full)` in the **Label** text field.

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

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** > **Eigenfrequency**.

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 2

Step 1: Eigenfrequency

1 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.

2 Select the **Desired number of eigenfrequencies** checkbox. In the associated text field, type `4`.

3 Locate the **Physics and Variables Selection** section. In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Solid Mechanics (Full) (solid)**.

To capture all possible eigenfrequencies, set up a sweep over the cyclic symmetry mode number m in the range from 0 to $N/2$, where N is the total number of sectors.

4 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
mn (Azimuthal mode number)		

7 Click **Range**.

8 In the **Range** dialog, type `0` in the **Start** text field.

9 In the **Step** text field, type `1`.

10 In the **Stop** text field, type `N/2`.

11 Click **Replace**.

12 In the **Model Builder** window, click **Study 2**.

13 In the **Settings** window for **Study**, type `Eigenfrequency Study (Sector)` in the **Label** text field.

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

RESULTS

Mode Shape (solid2)

Note a nearly eight times reduction in the number of degrees of freedom, and thus of the memory required to compute the reduced model.

However, the computational time is approximately the same because you need to perform a sweep over all values of the periodicity parameter.

Collect all the computed eigenfrequencies into tables.

Eigenfrequencies (Eigenfrequency Study (Full))

1 In the **Model Builder** window, click **Eigenfrequencies (Eigenfrequency Study (Full))**.

2 In the **Eigenfrequencies (Eigenfrequency Study (Full))** toolbar, click **Evaluate**.

   Note that the eigenfrequencies for the full geometry present groups of values very close to each other, eight frequencies in each group. This shows that vibrations of each of the eight blades of the impeller are only weakly coupled to the remaining structure, which is because the central part has significantly larger effective bending stiffness compared to that of each blade. Hence, the eigenfrequencies in each group are close to the natural frequencies of a single blade (if computed assuming a fully fixed footing).

Eigenfrequencies (Eigenfrequency Study (Sector))

1 In the **Model Builder** window, click **Eigenfrequencies (Eigenfrequency Study (Sector))**.

2 In the **Eigenfrequencies (Eigenfrequency Study (Sector))** toolbar, click **Evaluate**.

   Compare the values of the eigenfrequencies computed by using the periodicity conditions to those found for the full geometry.

Next, add a load representing a periodic pressure perturbation in the stream, and thus on all the external boundaries of the impeller.

SOLID MECHANICS (FULL) (SOLID)

Boundary Load 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
2 In the **Settings** window for **Boundary Load**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Load Boundaries**.
4 Locate the **Force** section. From the **Load type** list, choose **Pressure**.
5 In the p text field, type `p0*exp(-j*mn*atan2(Y,X))`.

SOLID MECHANICS (SECTOR) (SOLID2)

In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

1 In the **Settings** window for **Boundary Load**, locate the **Boundary Selection** section.
2 From the **Selection** list, choose **Load Boundaries**.
3 Locate the **Force** section. From the **Load type** list, choose **Pressure**.
4 In the p text field, type `p0*exp(-j*mn*atan2(Y,X))`.

ROOT

Set up and perform the frequency-response analysis, first for the full model, and then for a sector of periodicity.

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** > **Frequency Domain**.
4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Solid Mechanics (Sector) (solid2)**.
5 Click the **Add Study** button in the window toolbar.
6 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 3

Step 1: Frequency Domain

1 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

2 In the **Frequencies** text field, type 200.

3 Click to expand the **Store in Output** section. In the table, enter the following settings:

Interface	Output
Solid Mechanics (Full) (solid)	Physics controlled
Solid Mechanics (Sector) (solid2)	None

Switch off the generation of the default plot as that would be a plot of the von Mises stress, while you will be comparing the full and reduced structure responses in terms of displacements.

4 In the **Model Builder** window, click **Study 3**.

5 In the **Settings** window for **Study**, locate the **Study Settings** section.

6 Clear the **Generate default plots** checkbox.

7 In the **Label** text field, type `Frequency Domain Study (Full)`.

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

RESULT TEMPLATES

1 In the **Home** toolbar, click **Windows** and choose **Result Templates**.

2 Go to the **Result Templates** window.

3 In the tree, select **Frequency Domain Study (Full)/Solution 3 (sol3)** > **Solid Mechanics (Full)** > **Displacement (solid)**.

4 Click the **Add Result Template** button in the window toolbar.

RESULTS

Displacement (solid)

In the **Model Builder** window, expand the **Displacement (solid)** node.

Deformation

1 In the **Model Builder** window, expand the **Results** > **Displacement (solid)** > **Volume 1** node, then click **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** checkbox. In the associated text field, type 25.

4 In the **Displacement (solid)** toolbar, click **Plot**.

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 > Frequency Domain**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Solid Mechanics (Full) (solid)**.

5 Click the **Add Study** button in the window toolbar.

6 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 4

Step 1: Frequency Domain

1 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

2 In the **Frequencies** text field, type `200`.

3 Locate the **Store in Output** section. In the table, enter the following settings:

Interface	Output
Solid Mechanics (Full) (solid)	None
Solid Mechanics (Sector) (solid2)	Physics controlled

4 In the **Model Builder** window, click **Study 4**.

5 In the **Settings** window for **Study**, locate the **Study Settings** section.

6 Clear the **Generate default plots** checkbox.

7 In the **Label** text field, type `Frequency Domain Study (Sector)`.

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

For a frequency-response analysis, use of the reduced geometry gives significant gains in both the memory required and computational time needed.

RESULTS

Set up a displacement plot for the reduced geometry and compare it to that for the full geometry.

RESULT TEMPLATES

1 In the **Home** toolbar, click **Windows** and choose **Result Templates**.

2 Go to the **Result Templates** window.

3 In the tree, select **Frequency Domain Study (Sector)/Solution 4 (sol4)** > **Solid Mechanics (Sector)** > **Displacement, Full Geometry (solid2)**.

4 Click the **Add Result Template** button in the window toolbar.

RESULTS

Displacement, Full Geometry (solid2)

In the **Model Builder** window, expand the **Displacement, Full Geometry (solid2)** node.

Deformation

1 In the **Model Builder** window, expand the **Results** > **Displacement, Full Geometry (solid2)** > **Volume 1** node, then click **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** checkbox. In the associated text field, type 25.

4 In the **Displacement, Full Geometry (solid2)** toolbar, click **Plot**.