Created in COMSOL Multiphysics 5.6

Centrifugal Pump

Introduction

Centrifugal pumps are widely used in the industry and can be found in various applications. These pumps belong to the axisymmetric work-absorbing turbomachinery category for which fluid is transported through the conversion of rotational kinetic energy into hydrodynamic energy. In most applications, fluid enters the pump along the rotating axis and is accelerated by the impeller. The flow is expelled radially outward into a diffuser, or volute chamber, from where it exits. The rotational kinetic energy of the pump is typically supplied by an engine or a motor.

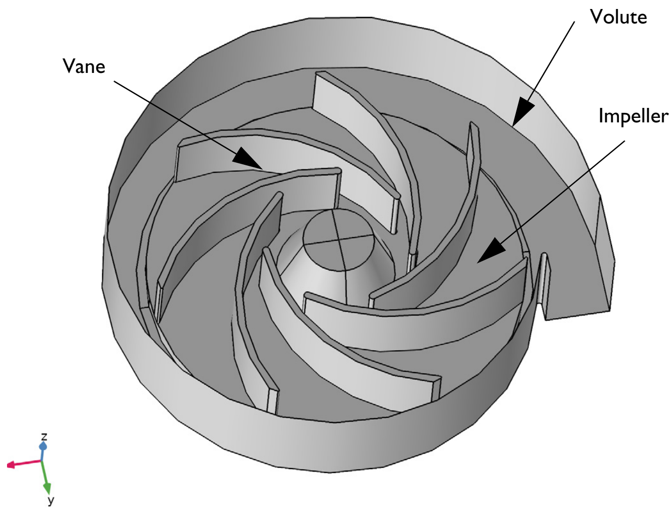


Figure 1: Geometry of the semiopen centrifugal pump.

The current model represents a semiopen centrifugal pump with seven vanes. For the semiopen impeller, the vanes are attached to the hub with a shroud on one side of the impeller. The volute has a spiral shape and the outer radius of the impeller is 10 cm. The size of the modeled pump is typical for automotive applications. The geometry in this work is highly parameterized, allowing straightforward modifications of the geometry to study different configurations of the centrifugal pump if needed.

Model Definition

This model shows how to set up rotating machinery simulations with the frozen rotor approach for centrifugal pumps. The equations that govern the physics are the Navier-Stokes equations and the continuity equation.

A frozen rotor is a cost and time efficient steady-state approximation where individual zones are assigned rotational different speeds. The flow in each of these zones is solved using the moving reference frame equations. In a sense, this approach can be described as freezing the motion of the moving part in a given position and then observing the resulting flow field with the rotor in that fixed position.

Turbulence is modeled with the k-ω model. This is a widely used model for turbomachinery simulations, with good performance for swirling flows and in the near-wall region.

The pressure condition at the inlet and outlet is set up using the `aveop` operator:

$$p_{inlet} = p_{\mathrm{tot}} - 0.5\rho \cdot \mathrm{aveop}(|\mathbf{u}|^2)$$

and

$$p_{outlet} = 0.5\rho \cdot \mathrm{aveop}(|\mathbf{u}|^2)$$

The problem is solved for different total pressure values, p_{tot}, at the inlet in order to obtain a pump curve for the specific geometry considered here.

Results and Discussion

The mass flow is monitored by two probe plots, one at the inlet and one at the outlet. Figure 2 shows that the mass flow at the inlet and the outlet are the equal, which means that mass conservation is achieved.

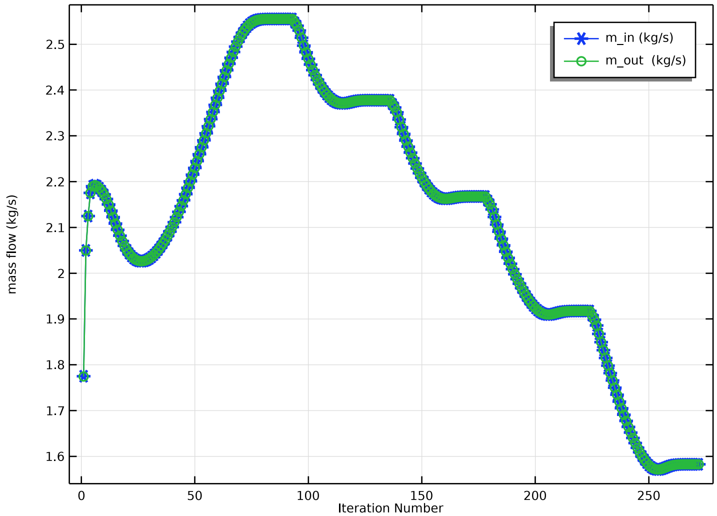


Figure 2: Mass flow probes at the inlet and the outlet.

Note that the five jumps in the curve represent a change in the given total pressure value at the inlet.

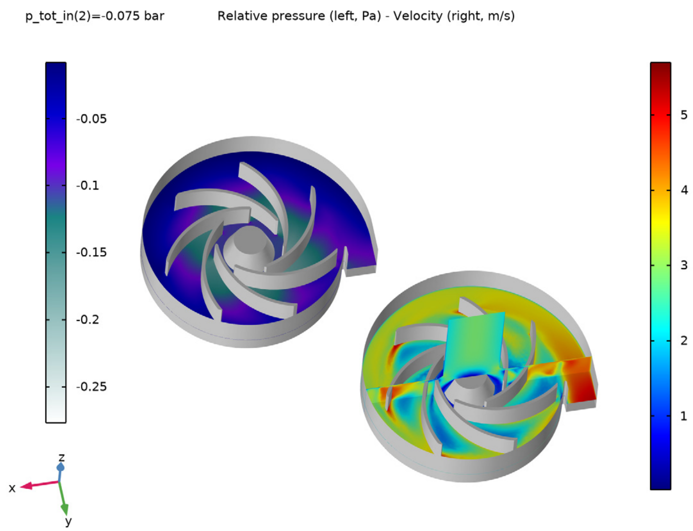


Figure 3: Distribution of the pressure and the velocity magnitude.

Examples of the pressure and velocity magnitude distributions are given in Figure 3. The solution clearly shows a rise in pressure and the corresponding change in velocity from the incoming (inlet) flow, radially toward the volute.

Finally, Figure 4 shows the pump performance curve. The total pressure at the inlet is expressed in terms of the pressure head, H, which is equal to

$$H = \frac{\Delta p_{\mathrm{tot}}}{\rho \cdot g}$$

This curve is central when designing a pump for a given application. Choosing the right pump configuration maximizes the pump and system efficiency, prolongs the life of the system and reduces operational costs.

Table 1 shows the relation between shaft power consumption and pump efficiency.

TABLE 1: PERFORMANCE DATA.

p_tot_in (bar)	Shaft power consumption (Nm/s)	Pump efficiency
-0.050	54.7	0.276
-0.075	52.6	0.382
-0.100	49.8	0.473

TABLE 1: PERFORMANCE DATA.

p_tot_in (bar)	Shaft power consumption (Nm/s)	Pump efficiency
-0.125	45.7	0.555
-0.150	39.6	0.620

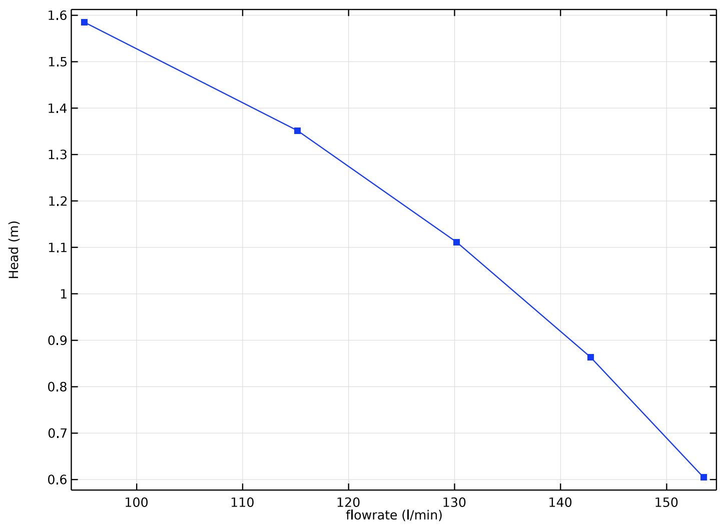


Figure 4: Pump curve.

Application Library path: `Mixer_Module/Tutorials/centrifugal_pump`

Modeling Instructions

Begin by loading the geometry file.

From the **File** menu, choose **Open**.

Browse to the model's Application Libraries folder and double-click the file `centrifugal_pump_geom_sequence.mph`.

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
p_tot_in	0.075[bar]	7500 Pa	Total pressure at the inlet
rot_rpm	1000[rpm]	16.667 1/s	Rotational speed
T_ref	20[degC]	293.15 K	Reference temperature

GEOMETRY 1

Partition Domains 1 (pard1)

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

2 In the **Settings** window for **Partition Domains**, locate the **Partition Domains** section.

3 From the **Partition with** list, choose **Extended faces**.

4 On the object **cmf2**, select Boundaries 13, 14, 73, and 94 only.

5 Find the **Domains to partition** subsection. Select the **Activate Selection** toggle button.

6 On the object **cmf2**, select Domain 1 only.

7 Click **Build Selected**.

ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Fluid Flow>Single-Phase Flow>Rotating Machinery, Fluid Flow>Turbulent Flow>Turbulent Flow, k-ω**.

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

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

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in>Water, liquid**.

4 Click **Add to Component** in the window toolbar.

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

DEFINITIONS

Boundary Probe 1 (bnd1)

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

2 In the **Settings** window for **Boundary Probe**, type `m_in` in the **Variable name** text field.

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

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

5 Click **Clear Selection**.

6 Select Boundary 58 only.

7 Locate the **Expression** section. In the **Expression** text field, type `-rhoRef*(u*nx+v*ny+w*nz)`.

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

Boundary Probe 2 (bnd2)

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

2 In the **Settings** window for **Boundary Probe**, type `m_out` in the **Variable name** text field.

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

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

5 Click **Clear Selection**.

6 Select Boundary 9 only.

7 Locate the **Expression** section. In the **Expression** text field, type `rhoRef*(u*nx+v*ny+w*nz)`.

8 Locate the **Table and Window Settings** section. From the **Plot window** list, choose **Probe Plot 1**.

Integration 1 (intop1)

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

2 In the **Settings** window for **Integration**, type `int_rot` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Walls 2**.

Integration 2 (intop2)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.
2 In the **Settings** window for **Integration**, type `int_in` in the **Operator name** text field.
3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4 Click **Paste Selection**.
5 In the **Paste Selection** dialog box, type `57` in the **Selection** text field.
6 Click **OK**.

Integration 3 (intop3)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.
2 In the **Settings** window for **Integration**, type `int_out` in the **Operator name** text field.
3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4 Click **Paste Selection**.
5 In the **Paste Selection** dialog box, type `6` in the **Selection** text field.
6 Click **OK**.

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 box, type `58` 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 **Point**.
4 Select Point 4 only.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.
2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
rhoRef	aveop2(spf.rhoref)	kg/m³	Reference density
delta_p	int_out(p)/int_out(1)-int_in(p)/int_in(1)	N/m²	Static pressure increase
delta_p_tot	((int_out(p+1/2*rhoRef*spf.U^2)/int_out(1)-int_in(p+1/2*rhoRef*spf.U^2)/int_in(1)))	N/m²	Total pressure increase
Torque	int_rot(+spf.T_stressx*y-spf.T_stressy*x)	N·m	Torque
Power	abs(int_rot(rot1.alphat)*Torque/int_rot(1))	N·m/s	Shaft power consumption
flowrate	int_in(u*nx+v*ny+w*nz)	m³/s	Flow rate
massflow	rhoRef*flowrate	kg/s	Mass flow
H_power	abs(massflow*delta_p_tot/rhoRef)	N·m/s	Power given to fluid
H	delta_p_tot/(rhoRef*g_const)	m	Head
eta	H_power/Power		Pump efficiency

Rotating Domain 1

1 In the **Model Builder** window, click **Rotating Domain 1**.

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

3 From the **Selection** list, choose **Rotating Domain 1**.

4 Locate the **Rotation** section. In the f text field, type rot_rpm.

5 Locate the **Axis** section. Specify the $\mathbf{u}_{\mathrm{rot}}$ vector as

0	X
0	Y
-1	Z

TURBULENT FLOW, K-ω (SPF)

Inlet 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Turbulent Flow, k-ω (spf)** and choose **Inlet**.

2 Select Boundary 58 only.

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

4 From the list, choose **Pressure**.

5 Locate the **Pressure Conditions** section. From the **Pressure** list, choose **Total**.

6 Select the **Average** check box.

7 In the p_0 text field, type `p_tot_in`.

8 Locate the **Turbulence Conditions** section. In the U_{ref} text field, type `3[m/s]`.

Outlet 1

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

2 In the **Settings** window for **Outlet**, locate the **Pressure Conditions** section.

3 From the **Pressure** list, choose **Total**.

4 Select Boundary 9 only.

Wall 2

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `64, 65, 87, 93` in the **Selection** text field.

5 Click **OK**.

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

7 From the **Translational velocity** list, choose **Zero (Fixed wall)**.

The **Translational velocity** is set to **Zero (Fixed Wall)** to ensure zero velocity at the lower wall. If set to **Automatic from frame**, it will rotate since it is adjacent to the **Rotating Domain**.

MESH 1

Size

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

Size 1

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

2 From the **Predefined** list, choose **Normal**.

Free Tetrahedral 1

1 In the **Model Builder** window, 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 box, type `1, 3, 4, 5` in the **Selection** text field.
6. Click **OK**.

Boundary Layers 1

1. In the **Model Builder** window, click **Boundary Layers 1**.
2. In the **Settings** window for **Boundary Layers**, locate the **Geometric Entity Selection** section.
3. Click **Clear Selection**.
4. Click **Paste Selection**.
5. In the **Paste Selection** dialog box, type `1, 3, 4, 5` in the **Selection** text field.
6. Click **OK**.
7. In the **Settings** window for **Boundary Layers**, click to expand the **Corner Settings** section.
8. In the **Minimum angle for trimming** text field, type `280`.

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 **Boundary Layer Properties** section.
3. In the **Number of boundary layers** text field, type `5`.
4. From the **Thickness of first layer** list, choose **Manual**.
5. In the **Thickness** text field, type `2.5e-4`.

Boundary Layer Properties 2

1. Right-click **Component 1 (comp1)>Mesh 1>Boundary Layers 1>Boundary Layer Properties 1** and choose **Duplicate**.
2. In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Layer Properties** section.
3. In the **Thickness** text field, type `6e-5`.
4. Select Boundaries 24, 27, 31, 36, 42, 45, 48, 75, and 105 only.

Boundary Layer Properties 3

1. Right-click **Boundary Layer Properties 2** and choose **Duplicate**.
2. Select Boundaries 15, 64–69, 87–90, 92, and 93 only.

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

4 In the **Thickness** text field, type `2e-4`.

Swept 1

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

Distribution 1

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

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

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

4 Click **Clear Selection**.

5 Click **Paste Selection**.

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

7 Click **OK**.

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

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

10 In the **Number of elements** text field, type `10`.

11 In the **Element ratio** text field, type `4`.

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 box, type `2` 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 `20`.

10 In the **Element ratio** text field, type `4`.

Use mapped mesh to improve the mesh quality.

Mapped 1

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

2 Select Boundaries 113 and 11 only.

Distribution 1

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

2 Select Edges 247 and 36 only.

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

4 In the **Number of elements** text field, type 20.

Distribution 2

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

2 Select Edges 245 and 16 only.

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

4 In the **Number of elements** text field, type 3.

Use mapped mesh to improve the mesh quality.

Mapped 2

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

2 Select Boundaries 39, 54, 55, 83, 84, and 97 only.

Distribution 1

1 In the **Model Builder** window, right-click **Mapped 2** and choose **Distribution**.

2 Select Edges 66 and 159 only.

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

4 In the **Number of elements** text field, type 20.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 2** and choose **Distribution**.

2 Select Edges 158, 189, and 190 only.

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

4 In the **Number of elements** text field, type 3.

Convert the mapped mesh to a triangular mesh.

Convert 1

1 In the **Mesh** toolbar, click **Modify** and choose **Elements>Convert**.

2 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

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 **Preset Studies for Selected Physics Interfaces>Frozen Rotor with Initialization**.

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

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

STUDY 1

Step 2: Frozen Rotor

1 In the **Model Builder** window, under **Study 1** click **Step 2: Frozen Rotor**.

2 In the **Settings** window for **Frozen Rotor**, click to expand the **Results While Solving** section.

3 From the **Probes** list, choose **None**.

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

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
p_tot_in (Total pressure at the inlet)	`range(-0.05,-0.1/4,-0.15)`	`bar`

The continuation solver works best for models with linear dependence on the parameter. A more robust alternative for nonlinear applications is to start from the solution for the previous parameter value.

7 From the **Run continuation for** list, choose **No parameter**.

8 From the **Reuse solution from previous step** list, choose **Yes**.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 2** node, then click **Segregated 1**.

4 In the **Settings** window for **Segregated**, click to expand the **Results While Solving** section.

5 From the **Probes** list, choose **All**.

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

7 In the **Settings** window for **Convergence Plot 2**, Click the right end of the **Quick Snapshot** split button in the window toolbar.

8 From the menu, choose **Zoom Extents**.

RESULTS

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results>Datasets** node, then click **Study 1/ Solution 1 (sol1)**.

Selection

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

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

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `1 3 4 5 6` in the **Selection** text field.

6 Click **OK**.

Exterior Walls

1 In the **Model Builder** window, click **Exterior Walls**.

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

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

Surface Average 1

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

2 Right-click **Derived Values** and choose **Average>Surface Average**.

3 In the **Settings** window for **Surface Average**, locate the **Selection** section.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `58` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Surface Average**, locate the **Expressions** section.

8 In the table, enter the following settings:

Expression	Unit	Description
`w`	`m/s`	`Velocity field, z component`

Surface Average 2

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Average>Surface Average**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type 58 in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Surface Average**, locate the **Expressions** section.

7 In the table, enter the following settings:

Expression	Unit	Description
p	bar	Pressure

Performance data

1 In the **Results** toolbar, click 8.5 **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type Performance data in the **Label** text field.

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

Expression	Unit	Description
delta_p	N/m^2	static pressure increase
delta_p_tot	N/m^2	total pressure increase
Torque	N*m	torque
Power	N*m/s	shaft power consumption
H_power	N*m/s	power given to fluid
eta	1	pump efficiency
H	1	Head
flowrate	l/min	flowrate

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

Performance data

1 In the **Model Builder** window, expand the **Results>Tables** node, then click **Table 2**.

2 In the **Settings** window for **Table**, type Performance data in the **Label** text field.

Pump Curve

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

2 In the **Settings** window for **1D Plot Group**, type Pump Curve in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **None**.

4 Click to expand the **Title** section.

Table Graph 1

1 Right-click **Pump Curve** and choose **Table Graph**.

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

3 From the **Table** list, choose **Performance data**.

4 From the **Plot columns** list, choose **Manual**.

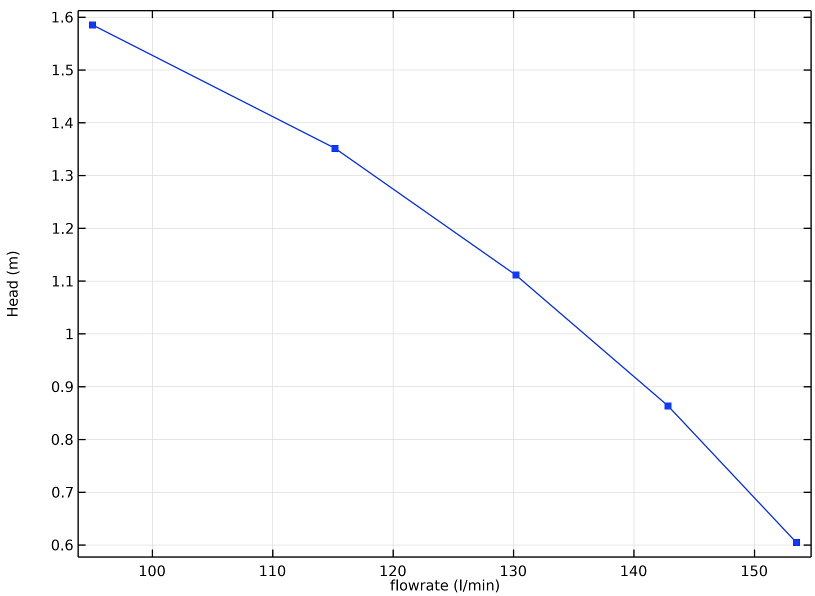
5 In the **Columns** list, select **Head (m)**.

6 From the **x-axis data** list, choose **flowrate (l/min)**.

7 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

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

9 In the **Pump Curve** toolbar, click **Plot**.



Velocity (spf)

1 In the **Model Builder** window, expand the **Results>Velocity (spf)** node, then click **Velocity (spf)**.

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

3. Clear the **Plot dataset edges** check box.
4. Locate the **Data** section. From the **Parameter value (p_tot_in (bar))** list, choose **-0.075**.
5. Click to expand the **Title** section. From the **Title type** list, choose **None**.

Slice

1. In the **Model Builder** window, click **Slice**.
2. In the **Settings** window for **Slice**, click to expand the **Title** section.
3. From the **Title type** list, choose **None**.
4. Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.
5. In the **Planes** text field, type `1`.
6. Select the **Interactive** check box.
7. In the **Shift** text field, type `-0.04`.
8. Locate the **Coloring and Style** section. Clear the **Color legend** check box.

Surface 1

1. In the **Model Builder** window, right-click **Velocity (spf)** and choose **Surface**.
2. In the **Settings** window for **Surface**, locate the **Data** section.
3. From the **Dataset** list, choose **Exterior Walls**.
4. Locate the **Expression** section. In the **Expression** text field, type `1`.
5. Click to expand the **Title** section. From the **Title type** list, choose **None**.
6. Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
7. From the **Color** list, choose **Gray**.
8. In the **Velocity (spf)** toolbar, click **Plot**.

Slice 2

1. Right-click **Velocity (spf)** and choose **Slice**.
2. In the **Settings** window for **Slice**, locate the **Expression** section.
3. In the **Expression** text field, type `p`.
4. From the **Unit** list, choose **bar**.
5. Locate the **Title** section. From the **Title type** list, choose **Manual**.
6. In the **Title** text area, type `Relative pressure (left, Pa) - Velocity (right, m/s)`.
7. Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.
8. In the **Planes** text field, type `1`.

9 Select the **Interactive** check box.

10 Locate the **Coloring and Style** section. From the **Color table** list, choose **AuroraAustralis**.

11 Locate the **Plane Data** section. In the **Shift** text field, type `-0.04`.

Deformation 1

1 Right-click **Slice 2** and choose **Deformation**.

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

3 In the **x component** text field, type `8[cm]*sqrt(2)`.

4 In the **y component** text field, type `-8[cm]*sqrt(2)`.

5 Locate the **Scale** section. Select the **Scale factor** check box.

6 In the associated text field, type `1`.

7 In the **Velocity (spf)** toolbar, click **Plot**.

Slice 3

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

2 In the **Settings** window for **Slice**, locate the **Title** section.

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

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

5 In the **Planes** text field, type `1`.

6 Select the **Interactive** check box.

7 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Slice**.

8 Locate the **Plane Data** section. In the **Shift** text field, type `0.006`.

Surface 2

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

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

3 From the **Dataset** list, choose **Exterior Walls**.

4 Locate the **Expression** section. In the **Expression** text field, type `1`.

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

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

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

Deformation 1

1 Right-click **Surface 2** and choose **Deformation**.

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

3 In the **x component** text field, type `8[cm]*sqrt(2)`.

4 In the **y component** text field, type `-8[cm]*sqrt(2)`.

5 Click to expand the **Title** section. Locate the **Scale** section. Select the **Scale factor** check box.

6 In the associated text field, type `1`.

Velocity (spf)

1 In the **Model Builder** window, click **Velocity (spf)**.

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

3 From the **Title type** list, choose **Automatic**.

4 Locate the **Color Legend** section. From the **Position** list, choose **Alternating**.

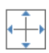
Slice

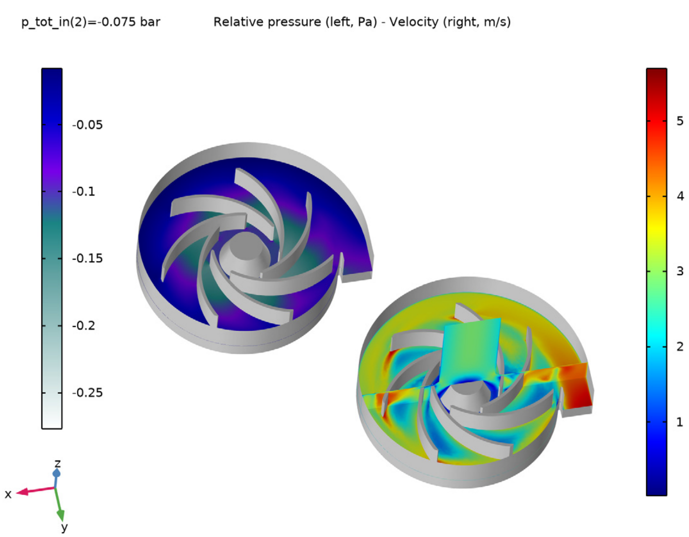
1 In the **Model Builder** window, click **Slice**.

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

3 Select the **Color legend** check box.

4 In the **Velocity (spf)** toolbar, click  **Plot**.

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



Pressure (spf)

1 In the **Model Builder** window, click **Pressure (spf)**.

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

3 From the **Parameter value (p_tot_in (bar))** list, choose **-0.075**.

Pressure

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

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

3 From the **Dataset** list, choose **Exterior Walls**.

4 Locate the **Expression** section. From the **Unit** list, choose **bar**.

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

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

Wall Resolution

1 In the **Model Builder** window, expand the **Wall Resolution (spf)** node, then click **Wall Resolution**.

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

3 From the **Dataset** list, choose **Exterior Walls**.

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

5 In the **Wall Resolution (spf)** toolbar, click **Plot**.

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 **Plot Settings** section.

3 Select the **y-axis label** check box.

4 In the associated text field, type `mass flow (kg/s)`.

Probe Table Graph 1

1 In the **Model Builder** window, expand the **Probe Plot Group 4** node, then click **Probe Table Graph 1**.

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

3 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

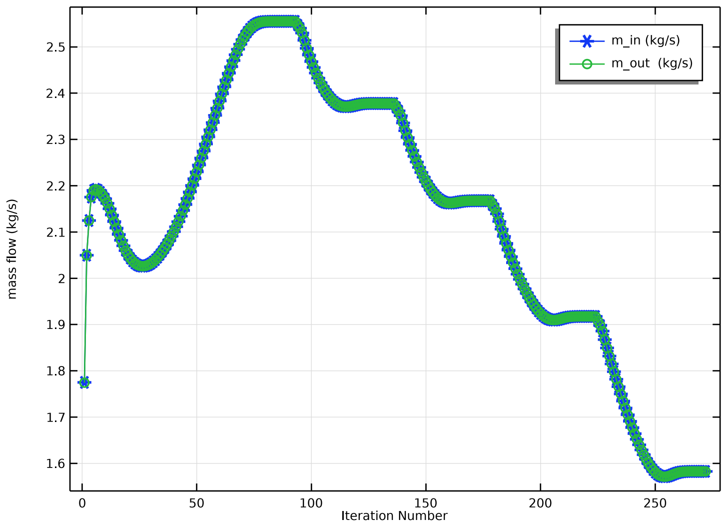
4 From the **Positioning** list, choose **In data points**.

5 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

6 In the table, enter the following settings:

Legends
`m_in (kg/s)`
`m_out  (kg/s)`

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



Exterior Walls 2

1 In the **Results** toolbar, click **More Datasets** and choose **Surface**.

2 In the **Settings** window for **Surface**, type `Exterior Walls  2` in the **Label** text field.

3 Select Boundaries 1–3, 5, 7, 8, 10, 11, 15, 25–38, 40–53, 56, 59–61, 64–78, 82, 85, 87–90, 92–94, 99, and 101–115 only.

Study 1/Solution 1 (4) (sol1)

In the **Results** toolbar, click **More Datasets** and choose **Solution**.

Cut Plane 1

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

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (4) (sol1)**.

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

5 In the **z-coordinate** text field, type `0.0125`.

3D Plot Group 6

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 **Study 1/Solution 1 (4) (sol1)**.

4 From the **Parameter value (p_tot_in (bar))** list, choose **-0.075**.

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Surface 1

1 Right-click **3D Plot Group 6** and choose **Surface**.

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

3 From the **Dataset** list, choose **Exterior Walls 2**.

4 From the **Parameter value (p_tot_in (bar))** list, choose **-0.075**.

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

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

Surface 2

1 In the **Model Builder** window, right-click **3D Plot Group 6** and choose **Surface**.

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

3 From the **Dataset** list, choose **Cut Plane 1**.

4 From the **Parameter value (p_tot_in (bar))** list, choose **-0.075**.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **JupiterAuroraBorealis**.

3D Plot Group 6

In the **Model Builder** window, click **3D Plot Group 6**.

Streamline Surface 1

1 In the **3D Plot Group 6** toolbar, click **More Plots** and choose **Streamline Surface**.

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

3 From the **Dataset** list, choose **Cut Plane 1**.

4 From the **Parameter value (p_tot_in (bar))** list, choose **-0.075**.

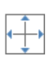
5 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Uniform density**.

6 In the **Separating distance** text field, type `0.01`.

7 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Tube**.

8. In the **Tube radius expression** text field, type `0.05`.
9. Select the **Radius scale factor** check box.
10. In the associated text field, type `0.005`.
11. Find the **Point style** subsection. From the **Color** list, choose **Custom**.
12. On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.
13. Click **Define custom colors**.
14. Set the RGB values to 105, 105, and 105, respectively.
15. Click **Add to custom colors**.
16. Click **Show color palette only** or **OK** on the cross-platform desktop.

Streamline 2

1. Right-click **3D Plot Group 6** and choose **Streamline**.
2. In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.
3. In the **Number** text field, type `14`.
4. Select Boundary 58 only.
5. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Tube**.
6. In the **Tube radius expression** text field, type `0.05`.
7. Select the **Radius scale factor** check box.
8. In the associated text field, type `0.005`.
9. Find the **Point style** subsection. From the **Color** list, choose **Custom**.
10. On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.
11. Click **Define custom colors**.
12. Set the RGB values to 105, 105, and 105, respectively.
13. Click **Add to custom colors**.
14. Click **Show color palette only** or **OK** on the cross-platform desktop.
15. In the **3D Plot Group 6** toolbar, click  **Plot**.
16. Click the  **Zoom Extents** button in the **Graphics** toolbar.