FLUENT - Flat Plate Boundary Layer

**This page has been moved to https://courses.ansys.com/index.php/courses/flat-plate-laminar-boundary-layer/
Click in the link above if you are not automatically redirected in 10 seconds.**

Author: Rajesh Bhaskaran, Cornell University

Problem Specification
1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Model Setup
5. Numerical Solution
6. Post Processing
7. Verification & Validation
8. Part II: Flat Plate Convection

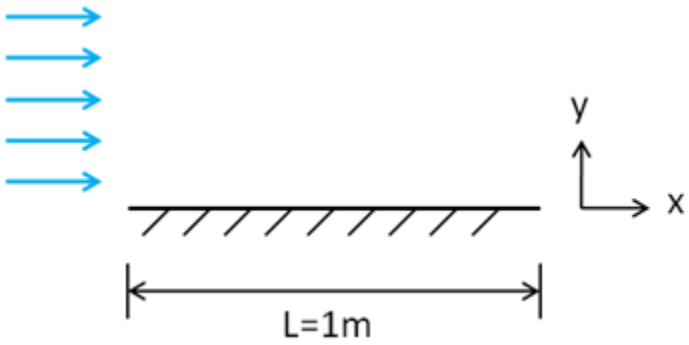
Frequently Asked Questions

Exercises

Comments

Flat Plate Boundary Layer

Problem Specification



Consider a fluid flowing across a flat plate, as illustrated above. Obtain the velocity and pressure distribution when the Reynolds number based on the plate length is 10,000. This Reynolds number is obtained by using the following settings. The plate length is 1 m. The incoming fluid is flowing in the x-direction with a velocity of 1 m/s. The density of the fluid is 1 kg/m^3 and the viscosity is 1 x 10 ^(-4) kg/(m-s). Note that these values are not necessarily physical. They have been picked to yield the desired Reynolds number.

Check your results by comparing the velocity and pressure distribution with classical boundary layer theory.

Handouts:

1. Slides used in the videos
2. ANSYS Fluent solution outline

Go to Step 1: Pre-Analysis & Start-Up

Go to all FLUENT Learning Modules