Model created in COMSOL Multiphysics 6.4

The Blasius Boundary Layer

Introduction

The incompressible boundary layer on a flat plate in the absence of a pressure gradient is usually referred to as the Blasius boundary layer (Ref. 1). The steady, laminar boundary layer developing downstream of the leading edge eventually becomes unstable to Tollmien–Schlichting waves and finally transitions to a fully turbulent boundary layer. Due to its fundamental importance, this type of flow has become the subject of numerous studies on boundary-layer flow, stability, transition, and turbulence. This application considers the first section of the plate where the boundary layer remains steady and laminar and compares results from incompressible, two-dimensional, single-phase-flow simulations obtained in COMSOL Multiphysics to the Blasius similarity solution. The solutions converge ideally with respect to both mesh refinement and discretization order.

Model Definition

Consider a homogeneous free-stream flow with speed U_0 parallel to an infinitely thin, flat plate located along the positive x-axis. The flow is assumed to be steady, symmetric with respect to y, and homogeneous in the z direction. Due to friction, the flow adjacent to the plate is retarded and a thin boundary layer, where the velocity gradually grows from zero to the free-stream value, develops downstream of the leading edge (see Figure 1).

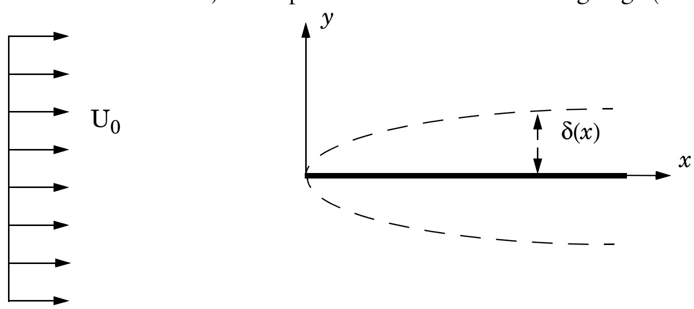

Figure 1: The boundary layer on a flat plate. $\delta(x)$ *is the boundary-layer thickness, such that* $u(x,\ \delta(x)) = U_0$.

A reasonably accurate solution for the flow field can be found by considering the boundary-layer approximation to the steady, incompressible Navier–Stokes equations

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = \upsilon\frac{\partial^2 u}{\partial y^2} \tag{1}$$

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{2}$$

Introducing a stream function,

$$u = \frac{\partial \Psi}{\partial y}, \quad v = -\frac{\partial \Psi}{\partial x}$$

and the similarity transformation,

$$\psi = \sqrt{\upsilon x U_0} f(\eta), \quad \eta = \frac{y}{\sqrt{\upsilon x / U_0}}$$

Equation 1 and Equation 2 reduce to the ODE

$$2f''' + ff'' = 0 \tag{3}$$

COMSOL solves Equation 3 on the interval $\eta \in [0, 10]$ with the boundary conditions

$$f(0) = 0, \quad f'(0) = 0$$
$$\lim_{\eta \to \infty} f'(\eta) = 1$$

by rewriting the equation as a system of two equations,

$$\begin{cases} f' = f_{\text{prime}} \\ f''_{\text{prime}} = -\frac{1}{2} f f'_{\text{prime}} \end{cases}$$

and implementing the system within the Coefficient Form PDE interface.

Using the Laminar Flow interface for single-phase flow, the model solves the steady, incompressible Navier-Stokes equation in a domain $(x, y) \in ([-1, 2.1], [0, 0.5])$ m with the leading edge of the plate located at $x = 0$ m. The working fluid is air at a temperature of $T = 20°\text{C}$ and $U_0 = 0.75$ m/s. The simulations uses discretizations with linear basis functions for velocity and pressure (P1+P1) on three different meshes.

Results and Discussion

Figure 2 shows the similarity solution $u/U_0 = f'(\eta)$. At $\eta = 4.99$, the deviation from the free-stream value is 1%. This value can be used to define the boundary-layer thickness,

$$\delta_{99}(x) = 4.99\sqrt{\frac{\upsilon x}{U_0}}$$

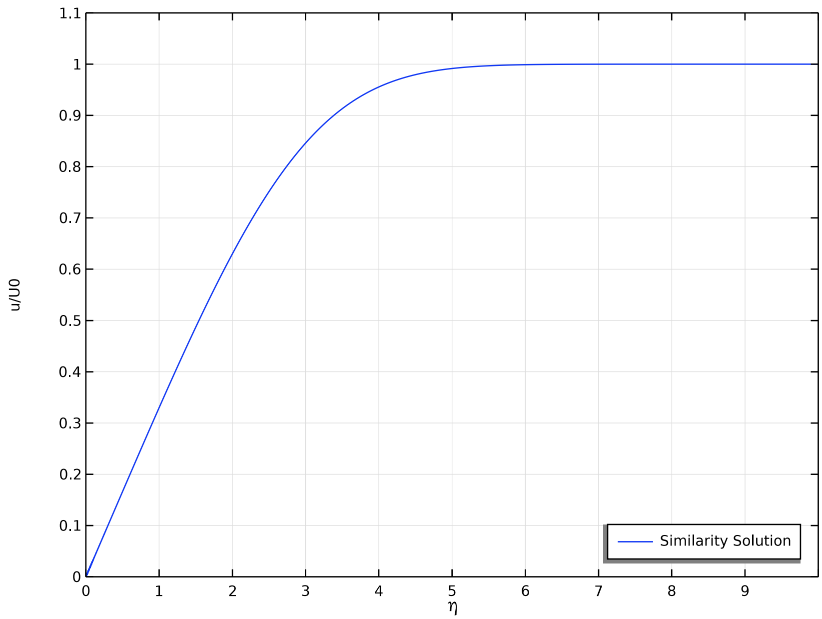

Figure 2: Similarity solution for the streamwise velocity component.

Figure 3 shows a comparison between the Blasius similarity solution and the results from the two-dimensional simulations at $x_E = 2$ m, corresponding to a Reynolds number of $\mathrm{Re}_x = 1.0 \cdot 10^5$. Only the results from the P1+P1 simulation on the coarse mesh show a significant deviation from the similarity solution. To quantify differences in the results, define the following measure,

$$\varepsilon = \sqrt{\int_0^{\eta_\infty} \left(\frac{u}{U_0} - f'\right)^2 d\eta}$$

Here, $\eta_\infty = 10$, for which the similarity solution has converged to its asymptotic value to within the numerical precision in the computations.

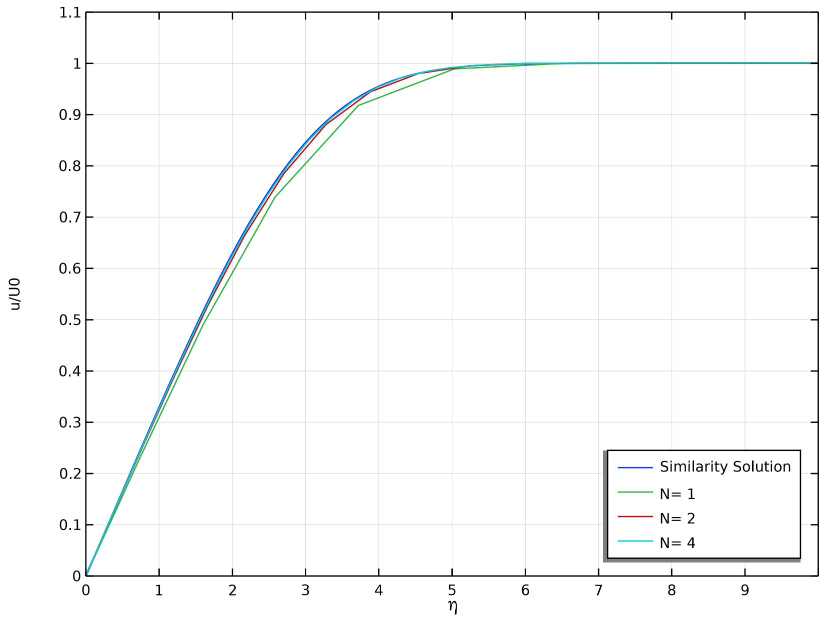

Figure 3: Comparison between the similarity solution and the two-dimensional simulations.

Table 1 displays deviations from the similarity solution together with the number of degrees of freedom (DOF) for the three simulations. The convergence is displayed in Figure 4 where the mesh size h is calculated as the maximum cell side in the mesh. The curve is close to straight line, which means that the model is in a mesh convergence regime; that is, the solution converges toward the correct solution when the mesh is refined.

TABLE 1: DEVIATION FROM THE BLASIUS SOLUTION.

1/H	**10**	**20**	**40**
ε	$6.10 \cdot 10^{-2}$	$3.33 \cdot 10^{-2}$	$1.68 \cdot 10^{-2}$
DOF	2016	7749	30375

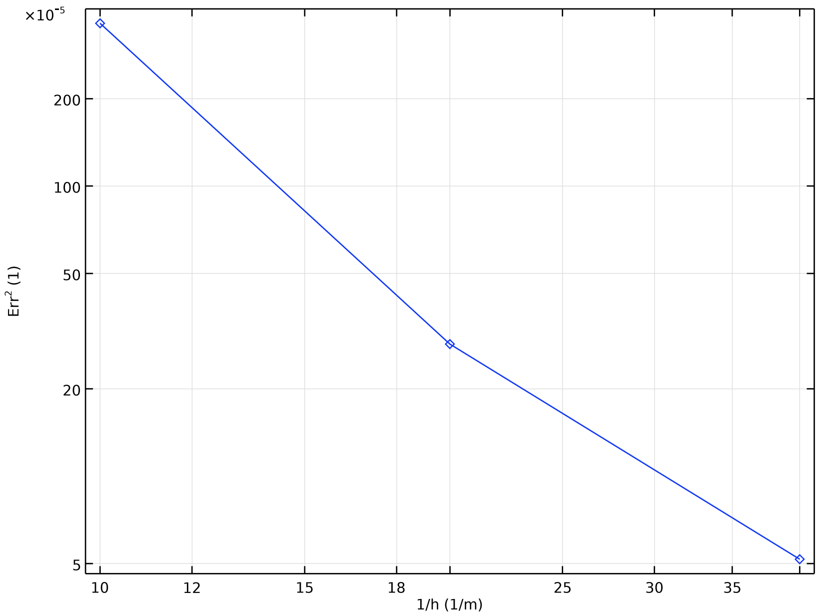

Figure 4: Convergence rate as a function of inverse maximum cell side.

Notes About the COMSOL Implementation

The relative tolerance is set to 10^{-5} in the study 2 solver to ensure that the equation system becomes well converged. All meshes have monotonically increasing element sizes away from the plate, with distributions employing geometric sequences. A nonlocal coupling is set up to enable evaluation of the similarity solution in the two-dimensional model.

Reference

1. H. Blasius, "Grenzschichten in Flüssigkeiten mit kleiner Reibung," *Z. Math. Phys.*, vol. 56, pp. 1–37, 1908 (English translation in *NACA TM* 1256).

Application Library path: `COMSOL_Multiphysics/Fluid_Dynamics/blasius_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 **1D**.

2 In the **Select Physics** tree, select **Mathematics** > **PDE Interfaces** > **Coefficient Form PDE (c)**.

3 Click **Add**.

4 In the **Field name (1)** text field, type `f`.

5 Click **Add Dependent Variable**.

6 In the **Dependent variables (1)** table, enter the following settings:

f
fprime

7 Click **Study**.

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

9 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
U0	0.75[m/s]	0.75 m/s	Inlet velocity
nu	1.506137e-5[m^2/s]	1.5061E-5 m²/s	Kinematic viscosity
xE	2[m]	2 m	Evaluation location
b0	nu/U0	2.0082E-5 m	B-L scale
N	1	1	Mesh refinement factor

GEOMETRY 1

Interval 1 (i1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Geometry 1** and choose **Interval**.

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

3 In the table, enter the following settings:

Coordinates (m)
0
10

COEFFICIENT FORM PDE (C)

Coefficient Form PDE 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Coefficient Form PDE (c)** click **Coefficient Form PDE 1**.

2 In the **Settings** window for **Coefficient Form PDE**, locate the **Diffusion Coefficient** section.

3 In the c text-field array, type `0` in the first column of the first row.

4 In the c text-field array, type `-2` in the second column of the second row.

5 Locate the **Absorption Coefficient** section. In the a text-field array, type `1` in the second column of the first row.

6 Locate the **Source Term** section. In the f text-field array, type `0` on the first row.

7 In the f text-field array, type `0` on the second row.

8 Locate the **Damping or Mass Coefficient** section. In the d_a text-field array, type `0` in the first column of the first row.

9 In the d_a text-field array, type `0` in the second column of the second row.

10 Click to expand the **Convection Coefficient** section. In the β text-field array, type `-1` in the first column of the first row.

11 In the β text-field array, type `f` in the second column of the second row.

Dirichlet Boundary Condition 1

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

2 Select Boundary 1 only.

Dirichlet Boundary Condition 2

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

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

3 Select Boundary 2 only.

4 In the **Settings** window for **Dirichlet Boundary Condition**, locate the **Dirichlet Boundary Condition** section.

5 Clear the **Prescribed value of f** checkbox.

6 In the r_2 text field, type `1`.

MESH 1

Edge 1

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

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

3 From the **Geometric entity level** list, choose **Entire geometry**.

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 `10000`.

5 In the **Element ratio** text field, type `100`.

6 From the **Growth rate** list, choose **Exponential**.

7 Click **Build Selected**.

STUDY 1

Solution 1 (sol1)

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

Step 1: Stationary

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

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

3 From the **Tolerance** list, choose **User controlled**.

4 In the **Relative tolerance** text field, type `1e-6`.

Solution 1 (sol1)

Allocate more memory than the default suggestion to avoid a warning message. The solver will automatically increase the allocation factor when needed, but changing it manually is more computationally efficient.

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

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

3 In the **Settings** window for **Direct**, locate the **General** section.

4 In the **Memory allocation factor** text field, type `1.5`.

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

6 In the **Model Builder** window, right-click **Study 1** and choose **Rename**.

7 In the **Rename Study** dialog, type `Similarity Solution` in the **New label** text field.

8 Click **OK**.

RESULTS

Line Graph 1

1 In the **Model Builder** window, expand the **Coefficient Form PDE** node, then click **Line Graph 1**.

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

3 In the **Expression** text field, type `fprime`.

4 Click to expand the **Legends** section. Select the **Show legends** checkbox.

5 From the **Legends** list, choose **Manual**.

6 In the table, enter the following settings:

Legends
`Similarity Solution`

Coefficient Form PDE

1 In the **Model Builder** window, click **Coefficient Form PDE**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

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

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** checkbox. In the associated text field, type `\eta`.

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

7 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

8 In the **x minimum** text field, type `0`.

9 In the **x maximum** text field, type `10`.

10 In the **y minimum** text field, type `0`.

11 In the **y maximum** text field, type `1.1`.

12 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

13 In the **Coefficient Form PDE** toolbar, click **Plot**.

DEFINITIONS

Set up a nonlocal coupling to be able to evaluate the similarity solution in the upcoming 2D model.

General Extrusion 1 (genext1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** node.

2 Right-click **Definitions** and choose **Nonlocal Couplings** > **General Extrusion**.

3 Select Domain 1 only.

4 In the **Settings** window for **General Extrusion**, locate the **Destination Map** section.

5 In the **x-expression** text field, type `root.y/sqrt(b0*root.x)`.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component** > **2D**.

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** > **Laminar Flow (spf)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Similarity Solution**.

5 Click the **Add to Component 2** button in the window toolbar.

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

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 **General Studies** > **Stationary**.

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

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

GEOMETRY 2

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 `3.1`.

4 In the **Height** text field, type `0.5`.

5 Locate the **Position** section. In the **x** text field, type `-1`.

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

Point 1 (pt1)

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

2 Click **Build All**.

Point 2 (pt2)

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

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

3 In the **y** text field, type `0.5`.

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

ADD MATERIAL

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

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

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

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

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

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 **Velocity** section.

4 In the U_0 text field, type `U0`.

Open Boundary 1

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

2 Select Boundaries 3 and 5 only.

Outlet 1

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

2 Select Boundary 6 only.

Symmetry 1

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

2 Select Boundary 2 only.

MESH 2

Mapped 1

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

Distribution 1

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

2 Select Boundaries 4 and 5 only.

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

4 In the **Number of elements** text field, type `21*N`.

Distribution 2

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

2 Select Boundaries 2 and 3 only.

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

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

Distribution 3

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

2 Select Boundaries 1 and 6 only.

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

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

5 In the **Number of elements** text field, type `20*N`.

6 In the **Element ratio** text field, type `15`.

7 From the **Growth rate** list, choose **Exponential**.

8 Click **Build Selected**.

STUDY 2

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list
N (Mesh refinement factor)	1 2 4

Step 1: Stationary

1 In the **Model Builder** window, click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Values of Dependent Variables** section.

3 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

4 From the **Method** list, choose **Solution**.

5 From the **Study** list, choose **Similarity Solution, Stationary**.

6 Locate the **Study Settings** section. From the **Tolerance** list, choose **User controlled**.

7 In the **Relative tolerance** text field, type `1e-5`.

Solution 2 (sol2)

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2** > **Solver Configurations** > **Solution 2 (sol2)** > **Stationary Solver 1** node, then click **Fully Coupled 1**.

4 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

5 In the **Maximum number of iterations** text field, type `50`.

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

RESULTS

Cut Line 2D 1

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

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

3 From the **Dataset** list, choose **Study 2/Parametric Solutions 1 (5) (sol3)**.

4 Locate the **Line Data** section. In row **Point 1**, set **x** to `xE`.

5 In row **Point 2**, set **x** to `xE` and **y** to `10*sqrt(b0*xE)`.

Line Graph 2

1 In the **Model Builder** window, under **Results** > **Coefficient Form PDE** right-click **Line Graph 1** and choose **Duplicate**.

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 **Data** section. From the **Dataset** list, choose **Cut Line 2D 1**.

5 Locate the **x-Axis Data** section. In the **Expression** text field, type `y/sqrt(b0*x)`.

6 Locate the **Legends** section. From the **Legends** list, choose **Automatic**.

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

8 In the **Coefficient Form PDE** toolbar, click **Plot**.

Line Integration 1

1 In the **Model Builder** window, under **Results** right-click **Derived Values** and choose **Integration** > **Line Integration**.

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

3 From the **Dataset** list, choose **Cut Line 2D 1**.

4 Locate the **Expressions** section. Click **Clear Table**.

5 In the table, enter the following settings:

Expression	Unit	Description
`(u/U0-comp1.genext1(fprime))^2/sqrt(b0*x)`	1	`Err^2`

6 Click **Evaluate**.

Surface Minimum 1

1 Right-click **Derived Values** and choose **Minimum** > **Surface Minimum**.

2 Select Domain 1 only.

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

4 In the table, enter the following settings:

Expression	Unit
1/h	1/m

5 Locate the **Data** section. From the **Dataset** list, choose **Study 2/ Parametric Solutions 1 (5) (sol3)**.

6 Click the arrow next to the **Evaluate** button and choose **Table 1 - Line Integration 1 ((u/ U0-comp1.genext1(fprime))^2/sqrt(b0*x))**.

TABLE 1

1 Go to the **Table 1** window.

2 Click the **Table Graph** button in the window toolbar.

RESULTS

Table Graph 1

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

2 From the **x-axis data** list, choose **1/h (1/m)**.

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

4 In the **Columns** list box, select **Err^2 (1)**.

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

6 Click the **y-Axis Log Scale** button in the **Graphics** toolbar.

7 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.