Model created in COMSOL Multiphysics 6.4

FDA Benchmark Blood Pump

Introduction

Centrifugal pumps are widely used in medical applications. Fluid-structure interaction is a vital aspect of a variety of medical devices. A centrifugal blood pump is one such device that uses a rotating impeller to pump blood.

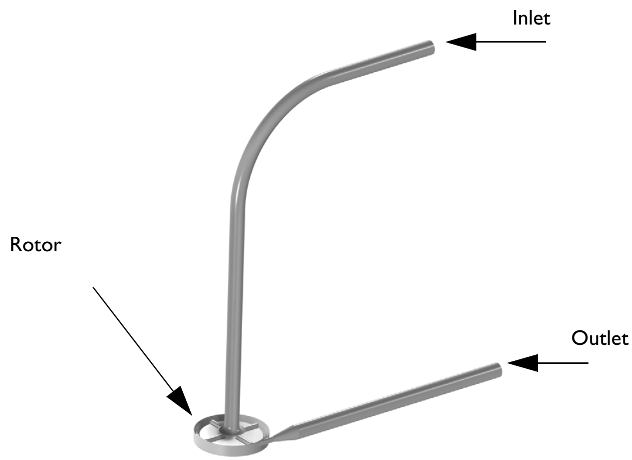


Figure 1: Geometry of the centrifugal blood pump.

This model features a centrifugal blood pump with four filleted blades (3 mm tall and 3 mm wide) placed at 90° angle on a 4 mm thick rotor base; see Figure 1. The CAD geometry used in this model is taken from Ref. 1.

Disclaimer: *CFD and blood damage validation studies were performed at the Food and Drug Administration (FDA) by employees of the Federal Government in the course of their official duties. Pursuant to Title 17, Section 105 of the United States Code, the CAD file obtained from the FDA is not subject to copyright protection and is in the public domain. Permission is hereby granted, free of charge, to any person obtaining a copy of the study results, to deal in such CAD file without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, or sell copies of such CAD file or derivatives, and to permit persons to whom such CAD file is furnished to do so. FDA assumes no responsibility whatsoever for use by other parties of the such CAD file, its contents or documentation, and makes no guarantees, expressed or implied, about its quality, reliability, or any other characteristic. Further, use of this CAD file in no way implies endorsement by the FDA or confers any advantage in* *regulatory decisions. Although this CAD file can be redistributed and/or modified freely, we ask that any derivative works bear some notice that they are derived from it, and any modified versions bear some notice that they have been modified.*

Model Definition

Following the FDA guidelines, the blood is assumed to be a Newtonian fluid. The k-ε turbulence model is used to obtain an initial flow solution for the shear stress transport (SST) model, which is subsequently used to obtain flow solutions with higher accuracy. Inlet flow rates of 2.5–7 L/min at a pump speed of 3500 rpm are simulated. A frozen rotor approach is used to compute the (pseudo) steady-state solution.

Results and Discussion

Table 1 shows a summary of wall shear stress and torque values. The maximum wall shear stress magnitude is computed over the housing rim (the 180° segment nearest the outlet) and also wall maximum shear stress magnitude is computed over the fillet around cutwater. The shaft torque value is computed from the integral of the total stress.

TABLE 1: PRESSURE HEAD (BETWEEN INLET AND OUTLET), WALL SHEAR STRESS MAGNITUDE OVER THE HOUSING RIM, AND SHAFT TORQUE FOR ALL SIMULATION CASES.

Case	**Inlet flow rate (L/m)**	**Maximum wall shear stress,housing rim (N·m^2)**	**Maximum wall shear stress, fillet (N·m^2)**	**Shaft torque (nN·m)**
1	2.5	318.59	302.82	28480
2	4.5	562.24	282.27	33841
4	7	668.15	270.06	36281

In Ref. 2, FDA published its benchmark centrifugal blood pump study, reporting results from computational studies as well as the data from its experiments. The experimental data is extracted from the graphs in the publication and compared with the results of the centrifugal benchmark blood pump model in order to validate the CFD results.

The pressure head across the pump is computed at a pump speed of 3500 rpm for several flow rates. The computed results show good agreement with the experimental results from Ref. 2, as is shown in Figure 2.

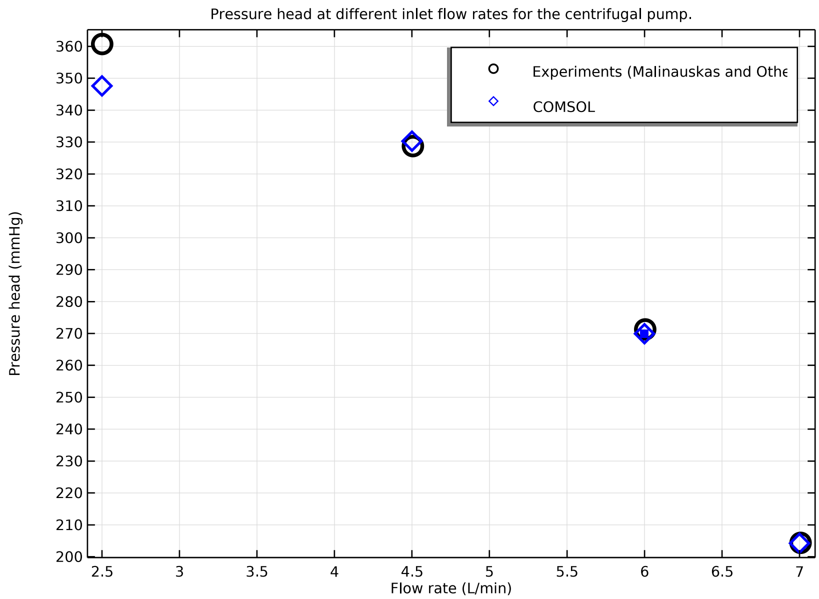


Figure 2: Pressure head at different inlet flow rates for the centrifugal pump operating at 3500 rpm.

The velocity magnitude at pump condition of 6 L/min and 3500 rpm is computed at the upper blade plane. The velocity magnitude, inside the blood pump model, based on the x- and y-velocity components along the radial cut line (see Figure 3) match the measurements qualitatively and are consistent with what is reported in other CFD studies Ref. 2. The results of the comparison are presented in Figure 4.

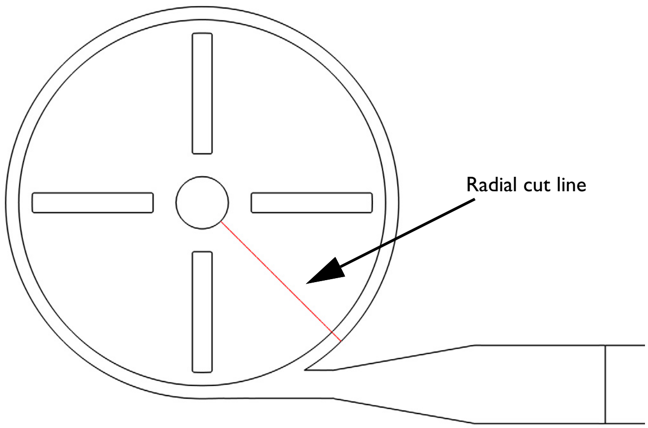


Figure 3: Radial cut line at upper blade plane.

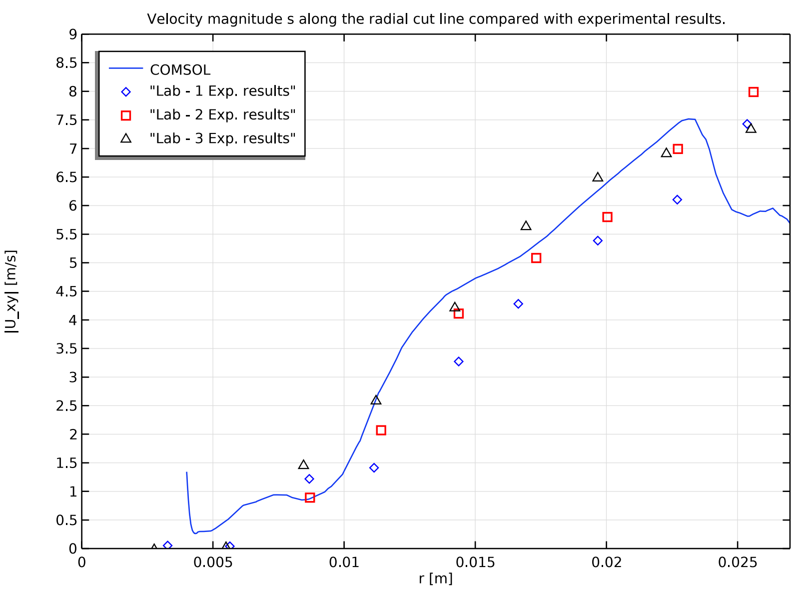


Figure 4: The velocity magnitude based on the x- and y-velocity components along the radial cut line compared with experimental data Ref. 2.

Similarly, the velocity profile was computed at x = 0.035 m in the diffuser region Figure 5 at a pump condition of 6 L/min and 3500 rpm. Figure 6 shows that the computed velocity magnitudes match the measurements qualitatively and are consistent with what is reported in Ref. 2.

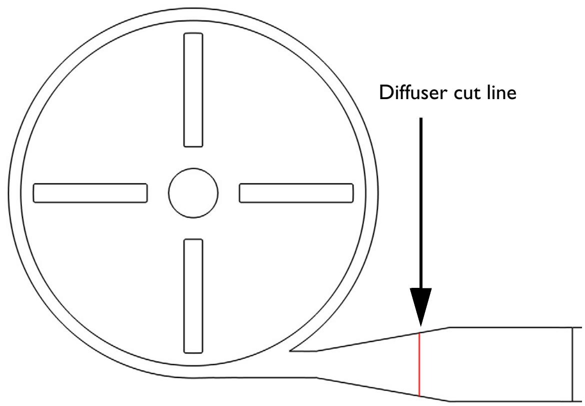


Figure 5: Diffuser cut line.

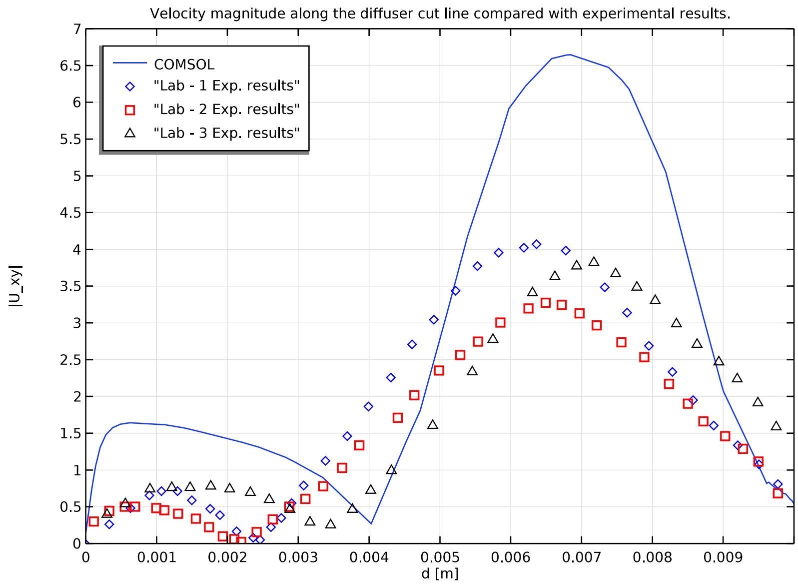


Figure 6: Velocity magnitude inside the blood pump based on the x- and y-velocity components along the diffuser cut line compared with experimental results from Ref. 2.

The interpolated contours of the three-dimensional velocity magnitude for the upper-blade passage plane under operating condition of 3500 rpm and an inlet flow rate of 6 L/min are plotted in Figure 7. The results qualitatively match those in Ref. 2.

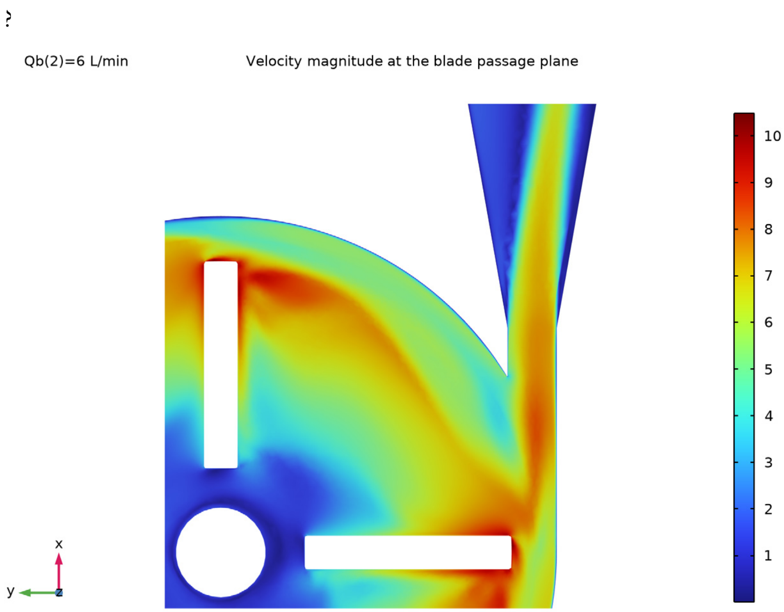


Figure 7: Velocity magnitude at the blade passage plane under operating condition of 3500 rpm and an inlet flow rate of 6 L/min.

References

1. Computational Fluid Dynamics Round Robin Study: github.com/OSEL-DAM/CFD-and-Blood-Damage-Benchmarks/tree/main.

2. R.A. Malinauskas and others, "FDA benchmark medical device flow models for CFD validation," *ASAIO Journal*, vol. 63, no. 2, pp. 150–160, 2017.

Application Library path: `CFD_Module/Verification_Examples/fda_benchmark_blood_pump`

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 **Fluid Flow** > **Single-Phase Flow** > **Turbulent Flow** > **Turbulent Flow, k-ε** (spf).
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Empty Study**.
6 Click **Done**.

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
TIME	0[s]	0 s	
rho_f	1035[kg/m^3]	1035 kg/m³	Blood density
mu_f	0.0035[Pa*s]	0.0035 Pa·s	Blood viscosity
Qb	6.0[L/min]	1E-4 m³/s	Inlet flow rate
rp	3500[rpm]	58.333 1/s	Impeller rpm

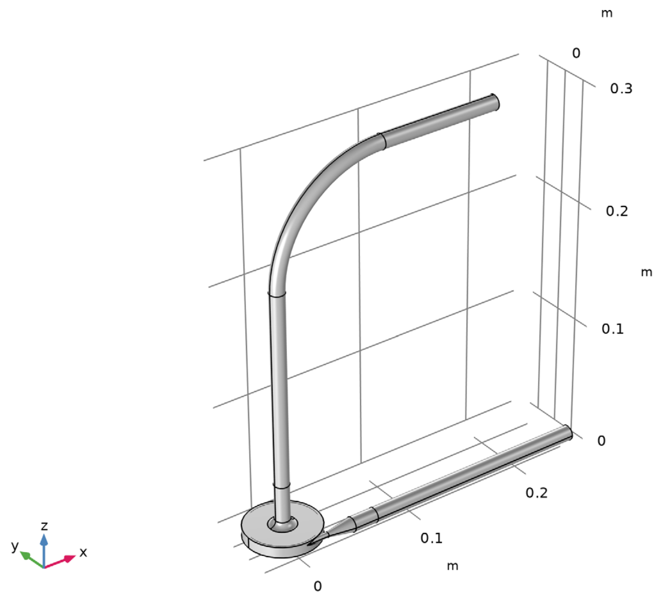
GEOMETRY 1

Create the geometry. To simplify this step, insert a prepared geometry sequence.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2 In the **Settings** window for **Geometry**, locate the **Advanced** section.
3 From the **Geometry representation** list, choose **CAD kernel**.
4 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.
5 Browse to the model's Application Libraries folder and double-click the file `fda_benchmark_blood_pump_geom_sequence.mph`.

   Full geometry instructions can be found at the end of the document.

6 In the **Geometry** toolbar, click **Build All**.



Disable the analysis of the geometry as the remaining small geometric details are needed.

7 In the **Model Builder** window, click **Geometry 1**.

8 Locate the **Cleanup** section. Clear the **Automatic detection of small details** checkbox.

MATERIALS

Blood

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

Property	Variable	Value	Unit	Property group
Density	rho	`rho_f`	kg/m³	Basic
Dynamic viscosity	mu	`mu_f`	Pa·s	Basic

COMPONENT 1 (COMP1)

A Moving Mesh is used to rotate the impeller/rotor.

Rotating Domain 1

1. In the **Physics** toolbar, click **Moving Mesh** and choose **Rotating Domain**.
2. In the **Settings** window for **Rotating Domain**, locate the **Domain Selection** section.
3. Click **Clear Selection**.
4. Click **Paste Selection**.
5. In the **Paste Selection** dialog, type `2` in the **Selection** text field.
6. Click **OK**.
7. In the **Settings** window for **Rotating Domain**, locate the **Rotation** section.
8. From the **Rotation type** list, choose **Specified rotational velocity**.
9. From the **Rotational velocity expression** list, choose **Constant revolutions per time**.
10. In the *f* text field, type `rp`.

The turbulent flow in the pump is resolved using a RANS model. To obtain initial values for the SST model, a k-ε model is first solved in Study 1.

TURBULENT FLOW, K-Ε- PRELIM

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, k-ε (spf)**.
2. In the **Settings** window for **Turbulent Flow, k-ε**, type `Turbulent Flow, k-[epsilon ]-Prelim` in the **Label** text field.

Wall 2

1. In the **Model Builder** window, expand the **Component 1 (comp1)** > **Turbulent Flow, k-ε** - Prelim (spf) node.
2. Right-click **Component 1 (comp1)** > **Turbulent Flow, k-ε** - Prelim (spf) and choose **Wall**.
3. In the **Settings** window for **Wall**, click to expand the **Wall Movement** section.
4. From the **Translational velocity** list, choose **Zero (Fixed wall)**.
5. Locate the **Boundary Selection** section. Click **Paste Selection**.
6. In the **Paste Selection** dialog, type `7` in the **Selection** text field.
7. Click **OK**.

Inlet 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2. In the **Settings** window for **Inlet**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `54` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.

7 From the list, choose **Fully developed flow**.

8 Locate the **Fully Developed Flow** section. Click the **Flow rate** button.

9 In the V_0 text field, type `Qb`.

Outlet 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `55` in the **Selection** text field.

5 Click **OK**.

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 In the table, clear the **Use** checkbox for **Geometric Analysis, Detail Size**.

4 Locate the **Sequence Type** section. From the list, choose **User-controlled mesh**.

Size

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

2 In the **Settings** window for **Size**, click to expand the **Element Size Parameters** section.

3 Locate the **Element Size** section. From the **Predefined** list, choose **Fine**.

4 Locate the **Element Size Parameters** section. In the **Minimum element size** text field, type `1e-3`.

Size 1

In the **Model Builder** window, right-click **Size 1** and choose **Delete**. Click **Yes** to confirm.

Corner Refinement 1

Right-click **Corner Refinement 1** and choose **Delete**. Click **Yes** to confirm.

Free Triangular 1

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

2 Select Boundary 40 only.

3 Right-click **Free Triangular 1** and choose **Move Up**.

4 Right-click **Free Triangular 1** and choose **Move Up**.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Calibrate for** list, choose **Fluid dynamics**.

4 From the **Predefined** list, choose **Finer**.

5 Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** checkbox. In the associated text field, type `1e-4`.

8 Select the **Minimum element size** checkbox. In the associated text field, type `1e-7`.

Free Tetrahedral 1

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

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

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `1 2` in the **Selection** text field.

6 Click **OK**.

Size 1

1 In the **Model Builder** window, right-click **Free Tetrahedral 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `2` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Size**, locate the **Element Size** section.

8 From the **Calibrate for** list, choose **Fluid dynamics**.

9 Click the **Custom** button.

10 Locate the **Element Size Parameters** section.

11 Select the **Maximum element size** checkbox. In the associated text field, type `1e-3`.

12 Select the **Minimum element size** checkbox. In the associated text field, type `4e-4`.

Size 2

1 In the **Model Builder** window, right-click **Free Tetrahedral 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `1` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Size**, locate the **Element Size** section.

8 From the **Calibrate for** list, choose **Fluid dynamics**.

9 Click the **Custom** button.

10 Locate the **Element Size Parameters** section.

11 Select the **Maximum element size** checkbox. In the associated text field, type `1e-3`.

12 Select the **Minimum element size** checkbox. In the associated text field, type `1e-7`.

13 Select the **Maximum element growth rate** checkbox. In the associated text field, type `1.4`.

14 Select the **Curvature factor** checkbox.

15 Select the **Resolution of narrow regions** checkbox.

Free Tetrahedral 1

In the **Model Builder** window, right-click **Free Tetrahedral 1** and choose **Size**.

Size 3

1 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `9-13, 28-33` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Size**, locate the **Element Size** section.

7 From the **Calibrate for** list, choose **Fluid dynamics**.

8 From the **Predefined** list, choose **Finer**.

9 Click the **Custom** button.

10 Locate the **Element Size Parameters** section.

11 Select the **Maximum element size** checkbox. In the associated text field, type `1e-3`.

12 Select the **Minimum element size** checkbox. In the associated text field, type `4e-4`.

13 Click **Build Selected**.

Swept 1

1 In the **Mesh** toolbar, click **Swept**.

2 Drag and drop below **Free Tetrahedral 1**.

3 In the **Settings** window for **Swept**, locate the **Domain Selection** section.

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

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog, type `3 4 5 6` in the **Selection** text field.

7 Click **OK**.

Distribution 1

1 In the **Model Builder** window, right-click **Swept 1** and choose **Distribution**.

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

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `3` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Distribution**, locate the **Distribution** section.

8 From the **Distribution type** list, choose **Predefined**.

9 In the **Number of elements** text field, type `60`.

10 In the **Element ratio** text field, type `3`.

11 Select the **Symmetric distribution** checkbox.

Distribution 2

1 In the **Model Builder** window, right-click **Swept 1** and choose **Distribution**.

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

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `4` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Distribution**, locate the **Distribution** section.

8 From the **Distribution type** list, choose **Predefined**.

9 In the **Number of elements** text field, type 70.

10 In the **Element ratio** text field, type 1.

Distribution 3

1 Right-click **Swept 1** and choose **Distribution**.

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

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type 5 in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Distribution**, locate the **Distribution** section.

8 From the **Distribution type** list, choose **Predefined**.

9 In the **Number of elements** text field, type 100.

10 In the **Element ratio** text field, type 2.

Distribution 4

1 Right-click **Swept 1** and choose **Distribution**.

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

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type 6 in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Distribution**, locate the **Distribution** section.

8 From the **Distribution type** list, choose **Predefined**.

9 In the **Number of elements** text field, type 30.

10 In the **Element ratio** text field, type 2.

Boundary Layers 1

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

2 In the **Settings** window for **Boundary Layers**, locate the **Geometric Entity Selection** section.

3 In the list box, select **1**.

4 From the **Geometric entity level** list, choose **Entire geometry**.

5 Click to expand the **Corner Settings** section. From the **Handling of sharp edges** list, choose **Splitting**.

Boundary Layer Properties 1

1 In the **Model Builder** window, expand the **Boundary Layers 1** node, then click **Boundary Layer Properties 1**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.

3 In the **Number of layers** text field, type `8`.

4 In the **Thickness adjustment factor** text field, type `1.1`.

5 Click **Build All**.

STUDY 1

Step 1: Frozen Rotor

1 In the **Study** toolbar, click **More Study Steps** and choose **Stationary** > **Frozen Rotor**.

A Frozen Rotor study is used to compute the steady-state flow involving the rotating machinery.

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

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

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

5 In the **Label** text field, type `Study 1 - k-epsilon`.

Solution 1 (sol1)

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

2 Click **Compute**.

DEFINITIONS

Wall Boundaries

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** node.

2 Right-click **Definitions** and choose **Selections** > **Explicit**.

3 In the **Settings** window for **Explicit**, type `Wall Boundaries` in the **Label** text field.

4 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog, type `1-4, 7, 9-19, 22, 23, 28-33, 40-51` in the **Selection** text field.

7 Click **OK**.

Inlet Pressure

1 In the **Definitions** toolbar, click **Probes** and choose **Boundary Probe**.

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

3 From the **Selection** list, choose **Manual**.

4 Click **Clear Selection**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog, type `54` in the **Selection** text field.

7 Click **OK**.

8 In the **Settings** window for **Boundary Probe**, type `Inlet Pressure` in the **Label** text field.

9 In the **Variable name** text field, type `p_in`.

10 Locate the **Expression** section. In the **Expression** text field, type `spf2.pA`.

11 In the **Table and plot unit** field, type `Pa`.

Outlet Pressure

1 In the **Definitions** toolbar, click **Probes** and choose **Boundary Probe**.

2 In the **Settings** window for **Boundary Probe**, type `Outlet Pressure` in the **Label** text field.

3 In the **Variable name** text field, type `p_out`.

4 Locate the **Source Selection** section. Click **Clear Selection**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog, type `55` in the **Selection** text field.

7 Click **OK**.

8 In the **Settings** window for **Boundary Probe**, locate the **Expression** section.

9 In the **Expression** text field, type `spf2.pA`.

10 In the **Table and plot unit** field, type `Pa`.

11 Click to expand the **Table and Window Settings** section. Click **Add Plot Window**.

An important indicator of less trauma to red blood cells and a lower systemic inflammatory response is shear stress.

Maximum Wall Shear Stress - Housing Rim

1 In the **Definitions** toolbar, click **Probes** and choose **Boundary Probe**.

2 In the **Settings** window for **Boundary Probe**, type `Maximum Wall Shear Stress - Housing Rim` in the **Label** text field.

3 In the **Variable name** text field, type `tau_housing_max`.

4 Locate the **Probe Type** section. From the **Type** list, choose **Maximum**.

5 Locate the **Source Selection** section. Click **Clear Selection**.

6 Click **Copy Selection**.

7 Click **Paste Selection**.

8 In the **Paste Selection** dialog, type `1` in the **Selection** text field.

9 Click **OK**.

10 In the **Settings** window for **Boundary Probe**, locate the **Expression** section.

11 In the **Expression** text field, type `spf2.u_tauWall^2*spf2.rho`.

12 In the **Table and plot unit** field, type `N/m^2`.

13 Click to expand the **Table and Window Settings** section. Click **Add Plot Window**.

Maximum Wall Shear Stress - Fillet

1 In the **Definitions** toolbar, click **Probes** and choose **Boundary Probe**.

2 In the **Settings** window for **Boundary Probe**, type `Maximum Wall Shear Stress - Fillet` in the **Label** text field.

3 In the **Variable name** text field, type `tau_fillet_max`.

4 Locate the **Probe Type** section. From the **Type** list, choose **Maximum**.

5 Locate the **Source Selection** section. From the **Selection** list, choose **Manual**.

6 Click **Clear Selection**.

7 Select Boundary 40 only.

8 Locate the **Expression** section. In the **Expression** text field, type `spf2.u_tauWall^2*spf2.rho`.

9 Click to expand the **Table and Window Settings** section. Click **Add Plot Window**.

Shaft Torque

1 In the **Definitions** toolbar, click **Probes** and choose **Boundary Probe**.

2 In the **Settings** window for **Boundary Probe**, type `Shaft Torque` in the **Label** text field.

3 In the **Variable name** text field, type `shaft_torque`.

4 Locate the **Probe Type** section. From the **Type** list, choose **Integral**.

5 Locate the **Source Selection** section. Click **Clear Selection**.

6 Click **Paste Selection**.

7 In the **Paste Selection** dialog, type `28-31` in the **Selection** text field.

8 Click **OK**.

9 In the **Settings** window for **Boundary Probe**, locate the **Expression** section.

10 In the **Expression** text field, type `x*spf2.T_tracy-y*spf2.T_tracx`.

11 In the **Table and plot unit** field, type `nN*m`.

12 Locate the **Table and Window Settings** section. Click **Add Plot Window**.

Average 1 (aveop1)

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

2 In the **Settings** window for **Average**, locate the **Source Selection** section.

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `21` in the **Selection** text field.

6 Click **OK**.

Average 2 (aveop2)

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

2 In the **Settings** window for **Average**, locate the **Source Selection** section.

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `55` in the **Selection** text field.

6 Click **OK**.

TURBULENT FLOW, K-ε- PRELIM (SPF)

Generate New Turbulence Model Interface 1

1 In the **Physics** toolbar, click **Global** and choose **Generate New Turbulence Model Interface**.

2 In the **Settings** window for **Generate New Turbulence Model Interface**, locate the **Study** section.

3 From the **Initial value from study** list, choose **Study 1 - k-epsilon**.

4 Locate the **Turbulence Model Interface** section. From the list, choose **Turbulent Flow, SST**.

5 Locate the **Model Generation** section. Click **Create**.

TURBULENT FLOW, SST 2 (SPF2)

Wall 1

1 In the **Model Builder** window, expand the **Turbulent Flow, SST 2 (spf2)** node, then click **Wall 1**.

2 In the **Settings** window for **Wall**, click to expand the **Wall Movement** section.

3 From the **Translational velocity** list, choose **Automatic from frame**.

STUDY 2 - SST

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

2 In the **Settings** window for **Study**, type `Study 2 - SST` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

4 In the **Home** toolbar, click **Compute**.

The **Generate New Turbulence Model** option is used to create a turbulent flow interface using the SST model solved with Study 2. This also generates initial values for the new interface based on the solution from Study 1.

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 **Empty Study**.

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

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

STUDY 3

Step 1: Frozen Rotor

1 In the **Study** toolbar, click **More Study Steps** and choose **Stationary** > **Frozen Rotor**.

2 In the **Settings** window for **Frozen Rotor**, locate the **Physics and Variables Selection** section.

3 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Turbulent Flow, k-ε - Prelim (spf)**.

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

5 In the **Settings** window for **Study**, type `Study 3 - SST - Qb Sweep` in the **Label** text field.

6 In the **Model Builder** window, click **Step 1: Frozen Rotor**.

7 In the **Settings** window for **Frozen Rotor**, click to expand the **Values of Dependent Variables** section.

8 Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

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

10 From the **Study** list, choose **Study 2 - SST, Frozen Rotor**.

11 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

12 From the **Study** list, choose **Study 2 - SST, Frozen Rotor**.

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

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

15 Right-click and choose **Add**.

16 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Qb (Inlet flow rate)	`7 4.5 2.5`	`L/min`

Solution 4 (sol4)

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

2 Click **Compute**.

RESULTS

Exterior Walls, Study 3 - SST - Qb Sweep/Solution 4

1 In the **Model Builder** window, expand the **Results** > **Datasets** node.

2 Right-click **Results** > **Datasets** and choose **Surface**.

3 In the **Settings** window for **Surface**, type `Exterior Walls, Study 3 - SST - Qb Sweep/Solution 4` in the **Label** text field.

4 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - SST - Qb Sweep/Solution 4 (sol4)**.

5 Locate the **Selection** section. From the **Selection** list, choose **Wall Boundaries**.

Exterior Walls, Study 2 - SST/Solution 2

1 Right-click **Exterior Walls, Study 3 - SST - Qb Sweep/Solution 4** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `Exterior Walls, Study 2 - SST/Solution 2` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - SST/Solution 2 (sol2)**.

4 Locate the **Selection** section. From the **Selection** list, choose **Wall Boundaries**.

Blade Passage Plane

1 In the **Results** toolbar, click **Cut Plane**.

2 In the **Settings** window for **Cut Plane**, type `Blade Passage Plane` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - SST/Solution 2 (sol2)**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.

5 In the **z-coordinate** text field, type `0.006562`.

Lower Gap Plane

1 In the **Results** toolbar, click **Cut Plane**.

2 In the **Settings** window for **Cut Plane**, type `Lower Gap Plane` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - SST/Solution 2 (sol2)**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.

5 In the **z-coordinate** text field, type `0.0005`.

zx Outlet Plane

1 In the **Results** toolbar, click **Cut Plane**.

2 In the **Settings** window for **Cut Plane**, type `zx Outlet Plane` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - SST/Solution 2 (sol2)**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.

5 In the **y-coordinate** text field, type `-0.027805`.

Upper Gap Plane

1 In the **Results** toolbar, click **Cut Plane**.

2 In the **Settings** window for **Cut Plane**, type `Upper Gap Plane` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - SST/Solution 2 (sol2)**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.

5 In the **z-coordinate** text field, type `0.0085`.

zx Inlet Plane

1 In the **Results** toolbar, click **Cut Plane**.

2 In the **Settings** window for **Cut Plane**, type `zx Inlet Plane` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - SST/Solution 2 (sol2)**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.

yz Inlet Plane

1 In the **Results** toolbar, click **Cut Plane**.

2 In the **Settings** window for **Cut Plane**, type `yz Inlet Plane` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - SST/Solution 2 (sol2)**.

Upper Blade Plane

1 In the **Results** toolbar, click **Cut Plane**.

2 In the **Settings** window for **Cut Plane**, type `Upper Blade Plane` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - SST/Solution 2 (sol2)**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.

5 In the **z-coordinate** text field, type `8[mm]-1.2[mm]`.

Radial Cut Line 2D

1 In the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, type `Radial Cut Line 2D` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Upper Blade Plane**.

4 Locate the **Line Data** section. In row **Point 2**, set **x** to `0.03*cos(45[deg])`.

5 In row **Point 2**, set **y** to `-0.03*cos(45[deg])`.

Diffuser Cut Line 2D

1 In the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, type `Diffuser Cut Line 2D` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Upper Blade Plane**.

4 Locate the **Line Data** section. In row **Point 1**, set **x** to `0.0367`.

5 In row **Point 2**, set **x** to `0.0367`.

6 In row **Point 2**, set **y** to `-0.04`.

Imported Experimental Results - Radial Cut Line 1

1 In the **Results** toolbar, click **Table**.

2 In the **Settings** window for **Table**, type `Imported Experimental Results - Radial Cut Line 1` in the **Label** text field.

3 Locate the **Data** section. Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `fda_benchmark_blood_pump_uxy_1_lab1.txt`.

5 Locate the **Column Headers** section. In the table, enter the following settings:

Column	Header
1	`r [m]`
2	`|U_xy| [m/s]`

Imported Experimental Results - Radial Cut Line 2

1 In the **Results** toolbar, click **Table**.

2 In the **Settings** window for **Table**, type `Imported Experimental Results - Radial Cut Line 2` in the **Label** text field.

3 Locate the **Data** section. Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `fda_benchmark_blood_pump_uxy_1_lab2.txt`.

5 Locate the **Column Headers** section. In the table, enter the following settings:

Column	Header
1	`r [m]`
2	`|U_xy| [m/s]`

Imported Experimental Results - Radial Cut Line 3

1 In the **Results** toolbar, click **Table**.

2 In the **Settings** window for **Table**, type `Imported Experimental Results - Radial Cut Line 3` in the **Label** text field.

3 Locate the **Data** section. Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `fda_benchmark_blood_pump_uxy_1_lab3a.txt`.

5 Locate the **Column Headers** section. In the table, enter the following settings:

Column	Header
1	`r [m]`
2	`|U_xy| [m/s]`

Imported Experimental Results -Diffuser Cut Line 1

1 In the **Results** toolbar, click **Table**.

2 In the **Settings** window for **Table**, type `Imported Experimental Results -Diffuser Cut Line 1` in the **Label** text field.

3 Locate the **Data** section. Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `fda_benchmark_blood_pump_uxy_2_lab1.txt`.

5 Locate the **Column Headers** section. In the table, enter the following settings:

Column	Header
1	d [m]
2	|U_xy| [m/s]

Imported Experimental Results -Diffuser Cut Line 2

1 In the **Results** toolbar, click **Table**.

2 In the **Settings** window for **Table**, type `Imported Experimental Results -Diffuser Cut Line 2` in the **Label** text field.

3 Locate the **Data** section. Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `fda_benchmark_blood_pump_uxy_2_lab2.txt`.

5 Locate the **Column Headers** section. In the table, enter the following settings:

Column	Header
1	d [m]
2	|U_xy| [m/s]

Imported Experimental Results -Diffuser Cut Line 3

1 In the **Results** toolbar, click **Table**.

2 In the **Settings** window for **Table**, type `Imported Experimental Results -Diffuser Cut Line 3` in the **Label** text field.

3 Locate the **Data** section. Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `fda_benchmark_blood_pump_uxy_2_lab3a.txt`.

5 Locate the **Column Headers** section. In the table, enter the following settings:

Column	Header
1	d [m]
2	|U_xy| [m/s]

Imported - Experimental Results - Pressure Head

1 In the **Results** toolbar, click **Table**.

2 In the **Settings** window for **Table**, type `Imported - Experimental Results - Pressure Head` in the **Label** text field.

3 Locate the **Data** section. Click **Import**.

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

5 Locate the **Column Headers** section. In the table, enter the following settings:

Column	Header
1	`Flow rate(L/min)`
2	`Pressure Head(mmHg)`

Pressure Head - Study 3

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Pressure Head - Study 3` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - SST - Qb Sweep/Solution 4 (sol4)**.

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

Expression	Unit	Description
`abs(aveop1(p2)-aveop2(p2))`	`mmHg`	

5 Click next to **Evaluate**, then choose **New Table**.

Pressure Head - Study 3

1 In the **Model Builder** window, under **Results** > **Tables** click **Table 9**.

2 In the **Settings** window for **Table**, type `Pressure Head - Study 3` in the **Label** text field.

Pressure Head - Study 2

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Pressure Head - Study 2` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - SST/Solution 2 (sol2)**.

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

Expression	Unit	Description
`Qb`	`L/min`	`Inlet flow rate`
`abs(aveop1(p2)-aveop2(p2))`	`mmHg`	

5 Click ▾ next to **Evaluate**, then choose **New Table**.

Pressure Head - Study 2

1 In the **Model Builder** window, under **Results** > **Tables** click **Table 10**.

2 In the **Settings** window for **Table**, type `Pressure Head - Study 2` in the **Label** text field.

Surface Integration 1 - Inflow

1 In the **Results** toolbar, click **More Derived Values** and choose **Integration** > **Surface Integration**.

2 In the **Settings** window for **Surface Integration**, type `Surface Integration 1 - Inflow` in the **Label** text field.

3 Select Boundary 52 only.

4 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - SST/Solution 2 (sol2)**.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
spf2.rho*(u2*nx+v2*ny+w2*nz)	kg/s	

Surface Integration 1 - Outflow

1 In the **Results** toolbar, click **More Derived Values** and choose **Integration** > **Surface Integration**.

2 In the **Settings** window for **Surface Integration**, type `Surface Integration 1 - Outflow` in the **Label** text field.

3 Select Boundary 53 only.

4 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - SST/Solution 2 (sol2)**.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
spf2.rho*(u2*nx+v2*ny+w2*nz)	kg/s	

Probe Plot Group 1

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 1**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 Clear the **Show legends** checkbox.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** checkbox. In the associated text field, type `Absolute pressure (Pa)`.

6 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

7 In the **Title** text area, type `Inlet Pressure`.

Probe Plot Group 2

1 In the **Model Builder** window, click **Probe Plot Group 2**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 Clear the **Show legends** checkbox.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** checkbox. In the associated text field, type `Absolute pressure (Pa)`.

6 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

7 In the **Title** text area, type `Outlet Pressure`.

Probe Plot Group 3

1 In the **Model Builder** window, expand the **Probe Plot Group 2** node, then click **Results** > **Probe Plot Group 3**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 Clear the **Show legends** checkbox.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** checkbox. In the associated text field, type `Maximum Wall Shear Stress (N/m^2)`.

6 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

7 In the **Title** text area, type `Maximum Wall Shear Stress on Housing Rim`.

Probe Plot Group 4

1 In the **Model Builder** window, click **Probe Plot Group 4**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 Clear the **Show legends** checkbox.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** checkbox. In the associated text field, type `Maximum Wall Shear Stress`.

6 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

7 In the **Title** text area, type `Maximum Wall Shear Stress on Fillet`.

8 In the **Probe Plot Group 4** toolbar, click **Plot**.

Probe Plot Group 5

1 In the **Model Builder** window, click **Probe Plot Group 5**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 Clear the **Show legends** checkbox.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** checkbox. In the associated text field, type `Shaft Torque (nN*m)`.

DEFINITIONS

View 8

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **View**.

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

3 Select the **Show axis units** checkbox.

4 Clear the **Show grid** checkbox.

Hide for Geometry 1

1 Right-click **View 8** and choose **Hide for Geometry**.

2 In the **Settings** window for **Hide for Geometry**, locate the **Selection** section.

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

4 Click the  **Paste Selection** button for **Selection**.

5 In the **Paste Selection** dialog, type `4-6, 8, 14, 16, 18, 20-22, 24-27, 34-37, 39, 42, 45, 47, 48, 50-52, 54, 55` in the **Selection** text field.

6 Click **OK**.

RESULTS

Velocity (spf2)

1 In the **Model Builder** window, under **Results** click **Velocity (spf2)**.

2 In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

5 From the **View** list, choose **View 8**.

6 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - SST/Solution 2 (sol2)**.

Surface 1

1 Right-click **Velocity (spf2)** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 Clear the **Color legend** checkbox.

4 Locate the **Data** section. From the **Dataset** list, choose **Blade Passage Plane**.

5 Locate the **Expression** section. In the **Expression** text field, type `spf2.U`.

Transparency 1

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

Surface 2

1 In the **Model Builder** window, right-click **Velocity (spf2)** and choose **Surface**.

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

3 From the **Dataset** list, choose **zx Outlet Plane**.

4 Locate the **Expression** section. In the **Expression** text field, type `spf2.U`.

5 Locate the **Coloring and Style** section. Clear the **Color legend** checkbox.

Transparency 1

Right-click **Surface 2** and choose **Transparency**.

Surface 3

1 In the **Model Builder** window, right-click **Velocity (spf2)** and choose **Surface**.

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

3 From the **Dataset** list, choose **zx Inlet Plane**.

4 Locate the **Expression** section. In the **Expression** text field, type `spf2.U`.

5 Locate the **Coloring and Style** section. Clear the **Color legend** checkbox.

Transparency 1

Right-click **Surface 3** and choose **Transparency**.

Velocity (spf2)

Right-click **Surface 3** > **Transparency 1** and choose **Surface**.

Surface 4

1 In the **Settings** window for **Surface**, locate the **Data** section.

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

3 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

4 From the **Color** list, choose **Gray**.

5 In the **Velocity (spf2)** toolbar, click **Plot**.

Pressure (spf2)

1 In the **Model Builder** window, under **Results** click **Pressure (spf2)**.

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

3 From the **Dataset** list, choose **Exterior Walls, Study 2 - SST/Solution 2**.

Surface

1 In the **Model Builder** window, expand the **Pressure (spf2)** node, then click **Surface**.

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

3 From the **Dataset** list, choose **Blade Passage Plane**.

4 Locate the **Expression** section. In the **Expression** text field, type `p2`.

Pressure (spf2)

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

Surface 2

1 In the **Settings** window for **Surface**, locate the **Data** section.

2 From the **Dataset** list, choose **zx Outlet Plane**.

3 Locate the **Expression** section. In the **Expression** text field, type `p2`.

4 Locate the **Coloring and Style** section. Clear the **Color legend** checkbox.

5 In the **Pressure (spf2)** toolbar, click **Plot**.

Wall Resolution (spf2)

1 In the **Model Builder** window, under **Results** click **Wall Resolution (spf2)**.

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

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

Turbulent Viscosity

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

2 In the **Settings** window for **3D Plot Group**, type `Turbulent Viscosity` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - SST/Solution 2 (sol2)**.

Surface 1

1 Right-click **Turbulent Viscosity** and choose **Surface**.

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

3 From the **Dataset** list, choose **Blade Passage Plane**.

4 Locate the **Expression** section. In the **Expression** text field, type `spf2.muT`.

Surface 2

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

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

3 From the **Dataset** list, choose **zx Outlet Plane**.

4 Locate the **Expression** section. In the **Expression** text field, type `spf2.muT`.

5 Locate the **Coloring and Style** section. Clear the **Color legend** checkbox.

Turbulent Viscosity

1 In the **Model Builder** window, click **Turbulent Viscosity**.

2 In the **Turbulent Viscosity** toolbar, click **Plot**.

Magnitude of Uxy - Radial

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

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **x-axis label** checkbox.

4 Select the **y-axis label** checkbox.

5 In the **x-axis label** text field, type `r [m]`.

6 In the **y-axis label** text field, type `|U_xy| [m/s]`.

7 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

8 In the **Title** text area, type `Velocity magnitude s along the radial cut line compared with experimental results.`.

9 In the **Label** text field, type `Magnitude of Uxy - Radial`.

10 Locate the **Data** section. From the **Dataset** list, choose **Radial Cut Line 2D**.

11 Locate the **Plot Settings** section. Select the **x-axis label** checkbox.

12 Select the **y-axis label** checkbox.

13 In the **x-axis label** text field, type `r [m]`.

14 In the **y-axis label** text field, type `|U_xy| [m/s]`.

15 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

16 In the **Title** text area, type `Velocity magnitude s along the radial cut line compared with experimental results.`.

17 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

18 In the **x minimum** text field, type `0`.

19 In the **x maximum** text field, type 0.027.

20 In the **y minimum** text field, type 0.

21 In the **y maximum** text field, type 9.

22 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Line Graph 1

1 Right-click **Magnitude of Uxy - Radial** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `sqrt(u2^2+v2^2)`.

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `sqrt(x^2+y^2)`.

6 Click to expand the **Legends** section. Select the **Show legends** checkbox.

7 From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
COMSOL

9 In the **Magnitude of Uxy - Radial** toolbar, click **Plot**.

Table Graph 1

1 In the **Model Builder** window, right-click **Magnitude of Uxy - Radial** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Table** list, choose **Imported Experimental Results - Radial Cut Line 1**.

4 From the **x-axis data** list, choose **r [m]**.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

6 From the **Color** list, choose **Blue**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Diamond**.

8 From the **Positioning** list, choose **In data points**.

9 Click to expand the **Legends** section. Select the **Show legends** checkbox.

10 From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
"Lab - 1 Exp. results"

12 In the **Magnitude of Uxy - Radial** toolbar, click **Plot**.

Table Graph 2

1 Right-click **Magnitude of Uxy - Radial** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Table** list, choose **Imported Experimental Results - Radial Cut Line 2**.

4 From the **x-axis data** list, choose **r [m]**.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

6 From the **Color** list, choose **Red**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Square**.

8 From the **Positioning** list, choose **In data points**.

9 Click to expand the **Legends** section. Select the **Show legends** checkbox.

10 From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
"Lab - 2 Exp. results"

12 In the **Magnitude of Uxy - Radial** toolbar, click **Plot**.

Table Graph 3

1 Right-click **Magnitude of Uxy - Radial** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Table** list, choose **Imported Experimental Results - Radial Cut Line 3**.

4 From the **x-axis data** list, choose **r [m]**.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

6 From the **Color** list, choose **Black**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Triangle**.

8 From the **Positioning** list, choose **In data points**.

9 Click to expand the **Legends** section. Select the **Show legends** checkbox.

10 From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
"Lab - 3 Exp. results"

12 In the **Magnitude of Uxy - Radial** toolbar, click **Plot**.

Magnitude of Uxy - Diffuser

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

2 In the **Settings** window for **1D Plot Group**, type `Magnitude of Uxy - Diffuser` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Diffuser Cut Line 2D**.

4 Locate the **Plot Settings** section. Select the **x-axis label** checkbox.

5 Select the **y-axis label** checkbox.

6 In the **x-axis label** text field, type `d [m]`.

7 In the **y-axis label** text field, type `|U_xy|`.

8 Locate the **Title** section. From the **Title type** list, choose **Manual**.

9 In the **Title** text area, type `Velocity magnitude along the diffuser cut line compared with experimental results.`.

10 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

11 In the **x minimum** text field, type `0`.

12 In the **x maximum** text field, type `0.01`.

13 In the **y minimum** text field, type `0`.

14 In the **y maximum** text field, type `7`.

15 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Line Graph 1

1 Right-click **Magnitude of Uxy - Diffuser** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `sqrt(u2^2+v2^2)`.

4 Click to expand the **Legends** section. Select the **Show legends** checkbox.

5 From the **Legends** list, choose **Manual**.

6 In the table, enter the following settings:

Legends
COMSOL

7 In the **Magnitude of Uxy - Diffuser** toolbar, click **Plot**.

Table Graph 1

1 In the **Model Builder** window, right-click **Magnitude of Uxy - Diffuser** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Table** list, choose **Imported Experimental Results -Diffuser Cut Line 1**.

4 From the **x-axis data** list, choose **d [m]**.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

6 From the **Color** list, choose **Blue**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Diamond**.

8 From the **Positioning** list, choose **In data points**.

9 Click to expand the **Legends** section. Select the **Show legends** checkbox.

10 From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
"Lab - 1 Exp. results"

12 In the **Magnitude of Uxy - Diffuser** toolbar, click **Plot**.

Table Graph 2

1 Right-click **Magnitude of Uxy - Diffuser** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Table** list, choose **Imported Experimental Results -Diffuser Cut Line 2**.

4 From the **x-axis data** list, choose **d [m]**.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

6 From the **Color** list, choose **Red**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Square**.

8 From the **Positioning** list, choose **In data points**.

9 Click to expand the **Legends** section. Select the **Show legends** checkbox.

10 From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
"Lab - 2 Exp. results"

12 In the **Magnitude of Uxy - Diffuser** toolbar, click **Plot**.

Table Graph 3

1 Right-click **Magnitude of Uxy - Diffuser** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Table** list, choose **Imported Experimental Results -Diffuser Cut Line 3**.

4 From the **x-axis data** list, choose **d [m]**.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

6 From the **Color** list, choose **Black**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Triangle**.

8 From the **Positioning** list, choose **In data points**.

9 Click to expand the **Legends** section. Select the **Show legends** checkbox.

10 From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
"Lab - 3 Exp. results"

12 In the **Magnitude of Uxy - Diffuser** toolbar, click **Plot**.

Pressure Head - Exp. Comparison

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

2 In the **Settings** window for **1D Plot Group**, type `Pressure Head - Exp. Comparison` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Pressure head at different inlet flow rates for the centrifugal pump.`.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** checkbox. In the associated text field, type `Flow rate (L/min)`.

7 Select the **y-axis label** checkbox. In the associated text field, type `Pressure head (mmHg)`.

Table Graph 1

1 Right-click **Pressure Head - Exp. Comparison** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Table** list, choose **Imported - Experimental Results - Pressure Head**.

4 From the **x-axis data** list, choose **Flow rate(L/min)**.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

6 From the **Color** list, choose **Black**.

7 From the **Width** list, choose **6**.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

9 Locate the **Legends** section. From the **Legends** list, choose **Manual**.

10 Select the **Show legends** checkbox.

11 In the table, enter the following settings:

Legends
`Experiments (Malinauskas and Others, ASAIO J., 2017)`

Table Graph 2

1 In the **Model Builder** window, right-click **Pressure Head - Exp. Comparison** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Table** list, choose **Pressure Head - Study 3**.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

5 From the **Color** list, choose **Blue**.

6 From the **Width** list, choose **6**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Diamond**.

8 Locate the **Legends** section. Select the **Show legends** checkbox.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
COMSOL

Table Graph 3

1 Right-click **Pressure Head - Exp. Comparison** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Table** list, choose **Pressure Head - Study 2**.

4 From the **x-axis data** list, choose **Inlet flow rate (L/min)**.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Solid**.

6 From the **Color** list, choose **Blue**.

7 From the **Width** list, choose **6**.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Diamond**.

9 In the **Pressure Head - Exp. Comparison** toolbar, click **Plot**.

Velocity magnitude at the blade passage plane

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

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

3 From the **Dataset** list, choose **Blade Passage Plane**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Velocity magnitude at the blade passage plane under operating condition of 3500 [rpm] and an inlet flow rate of 6 [L/min].`.

6 In the **Label** text field, type `Velocity magnitude at the blade passage plane`.

7 Locate the **Title** section. In the **Title** text area, type `Velocity magnitude at the blade passage plane`.

8 In the **Parameter indicator** text field, type `Qb(2)=6 L/min`.

9 Click to expand the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

10 From the **View** list, choose **New view**.

Surface 1

1 Right-click **Velocity magnitude at the blade passage plane** and choose **Surface**.

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

3 In the **Expression** text field, type `spf2.U`.

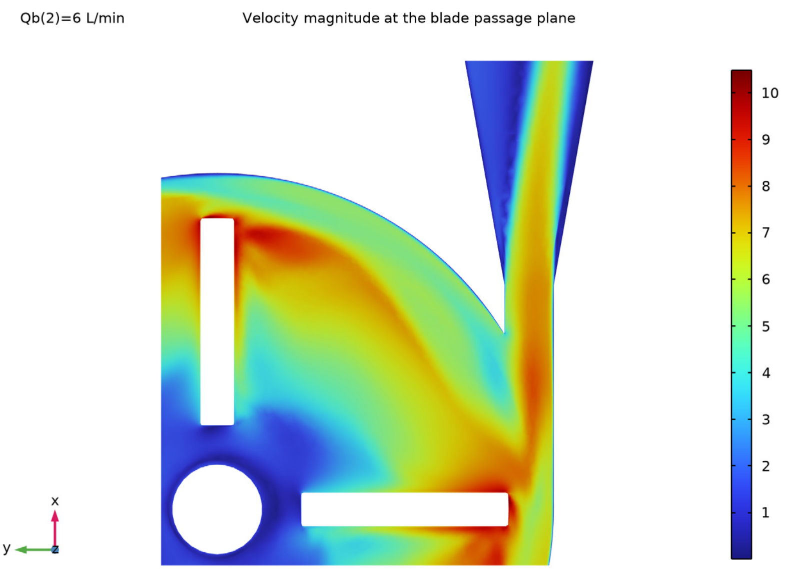
Filter 1

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

2 In the **Settings** window for **Filter**, locate the **Element Selection** section.

3 In the **Logical expression for inclusion** text field, type `x>-0.005 && x<0.04 && y<0.005`.

4 In the **Velocity magnitude at the blade passage plane** toolbar, click **Plot**.



Appendix — Geometry Modeling Instructions

ADD COMPONENT

In the **Home** toolbar, click **Add Component** and choose **3D**.

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 Click **Load from File**.

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

GEOMETRY 1

Check to see if the **Geometry representation** is set to **CAD kernel**. This is required to import the model geometry and requires the CAD Import Module. If the COMSOL kernel is selected proceed with the next step.

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2. In the **Settings** window for **Geometry**, locate the **Advanced** section.
3. From the **Geometry representation** list, choose **CAD kernel**.

Import 1 (imp1)

1. In the **Geometry** toolbar, click **Import**.
2. In the **Settings** window for **Import**, locate the **Source** section.
3. Click **Browse**.
4. Browse to the model's Application Libraries folder and double-click the file `fda_benchmark_blood_pump_rotor_cad_geometry.stp`.
5. Click **null**.
6. Click **Build Selected**.

Scale 1 (sca1)

1. In the **Geometry** toolbar, click **Transforms** and choose **Scale**.
2. Select the object **imp1** only.
3. In the **Settings** window for **Scale**, locate the **Scale Factor** section.
4. In the **Factor** text field, type `Sf`.
5. Click **Build Selected**.
6. Click the **Zoom Extents** button in the **Graphics** toolbar.

Work Plane 1 (wp1)

1. In the **Geometry** toolbar, click **Work Plane**.
2. In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.
3. From the **Plane type** list, choose **Face parallel**.
4. On the object **sca1**, select Boundary 16 only.
5. In the **Offset in normal direction** text field, type `z_os_rd`.

Work Plane 1 (wp1) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1) > Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `r_rd`.

4 Click **Build Selected**.

Extrude 1 (ext1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 From the **Specify** list, choose **Vertices to extrude to**.

4 On the object **sca1**, select Point 81 only.

5 Click **Build Selected**.

Work Plane 2 (wp2)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane type** list, choose **Face parallel**.

4 On the object **ext1**, select Boundary 4 only.

Work Plane 2 (wp2) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 2 (wp2) > Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `r_rd_h`.

Extrude 2 (ext2)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 From the **Specify** list, choose **Vertices to extrude to**.

4 On the object **sca1**, select Point 68 only.

Work Plane 3 (wp3)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane type** list, choose **Edge parallel**.

4 On the object **sca1**, select Edge 224 only.

Work Plane 3 (wp3) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 3 (wp3) > Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `r_rd_h`.

4 In the **Sector angle** text field, type `90`.

5 Locate the **Position** section. In the **yw** text field, type `r_h`.

6 Click **Build Selected**.

Revolve 1 (rev1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Revolve**.

2 In the **Settings** window for **Revolve**, locate the **Revolution Angles** section.

3 Click the **Angles** button.

4 Locate the **Revolution Axis** section. From the **Axis type** list, choose **3D**.

5 Find the **Direction of revolution axis** subsection. In the **y** text field, type `0`.

6 In the **z** text field, type `1`.

7 Click **Build Selected**.

Union 1 (uni1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **ext1**, **ext2**, and **rev1** only.

3 In the **Settings** window for **Union**, locate the **Union** section.

4 Clear the **Keep interior boundaries** checkbox.

Import 2 (imp2)

1 In the **Geometry** toolbar, click **Import**.

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

3 From the **Source** list, choose **3D CAD file**.

4 Locate the **Simplify and Repair** section. Clear the **Repair imported objects** checkbox.

5 In the **Absolute import tolerance** text field, type `1.0E-8`.

6 Locate the **Source** section. Click **Browse**.

7 Browse to the model's Application Libraries folder and double-click the file `fda_benchmark_blood_pump_housing_cad_geometry.stp`.

8 Click **null**.

9 In the **Geometry** toolbar, click **Defeaturing and Repair** and choose **Delete Fillets**.

10 In the **Model Builder** window, click **Geometry 1**.

11 In the **Tools** window for **Delete Fillets**, locate the **Delete Fillets** section.

12 Click the **Clear Selection** button for **Input objects**.

13 Select the object **imp2** only.

14 Click **Find Fillets**.

15 In the **Maximum fillet radius** text field, type `0.1`.

16 Click **Find Fillets**.

17 In the list box, select **Fillet 1**.

18 Click **Delete Selected**.

Scale 2 (sca2)

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

2 Select the object **dfi1** only.

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

4 In the **Factor** text field, type `Sf`.

5 Click **Build All Objects**.

Fillet 1 (fil1)

1 In the **Geometry** toolbar, click **Editing** and choose **Fillet**.

2 In the **Settings** window for **Fillet**, locate the **Radius** section.

3 In the **Radius** text field, type `0.00004`.

4 On the object **sca2**, select Edges 17 and 19 only.

5 Click **Build Selected**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the objects **fil1** and **uni1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Click to select the Activate Selection toggle button for **Objects to subtract**.

5 Select the object **sca1** only.

Work Plane 4 (wp4)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane type** list, choose **Edge parallel**.

4 On the object **dif1**, select Edge 129 only.

Partition Domains 1 (pard1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Domains**.

2 On the object **dif1**, select Domains 1 and 2 only.

Work Plane 5 (wp5)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane type** list, choose **Edge parallel**.

4 On the object **pard1**, select Edge 129 only.

5 In the **Offset in normal direction** text field, type `z_wp5`.

6 Select the **Reverse normal direction** checkbox.

Partition Domains 2 (pard2)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Domains**.

2 On the object **pard1**, select Domains 1 and 2 only.

Work Plane 6 (wp6)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane type** list, choose **Edge parallel**.

4 On the object **pard2**, select Edge 485 only.

5 In the **Offset in normal direction** text field, type `x_wp6`.

Partition Domains 3 (pard3)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Domains**.

2 On the object **pard2**, select Domains 1 and 2 only.

Extrude 3 (ext3)

1 In the **Geometry** toolbar, click **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **General** section.

3 From the **Extrude from** list, choose **Faces**.

4 On the object **pard3**, select Boundary 213 only.

5 Locate the **Distances** section. In the table, enter the following settings:

Distances (m)
x_in

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

Form Composite Faces 1 (cmf1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Faces**.

2 In the **Settings** window for **Form Composite Faces**, locate the **Input** section.

3 Click the **Paste Selection** button for **Faces to composite**.

4 In the **Paste Selection** dialog, type `18, 20, 22, 23, 26-29, 43, 48, 50, 51, 54, 55, 58, 59` in the **Selection** text field.

5 Click **OK**.

Form Composite Faces 2 (cmf2)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Faces**.

2 In the **Settings** window for **Form Composite Faces**, locate the **Input** section.

3 Click the **Paste Selection** button for **Faces to composite**.

4 In the **Paste Selection** dialog, type `40-43, 48, 49` in the **Selection** text field.

5 Click **OK**.

Form Composite Faces 3 (cmf3)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Faces**.

2 In the **Settings** window for **Form Composite Faces**, locate the **Input** section.

3 Click the **Paste Selection** button for **Faces to composite**.

4 In the **Paste Selection** dialog, type `68, 72, 78, 95-97, 102, 104, 121, 129, 144, 147` in the **Selection** text field.

5 Click **OK**.

Form Composite Faces 4 (cmf4)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Faces**.

2 In the **Settings** window for **Form Composite Faces**, locate the **Input** section.

3 Click the **Paste Selection** button for **Faces to composite**.

4 In the **Paste Selection** dialog, type `80, 87, 90, 119, 123, 134` in the **Selection** text field.

5 Click **OK**.

Form Composite Faces 5 (cmf5)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Faces**.

2 In the **Settings** window for **Form Composite Faces**, locate the **Input** section.

3 Click the **Paste Selection** button for **Faces to composite**.

4 In the **Paste Selection** dialog, type `143, 146, 148, 149, 151, 155, 156, 159, 160, 162, 164-169` in the **Selection** text field.

5 Click **OK**.

Form Composite Faces 6 (cmf6)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Faces**.

2 In the **Settings** window for **Form Composite Faces**, locate the **Input** section.

3 Click the **Paste Selection** button for **Faces to composite**.

4 In the **Paste Selection** dialog, type `149, 151-155` in the **Selection** text field.

5 Click **OK**.

Form Composite Faces 7 (cmf7)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Faces**.

2 In the **Settings** window for **Form Composite Faces**, locate the **Input** section.

3 Click the **Paste Selection** button for **Faces to composite**.

4 In the **Paste Selection** dialog, type `70, 72, 82, 83, 89, 94, 95, 97-99, 117, 118, 122, 126, 131, 137-139` in the **Selection** text field.

5 Click **OK**.

Form Composite Faces 8 (cmf8)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Faces**.

2 In the **Settings** window for **Form Composite Faces**, locate the **Input** section.

3 Click the Paste Selection button for **Faces to composite**.

4 In the **Paste Selection** dialog, type `83, 86, 88, 114, 116, 123` in the **Selection** text field.

5 Click **OK**.

Form Composite Faces 9 (cmf9)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Faces**.

2 In the **Settings** window for **Form Composite Faces**, locate the **Input** section.

3 Click the **Paste Selection** button for **Faces to composite**.

4 In the **Paste Selection** dialog, type `10-28, 30, 32, 39-47, 60, 65-71, 74-96, 108-146, 148, 149, 153-161` in the **Selection** text field.

5 Click **OK**.

Form Composite Faces 10 (cmf10)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Faces**.

2 In the **Settings** window for **Form Composite Faces**, locate the **Input** section.

3 Click the **Paste Selection** button for **Faces to composite**.

4 In the **Paste Selection** dialog, type `13-16, 48` in the **Selection** text field.

5 Click **OK**.

Form Composite Faces 11 (cmf11)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Faces**.

2 In the **Settings** window for **Form Composite Faces**, locate the **Input** section.

3 Click the **Paste Selection** button for **Faces to composite**.

4 In the **Paste Selection** dialog, type `40, 41, 44` in the **Selection** text field.

5 Click **OK**.