

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 Floquet periodicity 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 general extrusion coupling variables. 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.

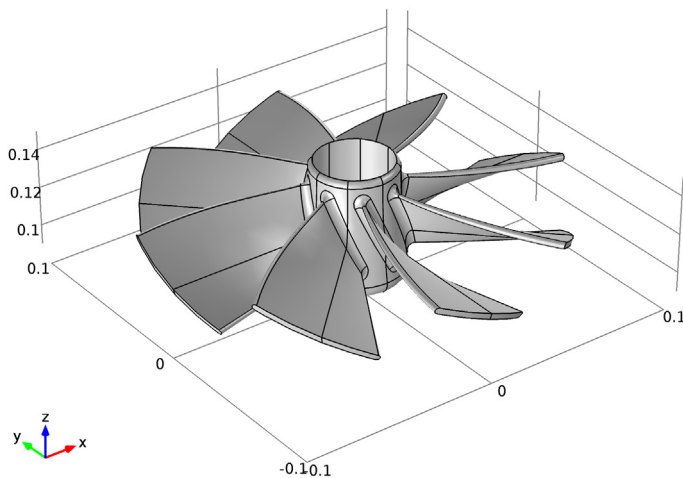


Figure 1: Impeller geometry.

The geometry can be divided into eight identical parts, each represented by a sector with an angle $\theta = \pi/4$ about the z -axis; see [Figure 2](#).

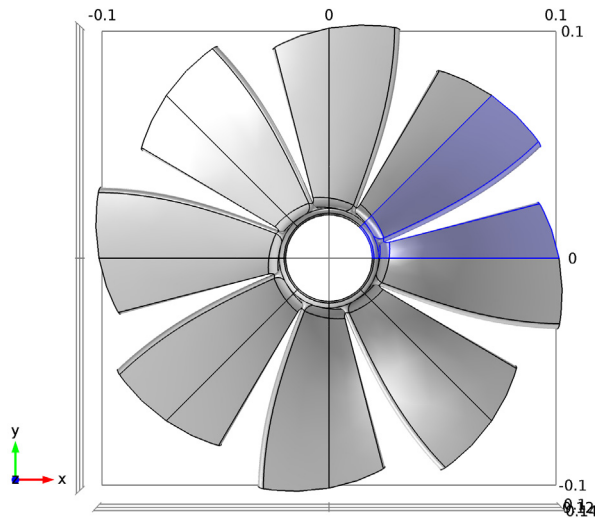


Figure 2: Sector of periodicity.

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 model.

If the problem is to determine the frequency response to a small-amplitude time-periodic excitation that also possesses cyclic symmetry, the theory states that the solution can be sought in form of a product of two functions. One follows the periodicity of the structure, while the other one follows the periodicity of the excitation. The problem can be solved in one sector of periodicity by applying the corresponding periodicity conditions to each of the two components in the product. The situation is often referred to as *dynamic cyclic symmetry*.

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 $\theta = 2\pi/N$.

You model the problem using the full solution without applying the above described multiplicative decomposition. For such a solution, the Floquet periodicity conditions at the sides of the sector of symmetry can be expressed as

$$\mathbf{u}_{\text{destination}} = e^{-j\varphi} \mathbf{R}_{\theta} \mathbf{u}_{\text{source}}^T \quad (1)$$

where the \mathbf{u} represents the displacement vectors with the components given in the default cartesian coordinates. Multiplication by the rotation matrix \mathbf{R} given by

$$\mathbf{R}_{\theta} = \begin{bmatrix} \cos(\theta) & -\sin(\theta) & 0 \\ \sin(\theta) & \cos(\theta) & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

makes the corresponding displacement components in the cylindrical coordinate system differ by the factor $\exp(-j\varphi)$ only. The angle φ represents either the periodicity of the eigenmode for an eigenfrequency analysis or the periodicity of the excitation signal in case of a frequency-response analysis.

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

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

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.

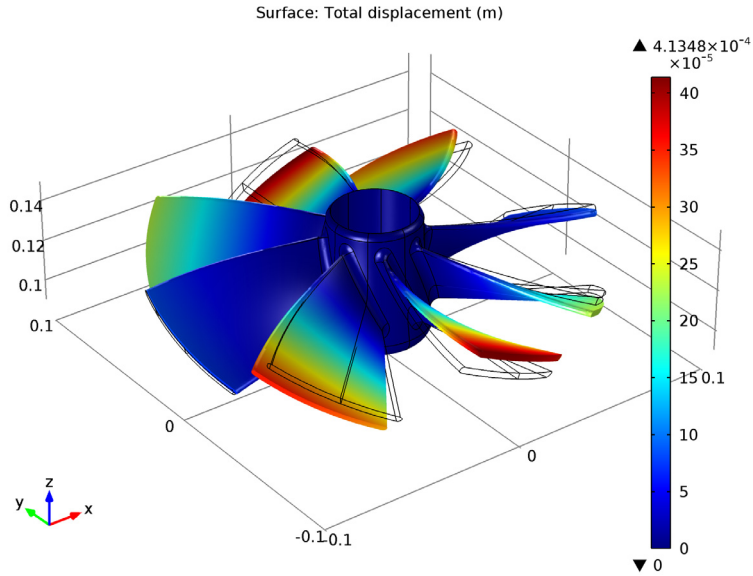


Figure 3: Frequency response computed on the sector of periodicity only, and then visualized over the full geometry.

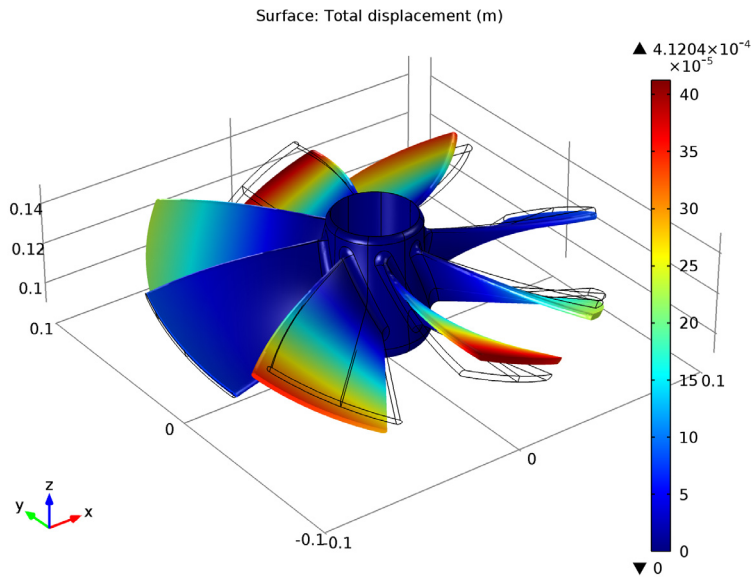


Figure 4: Frequency response computed for the full geometry.

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).

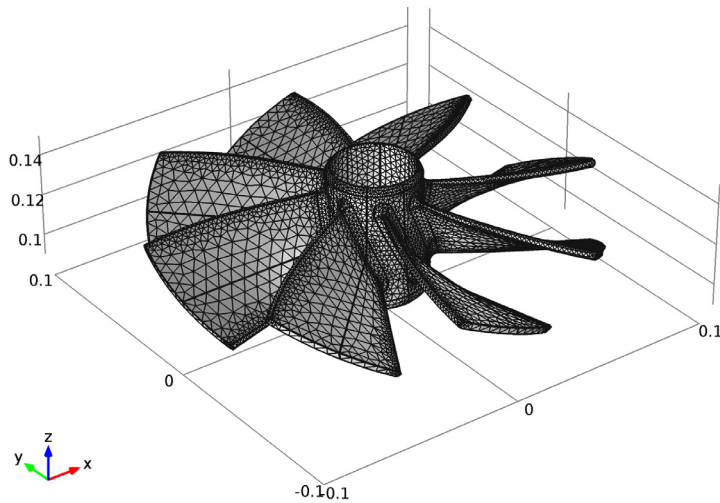


Figure 5: Meshed geometry.

FLOQUET PERIODICITY CONDITION

You set up the linear extrusion coupling variables to connect the boundaries of the sector, which is needed to set up the Floquet periodicity conditions [Equation 1](#).

Note that when entering the constraint expression, you also need to manually enter the constraint force expression i to ensure the Hermitian symmetry of the constraint force Jacobian matrix in the discretized model. See the [Modeling Instructions](#) section for more details.

POSTPROCESSING

You use the general extrusion coupling variables to propagate the results computed for one sector over the full geometry. The coordinate mapping is done as

$$\mathbf{r}_{\text{destination}} = R_s \mathbf{r}_{\text{source}}^T$$

where $\mathbf{r} = (X, Y, Z)$ is a vector of material coordinates. The rotation matrix is given by

$$R_s = \begin{bmatrix} \cos(n_s \theta) & -\sin(n_s \theta) & 0 \\ \sin(n_s \theta) & \cos(n_s \theta) & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

The number of the destination sector can be computed as

$$n_s = \text{floor} \left[\text{atan} \left(\frac{Y_{\text{destination}}}{X_{\text{destination}}} \right) / \theta \right]$$

where the **floor** function returns the largest integer that is less than or equal to the argument.

Reference

I. B. Lalanne and M. Touratier, "Aeroelastic Vibrations and Stability in Cyclic Symmetric Domains," *The international journal of rotating machinery*, vol. 6, no. 6, pp 445-452, 2000.

Model Library path: Structural_Mechanics_Module/Tutorial_Models/impeller

*Modeling Instructions***MODEL WIZARD**

- 1 Go to the **Model Wizard** window.
- 2 Click **Next**.
- 3 In the **Add physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
- 4 Click **Next**.

- 5 Find the **Studies** subsection. In the tree, select **Preset Studies>Eigenfrequency**.
- 6 Click **Finish**.

GEOMETRY I

Import the pre-build geometry for the impeller from a file.

Import I

- 1 In the **Model Builder** window, right-click **Model I>Geometry I** and choose **Import**.
- 2 Go to the **Settings** window for Import.
- 3 Locate the **Import** section. Click the **Browse** button.
- 4 Browse to the model's Model Library folder and double-click the file `impeller.mphbin`.
- 5 Click the **Import** button.
- 6 Click the **Go to Default 3D View** button on the Graphics toolbar.
- 7 In the **Model Builder** window, right-click **Geometry I** and choose **Build All**.
- 8 Click the **Go to Default 3D View** button on the Graphics toolbar.
- 9 Click the **Go to XY View** button on the Graphics toolbar.

The complete geometry should look similar to that shown in [Figure 1](#) and [Figure 2](#).

GLOBAL DEFINITIONS

Parameters

- 1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
- 2 Go to the **Settings** window for Parameters.
- 3 Locate the **Parameters** section. In the **Parameters** table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
N	8	Number of sectors
theta	$2 \cdot \pi / N$	Unit sector angle
m	3	Cyclic symmetry mode number
phi	$m \cdot \text{theta}$	Periodicity angle
p0	$1e4[\text{Pa}]$	Load magnitude

Variables I

- 1 In the **Model Builder** window, right-click **Global Definitions** and choose **Variables**.
- 2 Go to the **Settings** window for Variables.

3 Locate the **Variables** section. In the **Variables** table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
sector	$\text{floor}(\text{atan2}(Y,X)/(2*\pi/N))$	Sector selector
theta_sector	$\text{sector}*2*\pi/N$	Sector angle

DEFINITIONS

Define a linear extrusion coupling to connect the boundaries of the sector, which is needed when setting up the Floquet periodicity conditions.

Linear Extrusion 1

- 1 In the **Model Builder** window, right-click **Model 1 > Definitions** and choose **Model Couplings > Linear Extrusion**.
- 2 Go to the **Settings** window for Linear Extrusion.
- 3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundary 134 only.
- 5 Locate the **Source Vertices** section. Under **Source vertex 1**, click **Activate Selection**.
- 6 Select Point 135 only.
- 7 Under **Source vertex 2**, click **Activate Selection**.
- 8 Select Point 141 only.
- 9 Under **Source vertex 3**, click **Activate Selection**.
- 10 Select Point 153 only.
- 11 Under **Source vertex 4**, click **Activate Selection**.
- 12 Select Point 176 only.
- 13 Locate the **Destination Vertices** section. Under **Destination vertex 1**, click **Activate Selection**.
- 14 Select Point 115 only.
- 15 Under **Destination vertex 2**, click **Activate Selection**.
- 16 Select Point 121 only.
- 17 Under **Destination vertex 3**, click **Activate Selection**.
- 18 Select Point 130 only.
- 19 Under **Destination vertex 4**, click **Activate Selection**.
- 20 Select Point 166 only.

- 21 Click to expand the **Source** section.
- 22 From the **Source frame** list, choose **Material (X, Y, Z)**.
- 23 Click to expand the **Destination** section.
- 24 From the **Destination frame** list, choose **Material (X, Y, Z)**.

Add a general extrusion coupling to visualize the results computed for the sector over the full geometry.

General Extrusion 1

- 1 In the **Model Builder** window, right-click **Definitions** and choose **Model Couplings>General Extrusion**.
- 2 Select Domain 8 only.
- 3 Go to the **Settings** window for General Extrusion.
- 4 Locate the **Source** section. From the **Source frame** list, choose **Material (X, Y, Z)**.
- 5 Locate the **Destination Map** section. In the **x-expression** edit field, type $X*\cos(-\text{theta_sector})-Y*\sin(-\text{theta_sector})$.
- 6 In the **y-expression** edit field, type $X*\sin(-\text{theta_sector})+Y*\cos(-\text{theta_sector})$.
- 7 In the **z-expression** edit field, type Z.

MODEL 1

Add one more Solid Mechanics interface to use for the computations on the reduced geometry only.

MODEL WIZARD

- 1 In the **Model Builder** window, right-click **Model 1** and choose **Add Physics**.
- 2 Go to the **Model Wizard** window.
- 3 In the **Add physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
- 4 Click **Finish**.

SOLID MECHANICS 2

- 1 In the **Model Builder** window, click **Model 1>Solid Mechanics 2**.
- 2 Select Domain 8 only.

DEFINITIONS*Variables 2*

- 1 In the **Model Builder** window, right-click **Model 1 > Definitions** and choose **Variables**.
- 2 Right-click **Variables 2** and choose **Rename**.
- 3 Go to the **Rename Variables** dialog box and type Variables, postprocessing in the **New name** edit field.
- 4 Click **OK**.
- 5 Go to the **Settings** window for Variables.
- 6 Locate the **Variables** section. Click **Load from File**.
- 7 Browse to the model's Model Library folder and double-click the file `impeller_variables.txt`.

Variables 3

- 1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.
- 2 Right-click **Variables 3** and choose **Rename**.
- 3 Go to the **Rename Variables** dialog box and type Variables, source in the **New name** edit field.
- 4 Click **OK**.
- 5 Go to the **Settings** window for Variables.
- 6 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.
- 7 Select Boundary 134 only.
- 8 Locate the **Variables** section. In the **Variables** table, enter the following settings:

NAME	EXPRESSION
source_constr1	$(u2*\cos(\theta) - v2*\sin(\theta))$
source_constr2	$(u2*\sin(\theta) + v2*\cos(\theta))$
source_constr3	w2

Variables 4

- 1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.
- 2 Right-click **Variables 4** and choose **Rename**.
- 3 Go to the **Rename Variables** dialog box and type Variables, destination in the **New name** edit field.
- 4 Click **OK**.

- 5 Go to the **Settings** window for Variables.
- 6 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.
- 7 Select Boundary 112 only.

Variables, destination

- 1 In the **Model Builder** window, click **Variables, destination**.
- 2 Go to the **Settings** window for Variables.
- 3 Locate the **Variables** section. In the **Variables** table, enter the following settings:

NAME	EXPRESSION
constr1	linext1(source_constr1)
constr2	linext1(source_constr2)
constr3	linext1(source_constr3)

MATERIALS

- 1 In the **Model Builder** window, right-click **Model 1>Materials** and choose **Open Material Browser**.
- 2 Go to the **Material Browser** window.
- 3 Locate the **Materials** section. In the **Materials** tree, select **Built-In>Aluminum**.
- 4 Right-click and choose **Add Material to Model** from the menu.

SOLID MECHANICS

Fixed Constraint 1

- 1 In the **Model Builder** window, right-click **Model 1>Solid Mechanics** and choose **Fixed Constraint**.
- 2 Select Boundaries 54, 55, 67, 68, 85, 87, 109, and 113 only.

For a reduced geometry, you set up the Floquet periodicity conditions on the sector boundaries as pointwise constraints. To get access to this constraint setting, enable the advanced physics interface options.

- 3 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Interface Options** in the menu.

SOLID MECHANICS 2

- 1 In the **Model Builder** window, right-click **2** and choose **Fixed Constraint**.
- 2 Select Boundary 113 only.

Pointwise Constraint 1

- 1 In the **Model Builder** window, right-click **Solid Mechanics 2** and choose the boundary condition **More>Pointwise Constraint**.
- 2 Select Boundary 112 only.
- 3 Go to the **Settings** window for Pointwise Constraint.
- 4 Locate the **Pointwise Constraint** section. From the **Constraint type** list, choose **User defined**.
- 5 In the **Constraint expression** edit field, type $\text{constr1} \cdot \exp(-i \cdot \text{phi}) - u2$.
- 6 In the **Constraint force expression** edit field, type $\text{test}(\text{constr1} \cdot \exp(i \cdot \text{phi}) - u2)$.
Note that this setting imposes Hermitian symmetry on the constraint force Jacobian matrix in the discretized model.

Pointwise Constraint 2

- 1 Right-click **Pointwise Constraint 1** and choose **Duplicate**.
- 2 Go to the **Settings** window for Pointwise Constraint.
- 3 Locate the **Pointwise Constraint** section. In the **Constraint expression** edit field, type $\text{constr2} \cdot \exp(-i \cdot \text{phi}) - v2$.
- 4 In the **Constraint force expression** edit field, type $\text{test}(\text{constr2} \cdot \exp(i \cdot \text{phi}) - v2)$.

Pointwise Constraint 3

- 1 Right-click **Pointwise Constraint 2** and choose **Duplicate**.
- 2 Go to the **Settings** window for Pointwise Constraint.
- 3 Locate the **Pointwise Constraint** section. In the **Constraint expression** edit field, type $\text{constr3} \cdot \exp(-i \cdot \text{phi}) - w2$.
- 4 In the **Constraint force expression** edit field, type $\text{test}(\text{constr3} \cdot \exp(i \cdot \text{phi}) - w2)$.

MESH 1

- 1 In the **Model Builder** window, click **Model 1>Mesh 1**.
- 2 Go to the **Settings** window for Mesh.
- 3 Locate the **Mesh Settings** section. From the **Element size** list, choose **Fine**.
- 4 Click the **Build All** button.
- 5 Click the **Select Domains** button on the Graphics toolbar.
- 6 Click the **Go to Default 3D View** button on the Graphics toolbar.

The resulting mesh should look similar to that shown in [Figure 3](#).

STUDY 1*Step 1: Eigenfrequency*

- 1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Eigenfrequency**.
- 2 Go to the **Settings** window for Eigenfrequency.
- 3 Locate the **Physics Selection** section. In the table, enter the following settings:

PHYSICS INTERFACE	USE
Solid Mechanics 2 (solid2)	×

- 4 Locate the **Study Settings** section. In the **Desired number of eigenfrequencies** edit field, type 32.

Solver 1

- 1 In the **Model Builder** window, right-click **Study 1** and choose **Show Default Solver**.
- 2 Expand the **Solver 1** node.
- 3 In the **Model Builder** window, expand the **Dependent Variables 1** node, then click **mod1.u2**.
- 4 Go to the **Settings** window for Field.
- 5 Locate the **General** section. Clear the **Store in output** check box.
- 6 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS**MODEL WIZARD**

- 1 In the **Model Builder** window, right-click the root node and choose **Add Study**.
- 2 Go to the **Model Wizard** window.
- 3 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Eigenfrequency**.
- 4 Find the **Selected physics** subsection. In the tree, select **Solid Mechanics (solid2)**.
- 5 Click **Finish**.

STUDY 2*Step 1: Eigenfrequency*

- 1 In the **Model Builder** window, click **Study 2>Step 1: Eigenfrequency**.
- 2 Go to the **Settings** window for Eigenfrequency.

3 Locate the **Physics Selection** section. In the table, enter the following settings:

PHYSICS INTERFACE	USE
Solid Mechanics (solid)	×

4 Locate the **Study Settings** section. In the **Desired number of eigenfrequencies** edit field, type 4.

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.

Parametric Sweep

- 1 In the **Model Builder** window, right-click **Study 2** and choose **Parametric Sweep**.
- 2 Go to the **Settings** window for Parametric Sweep.
- 3 Locate the **Study Settings** section. Under **Parameter names**, click **Add**.
- 4 Go to the **Add** dialog box.
- 5 In the **Parameter names** list, select **m (Cyclic symmetry mode number)**.
- 6 Click the **OK** button.
- 7 Go to the **Settings** window for Parametric Sweep.
- 8 Locate the **Study Settings** section. Click the **Range** button.
- 9 Go to the **Range** dialog box.
- 10 In the **Start** edit field, type 0.
- 11 In the **Stop** edit field, type $N/2$.
- 12 In the **Step** edit field, type 1.
- 13 Click the **Add** button.

Solver 2

- 1 In the **Model Builder** window, right-click **Study 2** and choose **Show Default Solver**.
- 2 Expand the **Solver 2** node.
- 3 In the **Model Builder** window, expand the **Dependent Variables 1** node, then click **mod1.u**.
- 4 Go to the **Settings** window for Field.
- 5 Locate the **General** section. Clear the **Store in output** check box.
- 6 In the **Model Builder** window, right-click **Study 2** and choose **Compute**.

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.

Modify the default plot of the eigenmode shape for the reduced model using the extrusion coupling variables to visualize the effective results over the full geometry.

RESULTS

Mode Shape (solid2)

- 1 In the **Model Builder** window, expand the **Mode Shape (solid2)** node, then click **Surface 1**.
- 2 Go to the **Settings** window for Surface.
- 3 Locate the **Expression** section. In the **Expression** edit field, type `Disp2`.
- 4 In the **Model Builder** window, expand the **Surface 1** node, then click **Deformation**.
- 5 Go to the **Settings** window for Deformation.
- 6 Locate the **Expression** section. In the **X component** edit field, type `U2`.
- 7 In the **Y component** edit field, type `V2`.
- 8 In the **Z component** edit field, type `W2`.
- 9 Click the **Plot** button.

Derived Values

Collect all the computed eigenfrequencies into tables.

- 1 In the **Model Builder** window, right-click **Results>Derived Values** and choose **Global Evaluation**.
- 2 Go to the **Settings** window for Global Evaluation.
- 3 Locate the **Expression** section. In the **Expression** edit field, type `freq`.
- 4 Right-click **Global Evaluation 1** and choose **Evaluate>New Table**.

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 happens 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).

- 1 Right-click **Derived Values** and choose **Global Evaluation**.
- 2 Go to the **Settings** window for Global Evaluation.

- 3 Locate the **Data** section. From the **Data set** list, choose **Solution 3**.
- 4 From the **Table rows** list, choose **Outer solutions**.
- 5 Locate the **Expression** section. In the **Expression** edit field, type f_{req} .
- 6 Right-click **Global Evaluation 2** and choose **Evaluate>New Table**.
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

Boundary Load 1

- 1 In the **Model Builder** window, right-click **Model 1>Solid Mechanics** and choose **Boundary Load**.
- 2 Go to the **Settings** window for Boundary Load.
- 3 Locate the **Boundary Selection** section. 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 selecting all boundaries first, and then removing from the selection all the constraint boundaries and all the internal boundaries of the periodicity sectors.
- 5 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Boundary System 1**.
- 6 Locate the **Force** section. Specify the \mathbf{F}_A vector as

0	$t1$
0	$t2$
$-p0 \cdot \exp(-j \cdot m \cdot \text{atan2}(Y, X))$	n

SOLID MECHANICS 2

- 1 In the **Model Builder** window, right-click **2** and choose **Boundary Load**.
- 2 Select Boundaries 114, 115, 119, 123, 125, 130, 131, 133, 136, 137, 142–146, 150, and 151 only.
- 3 Go to the **Settings** window for Boundary Load.

- 4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Boundary System 1**.
- 5 Locate the **Force** section. Specify the \mathbf{F}_A vector as

0	t1
0	t2
$-p0 * \exp(-j * m * \text{atan2}(Y, X))$	n

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

MODEL WIZARD

- 1 In the **Model Builder** window, right-click the root node and choose **Add Study**.
- 2 Go to the **Model Wizard** window.
- 3 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Frequency Domain**.
- 4 Click **Finish**.

STUDY 3

Step 1: Frequency Domain

- 1 In the **Model Builder** window, click **Study 3>Step 1: Frequency Domain**.
- 2 Go to the **Settings** window for Frequency Domain.
- 3 Locate the **Study Settings** section. In the **Frequencies** edit field, type 200.
- 4 Locate the **Physics Selection** section. In the table, enter the following settings:

PHYSICS INTERFACE	USE
Solid Mechanics 2 (solid2)	×

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.

- 5 In the **Model Builder** window, click **Study 3**.
- 6 Go to the **Settings** window for Study.
- 7 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Solver 9

- 1 Right-click **Study 3** and choose **Show Default Solver**.

- 2 In the **Model Builder** window, expand the **Solver 9** node.
- 3 In the **Model Builder** window, expand the **Dependent Variables 1** node, then click **mod1.u2**.
- 4 Go to the **Settings** window for Field.
- 5 Locate the **General** section. Clear the **Store in output** check box.
- 6 In the **Model Builder** window, right-click **Study 3** and choose **Compute**.

RESULTS

3D Plot Group 3

- 1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.
- 2 Go to the **Settings** window for 3D Plot Group.
- 3 Locate the **Data** section. From the **Data set** list, choose **Solution 4**.
- 4 Right-click **Results>3D Plot Group 3** and choose **Rename**.
- 5 Go to the **Rename 3D Plot Group** dialog box and type **Displacement (solid)** in the **New name** edit field.
- 6 Click **OK**.

Displacement (solid)

- 1 Right-click **Results>3D Plot Group 3** and choose **Surface**.
- 2 In the **Model Builder** window, right-click **Results>Displacement (solid)>Surface 1** and choose **Deformation**.
- 3 Go to the **Settings** window for Deformation.
- 4 Locate the **Scale** section. Select the **Scale factor** check box.
- 5 In the associated edit field, type 25.
- 6 Click the **Plot** button.

MODEL WIZARD

- 1 In the **Model Builder** window, right-click the root node and choose **Add Study**.
- 2 Go to the **Model Wizard** window.
- 3 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Frequency Domain**.
- 4 Click **Finish**.

STUDY 4*Step 1: Frequency Domain*

- 1 In the **Model Builder** window, click **Study 4>Step 1: Frequency Domain**.
- 2 Go to the **Settings** window for Frequency Domain.
- 3 Locate the **Physics Selection** section. In the table, enter the following settings:

PHYSICS INTERFACE	USE
Solid Mechanics (solid)	x

- 4 Locate the **Study Settings** section. In the **Frequencies** edit field, type 200.
- 5 In the **Model Builder** window, click **Study 4**.
- 6 Go to the **Settings** window for Study.
- 7 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Solver 10

- 1 Right-click **Study 4** and choose **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solver 10** node.
- 3 In the **Model Builder** window, expand the **Dependent Variables 1** node, then click **mod1.u**.
- 4 Go to the **Settings** window for Field.
- 5 Locate the **General** section. Clear the **Store in output** check box.
- 6 In the **Model Builder** window, right-click **Study 4** and choose **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.

3D Plot Group 4

- 1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.
- 2 Go to the **Settings** window for 3D Plot Group.
- 3 Locate the **Data** section. From the **Data set** list, choose **Solution 5**.
- 4 Right-click **Results>3D Plot Group 4** and choose **Rename**.
- 5 Go to the **Rename 3D Plot Group** dialog box and type **Displacement (solid2)** in the **New name** edit field.

6 Click **OK**.

Displacement (solid2)

- 1** Right-click **Results>3D Plot Group 4** and choose **Surface**.
- 2** Go to the **Settings** window for Surface.
- 3** Locate the **Expression** section. In the **Expression** edit field, type `Disp2`.
- 4** Right-click **Results>Displacement (solid2)>Surface 1** and choose **Deformation**.
- 5** Go to the **Settings** window for Deformation.
- 6** Locate the **Scale** section. Select the **Scale factor** check box.
- 7** In the associated edit field, type `25`.
- 8** Locate the **Expression** section. In the **X component** edit field, type `U2`.
- 9** In the **Y component** edit field, type `V2`.
- 10** In the **Z component** edit field, type `W2`.
- 11** Click the **Plot** button.