Finite Volume Modeling of Boundary Layer Development

Matthew Shorter

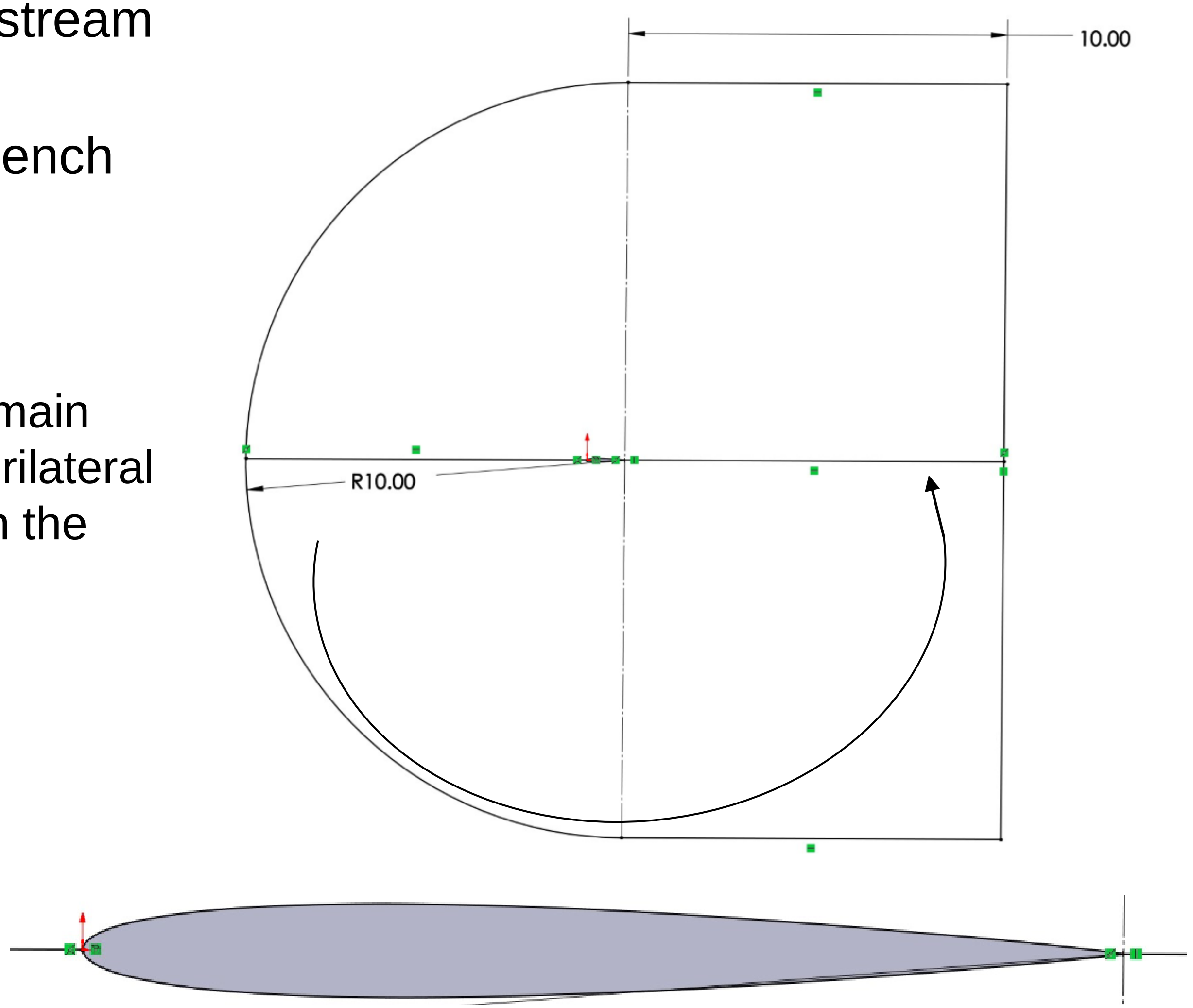
2.29 Final Project

Problem Overview

- Boundary layers are both fundamental to engineering fluids problems (e.g. drag and stall characteristics of an aircraft wing) and difficult to model analytically or numerically
- Approaches include wind-tunnel testing, panel methods such as Professor Drela's "XFOIL", and finite volume software such as the commercial Fluent package
- Airfoil examined was NACA 0009: slender airfoil with no camber. Relatively crude airfoil
- Reynolds number is 10^5 : this is in the transition regime where flow over airfoil is not completely turbulent over the entire chord.
- Incompressible fluid with constant viscosity:
  - Mach number effects on boundary layers are generally small even at $M > 1$ in the absence of shocks
  - Many relevant flows are at $M < 0.3$
- 2D steady computation
  - Physical flows are 3D and unsteady, but flow separation on an 2D wing is driven by 2D effects, and turbulence models allow for steady state simulation of steady mean flow

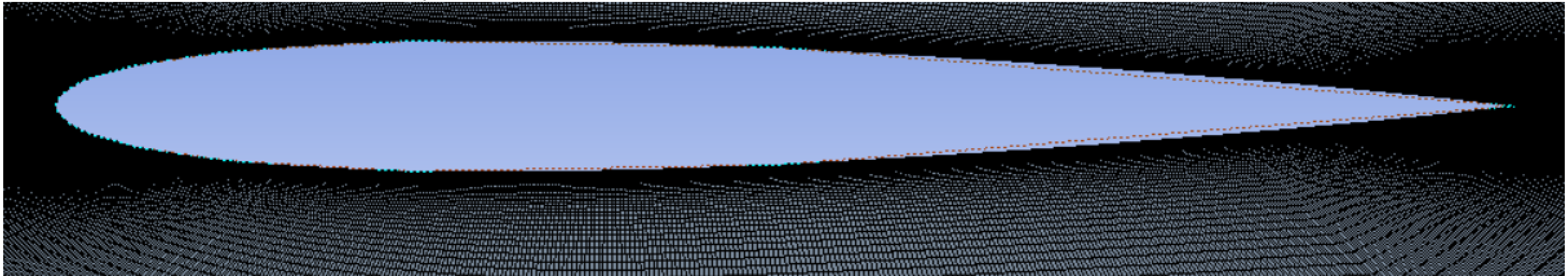
Computational Domain

- Per CFD best practices, semicircular upstream domain with radius 10x airfoil chord and downstream rectangle
- Mapped mesh generated using ANSYS workbench mesher
  - Arrow shows mesh sweep
  - Although Fluent is an unstructured grid solver, sweeping the upper and lower halves of the domain allows for improved mesh quality, a 100% quadrilateral mesh, and finer control over mesh resolution on the airfoil



Flow physics drive mesh resolution at wall

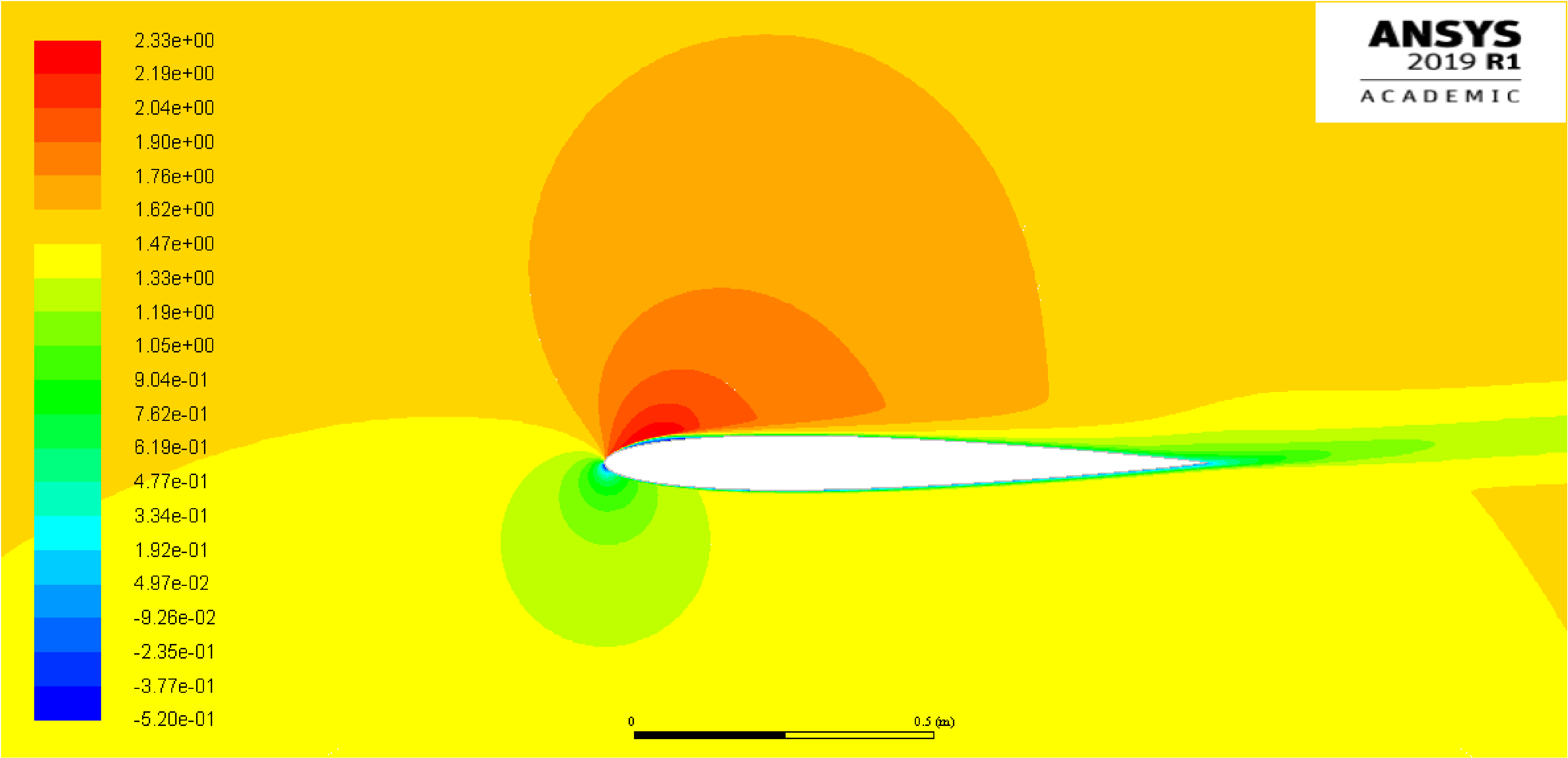
- Boundary layer theory defines the length scale y^+ as the thickness of the linear viscous shear layer near a wall
- The two length scales for this problem are the chord and the boundary layer thickness: these differ by orders of magnitude
- For a 1m flat plate in air with no adverse pressure gradient at Re = 1e5, y^+ is $2*10^{-4}$ m
- This is only an approximation for the shear layer thickness on an airfoil: the leading edge is more similar to impingement on a wall and there are strong adverse and favorable pressure gradients in the flow. The mesh was refined until $y^+ < 1$ everywhere except very close to the blunt leading edge
- Commercial tools such as Fluent generally include "wall functions" that act as fudge factors if the resolution is not high enough, but those do not work for the transition model, and the y+<1 models run very quickly for all cases
- Flow domain has 110,000 elements



Transition from Laminar to Turbulent flow

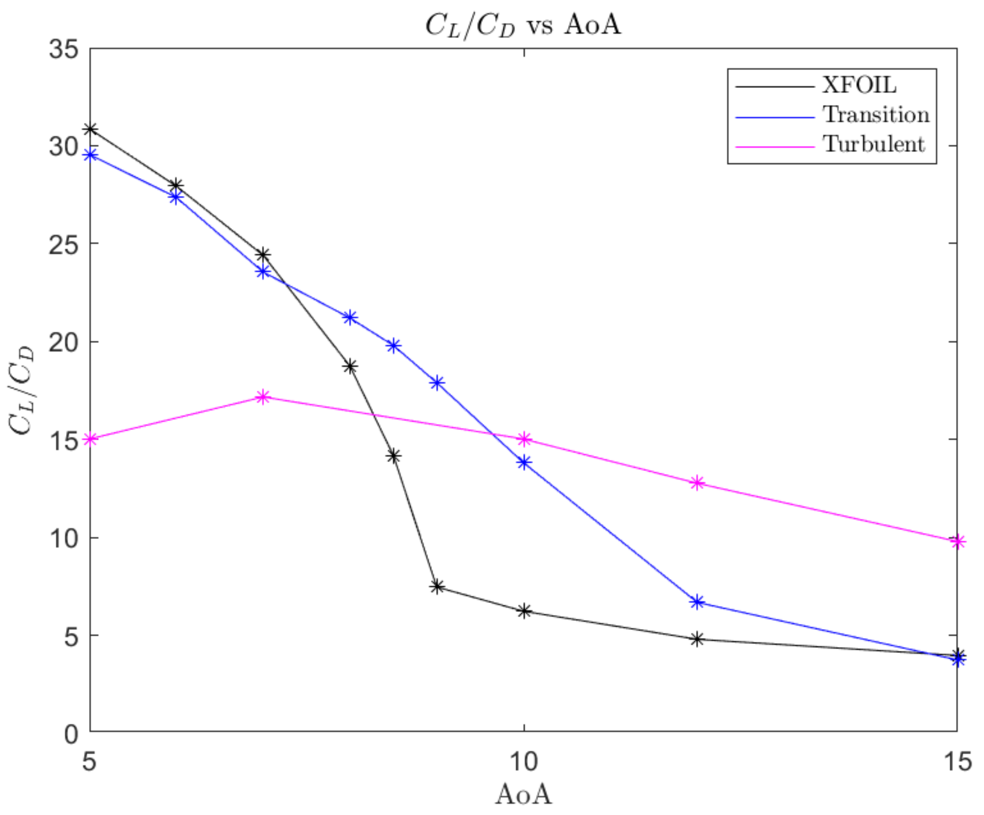
- Due to adverse pressure gradient on suction (upper) side of airfoil, the flow transitions from laminar to turbulent
- The turbulence in the free stream has a significant effect on when this transition occurs
- XFOIL uses an envelope method which accumulates the effect of the strongest disturbance frequencies until Ncrit is reached. Literature shows that the disturbances for an unswept wing can be well treated by a 2D approach
- FLUENT uses a method based on the k-omega RANS model that tracks production, destruction, and transport of turbulence starting with a laminar flow
- Challenge for designer or analyst is that the inlet boundary condition (turbulent intensity or Ncrit value) has a leading order effect on how the turbulent boundary layer develops

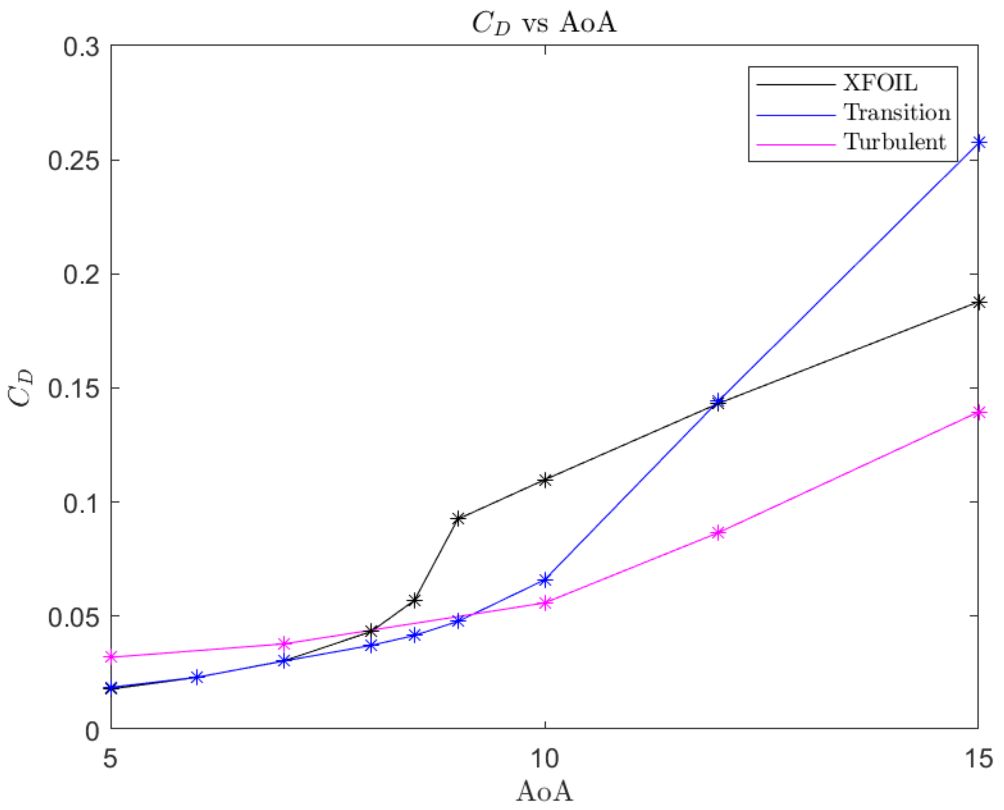
Postprocessing challenge: from "Colorful Fluid Dynamics" to useful engineering insight



Lift and Drag coefficients for Fluent and XFOIL

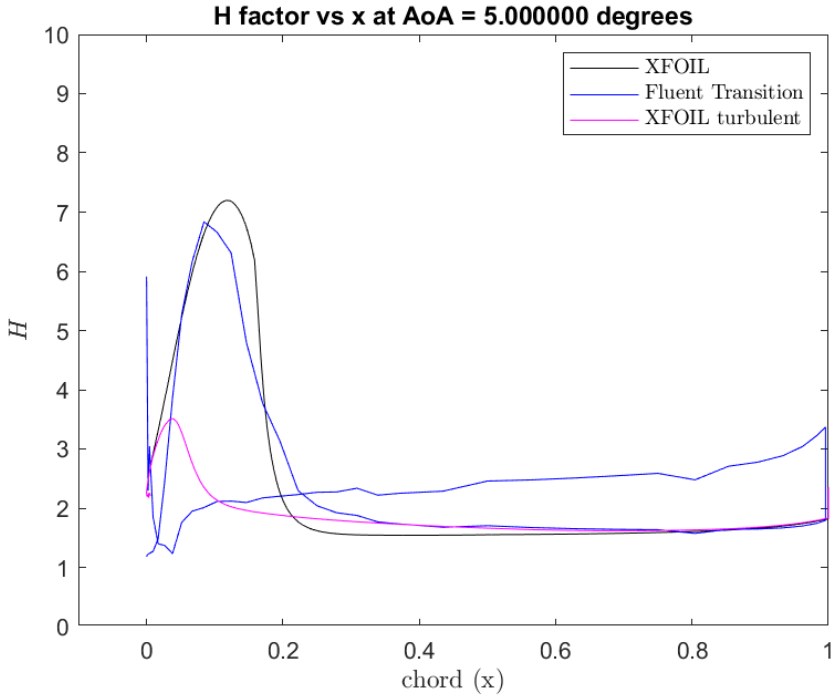
- Normalized Lift and Drag are the two primary deliverables of the airfoil
- Boundary layer determines wall shear stress
- Thick or separated boundary layers affect surrounding potential flow, which alter lift

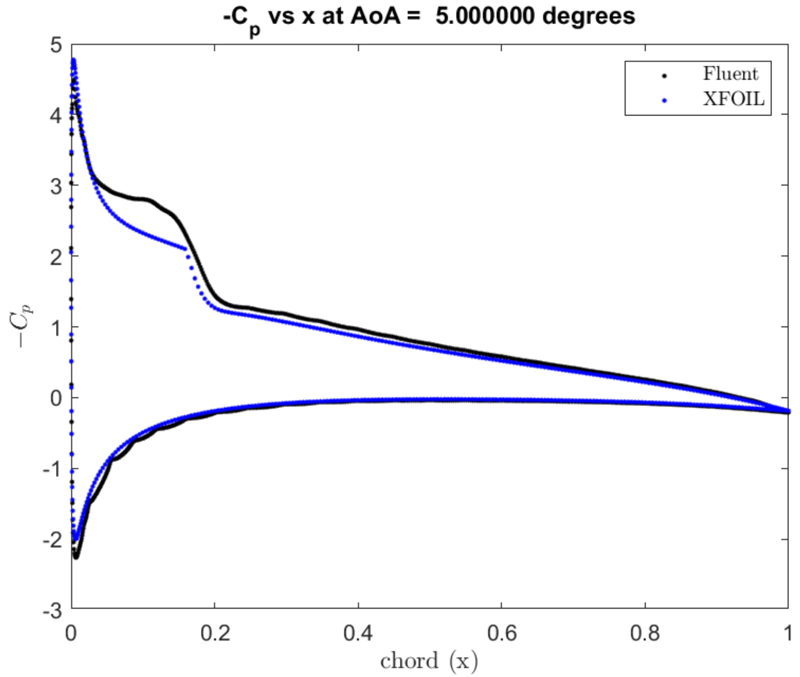




Health of boundary layer along airfoil chord

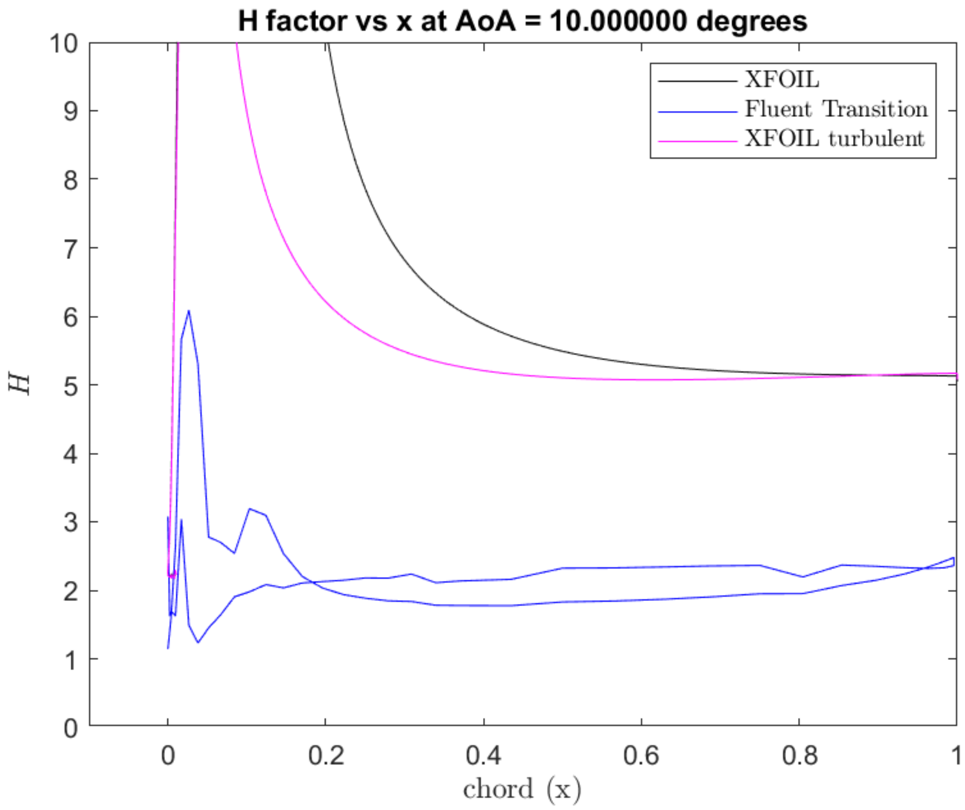
- The H factor is defined as the displacement thickness/momentum thickness
- We assume that a boundary layer with a given "H" is identical to a self similar boundary layer with the same "H"
- H = 2.6: Blasius flow (no pressure gradient)
- H < 2.6: favorable pressure gradient
- H > 2.6: adverse pressure gradient
- H = 4: incipient separation (no shear at the wall)
- At 5 degrees, both standard XFOIL and Fluent transition model predict separation bubble at ~0.1 chord
- Separation is visible on plots of pressure coefficient

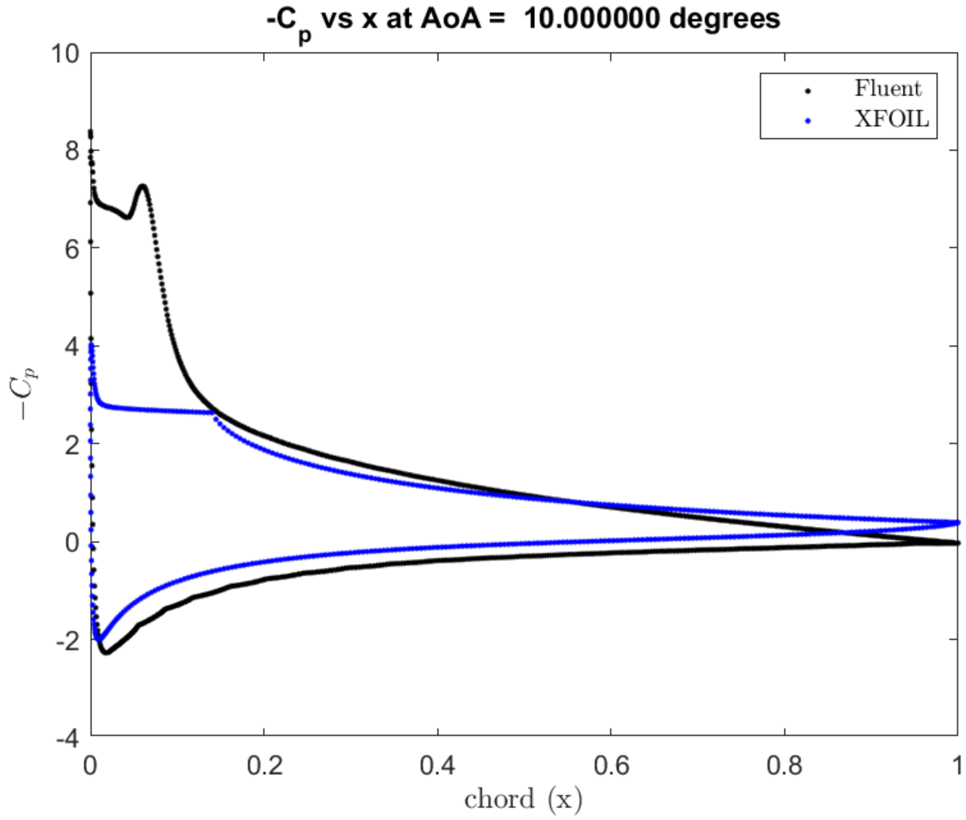




At higher angle of attack, correspondence between XFOIL and Fluent weakens

- Transition model predicts later separation bubble than the turbulent model, but predicts that the flow reattaches after the bubble
- Adverse pressure gradient is much weaker for Fluent case





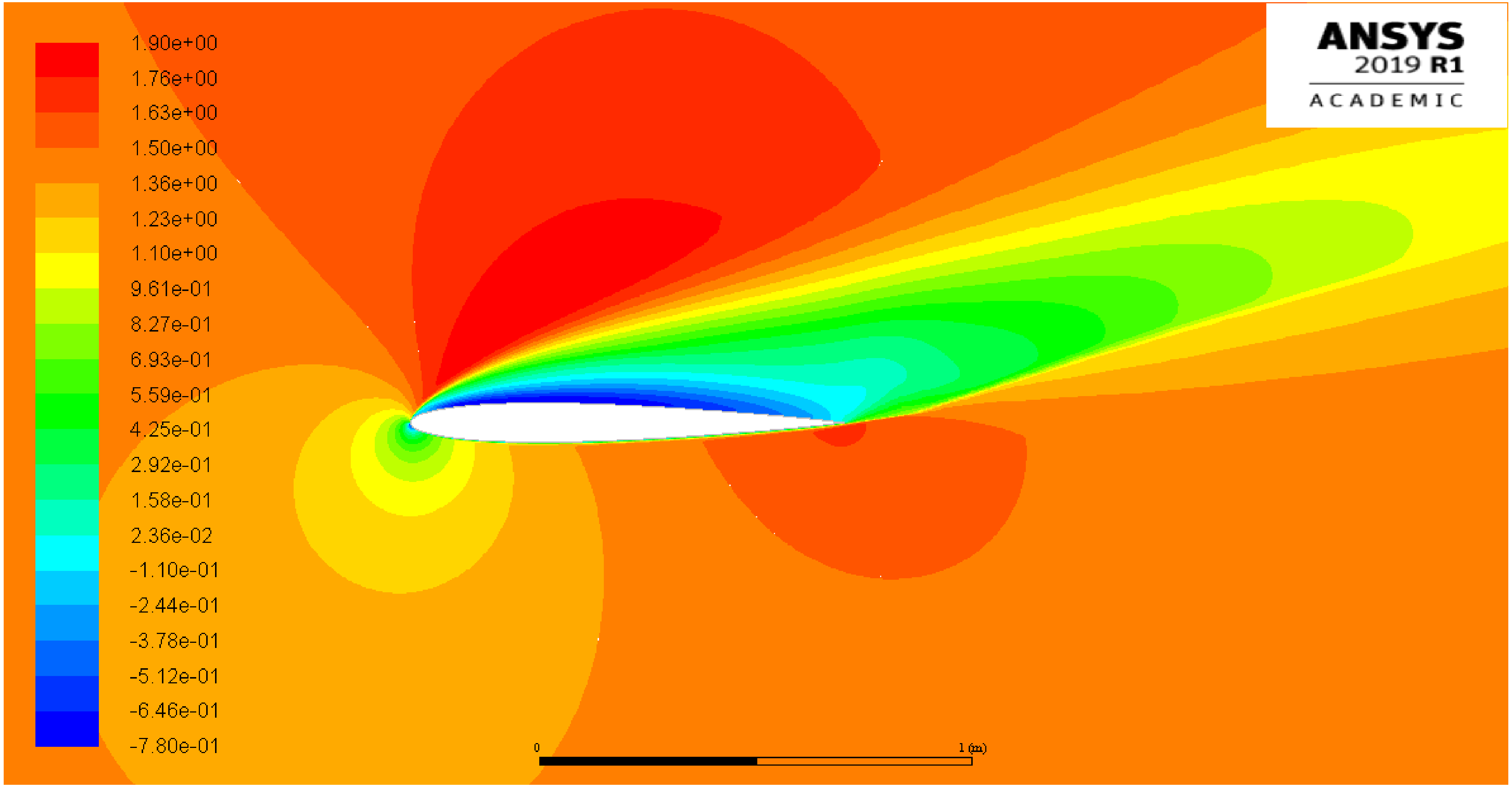
Conclusions and future work

- Transition model is likely very sensitive to the freestream conditions: literature shows that transition behaves differently in internal vs external flows
- XFOIL is very robust in region where flow is attached over most of airfoil, but panel method may be limited where the flow is separated over nearly the entire chord
- Finite volume solvers remain the best way to predict flow behavior in a system rather than XFOIL which is limited to a single 2D steady state airfoil
- This work is easily extensible to transonic flows
- A better test case instead of NACA0009 should be used

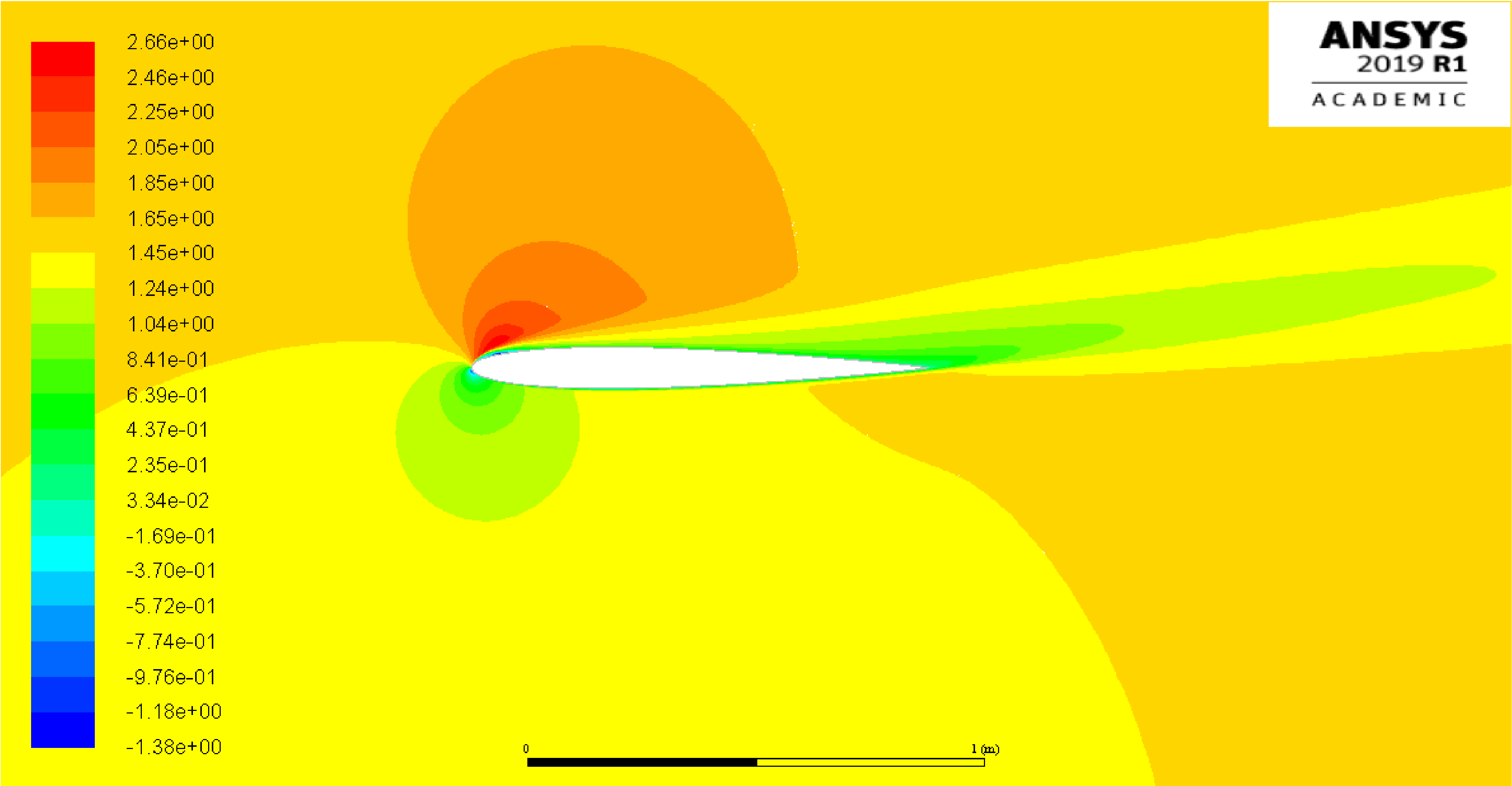
References

- FLUENT theory manual via https://www.sharcnet.ca/Software/Ansys/17.0/en-us/help/flu_th/flu_th.html
- XFOIL documentation https://web.mit.edu/drela/Public/web/xfoil/xfoil_doc.txt
- Aerodynamics of Viscous Flow, Mark Drela (unpublished, available through 16.13)

Backup slides: flow at 15 degrees: separated over nearly all of suction surface

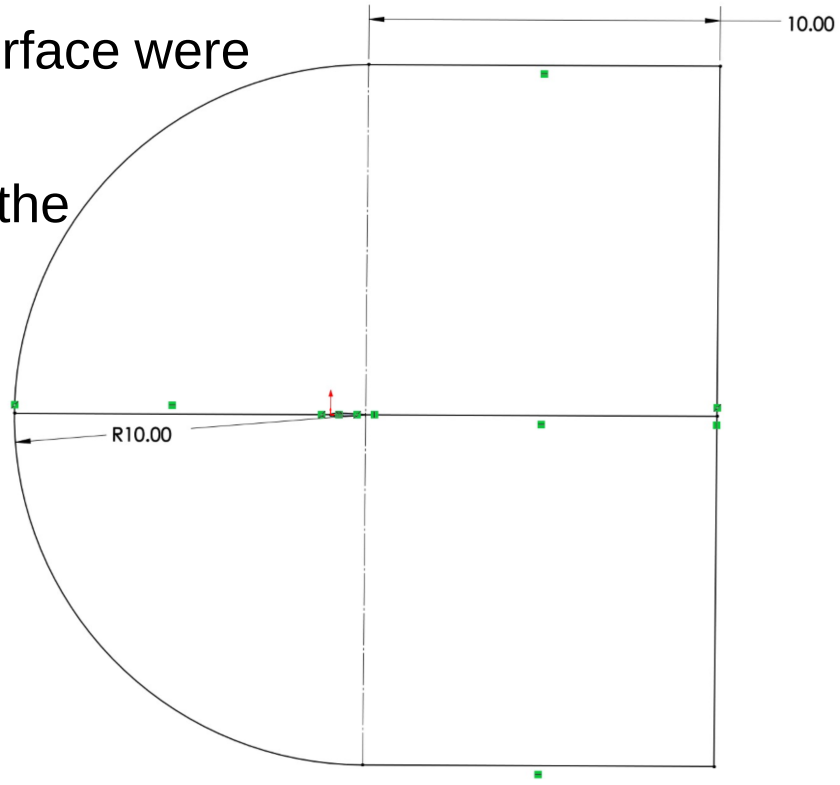


Flow at 10 degrees



Solution methods and comments

- 2^{nd} order upwind for transport of turbulence variables as well as momentum
- 2^{nd} order pressure
- This is generally a fairly standard commercial finite volume solver
- Coupled pressure based (this is the projection method seen in class) as recommended by theory manual
- To set the inlet BC, the semicircle and bottom horizontal surface were given a prescribed velocity vector
- The outlet was defined by a given pressure averaged over the boundary
- The airfoil wall was the no-slip condition
- FLUENT uses pseudo-time marching



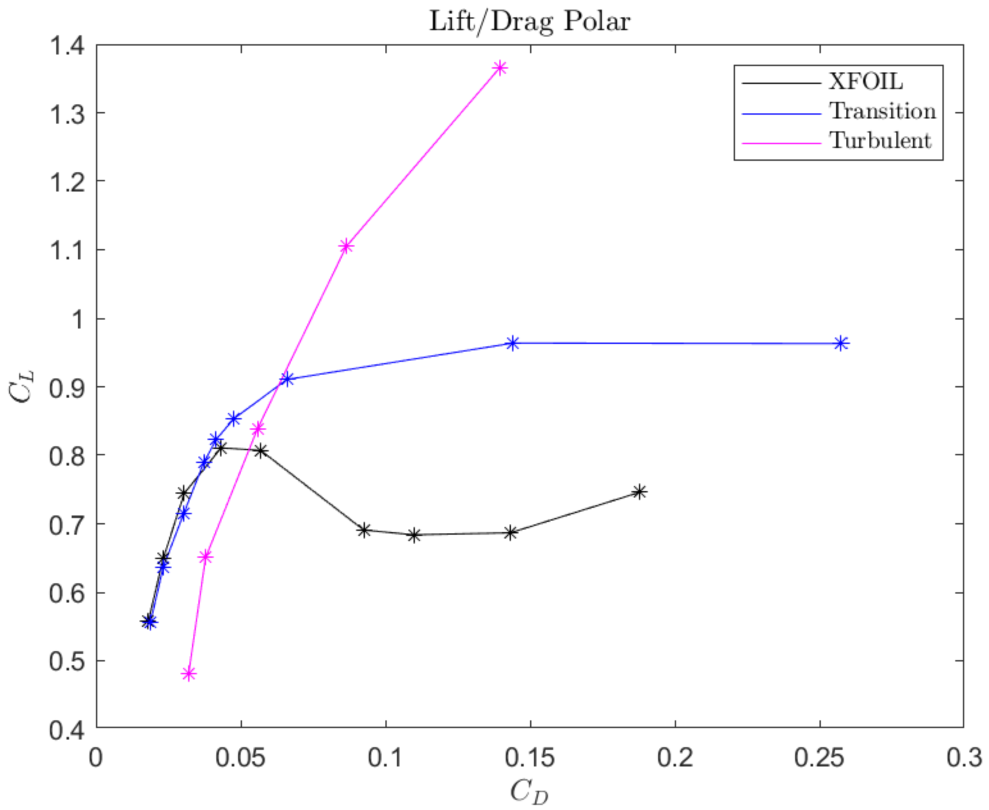
Calculating H factor

- Workflow was to
  - find the normal vectors at various arc lengths on the airfoil using central differences
  - Extract velocity along each of those vectors using tecplot
  - Import to MATLAB and integrate across the boundary layer to find the displacement and momentum thicknesses

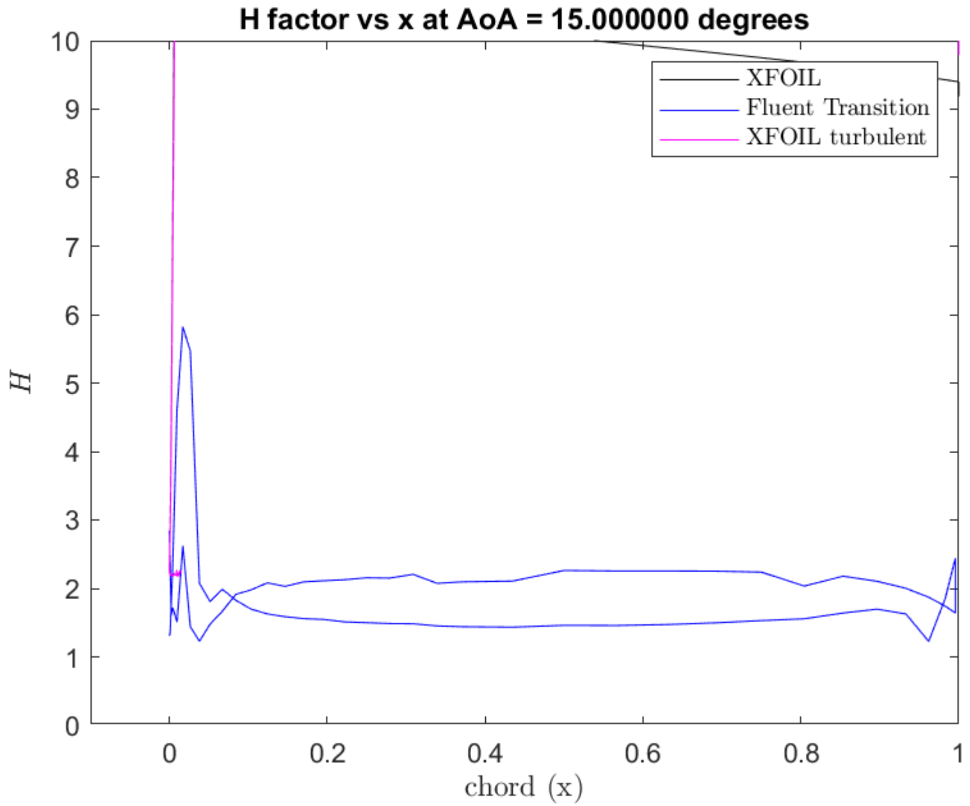
Why Fluent?

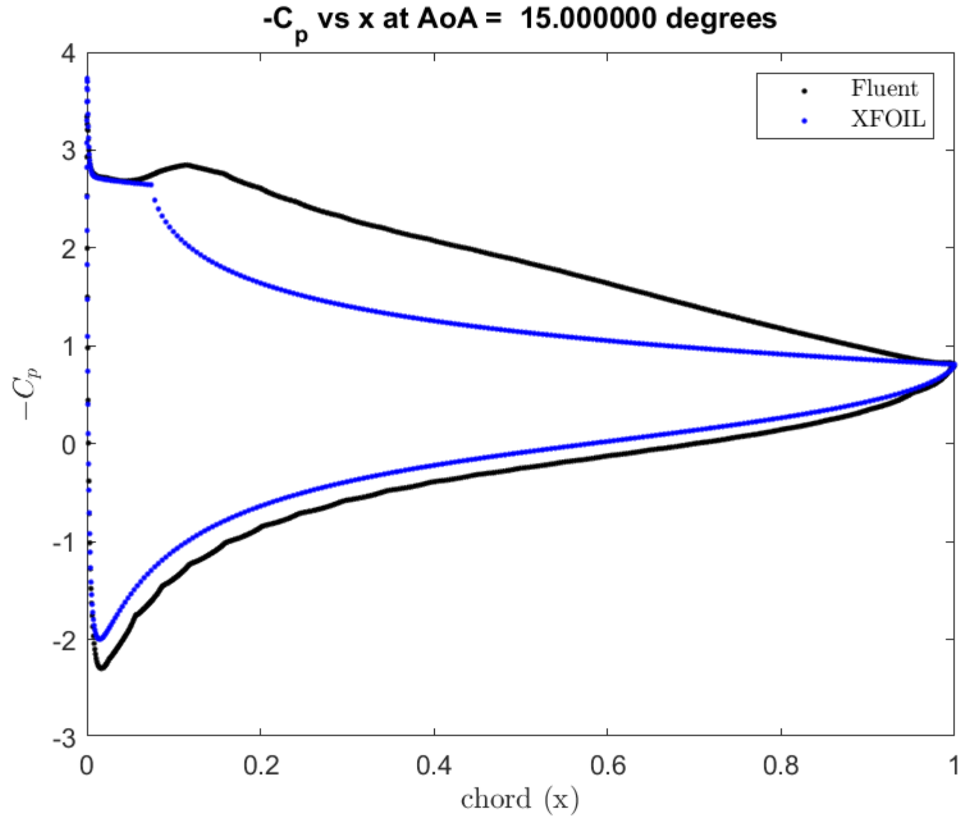
- Fluent is easy to learn and iterate on
- Biggest advantage by far is easy grid generation with ANSYS Workbench
  - Granular control of mesh is critical to successful CFD
- OpenFOAM and other solvers each have comparative advantages or disadvantages
- Virtually all post-processing done by exporting case/data to Tecplot, using macros to write out data files, and preparing plots in MATLAB
- Fluent has a free license for education. 110,000 elements is within the limit and easily run on my laptop machine (order of minutes)

Lift/Drag polar chart



H and C_p at high angle of attack (15 degrees)





XFOIL sample plot of C_p, velocity profiles