Hartmann Boundary Layer

Introduction

The velocity profile of a Hartmann boundary layer is a classical Magnetohydrodynamics problem with an analytical solution, making it appropriate for benchmarking of numerical models. This model consists of an electrically conducting liquid flowing between two no-slip surfaces immersed in an externally applied constant magnetic field. The electromotive force, as seen in the fluid frame of reference, induces a volumetric current, which in turn generates a Lorentz force opposing the flow velocity.

Model Definition

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

ANALYTICAL SOLUTION

The flow profile between the two plates can be found if the material properties are assumed constant, and the no-slip condition is applied at both walls. The velocity is

$$U(Y) = \frac{\cosh(\mathrm{Ha}) - \cosh(\mathrm{Ha}Y)}{\cosh(\mathrm{Ha}) - \frac{1}{\mathrm{Ha}}\sinh(\mathrm{Ha})}$$

where $U = u/U_0$ is the velocity u normalized to the average inlet velocity U_0, $Y = y/d$ is the position normalized to the distance between the no-slip planes $2d$, and the Hartmann number $\mathrm{Ha} = \mu H_0 d\sqrt{\sigma/(\rho\upsilon)}$ is the ratio of electromagnetic to viscous forces where μ is the magnetic permeability, H_0 the imposed magnetic field strength, σ the electrical conductivity, ρ the mass density, and υ the kinematic viscosity.

Results

Figure 1 compares the simulation results for three different values of the Hartmann number with the corresponding analytic solutions.

Figure 2 shows the absolute error of the numeric solution for the normalized velocity for the same Hartmann number values.

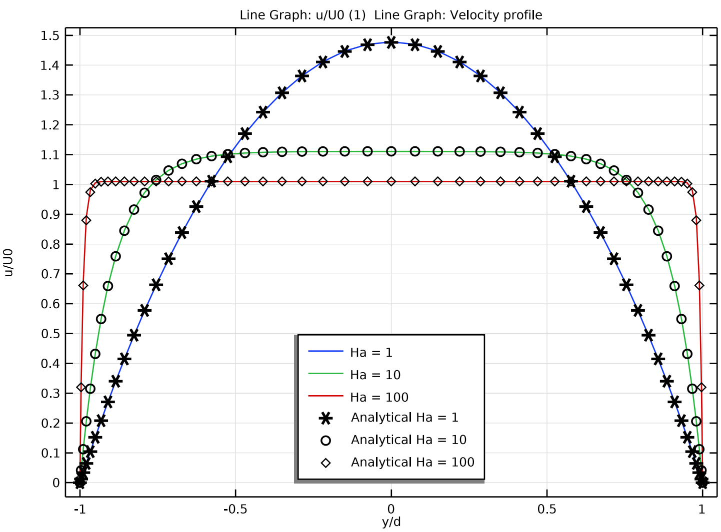


Figure 1: The normalized velocity profiles for three magnitudes of the Hartmann number plotted as functions of the y-coordinate compared with the corresponding analytical values.

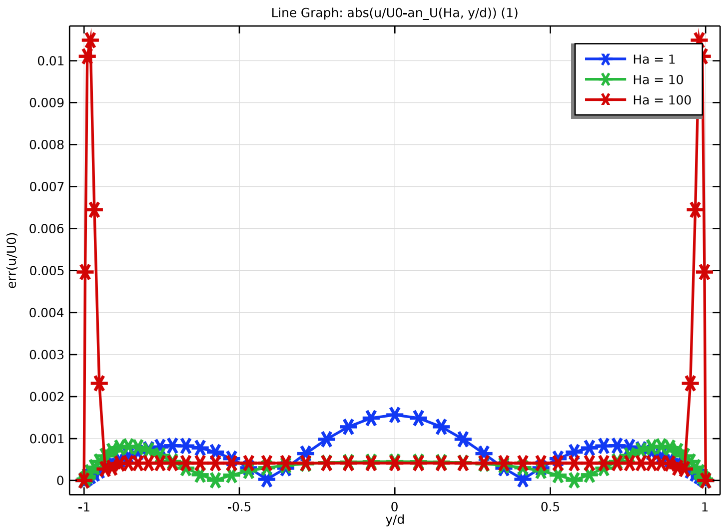


Figure 2: The absolute error of the normalized velocity plotted as a function of the y-coordinate for three magnitudes of the Hartmann number.

Application Library path: `ACDC_Module/Electromagnetics_and_Fluids/hartmann_boundary_layer`

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**.
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 **General Studies>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
Ha	10	10	Hartmann number
d	1[cm]	0.01 m	Half-distance between plates
dens	1000[kg/m^3]	1000 kg/m³	Fluid 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)	7957.7 A/m	Imposed magnetic field
B0	mu0_const*H0	0.01 T	Magnetic flux density
Re	dens*U0*2*d/visc	1000	Reynolds number
Exx	-mu0_const*H0*U0	-5E-4 V/m	Induced electric field
J0	sigma0*Exx	-5000 A/m²	Induced electric current density

Velocity profile

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global>Analytic**.

2 In the **Settings** window for **Analytic**, type `Velocity profile` in the **Label** text field.

3 In the **Function name** text field, type `an_U`.

4 Locate the **Definition** section. In the **Expression** text field, type `(cosh(Ha)-cosh(Ha*Y))/(cosh(Ha)-1/Ha*sinh(Ha))`.

5 In the **Arguments** text field, type `Ha, Y`.

6 Locate the **Plot Parameters** section. In the table, enter the following settings:

Argument	Lower limit	Upper limit	Unit
Ha	20	20	
Y	-1	1	

GEOMETRY 1

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 `80*d`.

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

5 Locate the **Position** section. In the **y** text field, type `-d`.

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

MATERIALS

Fluid

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

3 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	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	`sigma0`	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic
Density	rho	`dens`	kg/m³	Basic
Dynamic viscosity	mu	`visc`	Pa·s	Basic

LAMINAR FLOW (SPF)

Inlet 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Laminar Flow (spf)** 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 Clear the **Apply condition on each disjoint selection separately** check box.

6 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 4 only.

MAGNETIC FIELDS (MF)

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

2 In the **Settings** window for **Magnetic Fields**, locate the **Background Field** section.

3 From the **Solve for** list, choose **Reduced field**.

4 From the **Background field specification** list, choose **Uniform magnetic flux density**.

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

0	x
B0	y
0	z

External Magnetic Vector Potential 1

1 In the **Physics** toolbar, click **Boundaries** and choose **External Magnetic Vector Potential**.

2 Click in the **Graphics** window and then press Ctrl+A to select all boundaries.

MESH 1

Mapped 1

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

Distribution 1

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

Size 1

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

Distribution 1

1. Select Boundaries 1 and 4 only.
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 50.
5. In the **Element ratio** text field, type 20.
6. Select the **Symmetric distribution** check box.

Size 1

1. In the **Model Builder** window, click **Size 1**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. Click the **Custom** button.
4. Locate the **Element Size Parameters** section.
5. Select the **Maximum element size** check box. In the associated text field, type d.
6. Click **Build Selected**.

STUDY 1

Step 1: Stationary

1. In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.
2. In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.
3. Select the **Auxiliary sweep** check box.
4. Click **Add**.
5. In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Ha (Hartmann number)	1 10 100	

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

RESULTS

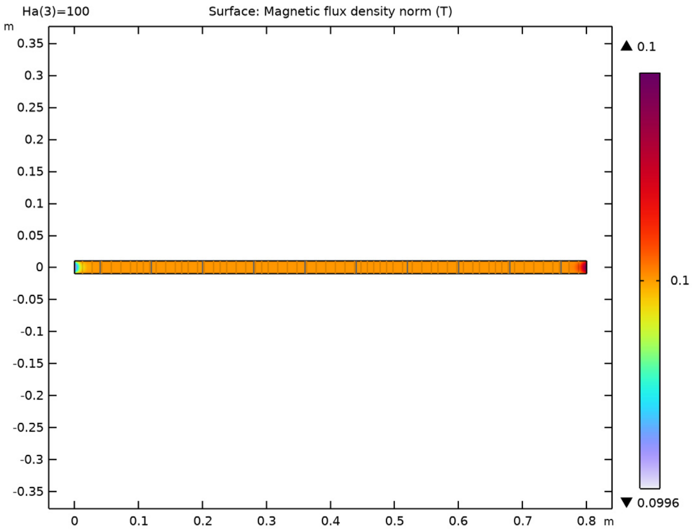
Streamline 1

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

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

3 In the **Separating distance** text field, type `0.015`.

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



Velocity (spf)

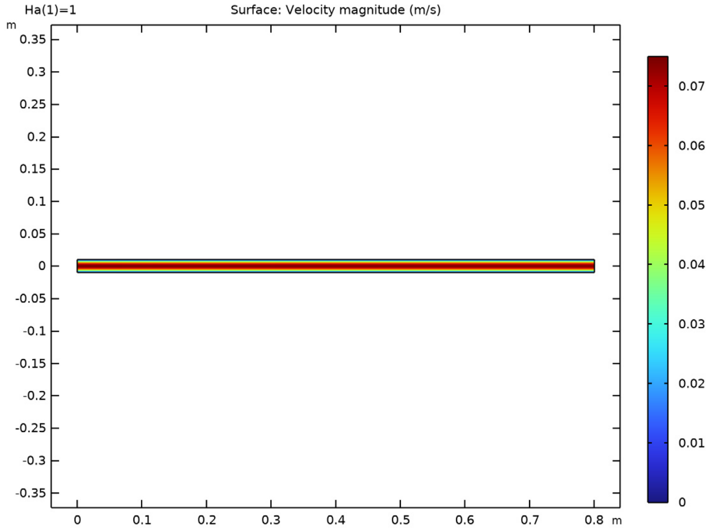
1 In the **Model Builder** window, under **Results** click **Velocity (spf)**.

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

3 From the **Parameter value (Ha)** list, choose **1**.

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

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



Contour

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

2 Right-click **Contour** and choose **Disable**.

Surface 1

1 In the **Model Builder** window, right-click **Pressure (spf)** 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>p - Pressure - Pa**.

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

Pressure (spf)

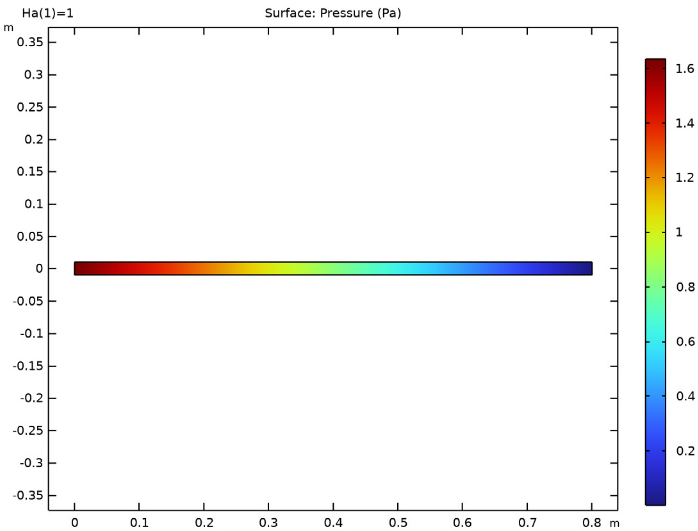
1 In the **Model Builder** window, click **Pressure (spf)**.

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

3 From the **Parameter value (Ha)** list, choose **1**.

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

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



Longitudinal section

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

2 Right-click **Results>Datasets** and choose **Cut Line 2D**.

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

4 Locate the **Line Data** section. In row **Point 2**, set **x** to `80*d`.

Cross section

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

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

3 In row **Point 1**, set **x** to `70*d`.

4 In row **Point 1**, set **y** to `-d`.

5 In row **Point 2**, set **x** to `70*d`.

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

7 In the **Label** text field, type `Cross section`.

Normalized velocity on the center line

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

2 In the **Settings** window for **1D Plot Group**, type `Normalized velocity on the center line` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Longitudinal section**.

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** check box. In the associated text field, type `x/d`.

6 Select the **y-axis label** check box. In the associated text field, type `u/U0`.

Line Graph 1

1 Right-click **Normalized velocity on the center line** and choose **Line Graph**.

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

3 From the **Dataset** list, choose **Longitudinal section**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `u/U0`.

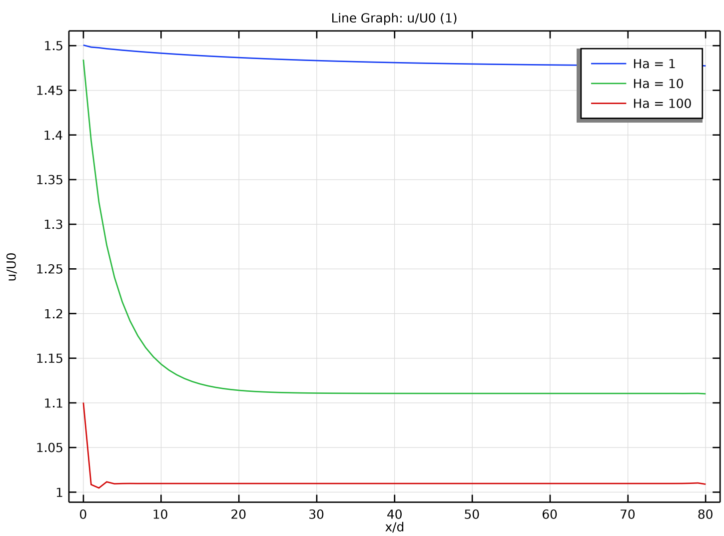
5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `x/d`.

7 Click to expand the **Legends** section. Select the **Show legends** check box.

8 Find the **Prefix and suffix** subsection. In the **Prefix** text field, type `Ha = `.

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



Normalized velocity profiles

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

2 In the **Settings** window for **1D Plot Group**, type `Normalized velocity profiles` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cross section**.

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

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

6 In the **x-axis label** text field, type `y/d`.

7 In the **y-axis label** text field, type `u/U0`.

8 Locate the **Legend** section. From the **Position** list, choose **Lower middle**.

Line Graph 1

1 Right-click **Normalized velocity profiles** 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 `u/U0`.

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

5 In the **Expression** text field, type `y/d`.

6 Locate the **Legends** section. Select the **Show legends** check box.

7 Find the **Prefix and suffix** subsection. In the **Prefix** text field, type `Ha = `.

Line Graph 2

1 In the **Model Builder** window, right-click **Normalized velocity profiles** 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 `an_U(Ha,y/d)`.

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

5 In the **Expression** text field, type `y/d`.

6 Locate the **Legends** section. Select the **Show legends** check box.

7 Find the **Prefix and suffix** subsection. In the **Prefix** text field, type `Analytical Ha = `.

8 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

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

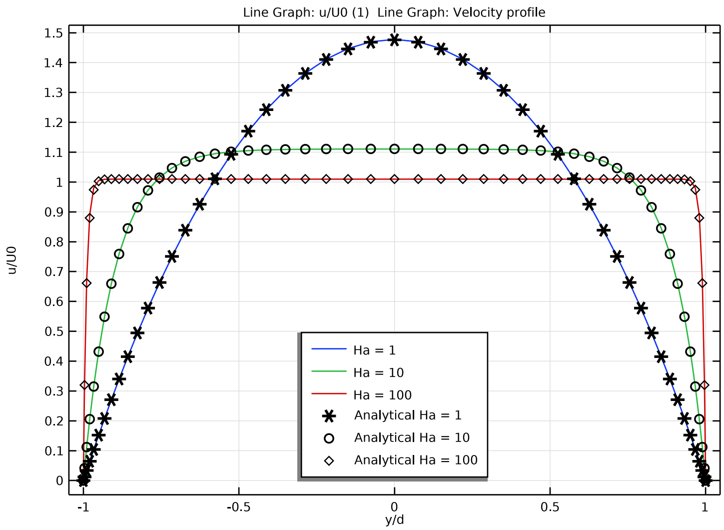
10 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

11 Click to expand the **Quality** section. From the **Resolution** list, choose **No refinement**.

12 From the **Smoothing** list, choose **None**.

13 In the **Normalized velocity profiles** toolbar, click **Plot**.

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



Normalized velocity error

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

2 In the **Settings** window for **1D Plot Group**, type `Normalized velocity error` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cross section**.

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** check box. In the associated text field, type `y/d`.

6 Select the **y-axis label** check box. In the associated text field, type `err(u/U0)`.

Line Graph 1

1 Right-click **Normalized velocity error** 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 `abs(u/U0-an_U(Ha, y/d))`.

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

5 In the **Expression** text field, type `y/d`.

6 Locate the **Coloring and Style** section. From the **Width** list, choose **2**.

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

8 Locate the **Legends** section. Select the **Show legends** check box.

9 Find the **Prefix and suffix** subsection. In the **Prefix** text field, type `Ha = `.

10 Locate the **Quality** section. From the **Resolution** list, choose **No refinement**.

11 From the **Smoothing** list, choose **None**.

12 In the **Normalized velocity error** toolbar, click **Plot**.

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