Fluid-Structure Interaction in a Network of Blood Vessels

Introduction

This example studies a portion of the vascular system, in particular the upper part of the aorta (Figure 1). The aorta and its ramified blood vessels are embedded in biological tissue, specifically the cardiac muscle. The flowing blood applies pressure to the artery’s internal surfaces and its branches, thereby deforming the tissue. The analysis consists of two distinct but coupled procedures: first, a fluid-dynamics analysis including a calculation of the velocity field and pressure distribution in the blood (variable in time and in space); second, a mechanical analysis of the deformation of the tissue and artery. Any change in the shape of the vessel walls does not influence the fluid domain, which implies that there is only a one-way fluid-structural coupling. However, in COMSOL Multiphysics it is possible to simulate a two-way coupling using the ALE (arbitrary Lagrangian-Eulerian) method.

Figure 1: The model domain consists of part of the aorta, its branches, and the surrounding tissue.

Model Definition

Figure 2 shows two views of the model domain, one with and one without the cardiac muscle. The mechanical analysis must consider the cardiac muscle because it presents a stiffness that resists artery deformation due to the applied pressure.
Figure 2: A view of the aorta and its ramification (branching vessels) with blood contained, shown both with (left) and without (right) the cardiac muscle.

The main characteristics of the analyses are:

- **Fluid dynamics analysis**
  Here the Navier-Stokes equations are solved in the blood domain. At each surface where the model brings a vessel to an abrupt end, it represents the load with a known pressure distribution.

- **Mechanical analysis**
  Only the domains related to the biological tissues are active in this analysis. The model represents the load with the total stress distribution it computes during the fluid-dynamics analysis.

**ANALYSIS OF RUBBER-LIKE TISSUE AND ARTERY MATERIAL MODELS**

Generally, the modeling of biological tissue is an advanced subject for several reasons:

- The material can undergo very large strains (finite deformations).
- The stress-strain relationship is generally nonlinear.
- Many hyperelastic materials are almost incompressible. You must then revise standard displacement-based finite element formulations in order to arrive at correct results (mixed formulations).

You must pay particular attention to the definition of stress and strain measures. In a geometrically nonlinear analysis, the assumptions about infinitesimal displacements are no longer valid. It is necessary to consider geographical nonlinearity in a model when:

- Significant rigid-body rotations occur (finite rotations).
The strains are no longer small (larger than a few percent).

The loading of the body depends on the deformation.

All of these issues are dealt with in the hyperelastic material model built-in the Nonlinear Structural Materials Module.

In this case, the displacements and strains are so small that it is sufficient to use a linear elastic material model. The material data is given for a neo-Hookean hyperelastic material, but in the small strain limit the interpretation of the material constants is the same for a linear elastic material.

MATERIALS

The following material properties are used:

- Blood
  - density = 1060 kg/m³
  - dynamic viscosity = 0.005 Ns/m²

- Artery
  - density = 960 kg/m³
  - Neo-Hookean hyperelastic behavior: the coefficient $\mu$ equals $6.20 \times 10^6$ N/m², while the bulk modulus equals $20\mu$ and corresponds to a value for Poisson’s ratio, $\nu$, of 0.45. An equivalent elastic modulus equals $1.0 \times 10^7$ N/m².

- Cardiac muscle
  - density = 1200 kg/m³
  - Neo-Hookean hyperelastic behavior: the coefficient $\mu$ equals $7.20 \times 10^6$ N/m², while the bulk modulus equals $20\mu$ and corresponds to a value for Poisson’s ratio, $\nu$, of 0.45. An equivalent elastic modulus equals $1.16 \times 10^6$ N/m².

FLUID DYNAMICS ANALYSIS

The fluid dynamics analysis considers the solution of the 3D Navier-Stokes equations. You can do so in both a stationary case or in the time domain. To establish the boundary conditions, six pressure conditions are applied with the configuration shown...
in Figure 3.

**Figure 3: Boundary conditions for the fluid-flow analysis.**

The pressure conditions are:

- Section 1: 126.09 mmHg
- Section 2: 125.91 mmHg
- Section 3: 125.415 mmHg
- Section 4: 125.415 mmHg
- Section 5: 125.415 mmHg
- Section 6: 125.1 mmHg

Those pressure values are the mean values over a heart beating cycle. During a cycle the pressure varies between a minimal and a maximal values which are calculated thanks to a relative amplitude $\alpha$. For the time-dependent analysis, a simple trigonometric function is used for varying the pressure distribution over time:

$$f(t) = \begin{cases} 
(1 - \alpha) \sin(\pi t) & 0 \leq t \leq 0.5s \\
1 - \alpha \cos(2\pi(t - 0.5)) & 0.5s \leq t \leq 1.5s 
\end{cases}$$

The first piece of function between 0 and 0.5 s has no physical significance, it is just a ramp that enable to calculate the initial state. The second piece of function makes the pressure vary between its minimal and maximal value during a 1 s cycle.

You implement this effect in COMSOL Multiphysics using Piecewise function.
Results and Discussion

The flow field at the time $t=1$ s is displayed in Figure 4 as a slice plot.

Figure 4: Velocity field in the aorta and its ramification (branching).

Figure 5 shows the total displacement at the peak load (after 1 s). The displacements are in the order of 4 μm, which suggests that the one-way multiphysics coupling is a reasonable approximation.
Figure 5: Displacements in the blood vessel.

Notes About the COMSOL Implementation

In this example, and many other cases, an analysis which is time dependent for one physics can be treated as quasi-static from the structural mechanics point of view. You can handle this by running the structural analysis as a parametric sweep over a number of static load cases, where the time is used as the parameter. This method is used here.

Application Library path: Structural_Mechanics_Module/Bioengineering/blood_vessel

Modeling Instructions

From the File menu, choose New.
NEW
1 In the New window, click Model Wizard.

MODEL WIZARD
1 In the Model Wizard window, click 3D.
2 In the Select physics tree, select Fluid Flow>Single-Phase Flow>Laminar Flow (spf).
3 Click Add.
4 In the Select physics tree, select Structural Mechanics>Solid Mechanics (solid).
5 Click Add.
6 Click Study.
7 In the Select study tree, select Preset Studies for Selected Physics Interfaces>Time Dependent.
8 Click Done.

GLOBAL DEFINITIONS

Parameters
1 On the Home toolbar, click Parameters.
2 In the Settings window for Parameters, locate the Parameters section.
3 In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Name</th>
<th>Expression</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>t</td>
<td>0[s]</td>
<td>0 s</td>
<td>Time continuation parameter</td>
</tr>
<tr>
<td>alpha</td>
<td>1/3</td>
<td>0.3333</td>
<td>Relative pressure amplitude during heart's beating</td>
</tr>
</tbody>
</table>

Piecewise 1 (pw1)
1 On the Home toolbar, click Functions and choose Global>Piecewise.
2 In the Settings window for Piecewise, type \( f \) in the Function name text field.
3 Locate the Definition section. In the Argument text field, type \( t \).
4 Find the Intervals subsection. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Start</th>
<th>End</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.5</td>
<td>((1-\text{alpha})\times\sin(\pi t))</td>
</tr>
<tr>
<td>0.5</td>
<td>1.5</td>
<td>((1-\text{alpha})\times\cos(2\pi(\text{t}-0.5)))</td>
</tr>
</tbody>
</table>
5 Locate the Units section. In the Arguments text field, type \( s \).
6. In the **Function** text field, type 1.

7. Click the **Plot** button.

**GEOMETRY 1**
The geometry for this model is available as an MPHBIN-file. Import this file as follows.

*Import 1 (imp1)*

1. On the **Home** toolbar, click **Import**.

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

3. Click **Browse**.

4. Browse to the application’s Application Library folder and double-click the file **blood_vessel.mphbin**.

5. Click **Import**.

   The length unit in the imported geometry is centimeters, while the default length unit in COMSOL Multiphysics is meters. Therefore, you need to rescale the geometry.

*Scale 1 (scal)*

1. On the **Geometry** toolbar, click **Transforms** and choose **Scale**.

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

3. In the **Factor** text field, type **0.01**.

4. Select the object **impl** only.

5. Right-click **Scale 1 (scal)** and choose **Build Selected**.

6. Click the **Go to Default 3D View** button on the **Graphics** toolbar.

*Form Union (fin)*

1. In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.
2. Click the **Transparency** button on the **Graphics** toolbar to see the interior.

DEFINITIONS
Next, define a number of selections as sets of geometric entities for use in setting up the model.

**Explicit 1**
1. On the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for Explicit, type **Blood** in the **Label** text field.
3. Select Domain 3 only.

**Explicit 2**
1. On the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for Explicit, type **Artery** in the **Label** text field.
3. Select Domain 2 only.

** Explicit 3**
1. On the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for Explicit, type **Muscle** in the **Label** text field.
3. Select Domain 1 only.
Explicit 4
1 On the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for Explicit, type **Inlet** in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Click **Paste Selection**.
5 In the **Paste Selection** dialog box, type **38** in the **Selection** text field.
6 Click **OK**.

Explicit 5
1 On the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for Explicit, type **Outlet 1** in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 19 only.

Explicit 6
1 On the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for Explicit, type **Outlet 2** in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 9 only.

Explicit 7
1 On the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for Explicit, type **Outlet 3** in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 41 only.

Explicit 8
1 On the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for Explicit, type **Outlet 4** in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 70 only.
Explicit 9
1. On the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for Explicit, type **Outlet 5** in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundary 86 only.

Explicit 10
1. On the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for Explicit, type **Roller boundaries** in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 1–6, 12, 26, 27, 30, 33, 64, 67, 85, and 87 only.
   - The roller boundaries are the free boundaries of muscle and artery that are neither in contact with each other nor with blood.

Explicit 11
1. On the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for Explicit, type **Loaded boundaries** in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select the **All boundaries** check box.
5. Select Boundaries 10, 11, 16, 17, 20, 21, 23, 24, 36, 37, 39, 40, 42, 43, 45, 46, 50–53, 58, 59, 61, 62, 68, 69, 71, 72, 75, 76, 79, 80, 82, and 83 only.
   - The loaded boundaries are the inner artery boundaries that are in contact with blood.

Explicit 12
1. On the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for Explicit, type **Artery walls** in the **Label** text field.
3. Select Domain 2 only.
4. Locate the **Output Entities** section. From the **Output entities** list, choose **Adjacent boundaries**.
5. Select the **Interior boundaries** check box.
**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 Blood.

**Inlet 1**

1. On the Physics toolbar, click Boundaries and choose Inlet.
2. In the Settings window for Inlet, locate the Boundary Selection section.
3. From the Selection list, choose Inlet.
4. Locate the Boundary Condition section. From the list, choose Pressure.
5. Locate the Pressure Conditions section. In the $p_0$ text field, type $126.09 \text{[mmHg]}*f(t)$.

**Outlet 1**

1. On the Physics toolbar, click Boundaries and choose Outlet.
2. In the Settings window for Outlet, locate the Boundary Selection section.
3. From the Selection list, choose Outlet 1.
4. Locate the Pressure Conditions section. In the $p_0$ text field, type $125.91 \text{[mmHg]}*f(t)$.

**Outlet 2**

1. On the Physics toolbar, click Boundaries and choose Outlet.
2. In the Settings window for Outlet, locate the Boundary Selection section.
3. From the Selection list, choose Outlet 2.
4. Locate the Pressure Conditions section. In the $p_0$ text field, type $125.415 \text{[mmHg]}*f(t)$.

**Outlet 3**

1. On the Physics toolbar, click Boundaries and choose Outlet.
2. In the Settings window for Outlet, locate the Boundary Selection section.
3. From the Selection list, choose Outlet 3.
4. Locate the Pressure Conditions section. In the $p_0$ text field, type $125.415 \text{[mmHg]}*f(t)$.

**Outlet 4**

1. On the Physics toolbar, click Boundaries and choose Outlet.
2. In the Settings window for Outlet, locate the Boundary Selection section.
3. From the Selection list, choose Outlet 4.

4. Locate the Pressure Conditions section. In the $p_0$ text field, type $125.415 \text{[mmHg]} * f(t)$.

Outlet 5

1. On the Physics toolbar, click Boundaries and choose Outlet.

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

3. From the Selection list, choose Outlet 5.

4. Locate the Pressure Conditions section. In the $p_0$ text field, type $125.1 \text{[mmHg]} * f(t)$.

SOLID MECHANICS (SOLID)

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

2. Select Domains 1 and 2 only.

3. In the Model Builder window’s toolbar, click the Show button and select Discretization in the menu.

4. In the Settings window for Solid Mechanics, click to expand the Discretization section.

5. From the Displacement field list, choose Linear.

   This is just to keep down the size of the model.

Linear Elastic Material 1

1. In the Model Builder window, under Component 1 (comp1) > Solid Mechanics (solid) click Linear Elastic Material 1.

2. In the Settings window for Linear Elastic Material, locate the Linear Elastic Material section.

3. From the Specify list, choose Lamé parameters.

4. In the Model Builder window, click Solid Mechanics (solid).

Roller 1

1. On the Physics toolbar, click Boundaries and choose Roller.

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

3. From the Selection list, choose Roller boundaries.
**Boundary Load I**

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

   The Laminar Flow interface defines variables for the total stress applied to the fluid at the wall (spf.T_stressx, spf.T_stressy, and spf.T_stressz). You can use these variables to define the fluid forces applied to the blood vessel wall.

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

3. From the **Selection** list, choose **Loaded boundaries**.

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

   - spf.T_stressx \( x \)
   - spf.T_stressy \( y \)
   - spf.T_stressz \( z \)

**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, type **Blood** in the **Label** text field.

3. Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Blood**.

4. Locate the **Material Contents** section. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Property</th>
<th>Name</th>
<th>Value</th>
<th>Unit</th>
<th>Property group</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>rho</td>
<td>1060</td>
<td>kg/m³</td>
<td>Basic</td>
</tr>
<tr>
<td>Dynamic viscosity</td>
<td>mu</td>
<td>0.005</td>
<td>Pa·s</td>
<td>Basic</td>
</tr>
</tbody>
</table>

**Material 2 (mat2)**

1. Right-click **Materials** and choose **Blank Material**.

2. In the **Settings** window for Material, type **Artery** in the **Label** text field.

3. Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Artery**.
4. Locate the **Material Contents** section. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Property</th>
<th>Name</th>
<th>Value</th>
<th>Unit</th>
<th>Property group</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lamé parameter $\lambda$</td>
<td>lambLame</td>
<td>$20*\muLame - 2*\muLame /3$</td>
<td>N/m²</td>
<td>Lamé parameters</td>
</tr>
<tr>
<td>Lamé parameter $\mu$</td>
<td>muLame</td>
<td>$6.20e6$</td>
<td>N/m²</td>
<td>Lamé parameters</td>
</tr>
<tr>
<td>Density</td>
<td>rho</td>
<td>960</td>
<td>kg/m³</td>
<td>Basic</td>
</tr>
</tbody>
</table>

**Material 3 (mat3)**

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

<table>
<thead>
<tr>
<th>Property</th>
<th>Name</th>
<th>Value</th>
<th>Unit</th>
<th>Property group</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lamé parameter $\lambda$</td>
<td>lambLame</td>
<td>$20*\muLame - 2*\muLame /3$</td>
<td>N/m²</td>
<td>Lamé parameters</td>
</tr>
<tr>
<td>Lamé parameter $\mu$</td>
<td>muLame</td>
<td>$7.20e6$</td>
<td>N/m²</td>
<td>Lamé parameters</td>
</tr>
<tr>
<td>Density</td>
<td>rho</td>
<td>1200</td>
<td>kg/m³</td>
<td>Basic</td>
</tr>
</tbody>
</table>

**MESH 1**

**Free Tetrahedral 1**

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.

**Size 1**

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Tetrahedral 1** and choose **Size**.
2. In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. From the **Selection** list, choose **Blood**.
5. Locate the **Element Size** section. Click the **Custom** button.
6. Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
7. In the associated text field, type $1e-3$.  

---

15 | **FLUID-STRUCTURE INTERACTION IN A NETWORK OF BLOOD VESSELS**
**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 **Fine**.
4. Click the **Build All** button.

**STUDY 1**

The structural problem is quasi-static, so you can use the time just as a parameter for the parametric solver, together with a stationary solver. Thus the whole study can be divided into two steps. First run the transient study for the fluid-mechanics part of the problem and then use the stationary solver to solve the structural part using the solution from first transient study.

**Step 1: Time Dependent**

1. In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.
2. In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
3. In the **Times** text field, type `range(0,0.05,1.5)`.
4 Locate the **Physics and Variables Selection** section. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Physics interface</th>
<th>Solve for</th>
<th>Discretization</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solid Mechanics</td>
<td></td>
<td>physics</td>
</tr>
</tbody>
</table>

**Stationary**

On the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

**Step 2: Stationary**

1 In the **Settings** window for Stationary, locate the **Physics and Variables Selection** section.

2 In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Physics interface</th>
<th>Solve for</th>
<th>Discretization</th>
</tr>
</thead>
<tbody>
<tr>
<td>Laminar Flow</td>
<td></td>
<td>physics</td>
</tr>
</tbody>
</table>

3 Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Parameter name</th>
<th>Parameter value list</th>
<th>Parameter unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>t</td>
<td>range(0,0.05,1.5)</td>
<td>s</td>
</tr>
</tbody>
</table>

6 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

7 From the **Method** list, choose **Solution**.

8 From the **Study** list, choose **Study 1, Time Dependent**.

9 From the **Selection** list, choose **All**.

10 On the **Study** toolbar, click **Compute**.
RESULTS

Velocity (spf)
1 Click the Transparency button on the Graphics toolbar to restore original transparency state.
   By default, you get a slice plot of the velocity and a contour plot of the fluid pressure on the wall surface. The plot in Figure 4 corresponds to the first default plot.
2 In the Model Builder window, click Velocity (spf).
3 In the Settings window for 3D Plot Group, locate the Data section.
4 From the Time (s) list, choose 1.
5 In the Model Builder window, under Results>Velocity (spf) click Slice 1.
6 In the Settings window for Slice, locate the Plane Data section.
7 From the Plane list, choose ZX-planes.
8 In the Planes text field, type 1.
9 On the Velocity (spf) toolbar, click Plot.
10 Click the Go to Default 3D View button on the Graphics toolbar.

Pressure (spf)
The default unit for pressure plot is Pascal. As the mmHg unit is not available in the selection list, type it directly in the text field.
1 In the Model Builder window, under Results click Pressure (spf).
2 In the Settings window for 3D Plot Group, click to expand the Title section.
3 In the Model Builder window, expand the Pressure (spf) node, then click Pressure.
4 In the Settings window for Contour, locate the Expression section.
5 In the Unit field, type mmHg.
6 On the Pressure (spf) toolbar, click Plot.

Data Sets
To reproduce the plot shown in Figure 5, begin by defining a selection for the solution data set to make interior boundaries visible in the plot.

Study 1/Solution 1 (2) (sol1)
On the Results toolbar, click More Data Sets and choose Solution.

Selection
On the Results toolbar, click Selection.
Data Sets  
1 In the Settings window for Selection, locate the Geometric Entity Selection section.  
2 From the Geometric entity level list, choose Boundary.  
3 From the Selection list, choose Artery walls.

Stress (solid)  
1 In the Model Builder window, under Results click Stress (solid).  
2 In the Settings window for 3D Plot Group, type Displacement (solid) in the Label text field.  
3 Locate the Data section. From the Data set list, choose Study 1/Solution 1 (2) (sol1).  
4 From the Time (s) list, choose 1.

Displacement (solid)  
1 In the Model Builder window, expand the Results>Displacement (solid) node, then click Surface 1.  
2 In the Settings window for Surface, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component 1>Solid Mechanics>Displacement>solid.disp - Total displacement.  
3 Locate the Expression section. From the Unit list, choose μm.  
4 In the Model Builder window, expand the Results>Displacement (solid)>Surface 1 node, then click Deformation.  
5 In the Settings window for Deformation, locate the Scale section.  
6 Select the Scale factor check box.  
7 In the associated text field, type 300.  
8 On the Displacement (solid) toolbar, click Plot.  
9 Click the Go to Default 3D View button on the Graphics toolbar.