



Model created in COMSOL Multiphysics 6.4

# 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 the 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

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

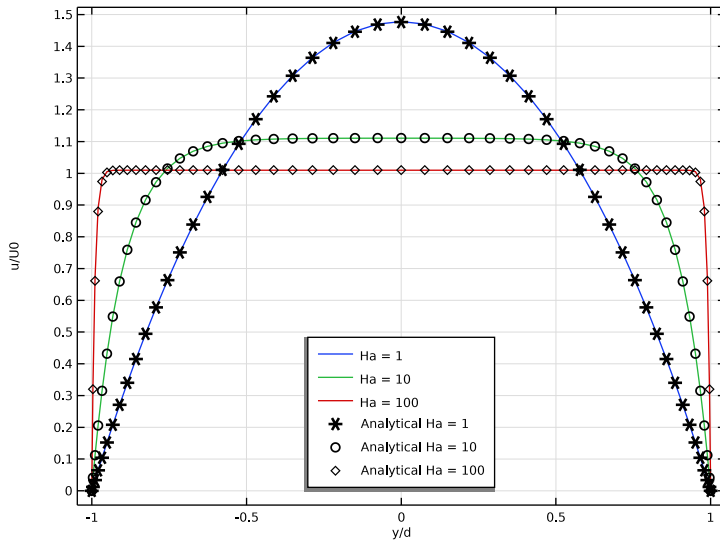
where  $U = u/U_0$  is the velocity  $u$  normalized to the average inlet velocity  $U_0$ ,  $Y = y/d$  is the position normalized to the distance between the no-slip planes  $2d$ , and the Hartmann number  $\text{Ha} = \mu H_0 d \sqrt{\sigma / (\rho \nu)}$  is the ratio of electromagnetic to viscous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

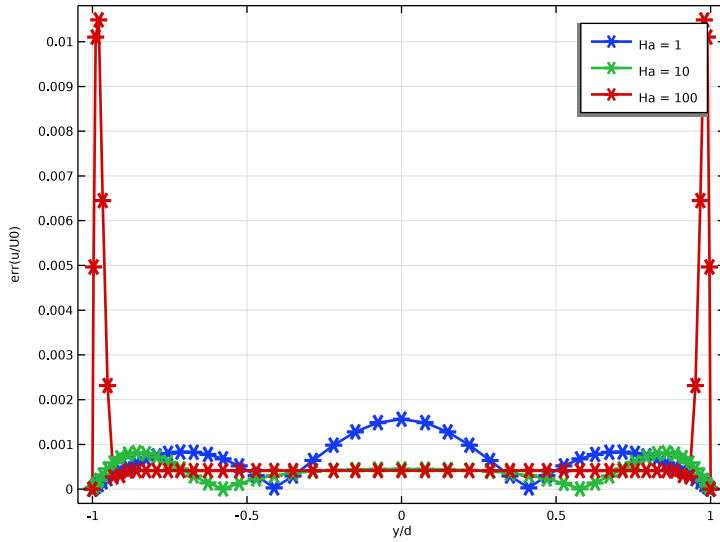


Figure 2: The absolute error of the normalized velocity plotted as a function of the  $y$ -coordinate for three magnitudes of the Hartmann number.

---

**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 <sup>3</sup> ]	1000 kg/m <sup>3</sup>	Fluid density
sigma0	1e7 [S/m]	1E7 S/m	Fluid electric 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	$\text{Ha}/\mu_0\text{-const}/d/\sqrt{\text{sigma}0/\text{visc}}$	7957.7 A/m	Imposed magnetic field
B0	$\mu_0\text{-const}*H0$	0.01 T	Magnetic flux density
Re	$\text{dens}*U0^2*d/\text{visc}$	1000	Reynolds number
Exx	$-\mu_0\text{-const}*H0*U0$	-5E-4 V/m	Induced electric field
J0	$\text{sigma}0*\text{Exx}$	-5000 A/m <sup>2</sup>	Induced electric current density

### Velocity profile

1 In the **Home** toolbar, click  **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  $(\cosh(\text{Ha}) - \cosh(\text{Ha} * \text{Y})) / (\cosh(\text{Ha}) - 1/\text{Ha} * \sinh(\text{Ha}))$ .



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

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

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	Ha	20	20	0	
√	Y	-1	1	0	

## GEOMETRY I

### 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	l	Basic
Electric conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	sigma0	S/m	Basic
Relative permittivity	epsilonIso ; epsilonrii = epsilonIso, epsilonrij = 0	1	l	Basic
Density	rho	dens	kg/m <sup>3</sup>	Basic
Dynamic viscosity	mu	visc	Pa·s	Basic

## 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 Clear the **Apply condition on each disjoint selection separately** checkbox.
- 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** checkbox.

### 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** checkbox. 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** checkbox.
- 4 Click **+ Add**.
- 5 In the table, enter the following settings:


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

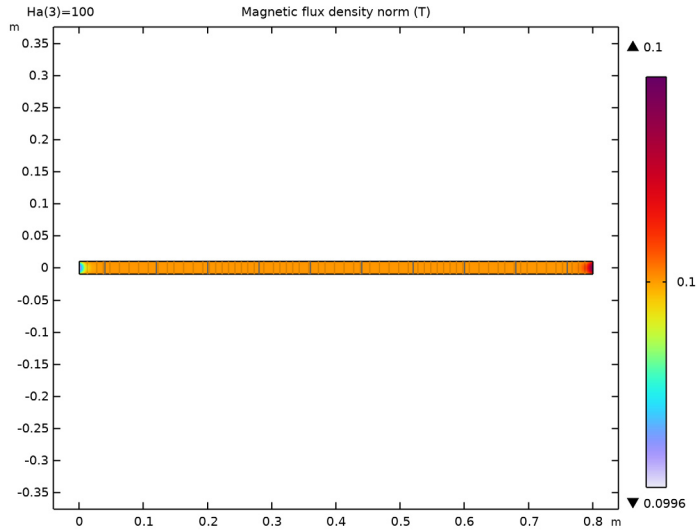
- 6 In the **Study** toolbar, click  **Compute**.

## RESULTS


### Streamline 1


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

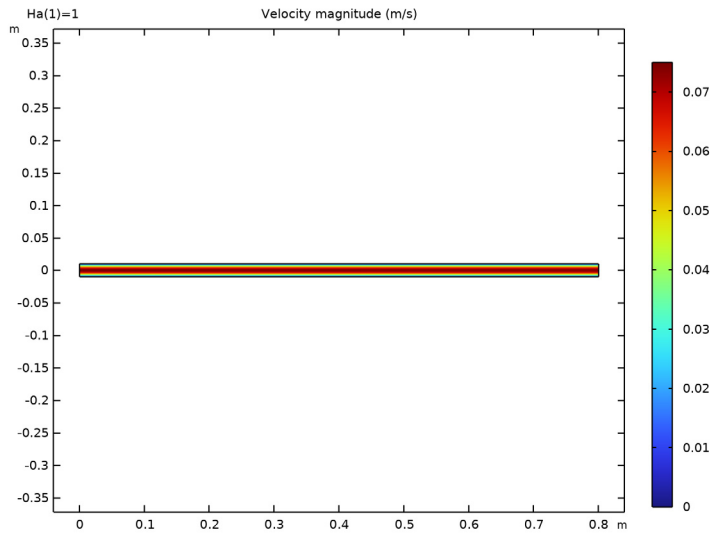
- 2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.
- 3 In the **Density level** text field, type 8.5.
- 4 In the **Magnetic Flux Density (mf)** toolbar, click  **Plot**.




*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**.

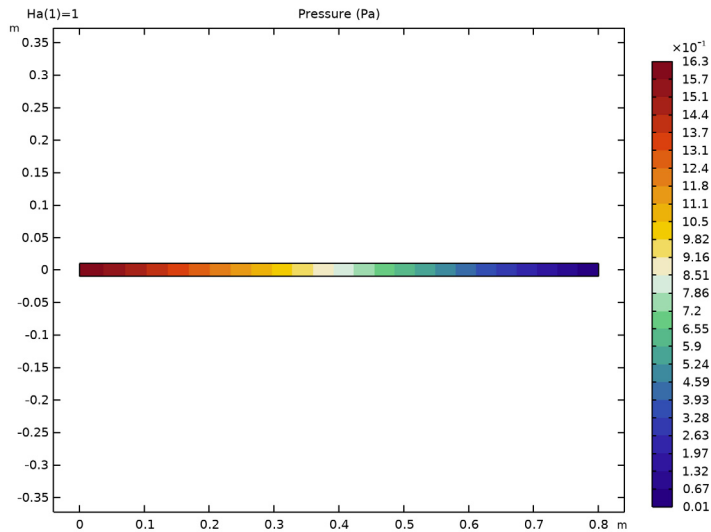
5 Click the  **Zoom Extents** button in the **Graphics** toolbar.



*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 Extends** 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**.

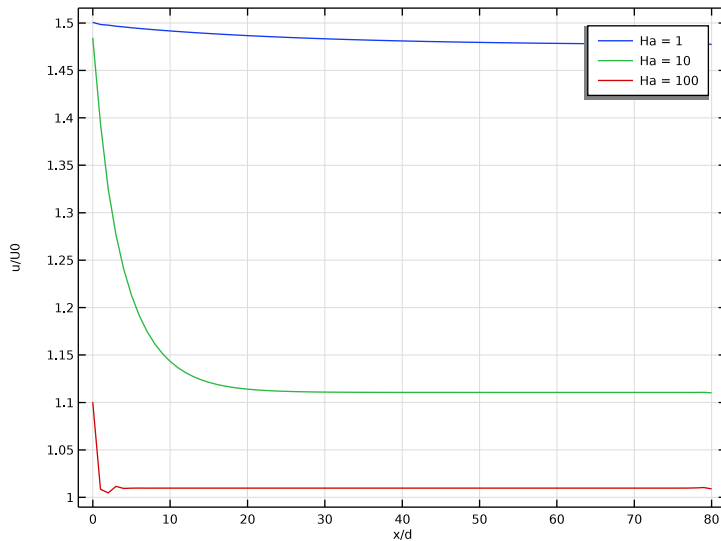
#### *Normalized velocity on the center line*

- 1 In the **Results** toolbar, click  **ID Plot Group**.


- 2 In the **Settings** window for **ID 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** checkbox. In the associated text field, type  $x/d$ .
- 6 Select the **y-axis label** checkbox. In the associated text field, type  $u/U_0$ .

*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  $u/U_0$ .
- 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** checkbox.
- 8 Find the **Prefix and suffix** subsection. In the **Prefix** text field, type  $Ha =$  .
- 9 Click to expand the **Title** section. From the **Title type** list, choose **None**.
- 10 Click the  **Zoom Extents** button in the **Graphics** toolbar.



### *Normalized velocity profiles*



- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID 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** checkbox.
- 5 Select the **y-axis label** checkbox.
- 6 In the **x-axis label** text field, type  $y/d$ .
- 7 In the **y-axis label** text field, type  $u/U_0$ .
- 8 Locate the **Legend** section. From the **Position** list, choose **Lower middle**.

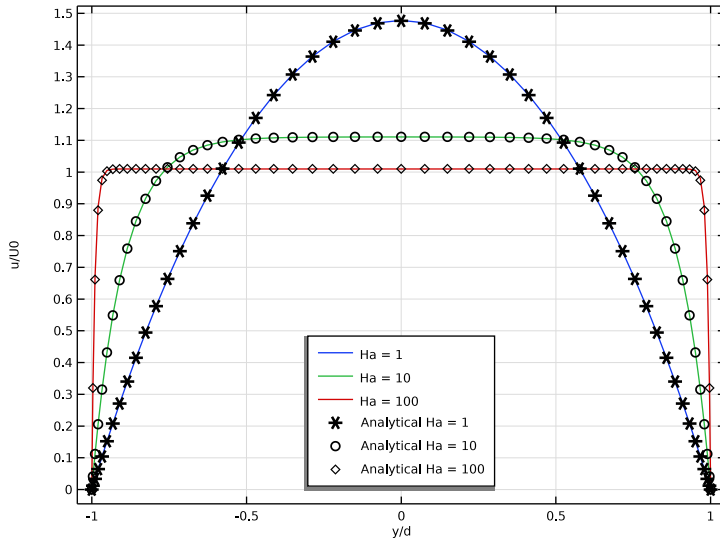
### *Line Graph 1*

- 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  $u/U_0$ .
- 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** checkbox.
- 7 Find the **Prefix and suffix** subsection. In the **Prefix** text field, type  $Ha = .$
- 8 Locate the **Title** section. From the **Title type** list, choose **None**.


### *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** checkbox.
- 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 **Evaluation settings** list, choose **Manual**.
- 12 From the **Resolution** list, choose **No refinement**.
- 13 From the **Smoothing** list, choose **None**.
- 14 Locate the **Title** section. From the **Title type** list, choose **None**.
- 15 In the **Normalized velocity profiles** toolbar, click  **Plot**.
- 16 Click the  **Zoom Extents** button in the **Graphics** toolbar.





### Normalized velocity error

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID 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** checkbox. In the associated text field, type  $y/d$ .
- 6 Select the **y-axis label** checkbox. In the associated text field, type  $\text{err}(u/U0)$ .

### 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  $\text{abs}(u/U0 - \text{an}_U(\text{Ha}, y/d))$ .

- 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** checkbox.
- 9 Find the **Prefix and suffix** subsection. In the **Prefix** text field, type  $Ha =$ .
- 10 Locate the **Quality** section. From the **Evaluation settings** list, choose **Manual**.
- 11 From the **Resolution** list, choose **No refinement**.
- 12 From the **Smoothing** list, choose **None**.
- 13 Locate the **Title** section. From the **Title type** list, choose **None**.
- 14 In the **Normalized velocity error** toolbar, click  **Plot**.
- 15 Click the  **Zoom Extents** button in the **Graphics** toolbar.

