



Vibrating Beam in Fluid Flow

Introduction

A classical flow pattern is the von Kármán vortex street that can form as fluid flows past an object. These vortices may induce vibrations in the object. This problem involves a fluid-structure interaction where the large deformation affects the flow path.

The magnitude and the frequencies of the oscillation generated by the fluid around the structure are computed and compared with the values proposed by Turek and Horn; see [Ref. 1](#).

Model Definition

The model geometry consists of a structure inside a channel with a fluid flow as represented in [Figure 1](#) below.

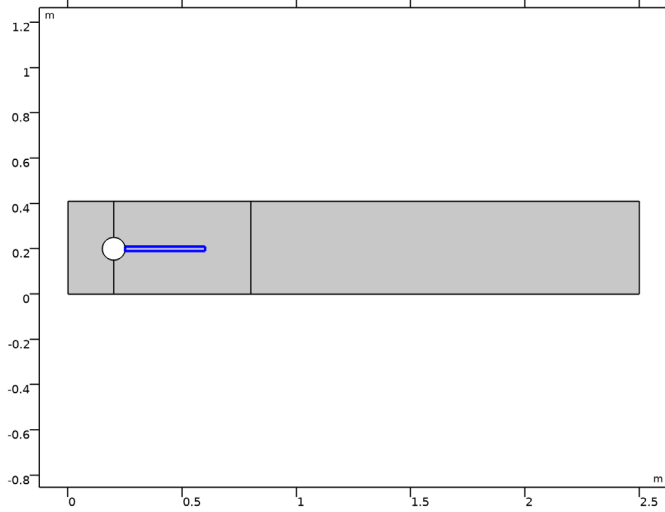


Figure 1: Model geometry including solid and fluid domains (blue and gray, respectively).

The fluid domain is a 2.5 m long and 0.41 m high channel. The structure is composed of a fixed circular domain with 0.05 m radius and centered at (0.2, 0.2). The second domain of the structure is a 0.35 m by 0.02 m rectangular beam made of elastic material.

The fluid enters the channel from the left with a mean velocity of 2 m/s, and the inlet velocity profile is assumed to be fully developed.

With the inlet boundary so close to the solid structure, one can expect the inlet velocity condition to affect the flow pattern. To avoid such an effect, one might need to increase

the distance between the inlet boundary and the solid structure. For the sake of comparison, the geometry in this model is kept as it is in the reference paper (Ref. 1).

The Reynolds number based on the diameter of the circle is about 200.

The fluid and solid properties are represented in the table below:

TABLE 1: FLUID AND SOLID MATERIAL PROPERTIES.

PARAMETER	VALUE
Fluid density	10^3 kg/m^3
Dynamic viscosity	$1 \text{ Pa}\cdot\text{s}$
Young's modulus	5.6 MPa
Poisson ratio	0.4

The quantities of interest are the beam rear tip displacements and the fluid forces acting on the structure. The magnitude and frequency targets (Ref. 1) are represented in the table below:

TABLE 2: TARGET RESULTS.

PARAMETER	MAGNITUDE	FREQUENCY
x-displacement	$-2.69 \pm 2.53 \text{ mm}$	10.9 Hz
y-displacement	$1.48 \pm 34.38 \text{ mm}$	5.3 Hz
Drag	$457.3 \pm 22.66 \text{ N}$	10.9 Hz
Lift	$2.22 \pm 149.78 \text{ N}$	5.3 Hz

Results and Discussion

Figure 2 shows the velocity field and the von Mises stress in the structure on the deformed shape at different times. Note the von Kármán vortex street past the structure, which is significantly deformed and affects the flow field.

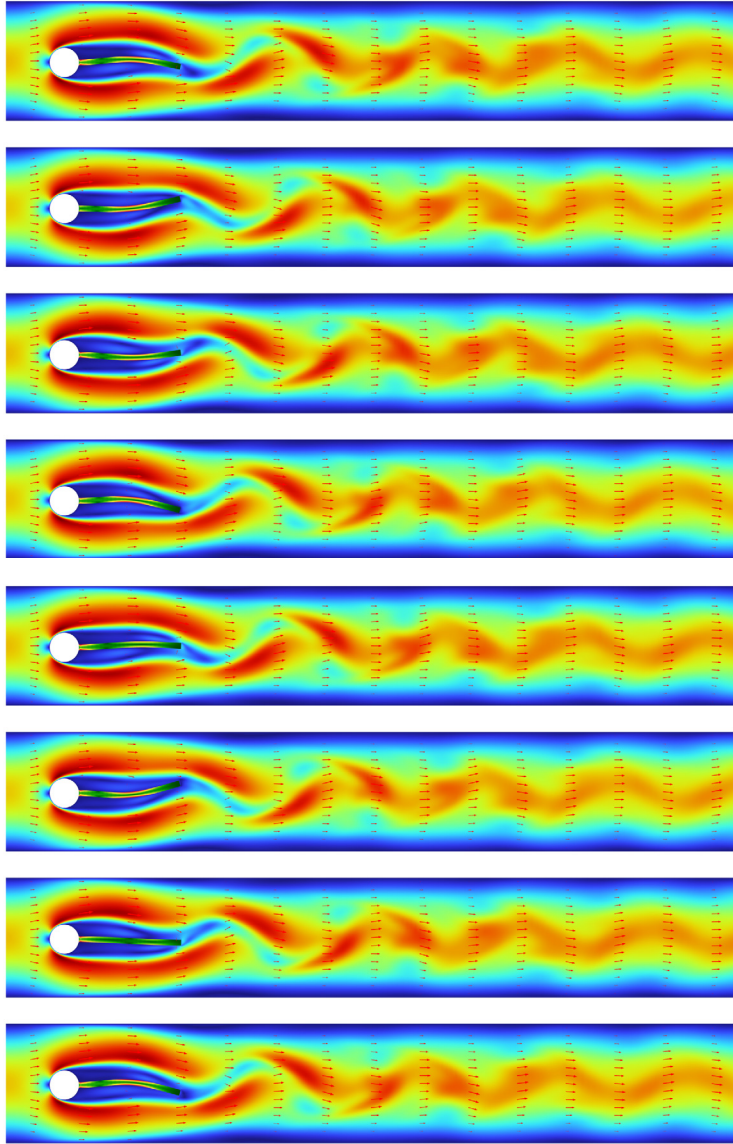


Figure 2: Velocity field in fluid and von Mises stress in structure for eight different time steps.

Figure 3 below shows the evolution of the fluid forces all along the time step. The oscillation is fully developed after $t = 3.5$ s. This is due to the external perturbation added at $t = 1.5$ s. Without this perturbation, the oscillation would develop after a longer time. Note that the oscillation can develop with some time shift due to nonlinearities in the model.

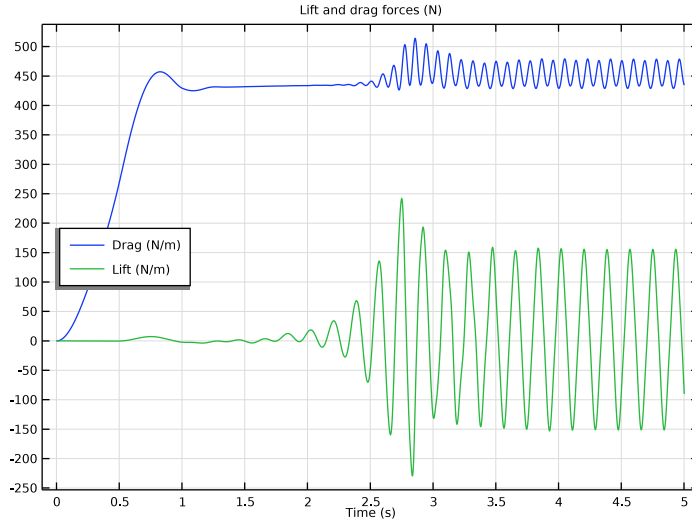


Figure 3: Drag and lift forces versus time.

Figure 4 shows the displacement of the tip of the beam in the x and y directions:

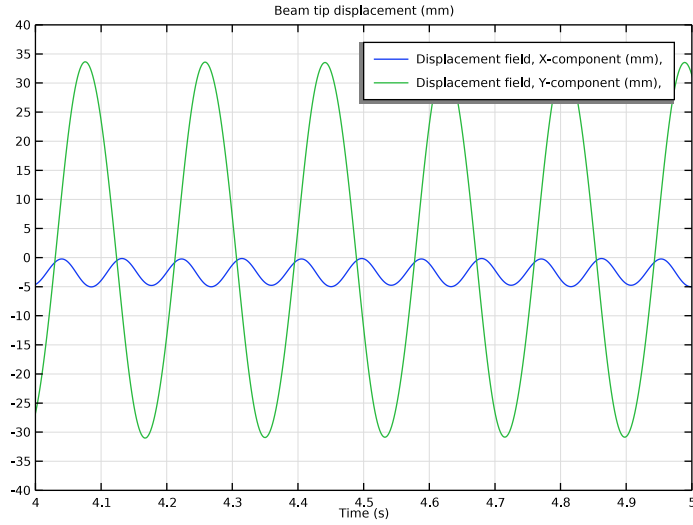


Figure 4: Tip displacement of the structure in the x and y directions (in green and blue respectively).

In the above figure, you can see that the magnitude of the x -displacement oscillation is about 2.5 mm around the average of -2.5 mm. The y -displacement varies around 1 mm with an oscillation magnitude of 33 mm, in good agreement with the targeted value.

The trajectory of the tip is shown in [Figure 6](#).

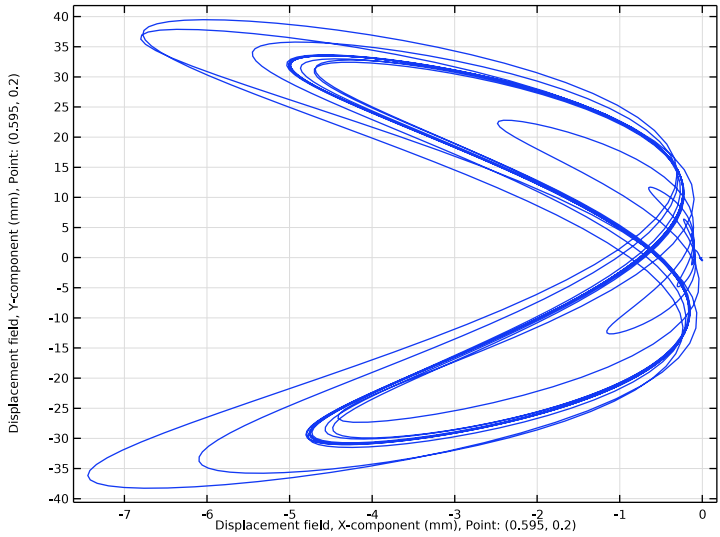


Figure 5: Beam tip trajectory. The origin corresponds to the initial position.

[Figure 6](#) below shows the frequency spectrum of the structure oscillation.

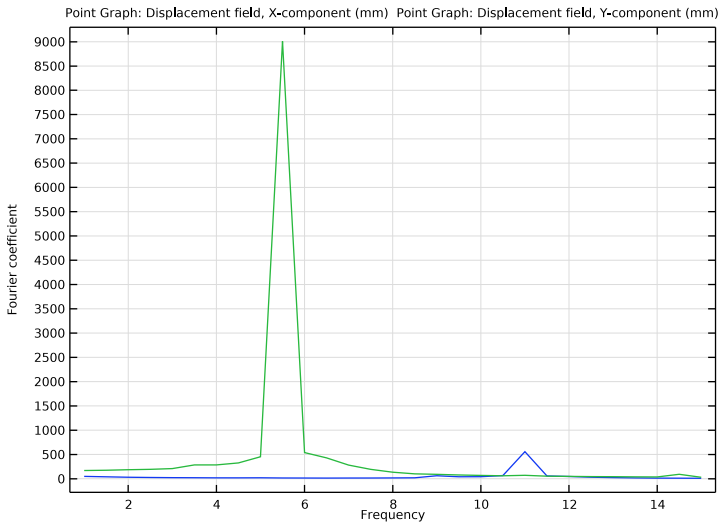


Figure 6: Frequency spectrum of the structure tip displacement.

The peaks show the main frequencies of the harmonic oscillation. For the x -displacement, the frequency is about 11 Hz, while for the y -displacement the main frequency is about 5.5 Hz, which agree well with the targeted results.

Figure 7 below shows the variations of the lift and drag forces applied to the structure:

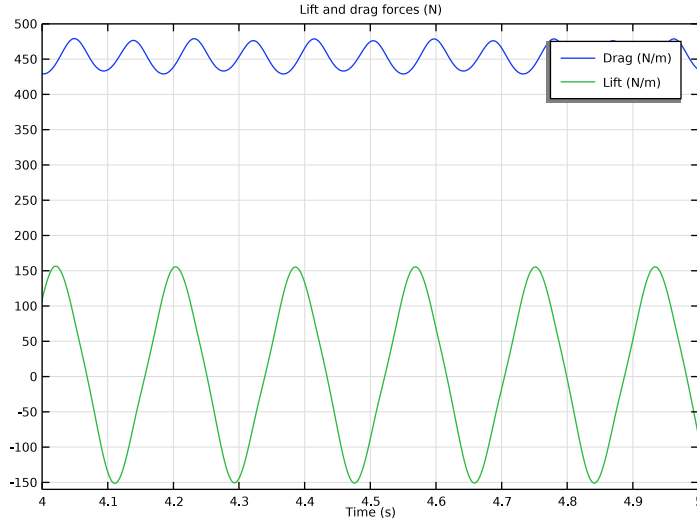


Figure 7: Lift and drag forces (green and blue curves, respectively) after the periodic oscillations have established.

The average of the total lift force is about 2 N with an oscillation magnitude of 154 N, while the drag force average is about 456 N with an oscillation magnitude of 26 N.

Notes About the COMSOL Implementation

The default discretization for the flow equations in the fluid-structure interface is based on P1+P1 elements. This means that linear order elements are used for the velocity variables. Such discretization is more stable for high Reynolds number but has lower accuracy especially in the forces evaluation. In this model, use P2+P2 elements to increase the accuracy for the flow equations.

Reference


1. S. Turek and J. Hron, *Proposal for numerical benchmarking of fluid-structure interaction between an elastic object and laminar incompressible flow*, Institute for Applied Mathematics and Numerics, University of Dortmund.

Application Library path: Structural_Mechanics_Module/
Verification_Examples/oscillating_fsi




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  **2D**.
- 2 In the **Select Physics** tree, select **Fluid Flow>Fluid–Structure Interaction>Fluid–Solid Interaction**.
- 3 Click **Add**.
- 4 Click  **Study**.
- 5 In the **Select Study** tree, select **General Studies>Time Dependent**.
- 6 Click  **Done**.

GEOMETRY I


Rectangle 1 (r1)

- 1 In the **Geometry** toolbar, click  **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 2.5.
- 4 In the **Height** text field, type 0.41.


Circle 1 (c1)

- 1 In the **Geometry** toolbar, click  **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type 0.05.
- 4 Locate the **Position** section. In the **x** text field, type 0.2.
- 5 In the **y** text field, type 0.2.



Rectangle 2 (r2)

- 1 In the **Geometry** toolbar, click  **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type $0.35+0.05$.
- 4 In the **Height** text field, type 0.02 .
- 5 Locate the **Position** section. From the **Base** list, choose **Center**.
- 6 In the **x** text field, type $0.2+0.4/2$.
- 7 In the **y** text field, type 0.2 .



Rectangle 3 (r3)

- 1 In the **Geometry** toolbar, click  **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 0.6 .
- 4 In the **Height** text field, type 0.41 .
- 5 Locate the **Position** section. In the **x** text field, type 0.2 .

Solid




- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Difference**.
- 2 In the **Settings** window for **Difference**, type **Solid** in the **Label** text field.
- 3 Select the object **r2** only.
- 4 Locate the **Difference** section. Click to select the  **Activate Selection** toggle button for **Objects to subtract**.
- 5 Select the object **c1** only.
- 6 Select the **Keep objects to add** check box.
- 7 Select the **Keep objects to subtract** check box.
- 8 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

Fluid

- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Difference**.
- 2 In the **Settings** window for **Difference**, type **Fluid** in the **Label** text field.
- 3 Select the objects **r1** and **r3** only.
- 4 Locate the **Difference** section. Click to select the  **Activate Selection** toggle button for **Objects to subtract**.
- 5 Select the objects **c1** and **r2** only.

- 6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

Form Union (fin)

- 1 In the **Geometry** toolbar, click  **Build All**.
- 2 In the **Model Builder** window, click **Form Union (fin)**.
- 3 In the **Settings** window for **Form Union/Assembly**, click  **Build Selected**.
- 4 Click the  **Zoom Extents** button in the **Graphics** toolbar.


MOVING MESH

Deforming Domain 1


- 1 In the **Model Builder** window, under **Component 1 (comp1)>Moving Mesh** click **Deforming Domain 1**.
- 2 Select Domain 2 only.

DEFINITIONS


Step 1 (step1)

- 1 In the **Home** toolbar, click  **Functions** and choose **Local>Step**.
- 2 In the **Settings** window for **Step**, locate the **Parameters** section.
- 3 In the **Location** text field, type 0.5[s].
- 4 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type 1.

Gaussian Pulse 1 (gp1)


- 1 In the **Home** toolbar, click  **Functions** and choose **Local>Gaussian Pulse**.
- 2 In the **Settings** window for **Gaussian Pulse**, locate the **Parameters** section.
- 3 In the **Location** text field, type 1.5[s].
- 4 In the **Standard deviation** text field, type 5e-2[s].
- 5 From the **Normalization** list, choose **Peak value**.

LAMINAR FLOW (SPF)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.
- 2 In the **Settings** window for **Laminar Flow**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Fluid**.
- 4 Click the  **Show More Options** button in the **Model Builder** toolbar.

- 5 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Stabilization**.
- 6 Click **OK**.
- 7 In the **Settings** window for **Laminar Flow**, click to expand the **Consistent Stabilization** section.
- 8 Find the **Navier-Stokes equations** subsection. Clear the **Crosswind diffusion** check box.
- 9 Click to expand the **Discretization** section. From the **Discretization of fluids** list, choose **P2+P2**.

Inlet 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Inlet**.
- 2 Select Boundary 1 only.
- 3 In the **Settings** window for **Inlet**, locate the **Velocity** section.
- 4 In the U_0 text field, type $1.5 \cdot 2 [\text{m/s}] * Y * (0.41 [\text{m}] - Y) / (0.41 [\text{m}] / 2)^2 * \text{step1}(t)$.


Outlet 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Outlet**.
- 2 Select Boundary 14 only.


SOLID MECHANICS (SOLID)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.
- 2 In the **Settings** window for **Solid Mechanics**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Solid**.

Fixed Constraint 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Fixed Constraint**.
- 2 Select Boundaries 19 and 20 only.

Point Load 1

- 1 In the **Physics** toolbar, click  **Points** and choose **Point Load**.
- 2 Select Point 11 only.
- 3 In the **Settings** window for **Point Load**, locate the **Force** section.
- 4 Specify the \mathbf{F}_P vector as

0	x
$1 [\text{N}] * \text{gp1}(t)$	y

MATERIALS

Material 1 (mat1)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
- 2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
- 3 From the **Selection** list, choose **Solid**.
- 4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	5.6 [MPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.4	I	Young's modulus and Poisson's ratio
Density	rho	1e3	kg/m ³	Basic

Material 2 (mat2)

- 1 Right-click **Materials** and choose **Blank Material**.
- 2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
- 3 From the **Selection** list, choose **Fluid**.
- 4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	1000	kg/m ³	Basic
Dynamic viscosity	mu	1	Pa·s	Basic

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Edit Physics-Induced Sequence**.

Size

- 1 In the **Model Builder** window, under **Component 1 (comp1)**>**Mesh 1** click **Size**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 From the **Predefined** list, choose **Coarse**.

Size 1


- 1 In the **Model Builder** window, click **Size 1**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.

- 3 From the **Predefined** list, choose **Normal**.

Free Triangular I

- 1 In the **Model Builder** window, click **Free Triangular I**.
- 2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domains 1–3 only.


Mapped I

- 1 In the **Mesh** toolbar, click  **Mapped**.
- 2 Right-click **Mapped I** and choose **Move Up**.
- 3 In the **Settings** window for **Mapped**, click to expand the **Control Entities** section.
- 4 Clear the **Smooth across removed control entities** check box.

Distribution I

- 1 Right-click **Mapped I** and choose **Distribution**.
- 2 Select Boundaries 12 and 13 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 From the **Distribution type** list, choose **Predefined**.
- 5 In the **Number of elements** text field, type 40.
- 6 In the **Element ratio** text field, type 5.
- 7 Select the **Reverse direction** check box.

Boundary Layers I

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Boundary Layers I**.
- 2 In the **Settings** window for **Boundary Layers**, click to expand the **Corner Settings** section.
- 3 From the **Handling of sharp corners** list, choose **No special handling**.
- 4 Click to expand the **Transition** section. Clear the **Smooth transition to interior mesh** check box.
- 5 Click  **Build All**.

You can now prepare the probe variables to display during the computation.



DEFINITIONS

Integration I (intop1)

- 1 In the **Definitions** toolbar, click  **Nonlocal Couplings** and choose **Integration**.

- 2 In the **Settings** window for **Integration**, locate the **Source Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundaries 8–10 and 15–18 only.

Global Variable Probe 1 (var1)

- 1 In the **Definitions** toolbar, click  **Probes** and choose **Global Variable Probe**.
- 2 In the **Settings** window for **Global Variable Probe**, type drag in the **Variable name** text field.
- 3 Locate the **Expression** section. In the **Expression** text field, type -
intop1(spf.T_stressx).
- 4 Select the **Description** check box. In the associated text field, type Drag.
- 5 Click to expand the **Table and Window Settings** section. Click  **Add Plot Window**.


Global Variable Probe 1 (drag)

Right-click **Global Variable Probe 1 (drag)** and choose **Duplicate**.

Global Variable Probe 2 (var2)


- 1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Global Variable Probe 2 (var2)**.
- 2 In the **Settings** window for **Global Variable Probe**, type lift in the **Variable name** text field.
- 3 Locate the **Expression** section. In the **Expression** text field, type -
intop1(spf.T_stressy).
- 4 In the **Description** text field, type Lift.

Domain Point Probe 1

- 1 In the **Definitions** toolbar, click  **Probes** and choose **Domain Point Probe**.
- 2 In the **Settings** window for **Domain Point Probe**, locate the **Point Selection** section.
- 3 From the **Frame** list, choose **Material**.
- 4 In row **Coordinates**, set **X** to 0.595.
- 5 In row **Coordinates**, set **Y** to 0.2.

Point Probe Expression 1 (ppb1)

- 1 In the **Model Builder** window, expand the **Domain Point Probe 1** node, then click **Point Probe Expression 1 (ppb1)**.
- 2 In the **Settings** window for **Point Probe Expression**, type u in the **Variable name** text field.
- 3 Locate the **Expression** section. In the **Expression** text field, type u_solid.

- 4 From the **Table and plot unit** list, choose **mm**.
- 5 Click to expand the **Table and Window Settings** section. Click  **Add Plot Window**.

Point Probe Expression 1 (u)

Right-click **Point Probe Expression 1 (u)** and choose **Duplicate**.

Point Probe Expression 2 (ppb2)


- 1 In the **Model Builder** window, click **Point Probe Expression 2 (ppb2)**.
- 2 In the **Settings** window for **Point Probe Expression**, locate the **Expression** section.
- 3 In the **Expression** text field, type `v_solid`.
- 4 In the **Variable name** text field, type `v`.


STUDY 1

Step 1: Time Dependent

- 1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.
- 2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
- 3 In the **Output times** text field, type `range(0,5e-2,5)`.
- 4 Click to expand the **Results While Solving** section. Select the **Plot** check box.
- 5 From the **Update at** list, choose **Time steps taken by solver**.

Solution 1 (sol1)

- 1 In the **Study** toolbar, click  **Show Default Solver**.
For this 2D model, use the fully coupled solver instead of the default segregated one, this will give a faster and more robust computation.
- 2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.
- 3 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** and choose **Fully Coupled**.
- 4 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.
- 5 In the **Damping factor** text field, type `0.9`.
- 6 From the **Jacobian update** list, choose **Once per time step**.
- 7 In the **Maximum number of iterations** text field, type `8`.
- 8 In the **Tolerance factor** text field, type `0.5`.
- 9 From the **Stabilization and acceleration** list, choose **Anderson acceleration**.
- 10 In the **Dimension of iteration space** text field, type `5`.


- 11 In the **Mixing parameter** text field, type 0.9.
- 12 In the **Iteration delay** text field, type 1.
- 13 In the **Study** toolbar, click  **Compute**.

RESULTS

Velocity (spf)

The first plot group shows the fluid velocity magnitude.

Surface 2

- 1 Right-click **Velocity (spf)** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type `solid.mises`.
- 4 Locate the **Coloring and Style** section. Click  **Change Color Table**.
- 5 In the **Color Table** dialog box, select **Traffic>Traffic** in the tree.
- 6 Click **OK**.


Arrow Surface 1

Right-click **Velocity (spf)** and choose **Arrow Surface**.

Animation 1


In the **Velocity (spf)** toolbar, click  **Animation** and choose **Player**.

Lift and Drag Forces


- 1 In the **Settings** window for **ID Plot Group**, type `Lift` and `Drag Forces` in the **Label** text field.
- 2 Locate the **Plot Settings** section. Select the **x-axis label** check box.
- 3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
- 4 In the **Title** text area, type `Lift` and `drag forces (N)`.
- 5 Locate the **Legend** section. From the **Position** list, choose **Middle left**.
- 6 In the **Lift and Drag Forces** toolbar, click  **Plot**.
- 7 Locate the **Axis** section. Select the **Manual axis limits** check box.
- 8 In the **x minimum** text field, type 4.
- 9 In the **x maximum** text field, type 5.
- 10 In the **y minimum** text field, type -160.
- 11 In the **y maximum** text field, type 500.
- 12 Locate the **Legend** section. From the **Position** list, choose **Upper right**.

13 In the **Lift and Drag Forces** toolbar, click  **Plot**.

Beam Tip Displacement

- 1** In the **Model Builder** window, under **Results** click **Probe Plot Group 2**.
- 2** In the **Settings** window for **ID Plot Group**, type Beam Tip Displacement in the **Label** text field.
- 3** Locate the **Plot Settings** section. Select the **x-axis label** check box.
- 4** Locate the **Axis** section. Select the **Manual axis limits** check box.
- 5** In the **x minimum** text field, type 4.
- 6** In the **x maximum** text field, type 5.
- 7** In the **y minimum** text field, type -40.
- 8** In the **y maximum** text field, type 40.
- 9** Locate the **Title** section. From the **Title type** list, choose **Manual**.
- 10** In the **Title** text area, type Beam tip displacement (mm).
- 11** In the **Beam Tip Displacement** toolbar, click  **Plot**.

Frequency Spectrum


- 1** In the **Home** toolbar, click  **Add Plot Group** and choose **ID Plot Group**.
- 2** In the **Settings** window for **ID Plot Group**, type Frequency Spectrum in the **Label** text field.
- 3** Locate the **Data** section. From the **Dataset** list, choose **Domain Point Probe 1**.
- 4** From the **Time selection** list, choose **Interpolated**.
- 5** In the **Times (s)** text field, type range (3,5e-3,5).

Point Graph 1

- 1** Right-click **Frequency Spectrum** and choose **Point Graph**.
- 2** In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
- 3** In the **Expression** text field, type `u_solid`.
- 4** From the **Unit** list, choose **mm**.
- 5** Locate the **x-Axis Data** section. From the **Parameter** list, choose **Discrete Fourier transform**.
- 6** From the **Show** list, choose **Frequency spectrum**.
- 7** Select the **Frequency range** check box.
- 8** In the **Minimum** text field, type 1.
- 9** In the **Maximum** text field, type 15.

10 Right-click **Point Graph 1** and choose **Duplicate**.

Point Graph 2

- 1 In the **Model Builder** window, click **Point Graph 2**.
- 2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type `v_solid`.
- 4 In the **Frequency Spectrum** toolbar, click  **Plot**.

Beam Tip Trajectory



- 1 In the **Home** toolbar, click  **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type **Beam Tip Trajectory** in the **Label** text field.

Table Graph 1

- 1 Right-click **Beam Tip Trajectory** and choose **Table Graph**.
- 2 In the **Settings** window for **Table Graph**, locate the **Data** section.
- 3 From the **x-axis data** list, choose **Displacement field, X-component (mm), Point: (0.595, 0.2)**.
- 4 From the **Plot columns** list, choose **Manual**.
- 5 In the **Columns** list, select **Displacement field, Y-component (mm), Point: (0.595, 0.2)**.
- 6 In the **Beam Tip Trajectory** toolbar, click  **Plot**.

