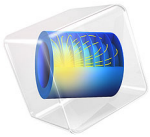


Magnetohydrodynamics Pump

Introduction

When an electrically conducting media is exposed to a time-varying magnetic field, eddy currents are induced that will counteract the change of magnetic flux and create a repelling force on the material. This magnetohydrodynamical principle can be utilized to create pumping action on a conducting liquid in a hermetically sealed column, without having to use moving parts.

Model Definition

The model is set up in a 2D axisymmetric geometry using the **Magnetic Fields** and **Laminar Flow** physics interfaces, coupled via the **Magnetohydrodynamics** multiphysics interface.

The model coupling relies on separate study types for the two physics interfaces, where the Magnetic Fields is solved in the frequency domain and the Laminar Flow is solved in the stationary domain. The cycle-averaged Lorentz force is employed in the fluid flow, and conversely the phase-dependent electromotive force is employed in the electromagnetic calculation. The cycle-averaged force on the liquid will be in the direction of the phase velocity of the magnetic field, where the latter is induced with a 3-phase coil setup. At both ends of the flow column there is a periodic condition for the pressure, fluid velocity, and magnetic vector potential, emulating an infinitely extended pump setup.

Results

Figure 1 shows the magnetic flux density norm on the 2D axisymmetric cross section of the pump.

Figure 2 shows the magnetic flux density as well as the fluid velocity norm on the partially revolved 2D axisymmetric geometry, with domain deformation illustrating the magnitude and direction of the fluid flow in the liquid column.

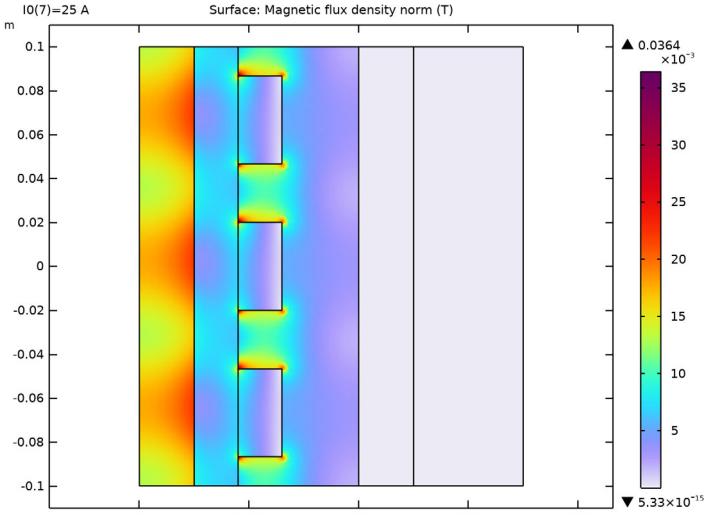


Figure 1: The magnetic flux density norm plotted on the 2D axisymmetric cross section of the pump.

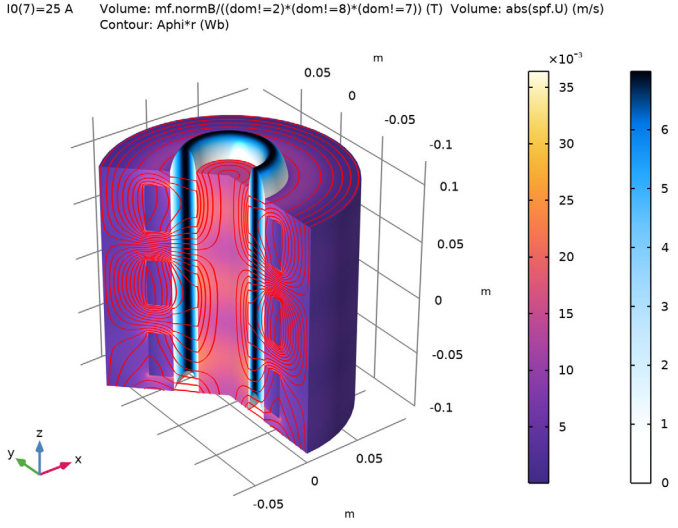


Figure 2: The velocity norm and the magnetic flux density norm plotted on the partially revolved 2D axisymmetric geometry.

Application Library path: ACDC_Module/Electromagnetics_and_Fluids/magnetohydrodynamics_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 **2D Axisymmetric**.

2 In the **Select Physics** tree, select **AC/DC>Electromagnetics and Fluids> Magnetohydrodynamics, Out-of-Plane Currents**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics>Frequency-Stationary**.

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
N	10	10	number or turns in coils
I0	1[A]	1 A	coil current magnitude
I1	I0*exp(-i*120[deg])	(-0.5-0.86603i) A	phase 1
I2	I0	1 A	phase 2
I3	I0*exp(i*120[deg])	(-0.5+0.86603i) A	phase 3

GEOMETRY 1

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 **Default repair tolerance** list, choose **Relative**.

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `0.175`.

4 In the **Height** text field, type `0.2`.

5 Locate the **Position** section. In the **z** text field, type `-0.1`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.05

7 Select the **Layers to the right** check box.

8 Clear the **Layers on bottom** check box.

9 Click **Build Selected**.

Rectangle 2 (r2)

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

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

3 In the **Width** text field, type `0.1`.

4 In the **Height** text field, type `0.2`.

5 Locate the **Position** section. In the **z** text field, type `-0.1`.

6 Click **Build Selected**.

Rectangle 3 (r3)

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

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

3 In the **Width** text field, type `0.02`.

4 In the **Height** text field, type `0.2`.

5 Locate the **Position** section. In the **r** text field, type `0.025`.

6 In the **z** text field, type `-0.1`.

7 Click **Build Selected**.

Rectangle 4 (r4)

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

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

3 In the **Width** text field, type `0.02`.

4 In the **Height** text field, type `0.04`.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 In the **r** text field, type `0.055`.

7 Click **Build Selected**.

Copy 1 (copy1)

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

2 Select the object **r4** only.

3 In the **Settings** window for **Copy**, locate the **Displacement** section.

4 In the **z** text field, type `2*0.1/3`.

5 Click **Build Selected**.

Copy 2 (copy2)

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

2 Select the object **r4** only.

3 In the **Settings** window for **Copy**, locate the **Displacement** section.

4 In the **z** text field, type `-2*0.1/3`.

5 Click **Build Selected**.

DEFINITIONS

Infinite Element Domain 1 (ie1)

1 In the **Definitions** toolbar, click **Infinite Element Domain**.

2 Select Domain 8 only.

3 In the **Settings** window for **Infinite Element Domain**, locate the **Geometry** section.

4 From the **Type** list, choose **Cylindrical**.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

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

3 Click **Clear Selection**.

4 Select Domain 2 only.

MATERIALS

Air

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 `Air` in the **Label** text field.

3 Select Domains 4–8 only.

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

Property	Variable	Value	Unit	Property group
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	I	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	I	Basic

Iron

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

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

3 Select Domains 1 and 3 only.

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

Property	Variable	Value	Unit	Property group
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1e3	I	Basic

Property	Variable	Value	Unit	Property group
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

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 **AC/DC>Liquid Metals>Lithium, 200 °C**.

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

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

MATERIALS

Lithium, 200 °C (mat3)

Select Domain 2 only.

MAGNETIC FIELDS (MF)

In the **Model Builder** window, under **Component 1 (comp1)** click **Magnetic Fields (mf)**.

Periodic Condition 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `2, 3, 5, 6, 8, 21, 26, 27, 29, 30` in the **Selection** text field.

5 Click **OK**.

Multi- Turn Coil 1

1 In the **Physics** toolbar, click **Domains** and choose **Coil**.

2 In the **Settings** window for **Coil**, type `Multi- Turn Coil 1` in the **Label** text field.

3 Locate the **Domain Selection** section. Click **Paste Selection**.

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

5 Click **OK**.

6 In the **Settings** window for **Coil**, locate the **Coil** section.

7 From the **Conductor model** list, choose **Homogenized multiturn**.

8 In the I_{coil} text field, type `I1`.

9 Locate the **Homogenized Multiturn Conductor** section. In the N text field, type `N`.

Multi- Turn Coil 2

1 Right-click **Multi- Turn Coil 1** and choose **Duplicate**.

2 In the **Settings** window for **Coil**, type `Multi- Turn Coil 2` in the **Label** text field.

3 Locate the **Domain Selection** section. Click  **Clear Selection**.

4 Select Domain 5 only.

5 Locate the **Coil** section. In the I_{coil} text field, type `I2`.

Multi- Turn Coil 3

1 Right-click **Multi- Turn Coil 2** and choose **Duplicate**.

2 In the **Settings** window for **Coil**, type `Multi- Turn Coil 3` in the **Label** text field.

3 Locate the **Domain Selection** section. Click  **Clear Selection**.

4 Select Domain 4 only.

5 Locate the **Coil** section. In the I_{coil} text field, type `I3`.

LAMINAR FLOW (SPF)

Fluid Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Laminar Flow (spf)** click **Fluid Properties 1**.

2 In the **Settings** window for **Fluid Properties**, locate the **Model Input** section.

3 From the T list, choose **User defined**. In the associated text field, type `T`.

Pressure Point Constraint 1

1 In the **Physics** toolbar, click  **Points** and choose **Pressure Point Constraint**.

2 In the **Settings** window for **Pressure Point Constraint**, locate the **Point Selection** section.

3 Click  **Paste Selection**.

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

5 Click **OK**.

Periodic Flow Condition 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Periodic Flow Condition**.

2 In the **Settings** window for **Periodic Flow Condition**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

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

5 Click **OK**.

MULTIPHYSICS

Magnetohydrodynamics 1 (mhd1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Magnetohydrodynamics 1 (mhd1)**.

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

3 Click **Clear Selection**.

4 Select Domain 2 only.

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

3 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**, locate the **Element Size** section.

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

4 Click **Build Selected**.

Size 1

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

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

3 From the **Predefined** list, choose **Extra fine**.

4 Click **Build Selected**.

Size 2

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Size 2** and choose **Delete**.

Size 3

In the **Model Builder** window, right-click **Size 3** and choose **Delete**.

Distribution 1

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

Edge 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Edge 1**.
2. In the **Settings** window for **Edge**, locate the **Boundary Selection** section.
3. Click  **Clear Selection**.
4. Click  **Paste Selection**.
5. In the **Paste Selection** dialog box, type `5-6` in the **Selection** text field.
6. Click **OK**.
7. In the **Settings** window for **Edge**, click to expand the **Control Entities** section.
8. In the **Number of iterations** text field, type `8`.
9. In the **Maximum element depth to process** text field, type `8`.

Distribution 1

1. Right-click **Edge 1** and choose **Distribution**.
2. In the **Settings** window for **Distribution**, locate the **Distribution** section.
3. From the **Distribution type** list, choose **Predefined**.
4. In the **Number of elements** text field, type `40`.
5. In the **Element ratio** text field, type `25`.
6. Select the **Symmetric distribution** check box.

Edge 1

Right-click **Edge 1** and choose **Build Selected**.

Edge 2

1. In the **Mesh** toolbar, click  **Edge**.
2. In the **Settings** window for **Edge**, locate the **Boundary Selection** section.
3. Click  **Paste Selection**.
4. In the **Paste Selection** dialog box, type `7, 9, 11, 13, 15, 17, 19` in the **Selection** text field.
5. Click **OK**.
6. In the **Settings** window for **Edge**, locate the **Control Entities** section.

7 In the **Number of iterations** text field, type 8.

8 In the **Maximum element depth to process** text field, type 8.

Size 1

1 Right-click **Edge 2** and choose **Size**.

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

3 From the **Predefined** list, choose **Extremely fine**.

Edge 2

In the **Model Builder** window, right-click **Edge 2** and choose **Build Selected**.

Copy Edge 1

1 In the **Mesh** toolbar, click **Copy** and choose **Copy Edge**.

2 In the **Settings** window for **Copy Edge**, locate the **Source Boundaries** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `7, 9, 11, 13, 15, 17, 19` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Copy Edge**, locate the **Destination Boundaries** section.

7 Click to select the **Activate Selection** toggle button.

8 Select Boundary 4 only.

9 Click to expand the **Control Entities** section. In the **Number of iterations** text field, type 8.

10 In the **Maximum element depth to process** text field, type 8.

11 Click **Build Selected**.

Mapped 2

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

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

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

4 Select Domain 2 only.

5 Click to expand the **Control Entities** section. In the **Number of iterations** text field, type 8.

6 In the **Maximum element depth to process** text field, type 8.

7 Click **Build Selected**.

Copy 1

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

Corner Refinement 1

In the **Model Builder** window, right-click **Corner Refinement 1** and choose **Delete**.

Free Triangular 1

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

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

3 In the list, select **2**.

4 Click — **Remove from Selection**.

5 Select Domains 1 and 3–7 only.

6 Click to expand the **Control Entities** section. In the **Number of iterations** text field, type `8`.

7 In the **Maximum element depth to process** text field, type `8`.

8 Click **Build Selected**.

Mapped 1

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

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

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

4 Locate the **Control Entities** section. In the **Number of iterations** text field, type `8`.

5 In the **Maximum element depth to process** text field, type `8`.

6 Click **Build Selected**.

Boundary Layers 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Boundary Layers 1** and choose **Delete**.

STUDY 1

Step 1: Frequency-Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Frequency-Stationary**.

2 In the **Settings** window for **Frequency-Stationary**, locate the **Study Settings** section.

3 In the **Frequency** text field, type `50`.

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
I0 (coil current magnitude)	`0.1 1 5 10 15 20 25`	A

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 1** node.

4 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** and choose **Segregated**.

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

6 In the **Settings** window for **Segregated Step**, locate the **General** section.

7 In the **Variables** list, choose **Pressure (comp1.p)** and **Velocity field (comp1.u)**.

8 Under **Variables**, click **Delete**.

9 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** right-click **Segregated 1** and choose **Segregated Step**.

10 In the **Settings** window for **Segregated Step**, locate the **General** section.

11 Under **Variables**, click + **Add**.

12 In the **Add** dialog box, in the **Variables** list, choose **Pressure (comp1.p)** and **Velocity field (comp1.u)**.

13 Click **OK**.

14 In the **Settings** window for **Segregated Step**, click to expand the **Method and Termination** section.

15 In the **Damping factor** text field, type `0.5`.

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

RESULTS

Study 1/Solution 1 (sol1)

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

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

3 From the **Frame** list, choose **Material (R, PHI, Z)**.

Magnetic Flux Density Norm (mf)

1 In the **Model Builder** window, expand the **Results>Magnetic Flux Density Norm (mf)** node, then click **Magnetic Flux Density Norm (mf)**.

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

3 From the **Frame** list, choose **Material (R, PHI, Z)**.

Streamline 1

In the **Model Builder** window, under **Results>Magnetic Flux Density Norm (mf)** right-click **Streamline 1** and choose **Delete**.

Contour 1

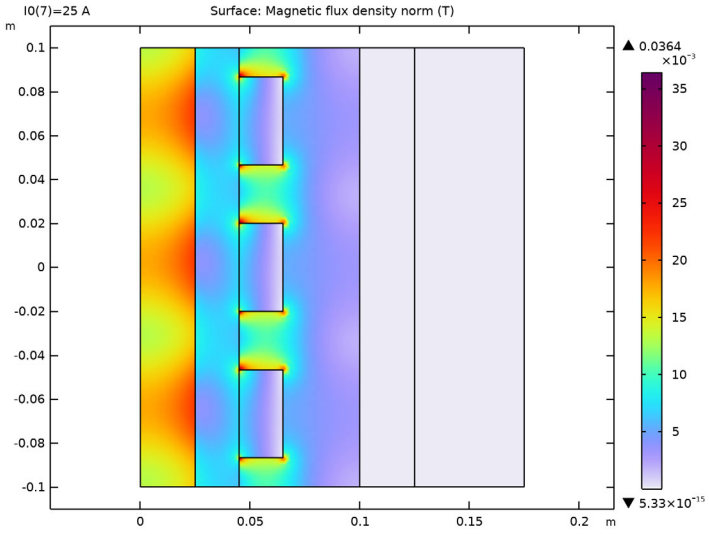
In the **Model Builder** window, right-click **Contour 1** and choose **Delete**.

Magnetic Flux Density Norm (mf)

1 In the **Model Builder** window, under **Results** click **Magnetic Flux Density Norm (mf)**.

2 In the **Magnetic Flux Density Norm (mf)** toolbar, click **Plot**.

3 Click the **Go to Default View** button in the **Graphics** toolbar.



Magnetic Flux Density Norm, Revolved Geometry (mf)

In the **Model Builder** window, under **Results** right-click **Magnetic Flux Density Norm, Revolved Geometry (mf)** and choose **Delete**.

Velocity (spf)

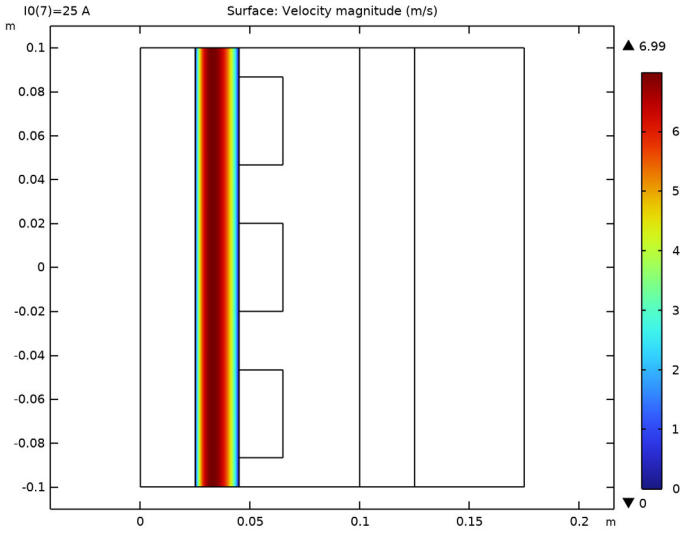
1 In the **Model Builder** window, under **Results** click **Velocity (spf)**.

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

3 Select the **Show maximum and minimum values** check box.

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

5 Click the **Go to Default View** button in the **Graphics** toolbar.



Pressure (spf)

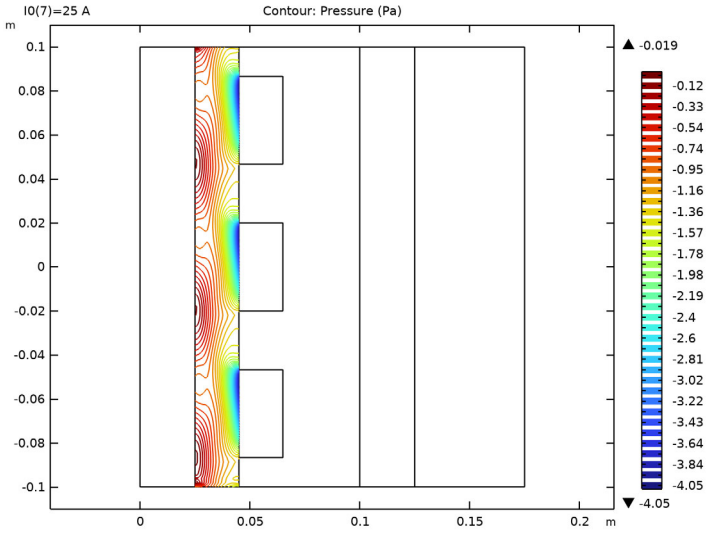
1 In the **Model Builder** window, expand the **Velocity (spf)** node, then click **Results> Pressure (spf)**.

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

3 Select the **Show maximum and minimum values** check box.

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

5 Click the **Go to Default View** button in the **Graphics** toolbar.



Velocity (spf) 1

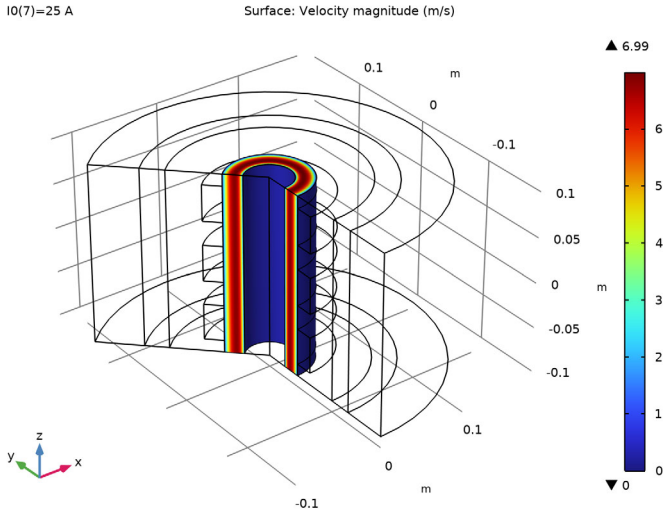
1 In the **Model Builder** window, expand the **Pressure (spf)** node, then click **Results>Velocity, 3D (spf)**.

2 In the **Settings** window for **3D Plot Group**, type `Velocity (spf) 1` in the **Label** text field.

3 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

4 In the **Velocity (spf) 1** toolbar, click **Plot**.

5 Click the **Go to Default View** button in the **Graphics** toolbar.



2D Plot Group 6

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

2 Right-click **Results>Velocity (spf) 1** and choose **2D Plot Group**.

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

4 Clear the **Plot dataset edges** check box.

5 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Contour 1

1 Right-click **2D Plot Group 6** and choose **Contour**.

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

3 In the **Expression** text field, type `r*Aphi`.

4 Locate the **Levels** section. Clear the **Round the levels** check box.

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

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

7 Clear the **Color legend** check box.

Arrow Surface 1

1 In the **Model Builder** window, right-click **2D Plot Group 6** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Laminar Flow>Velocity and pressure>u,w - Velocity field**.

3 Locate the **Arrow Positioning** section. Find the **R grid points** subsection. From the **Entry method** list, choose **Coordinates**.

4 In the **Coordinates** text field, type `range(0.0251,0.018/10,0.044)`.

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

Surface 1

1 Right-click **2D Plot Group 6** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Laminar Flow> Velocity and pressure>spf.U - Velocity magnitude - m/s**.

3 Locate the **Coloring and Style** section. Click  **Change Color Table**.

4 In the **Color Table** dialog box, select **Wave>WaveLight** in the tree.

5 Click **OK**.

Surface 2

1 Right-click **2D Plot Group 6** and choose **Surface**.

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

3 Click  **Change Color Table**.

4 In the **Color Table** dialog box, select **Rainbow>RainbowLight** in the tree.

5 Click **OK**.

Selection 1

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

2 Select Domains 1, 3, and 7 only.

Surface 3

1 In the **Model Builder** window, right-click **2D Plot Group 6** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Magnetic Fields> Currents and charge>Current density - A/m^2>mf.Jphi - Current density, phi-component**.

3 Locate the **Coloring and Style** section. Click  **Change Color Table**.

4 In the **Color Table** dialog box, select **Wave>WaveLight** in the tree.

5 Click **OK**.

Selection 1

1 Right-click **Surface 3** and choose **Selection**.

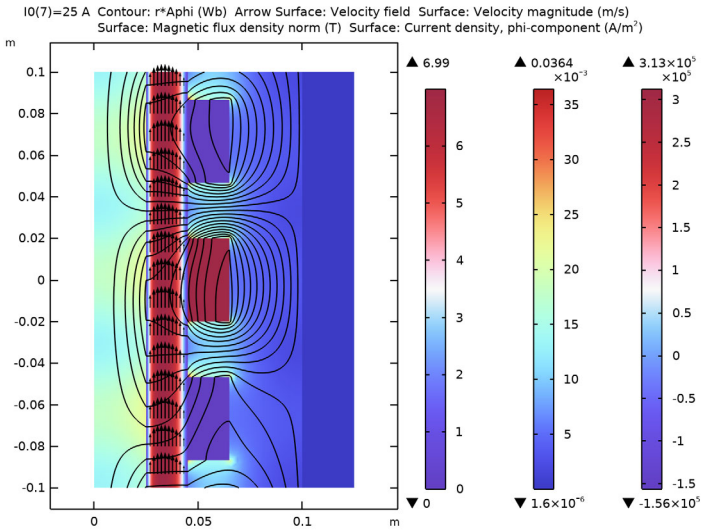
2 Select Domains 4–6 only.

2D Plot Group 6

1 In the **Model Builder** window, under **Results** click **2D Plot Group 6**.

2 In the **2D Plot Group 6** toolbar, click **Plot**.

3 Click the **Go to Default View** button in the **Graphics** toolbar.



2D Plot Group 7

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

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

3 Select the **Show maximum and minimum values** check box.

Surface 1

1 Right-click **2D Plot Group 7** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Magnetic Fields> Currents and charge>Current density - A/m²>mf.Jphi - Current density, phi-component**.

3 Locate the **Coloring and Style** section. Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Wave>WaveLight** in the tree.

5 Click **OK**.

Contour 1

1 In the **Model Builder** window, right-click **2D Plot Group 7** and choose **Contour**.

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

3 In the **Expression** text field, type `r*Aphi`.

4 Locate the **Levels** section. Clear the **Round the levels** check box.

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

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

7 Clear the **Color legend** check box.

Arrow Surface 1

1 Right-click **2D Plot Group 7** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Magnetic Fields>Mechanical>mf.FLtzavr,mf.FLtzavz - Lorentz force contribution, time average**.

3 Locate the **Arrow Positioning** section. Find the **R grid points** subsection. From the **Entry method** list, choose **Coordinates**.

4 In the **Coordinates** text field, type `range(0.0251,0.018/10,0.044)`.

5 Find the **Z grid points** subsection. In the **Points** text field, type 30.

6 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.

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

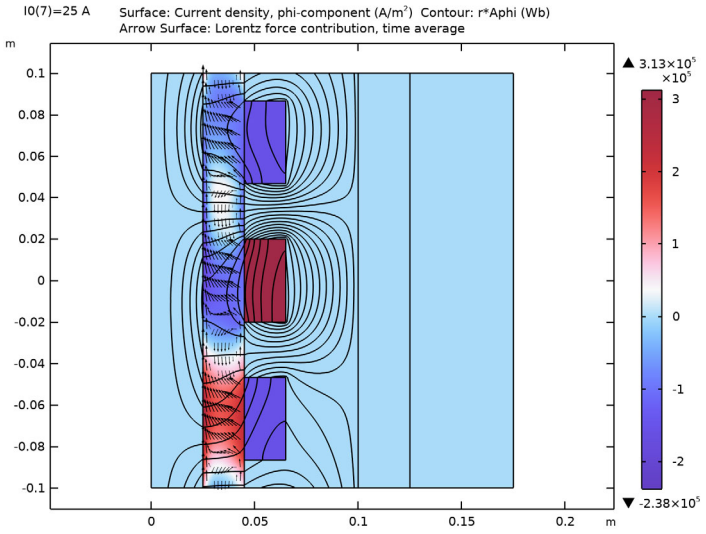
2D Plot Group 7

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

2 In the **Model Builder** window, click **2D Plot Group 7**.

3 In the **2D Plot Group 7** toolbar, click  **Plot**.

4 Click the **Go to Default View** button in the **Graphics** toolbar.



3D Plot Group 8

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 Clear the **Plot dataset edges** check box.

Volume 1

1 Right-click **3D Plot Group 8** and choose **Volume**.

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

3 In the **Expression** text field, type `mf.normB/((dom!=2)*(dom!=8)*(dom!=7))`.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Thermal>HeatCameraLight** in the tree.

6 Click **OK**.

Volume 2

1 In the **Model Builder** window, right-click **3D Plot Group 8** and choose **Volume**.

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

3 In the **Expression** text field, type `abs(spf.U)`.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Aurora>JupiterAuroraBorealis** in the tree.

6 Click **OK**.

7 In the **Settings** window for **Volume**, locate the **Coloring and Style** section.

8 From the **Color table transformation** list, choose **Reverse**.

Deformation 1

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

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

3 In the **R-component** text field, type `0`.

4 In the **PHI-component** text field, type `0`.

5 In the **Z-component** text field, type `abs(w)`.

6 Locate the **Scale** section.

7 Select the **Scale factor** check box. In the associated text field, type `0.00146045113569371*2`.

Contour 1

1 In the **Model Builder** window, right-click **3D Plot Group 8** and choose **Contour**.

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

3 In the **Expression** text field, type `Aphi*r`.

4 Locate the **Levels** section. Clear the **Round the levels** check box.

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

6 Clear the **Color legend** check box.

3D Plot Group 8

1 In the **Model Builder** window, click **3D Plot Group 8**.

2 In the **3D Plot Group 8** toolbar, click  **Plot**.

3 Click the **Go to Default View** button in the **Graphics** toolbar.