Created in COMSOL Multiphysics 5.6

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).

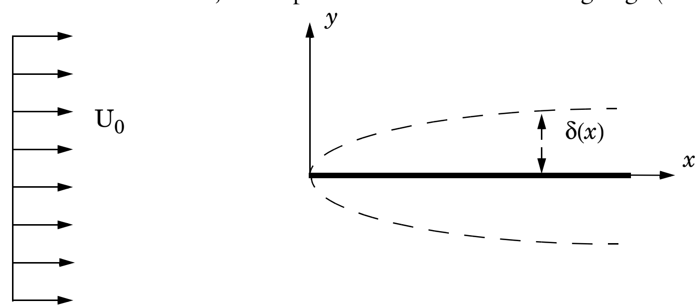

Figure 1: The boundary layer on a flat plate. $\delta(x)$ *is the boundary-layer thickness, such that* $u(x, \delta(x)) = U_0$.

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,

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

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 $T = 20$ °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,

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

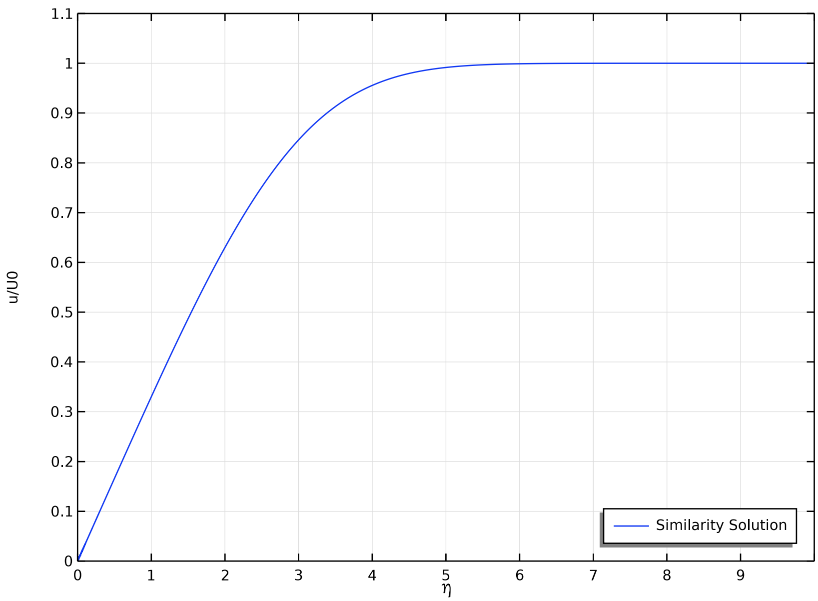

Figure 2: Similarity solution for the streamwise velocity component.

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 $Re_x = 1.0 \cdot 10^5$. 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.

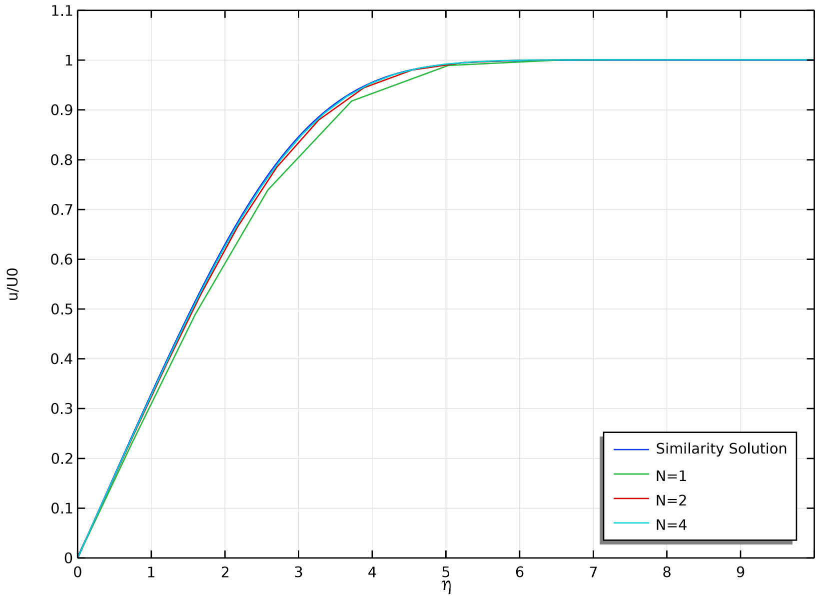

Figure 3: Comparison between the similarity solution and the two-dimensional simulations.

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.

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

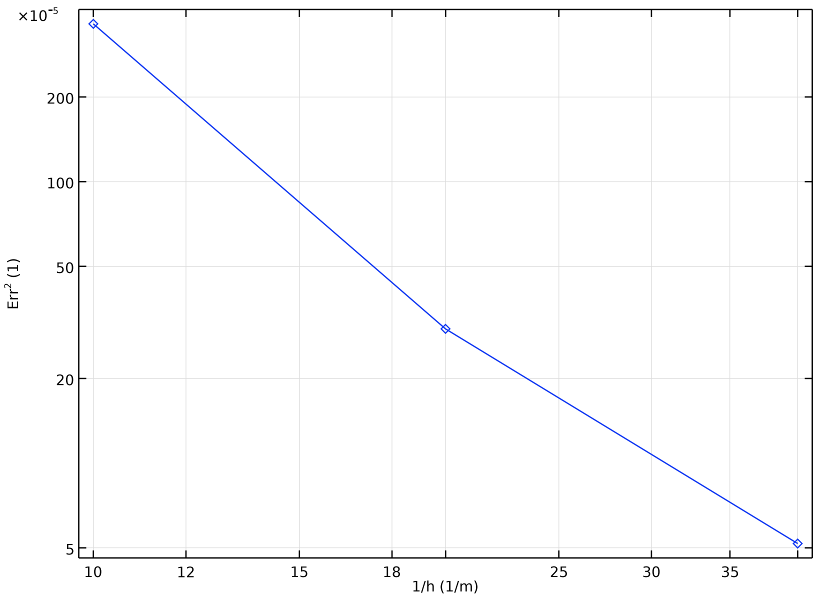

Figure 4: Convergence rate as a function of inverse maximum cell side.

Notes About the COMSOL Implementation

The relative tolerance is set to 10^{-4} in all the solvers to ensure that the equation systems become 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 (Engl. transl. 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** text field, type `f`.

5 Click **Add Dependent Variable**.

6 In the **Dependent variables** 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** check box.

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 formula** list, choose **Geometric sequence**.

7 Click **Build Selected**.

STUDY 1

Solution 1 (sol1)

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

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

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

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

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.

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

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

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

8 In the **Study** toolbar, click **= Compute**.

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

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

11 Click **OK**.

RESULTS

Line Graph 1

1 In the **Model Builder** window, expand the **1D Plot Group 1** 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** check box.

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

6 In the table, enter the following settings:

Legends
`Similarity Solution`

1D Plot Group 1

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

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. Select the **x-axis label** check box.

5 In the associated text field, type `\eta`.

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

7 In the associated text field, type `u/U0`.

8 Locate the **Axis** section. Select the **Manual axis limits** check box.

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

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

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

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

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

14 In the **1D Plot Group 1** 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** check box for **Similarity Solution**.
5. Click **Add to Component 2** 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 **Add Study** in the window toolbar.
5. In the **Model Builder** window, click the root node.
6. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

GEOMETRY 2

In the **Model Builder** window, under **Component 2 (comp2)** click **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 **Home** 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 **Add to Component** in the window toolbar.
5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

LAMINAR FLOW (SPF)

In the **Model Builder** window, under **Component 2 (comp2)** 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 **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 formula** list, choose **Geometric sequence**.

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**.

Solution 2 (sol2)

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Stationary Solver 1**.

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

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

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

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

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

8 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>1D Plot Group 1** 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. In the table, enter the following settings:

Legends
N=1
N=2
N=4

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

Line Integration 1

1 In the **Results** toolbar, click **More 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 In the **Model Builder** window, 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 Go to the **Table** window.

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

RESULTS

Table Graph 1

1 In the **Model Builder** window, under **Results>1D Plot Group 5** click **Table Graph 1**.

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

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

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

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

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

7 From the **Positioning** list, choose **In data points**.

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

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