Created in COMSOL Multiphysics 5.6

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.

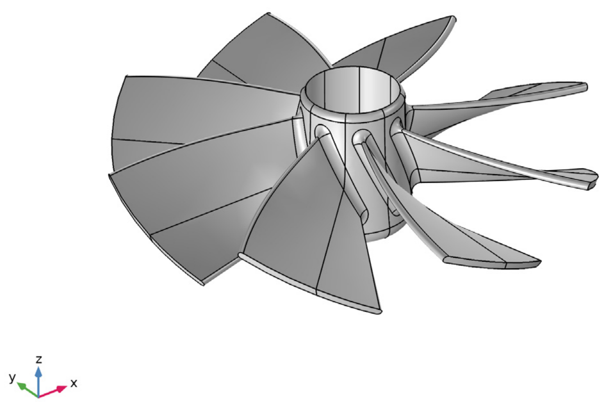

Figure 1: Impeller geometry.

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.

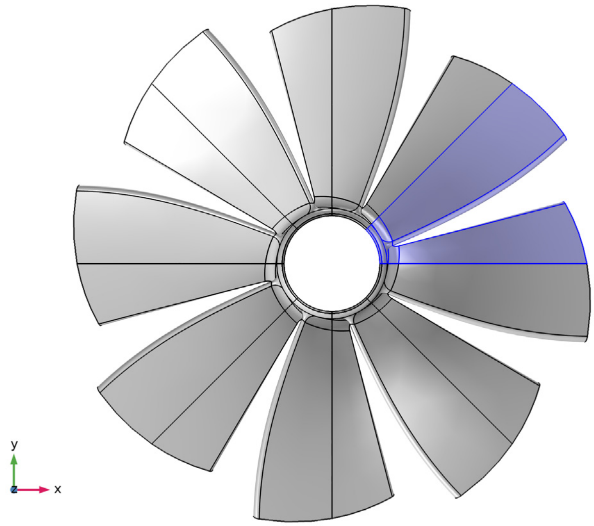

Figure 2: Sector of periodicity.

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.

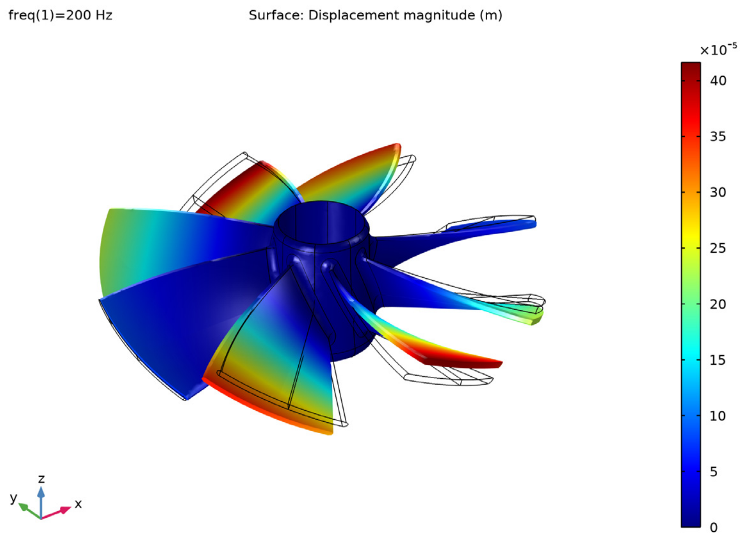

Figure 3: Frequency response computed on the sector of periodicity only, and then visualized over the full geometry.

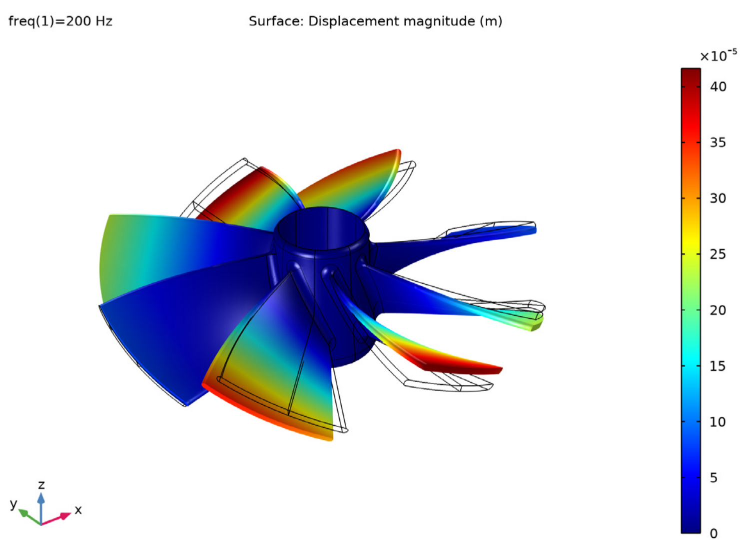

Figure 4: Frequency response computed for the full geometry.

Notes About the COMSOL Implementation

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).

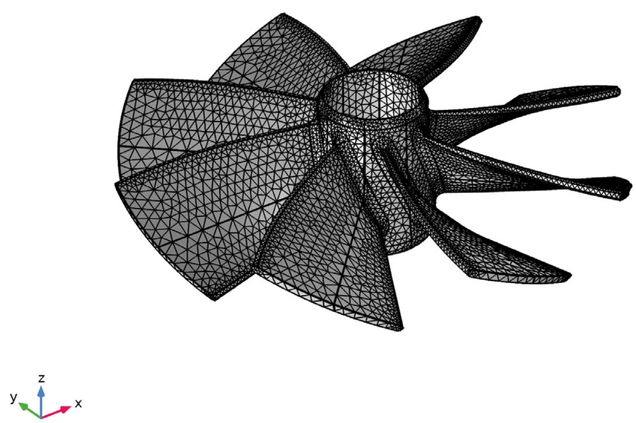

Figure 5: Meshed geometry.

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 **Home** toolbar, click **Import**.

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

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `impeller.mphbin`.

5 Click **Import**.

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

7 In the **Home** toolbar, click **Build All**.

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

9 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.

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

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 **Add to Component 1** in the window toolbar.

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

SOLID MECHANICS 2 (SOLID2)

Select Domain 8 only.

ADD MATERIAL

1 In the **Home** 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 **Add to Component** in the window toolbar.

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

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Fixed Constraint**.

2 Select Boundaries 54, 55, 67, 68, 85, 87, 109, and 113 only.

SOLID MECHANICS 2 (SOLID2)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics 2 (solid2)**.

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

Fixed Constraint 1

Select Boundary 113 only.

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`.

Destination Selection 1

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

2 In the **Settings** window for **Destination Selection**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 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**.

Free Triangular 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

2 Select Boundary 134 only.

Copy Face 1

1 In the **Mesh** toolbar, click **Copy** and choose **Copy Face**.

2 Select Boundary 134 only.

3 In the **Settings** window for **Copy Face**, locate the **Destination Boundaries** section.

4 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.

5 Click **Build Selected**.

Copy Domain 1

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

2 Select Domain 8 only.

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

4 Select the **Activate Selection** toggle button.

5 Select Domains 1–5 only.

6 Click **Build Selected**.

Copy Domain 2

1 Right-click **Mesh 1** and choose **More Operations>Copy Domain**.

2 Select Domain 1 only.

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

4 Select the **Activate Selection** toggle button.

5 Select Domains 6 and 7 only.

6 Right-click **Mesh 1** and choose **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** check box.

4 In the associated text field, type 32.

5 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

6 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Solid Mechanics 2 (solid2)**.

7 Click **Disable in Model**.

8 In the **Home** 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 **Add Study** 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** check box.

3 In the associated text field, type 4.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Solid Mechanics (solid)**.

6 Click **Disable in Model**.

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

7 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

8 Click **Add**.

9 In the table, enter the following settings:

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

10 Click **Range**.

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

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

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

14 Click **Replace**.

15 In the **Home** 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.

Add a new dataset to visualize over the full geometry the eigenmode shape for the reduced model.

Sector 3D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Sector 3D**.

2 In the **Settings** window for **Sector 3D**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **Symmetry** section. In the **Number of sectors** text field, type `8`.

5 Click to expand the **Advanced** section. In the **Azimuthal mode number** text field, type `solid2.pc1.mFloquet`.

Mode Shape (solid2)

1 In the **Model Builder** window, click **Mode Shape (solid2)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Sector 3D 1**.

4 In the **Mode Shape (solid2)** toolbar, click **Plot**.

Collect all the computed eigenfrequencies into tables.

Eigenfrequencies (Study 1)

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

2 In the **Eigenfrequencies (Study 1)** 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 (Study 2)

1 In the **Model Builder** window, click **Eigenfrequencies (Study 2)**.

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

Compare the values of the egenfrequencies 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 (SOLID)

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

4 Select Boundaries 1–3, 5–18, 20, 21, 23–53, 56–66, 69–75, 77–84, 88–107, 110, 111, 114–133, and 135–152 only.

You can do this by first selecting all boundaries, and then removing all the constraint boundaries and all the interior boundaries of the periodicity sectors from the selection .

5 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.

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

0	t1
0	t2
-p0*exp(-j*mn*atan2(Y,X))	n

SOLID MECHANICS 2 (SOLID2)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics 2 (solid2)**.

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

Boundary Load 1

1 Select Boundaries 114, 115, 119, 123, 125, 130, 131, 133, 136, 137, 142–146, 150, and 151 only.

2 In the **Settings** window for **Boundary Load**, locate the **Coordinate System Selection** section.

3 From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.

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

0	t1
0	t2
-p0*exp(-j*mn*atan2(Y,X))	n

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** check box for **Solid Mechanics 2 (solid2)**.

5 Click **Add Study** 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`.

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.

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

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

5 Clear the **Generate default plots** check box.

Solution 3 (sol3)

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

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node.

3 In the **Model Builder** window, expand the **Study 3>Solver Configurations> Solution 3 (sol3)>Dependent Variables 1** node, then click **Displacement field (comp1.u2)**.

4 In the **Settings** window for **Field**, locate the **General** section.

5 Clear the **Store in output** check box.

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

RESULTS

Displacement (solid)

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Displacement (solid)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3/Solution 3 (sol3)**.

Surface 1

Right-click **Displacement (solid)** and choose **Surface**.

Deformation 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Deformation**.

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

3 Select the **Scale factor** check box.

4 In the associated text field, type 25.

5 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** check box for **Solid Mechanics (solid)**.

5 Click **Add Study** 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 In the **Model Builder** window, click **Study 4**.

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

5 Clear the **Generate default plots** check box.

Solution 4 (sol4)

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

2 In the **Model Builder** window, expand the **Solution 4 (sol4)** node.

3 In the **Model Builder** window, expand the **Study 4>Solver Configurations> Solution 4 (sol4)>Dependent Variables 1** node, then click **Displacement field (comp1.u)**.

4 In the **Settings** window for **Field**, locate the **General** section.

5 Clear the **Store in output** check box.

6 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.

Sector 3D 2

1 In the **Results** toolbar, click **More Datasets** and choose **Sector 3D**.

2 In the **Settings** window for **Sector 3D**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 4/Solution 4 (sol4)**.

4 Locate the **Symmetry** section. In the **Number of sectors** text field, type `8`.

5 Locate the **Advanced** section. In the **Azimuthal mode number** text field, type `solid2.pc1.mFloquet`.

Displacement (solid2)

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

2 In the **Settings** window for **3D Plot Group**, type `Displacement (solid2)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Sector 3D 2**.

Surface 1

1 Right-click **Displacement (solid2)** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `solid2.disp`.

Deformation 1

1 Right-click **Surface 1** and choose **Deformation**.

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

3 Select the **Scale factor** check box.

4 In the associated text field, type `25`.

5 Locate the **Expression** section. In the **X component** text field, type `u2`.

6 In the **Y component** text field, type `v2`.

7 In the **Z component** text field, type `w2`.

8 In the **Displacement (solid2)** toolbar, click **Plot**.