Lab (28/2003)

a) After having completed the tutorial, http://courses.cit.cornell.edu/fluent/airfoil/step4.htm, save as airfoil_tut1.cas (/dat)

b) Airfoil_tut2

- Open airfoil_tut1.cas, save as airfoil_tut2.cas (or you may prefer to use your own airfoil geometry and mesh now).
- Re-setup some parameters in tutorial 2,
- Define->Models->Viscous->K-epsilon (keep the defaults)
- Change the inlet velocity magnitude from 50m/s to 10m/s. Define->Boundary conditions->farfield1->velocity inlet->set x-velocity 9.96m/s,y-velocity 0.872m/s, set turbulence specification method to intensity and viscosity ratio->0.1% and 1 respectively. Set the same parameters for the other inlet (farfield2).
- Re-initialize and run the computation.
- Get the computation converged.
- Compare the velocity vectors with those from tutorial 1.
- Save case and data files as airfoil_tut2.cas(dat).

c) Fluent tutorial 3 (or you may prefer to use your own airfoil geometry and mesh now).

Assuming the estimated maximum boundary layer thickness is 0.02m on the solid surface (for turbulent flow over a smooth plate, it is estimated that the BL thickness is $0.37x(U_\infty x/\nu)^{-0.2}$).

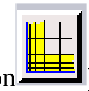
- Adapt->Boundary->Normal distance (highlight 'airfoil')->0.04m ->(have a check on Controls to make sure 'Hanging' and 'Refine' are chosen ->click Apply and Adapt.
- Adapt->Boundary->Normal distance (highlight 'airfoil')->0.02m ->click Apply and Adapt. (Within 0.02m BL there are approx 20 grid points).
- Re-run the computation (no need to re-initialize, which saves a lot CPU time for very large computations).
- Compare the velocity vectors with those from tutorial 2.

<u>This is an easy (but not the best) way to refine your mesh! Because hanging nodes may cause some errors.</u>

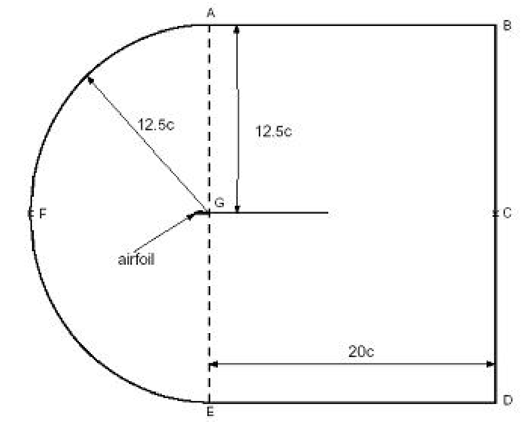
d) Gambit tutorial 3.

- Choose your own airfoil.
- Obtain the geometry. Obtain 2-D airfoil data from internet and import into Gambit. http://www.afm.ses.soton.ac.uk/~ztx/sess6011/cornell_changes.htm

  Design a domain.
- Estimate the maximum boundary layer thickness (e.g. for turbulent flow over a smooth plate, it is estimated that the BL thickness is $0.37x(U_\infty x/\nu)^{-0.2}$, where U_∞ freestream velocity, ν kinematic viscosity, x distance to the leading edge. See book by H Schlichting).
- Estimate the value of $y_1^+=(y_1 u_*)/\nu$, where y_1 is the distance of the first grid to the wall, u_* friction velocity, ν kinematic viscosity. (Again, for turbulent flow over a smooth plate, it is estimated that $(u_*/ U_\infty)^2 = 0.0296(U_\infty x/\nu)^{-0.2}$. See book by H Schlichting).
- Mesh it using unstructured mesh (all by the student himself/herself), with a boundary

layer (using Gambit BL function ) on the solid surfaces and a shear layer approx on the free shear layer.

<u>How to make a boundary layer mesh for unstructured grid:</u>



Mesh the airfoil surface and GC -> Operation, Mesh, Boundary Layer ->then you must make a calculation, what is your maximum boundary layer thickness, what is the first grid length, how many grids point within BL? For instance, they are 0.02m, 0.001m, 10 grid points-> Attachment->Edges->choose all the airfoil edges and choose GC twice (two BLs for GC's both sides)->First row, 0.001 ->Rows, 10 ->change the Growth Factor,e.g. 1.16, to force the Depth (D) around 0.02m, note don't have a large Growth Factor, e.g. >1.2 ->Transition pattern, 1:1.

You may want to split GC into two edges, then you don't need to setup BL mesh for the whole GC.

Then ->Mesh AG GE -> mesh BC and CD -> mesh AB and DE ->mesh AF and FE.

->mesh the 3 faces -> setup the BCs.

<u>If you want to setup a BL mesh for structured mesh, you should note below,</u>

Mesh the airfoil surface and GC -> build a BL mesh as above -> Mesh AG GE (for structured mesh, No. of grids on AG=No. of grids on GE for structured mesh) ->mesh BC and CD -> For structured mesh, No. of elements on AF=No. of elements on upper airfoil surface; No. of grids on FE=No. of grids on lower airfoil surface->…->for structured mesh, mesh face using 'Map' function.

e) Mesh quality check for unstructured mesh, using 'Examing Mesh'—'EquiAngle Skew'

<u>*EquiAngle Skew*</u>

The EquiAngle Skew (Q_{EAS}) is a normalized measure of skewness that is defined as follows:

$$Q_{EAS} = \max\left\{\frac{\theta_{\max} - \theta_{eq}}{180 - \theta_{eq}}, \frac{\theta_{eq} - \theta_{\min}}{\theta_{eq}}\right\}$$

where θ_{max} and θ_{min} are the maximum and minimum angles (in degrees) between the edges of the element, and θ_{eq} is the characteristic angle corresponding to an equilateral cell of similar form. For triangular and tetrahedral elements, $\theta_{eq} = 60$. For quadrilateral and hexahedral elements, $\theta_{eq} = 90$.

By definition,

$$0 \le Q_{EAS} \le 1$$

where $Q_{EAS} = 0$ describes an equilateral element, and $Q_{EAS} = 1$ describes a completely degenerate (poorly shaped) element.

<u>NOTE</u>: For pyramidal mesh elements, Q_{EAS} is equal to its maximum value for any of the five faces of the mesh element. In an ideal pyramidal mesh element, all four triangular faces are equilateral and the base of the pyramid is a square.

Table 3-1 outlines the overall relationship between Q_{EAS} and element quality.

Table 3-1: Q_{EAS} vs. Mesh Quality

Q_{EAS}	*Quality*
$Q_{EAS} = 0$	Equilateral (Perfect)
$0 < Q_{EAS} \le 0.25$	Excellent
$0.25 < Q_{EAS} \le 0.5$	Good
$0.5 < Q_{EAS} \le 0.75$	Fair

$0.75 < Q_{EAS} \le 0.9$	Poor
$0.9 < Q_{EAS} < 1$	Very poor (sliver)
$Q_{EAS} = 1$	Degenerate

In general, high-quality meshes contain elements that possess average Q_{EAS} values of 0.1 (2-D) and 0.4 (3-D).

f) Gambit tutorial 4.

- As above, but using structured mesh.
- Again, you must make a calculation, what is your maximum boundary layer thickness, what is the first grid length, how many grids point within BL?
- You may setup a boundary mesh over the airfoil for the structured mesh. But this is not compulsory for the coursework.
- You can simply setup a single structured mesh for the coursework, but with a reasonable resolution in the near wall/wake region as above.
- You may want to remind yourself from here, http://courses.cit.cornell.edu/fluent/airfoil/index.htm.