

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

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = v\frac{\partial^2 u}{\partial y^2}$$
(1)

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

2 | THE BLASIUS BOUNDARY LAYER

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

$$2f''' + ff'' = 0 (3)$$

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 °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 $\text{Re}_x = 1.0 \cdot 10^5$. 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.

I/H	10	20	40
ε	6.10·10 ⁻²	3.33·10 ⁻²	1.68·10 ⁻²
DOF	2016	7749	30375

TABLE I: DEVIATION FROM THE BLASIUS SOLUTION.



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 (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

- I In the Model Wizard window, click ID.
- 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 \bigcirc Study.
- 8 In the Select Study tree, select General Studies>Stationary.
- 9 Click 🗹 Done.

GLOBAL DEFINITIONS

Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** In the table, enter the following settings:

Name	Expression	Value	Description
UO	0.75[m/s]	0.75 m/s	Inlet velocity
nu	1.506137e-5[m^2/s]	1.5061E-5 m ² /s	Kinematic viscosity
хE	2[m]	2 m	Evaluation location
b0	nu/UO	2.0082E-5 m	B-L scale
Ν	1	I	Mesh refinement factor

GEOMETRY I

Interval I (i1)

- I In the Model Builder window, under Component I (comp1) right-click Geometry I 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

- I In the Model Builder window, under Component I (compl)>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.
- II In the β text-field array, type f in the second column of the second row.

Dirichlet Boundary Condition 1

- I In the Physics toolbar, click Boundaries and choose Dirichlet Boundary Condition.
- 2 Select Boundary 1 only.

Dirichlet Boundary Condition 2

I In the Physics toolbar, click — Boundaries and choose Dirichlet Boundary Condition.

- **2** Click the \leftarrow **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 I

Edge I

- I In the Mesh toolbar, click \triangle Edge.
- 2 In the Settings window for Edge, locate the Domain Selection section.
- **3** From the **Geometric entity level** list, choose **Entire geometry**.

Distribution I

- I Right-click Edge I 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 I

Solution 1 (soll)

- I In the Study toolbar, click **The Show Default Solver**.
- 2 In the Model Builder window, expand the Solution I (soll) node, then click Stationary Solver I.
- 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 I>Solver Configurations> Solution I (soll)>Stationary Solver I 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 I and choose Rename.
- 10 In the Rename Study dialog box, type Similarity Solution in the New label text field.

II Click OK.

RESULTS

Line Graph 1

- I In the Model Builder window, expand the ID Plot Group I node, then click Line Graph I.
- 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

ID Plot Group 1

- I In the Model Builder window, click ID Plot Group I.
- 2 In the Settings window for ID 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 check box. In the associated text field, type \eta.
- 6 Select the y-axis label check box. In the associated text field, type u/U0.
- 7 Locate the Axis section. Select the Manual axis limits check box.
- 8 In the **x minimum** text field, type 0.
- 9 In the **x maximum** text field, type 10.
- **IO** In the **y minimum** text field, type **0**.
- II In the **y maximum** text field, type 1.1.
- 12 Locate the Legend section. From the Position list, choose Lower right.
- **I3** In the **ID Plot Group I** toolbar, click **ID Plot**.

DEFINITIONS

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

General Extrusion 1 (genext1)

- I In the Model Builder window, expand the Component I (compl)>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

- I 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

- I In the Home toolbar, click $\sim\sim$ 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 $\stackrel{\text{res}}{\longrightarrow}$ 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)

I 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 I (ptl)

- I In the **Geometry** toolbar, click **Point**.
- 2 Click 📗 Build All.

Point 2 (pt2)

- I 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

- I 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

- I 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 I

- I In the Physics toolbar, click Boundaries and choose Open Boundary.
- **2** Select Boundaries 3 and 5 only.

Outlet I

- I In the Physics toolbar, click Boundaries and choose Outlet.
- 2 Select Boundary 6 only.

Symmetry I

- I In the Physics toolbar, click Boundaries and choose Symmetry.
- **2** Select Boundary 2 only.

MESH 2

Mapped I

In the Mesh toolbar, click Mapped.

Distribution I

- I Right-click Mapped I 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

- I In the Model Builder window, right-click Mapped I 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

- I Right-click Mapped I 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

- I 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)	124

Step 1: Stationary

- I In the Model Builder window, click Step I: 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)

- I In the Study toolbar, click **here** 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

I In the **Results** toolbar, click \frown **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 I (5) (sol3).
- 4 Locate the Line Data section. In row Point I, set x to xE.
- 5 In row Point 2, set x to xE and y to 10*sqrt(b0*xE).

Line Graph 2

- I In the Model Builder window, under Results>ID Plot Group I right-click Line Graph I 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 I.
- 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 ID Plot Group I toolbar, click 💿 Plot.

Line Integration 1

- I In the Results toolbar, click ^{8.85}_{e-12} 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 I.
- **4** Locate the **Expressions** section. Click **Clear Table**.
- **5** In the table, enter the following settings:

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

6 Click **= Evaluate**.

Surface Minimum 1

- I 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 I Line Integration I ((u/ U0-compl.genextl(fprime))^2/sqrt(b0*x)).

TABLE

- I Go to the Table window.
- 2 Click Table Graph in the window toolbar.

RESULTS

Table Graph 1

- I In the Model Builder window, under Results>ID Plot Group 5 click Table Graph I.
- 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 Err² (1).
- 6 Locate the Coloring and Style section. Find the Line markers subsection. From the Marker list, choose Diamond.
- 7 Click the y-Axis Log Scale button in the Graphics toolbar.
- 8 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.