Created in COMSOL Multiphysics 6.2

Hartmann Flow in Liquid Metal Blanket with Heat Transfer

Introduction

This model demonstrates the flow of Lithium through an electrically insulated duct in a simple tokamak blanket design. In the frame of reference of the flowing material, the electromotive force induces a volumetric current, which in turn generates a Lorentz force opposing the flow velocity owing to the ambient magnetic field. The induced current closes the loop by returning in a thin layer near the electrically insulated, no slip surface.

Model Definition

The model is set up in 3D using the **Magnetic and Electric Fields** and **Laminar Flow** physics interfaces coupled via a **Magnetohydrodynamics** multiphysics node, as well as with the **Heat Transfer in Fluids** interface coupled to **Laminar Flow** via a **Nonisothermal Flow** multiphysics node.

For the electromagnetic fields, an external magnetic flux density is defined on the duct boundaries contributing with electrical insulation, and the volumetric current is induced by the electromotive force as induced by the fluid velocity. The fluid flow is given no-slip wall conditions along the duct, and a pressure-driven base flow with a volumetric force contribution given by the Lorentz force. Heat transfer is treated by the convection-diffusion equation, where the convecting velocity is given by the flow field.

Results

Figure 1 shows the velocity magnitude in the liquid metal blanket and highlights the shear flow near the no-slip surfaces of the blanket with a surface deformation plot. The direction of the externally applied magnetic field is indicated with arrows.

Figure 2 shows the induced current density and circulating current paths in a blanket cross section, displaying the main decelerating current in the center of the blanket, and the thin layers of return current near the electrically insulated no-slip surfaces.

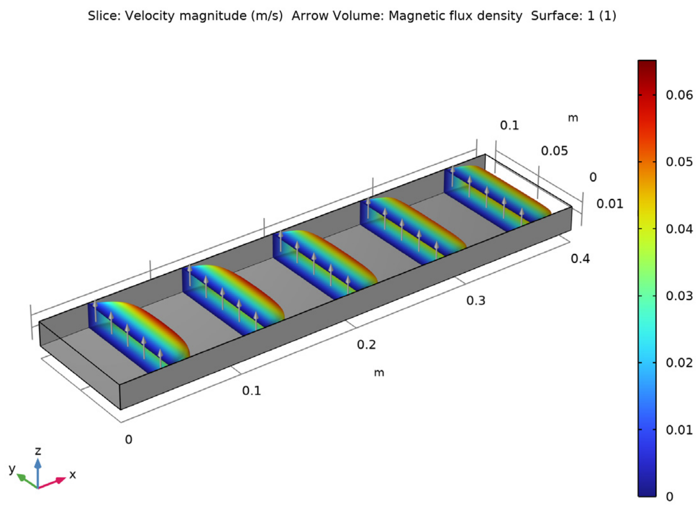


Figure 1: The velocity magnitude of the liquid metal flowing in the blanket, with arrows indicating the direction of the applied magnetic field.

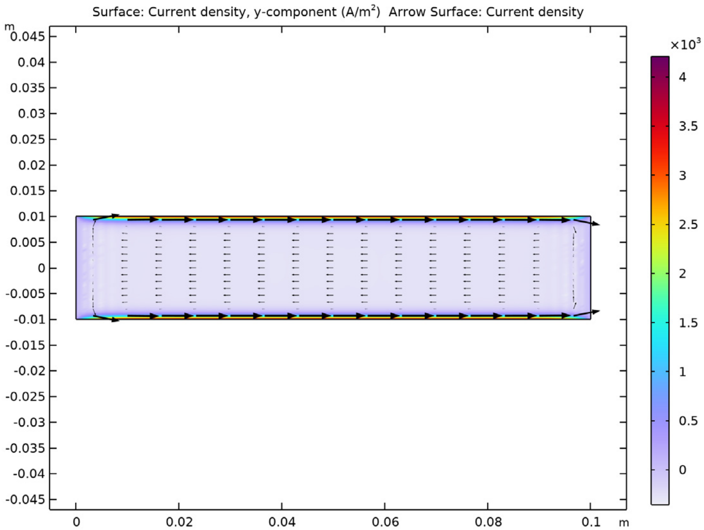


Figure 2: The current density y-component, and arrows displaying the current density direction.

Application Library path: `ACDC_Module/Electromagnetics_and_Fluids/hartmann_flow_in_liquid_metal_blanket`

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 **AC/DC>Electromagnetics and Fluids> Magnetohydrodynamics**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Fluids (ht)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **General Studies>Stationary**.

8 Click **Done**.

Add relevant physical parameters for the model.

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
Ha	20	20	Hartmann Number
d	1[cm]	0.01 m	Half-distance between planes
dens	1000[kg/m^3]	1000 kg/m³	Fluid mass density
sigma0	1e7[S/m]	1E7 S/m	Fluid electrical conductivity
visc	1e-3[Pa*s]	0.001 Pa·s	Fluid viscosity
U0	0.05[m/s]	0.05 m/s	Average inlet velocity
H0	Ha/mu0_const/d/sqrt(sigma0/visc)	15915 A/m	Imposed magnetic field
B0	mu0_const*H0	0.02 T	Magnetic flux density

GEOMETRY 1

Block 1 (blk1)

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

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

3 In the **Width** text field, type `40*d`.

4 In the **Depth** text field, type `10*d`.

5 In the **Height** text field, type `2*d`.

6 Locate the **Position** section. In the **z** text field, type `-d`.

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 (mat1)

1 In the **Settings** window for **Material**, locate the **Material Contents** section.
2 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Heat capacity at constant pressure	Cp	200	J/(kg·K)	Basic

Add a multiphysics coupling node to treat advective heat transfer correctly.

MULTIPHYSICS

Nonisothermal Flow 1 (nitf1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain>Nonisothermal Flow**.
2 In the **Settings** window for **Nonisothermal Flow**, locate the **Material Properties** section.
3 From the **Specify density** list, choose **Custom, linearized density**.
4 From the ρ_{ref} list, choose **From material**.
5 In the $\alpha_{p,0}$ text field, type `1e-3[1/K]`.

The background magnetic field is applied as a boundary condition for the tangential vector potential.

MAGNETIC AND ELECTRIC FIELDS (MEF)

Background Magnetic Flux Density 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Magnetic and Electric Fields (mef)** and choose **Background Magnetic Flux Density**.
2 In the **Settings** window for **Background Magnetic Flux Density**, locate the **Magnetic Flux Density** section.

3 Specify the $\mathbf{B}_b$ vector as

B0	Z

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **All boundaries**.

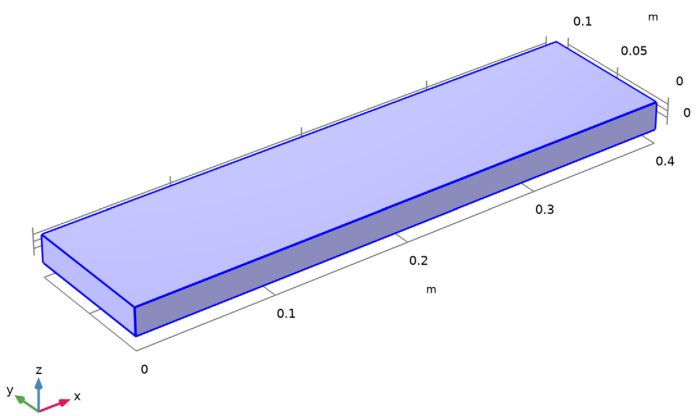
Electric Insulation 1

1 In the **Physics** toolbar, click **Attributes** and choose **Electric Insulation**.

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

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

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



LAMINAR FLOW (SPF)

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

Inlet 1

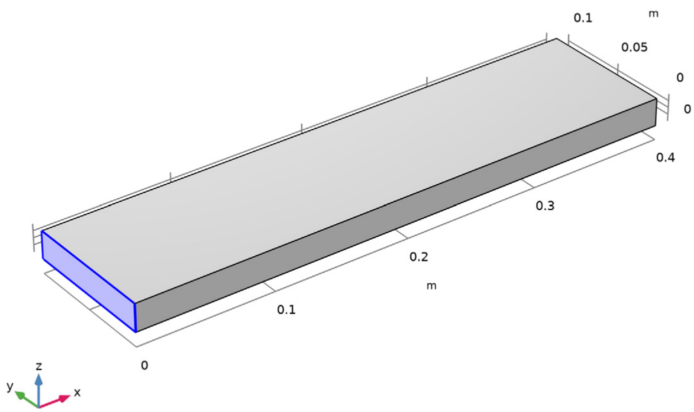
1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 1 only.

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

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

5 Locate the **Fully Developed Flow** section. In the U_{av} text field, type `U0`.



Outlet 1

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

2 Select Boundary 6 only.

HEAT TRANSFER IN FLUIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Fluids (ht)**.

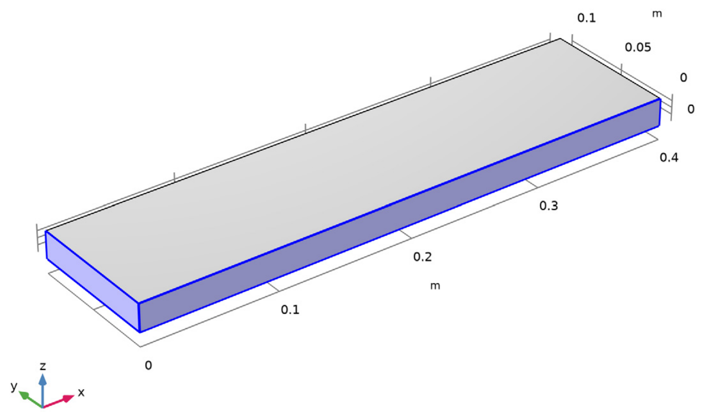
Temperature 1

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

2 Select Boundaries 1 and 2 only.

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

4 In the T_0 text field, type `450`.



Heat Source 1

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

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

3 From the **Selection** list, choose **All domains**.

4 Locate the **Heat Source** section. In the Q_0 text field, type `1e6*exp(-((y-0.1[m])/0.02[m])^2)`.

Refine the mesh near the boundary layers.

MESH 1

Mapped 1

In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.

Swept 1

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

Size

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

Mapped 1

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

2 Select Boundary 1 only.

Distribution 1

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

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

3 Click **Copy Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `1 2 4 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 `10`.

11 Select the **Symmetric distribution** check box.

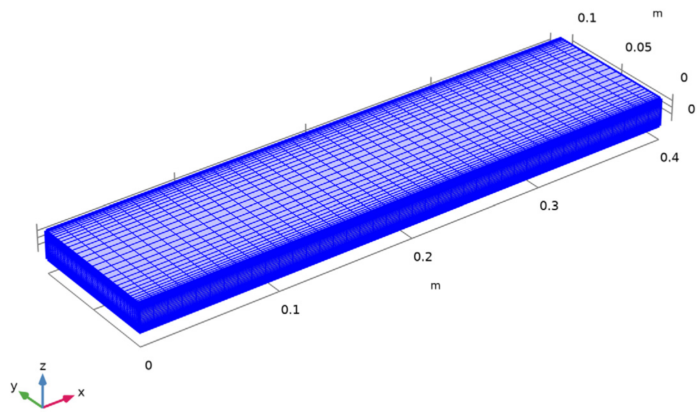
Distribution 1

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

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

3 In the **Number of elements** text field, type `40`.

4 Click **Build All**.

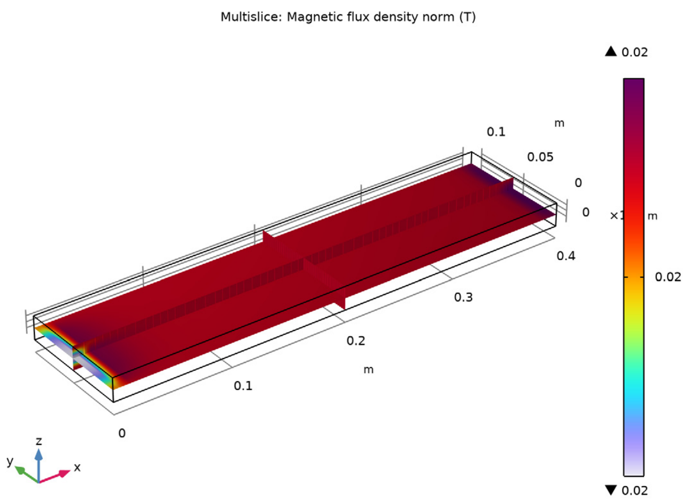


STUDY 1

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

RESULTS

Magnetic Flux Density Norm (mef)



Fluid velocity and Magnetic field

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

2 In the **Rename 3D Plot Group** dialog box, type `Fluid velocity and Magnetic field` in the **New label** text field.

3 Click **OK**.

Deformation 1

1 In the **Model Builder** window, expand the **Fluid velocity and Magnetic field** node.

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

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

4 In the **x-component** text field, type `u`.

5 In the **y-component** text field, type `v`.

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

7 Locate the **Scale** section.

8 Select the **Scale factor** check box. In the associated text field, type `0.5`.

Arrow Volume 1

1 In the **Model Builder** window, right-click **Fluid velocity and Magnetic field** and choose **Arrow Volume**.

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

3 Find the **x grid points** subsection. In the **Points** text field, type 5.

4 Find the **y grid points** subsection. In the **Points** text field, type 5.

5 Find the **z grid points** subsection. In the **Points** text field, type 1.

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

Exterior Walls

1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Exterior Walls**.

2 Select Boundaries 2, 3, and 5 only.

Surface 1

1 In the **Model Builder** window, right-click **Fluid velocity and Magnetic field** and choose **Surface**.

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

3 In the **Expression** text field, type 1.

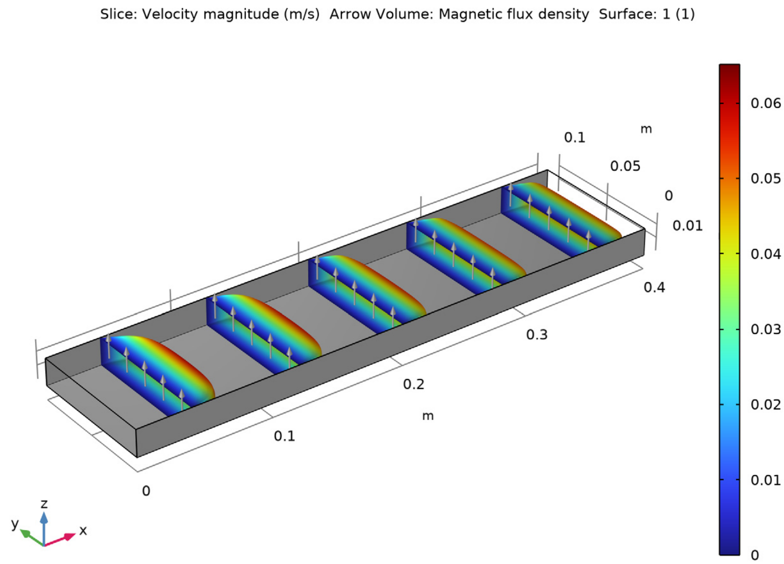
4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

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

6 Locate the **Data** section. From the **Dataset** list, choose **Exterior Walls**.

7 In the **Fluid velocity and Magnetic field** toolbar, click **Plot**.

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



Cut Plane 1

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

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

3 In the **x-coordinate** text field, type `35*d`.

Current Density and Pathways

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Current Density and Pathways` in the **Label** text field.

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

Surface 1

1 Right-click **Current Density and Pathways** and choose **Surface**.

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

3 In the **Expression** text field, type `mef.Jy`.

Arrow Surface 1

1 In the **Model Builder** window, right-click **Current Density and Pathways** and choose **Arrow Surface**.

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

3 In the **x-component** text field, type `mef.Jx`.

4 In the **y-component** text field, type `mef.Jy`.

5 In the **z-component** text field, type `mef.Jz`.

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

7 Select the **Scale factor** check box. In the associated text field, type `5.0E-6`.

8 In the **Current Density and Pathways** toolbar, click **Plot**.

Surface 1

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

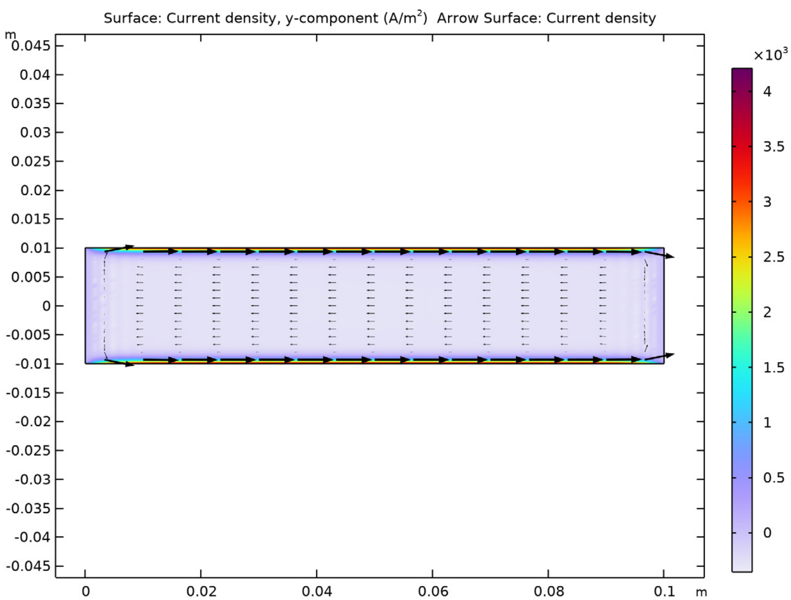
2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 Click **Change Color Table**.

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

5 Click **OK**.

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



Surface 1

1 In the **Model Builder** window, expand the **Results>Temperature (ht)** node.

2 Right-click **Temperature (ht)** and choose **Surface**.

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

4 In the **Expression** text field, type 1.

5 Locate the **Data** section. From the **Dataset** list, choose **Exterior Walls**.

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

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

Volume 1

In the **Model Builder** window, right-click **Volume 1** and choose **Disable**.

Temperature (ht)

In the **Model Builder** window, click **Temperature (ht)**.

Multislice 1

1 In the **Temperature (ht)** toolbar, click **More Plots** and choose **Multislice**.

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

3 In the **Expression** text field, type T.

4 Locate the **Multiplane Data** section. Find the **x-planes** subsection. In the **Planes** text field, type 5.

5 Find the **z-planes** subsection. In the **Planes** text field, type 0.

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

7 In the **Color Table** dialog box, select **Thermal>ThermalDark** in the tree.

8 Click **OK**.

Temperature (ht)

1 In the **Model Builder** window, click **Temperature (ht)**.

2 In the **Temperature (ht)** toolbar, click **Plot**.

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