

# Old (4/1/2020) Flat Plate Boundary Layer - Pre-Analysis & Start-Up

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](#)

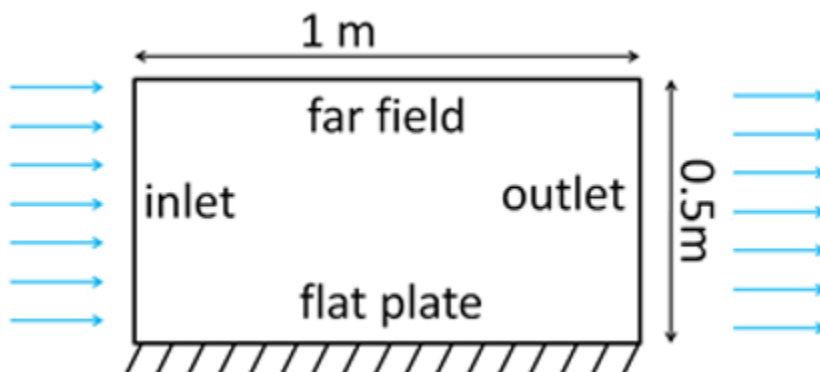
## Pre-Analysis and Start-Up

For an external flow like this, one needs to determine where to place the outer boundary. A rectangular domain will be used for this simulation. The height of the rectangular domain will be determined by the expected boundary layer thickness. For a flat plate boundary layer, one can use the following approximation to find the boundary layer thickness.

$$\frac{\delta_{99}}{x} = \frac{5}{\sqrt{Re_x}}$$

When  $x = L$ ,

$\delta_{99} = 0.05\text{m}$ . The height of the domain will be set to ten times the boundary layer thickness. Thus, the height of the boundary will be set to 0.5m. The following figure shows the dimensions of the domain, and the boundary labels which will be used to set the boundary conditions.

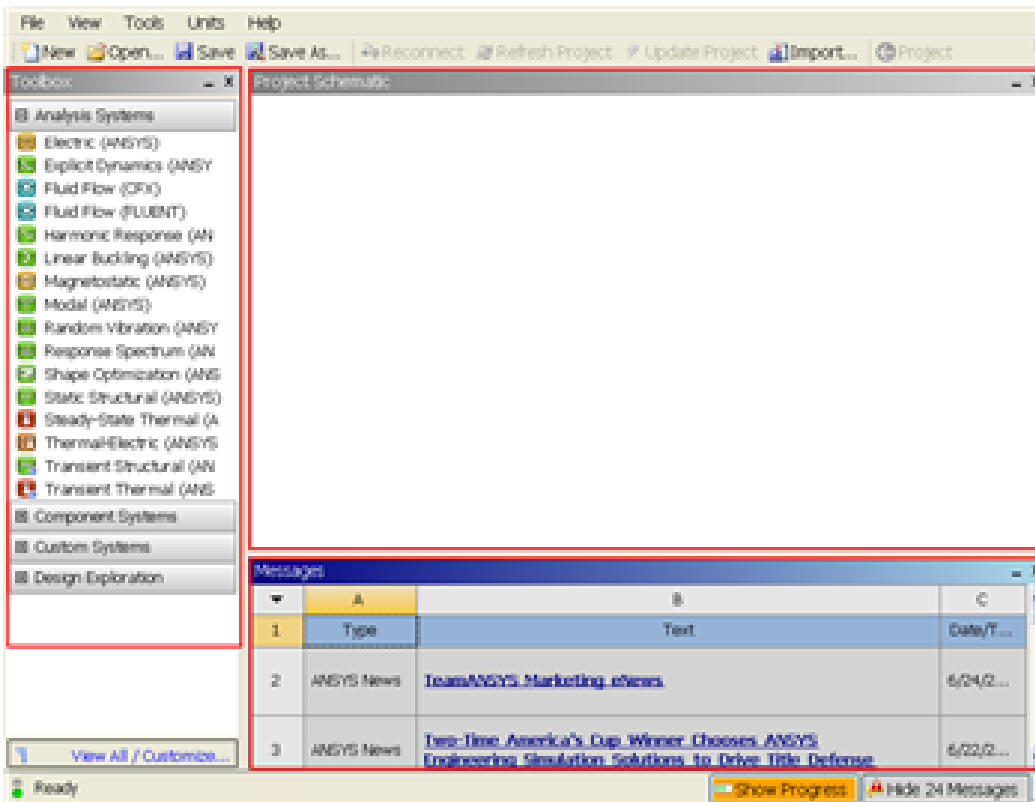


### Start ANSYS FLUENT

Prior to opening ANSYS, create a folder called FlatPlateCFD in a convenient location. We'll use this as the working folder in which files created during the session will be stored. For this simulation FLUENT will be run within the ANSYS Workbench Interface. Start ANSYS Workbench:

Start> All Programs> Ansys 12.1> Workbench

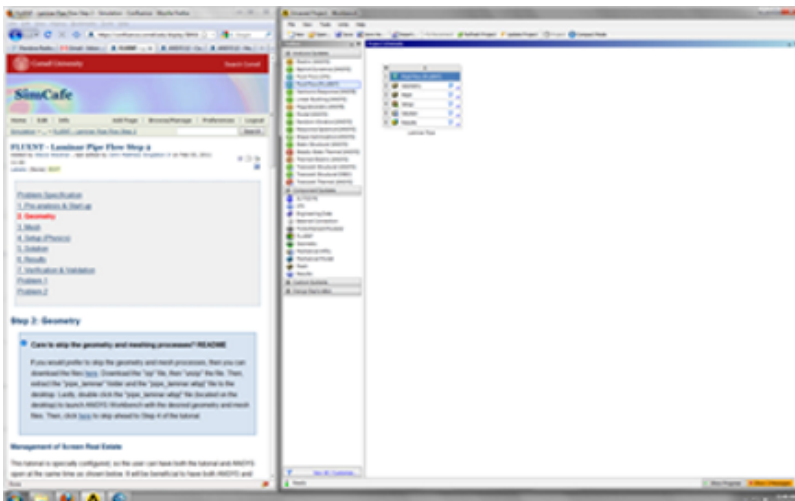
The following figure shows the workbench window.



<https://confluence.cornell.edu/download/attachments/111221567/Workbench.png>

## Management of Screen Real Estate

This tutorial is specially configured, so the user can have both the tutorial and ANSYS open at the same time as shown below. It will be beneficial to have both ANSYS and your internet browser displayed on your monitor simultaneously. Your internet browser should consume approximately one third of the screen width while ANSYS should take the other two thirds as shown below.



[https://confluence.cornell.edu/download/attachments/85624044/RealEstate\\_Full.png](https://confluence.cornell.edu/download/attachments/85624044/RealEstate_Full.png)

If the monitor you are using is insufficient in size, you can press the **Alt** and **Tab** keys simultaneously to toggle between ANSYS and your internet browser.

**[Go to Step 2: Geometry](#)**

[Go to all FLUENT Learning Modules](#)