Session 6: Incompressible Simulations

Beth Lee-Rausch

Learning Goals

- What this will teach you
  - How to set up and run steady-state incompressible turbulent simulations
    - Non-dimensionalization
    - Choosing the artificial compressibility parameter
    - Problem set up
    - Visualization
- What you will not learn
  - Rotorcraft (actuator disk), time-accurate: covered in follow-on sessions
  - Inviscid simulations
- What should you already know
  - Basic steady-state solver operation and control
  - Basic flow visualization

Setting

- Background
  - Incompressible capability extension of the compressible formulation- uses same solver mechanics
  - Target is external flow applications – no buoyancy terms
  - Non-dimensionalization is different from the compressible equations
  - Incompressible code is faster and has better convergence than low mach number compressible simulations
- Compatibility
  - Fully compatible for viscous steady-state flows: mixed elements, 2D/3D, time-accurate, adjoint design/ adaptation, actuator disk, SST turbulence model has a bug for incompressible
  - Overset grids and moving grids not currently compatible for incompressible flows
  - Has a limited sub-set of boundary conditions: Standard set is available

Incompressible Equations

- FUN3D uses the method of artificial compressibility (β)
  - See Anderson AIAA 95-1740 (website Publications section)
  - Pseudo-time derivative of pressure added to continuity eq.
  - Temperature and density assumed constant so no energy eq. is solved
  - At steady-state divergence of velocity approaches zero – coupling disappears
  - Steady solutions depends on β through inviscid flux eigenvectors where larger values correspond to increased dissipation (β range 1-15)

$$\frac{\partial(\mathbf{Q}V)}{\partial t} + \oint_{dV} (\mathbf{F}_i - \mathbf{F}_v) \cdot \hat{\mathbf{n}}\, dS = 0 \qquad \mathbf{Q} = [p, u, v, w]^T$$

$$\mathbf{F}_i = \begin{bmatrix} \beta(u - W_x) \\ u(u - W_x) + p \\ v(u - W_x) \\ w(u - W_x) \end{bmatrix} \hat{\mathbf{i}} + \begin{bmatrix} \beta(v - W_y) \\ u(v - W_y) \\ v(v - W_y) + p \\ w(v - W_y) \end{bmatrix} \hat{\mathbf{j}} + \begin{bmatrix} \beta(w - W_z) \\ u(w - W_z) \\ v(w - W_z) \\ w(w - W_z) + p \end{bmatrix} \hat{\mathbf{k}}$$

Nondimensionalization

- Notation: * indicates a dimensional variable, otherwise dimensionless; the reference flow state is ***usually*** free stream ("∞"), but need not be
- Define reference values:
  - L^*_{ref} = reference length of the physical problem (e.g. chord in ft)
  - L_{ref} = corresponding length in your grid (*dimensionless*)
  - ρ^*_{ref} = reference density (e.g. slug/ft^3)
  - μ^*_{ref} = reference molecular viscosity (e.g. slug/ft-s)
  - T^*_{ref} = reference temperature (e.g. ^{o}R, compressible only)
  - a^*_{ref} = reference sound speed (e.g. ft/s, compressible only)
  - U^*_{ref} = reference velocity (e.g. ft/s)
- Space and time are made dimensionless in FUN3D by:
  - $\vec{x} = \vec{x}^* /(L^*_{ref} / L_{ref})$ $\quad t = t^* a^*_{ref} /(L^*_{ref} / L_{ref})$ (compressible) $\quad t = t^* U^*_{ref} /(L^*_{ref} / L_{ref})$ (incompressible)

Nondimensionalization (cont)

- For the ***incompressible-flow*** equations the dimensionless variables are:
  - ρ and T are ***constant*** (no buoyancy terms)
  - $\vec{u} = \vec{u}^* / U^*_{ref}$ so $|\vec{u}|_{ref} = 1$
  - $P = (P^* - P^*_{norm}) / (\rho^*_{ref} U^{*2}_{ref})$

    so the nondimensional free stream pressure is equal to 1

$$P^*_{norm} = P^*_\infty - \rho^*_{ref} U^{*2}_{ref} \qquad C_p = 2(P - 1)$$

- It is possible for the non-dimensional pressure to be negative
- The input Reynolds number has the same definition for both compressible and incompressible flow
- The nondimensional time in FUN3D is related to physical time by:
  - $t = t^* U^*_{ref} (L_{ref}/L^*_{ref})$ (incompressible)
  - *Usually* you have $L_{ref}/L^*_{ref} = 1^*$, but need not - e.g. typical 2D airfoil
  - L_{ref}/L^*_{ref} factor results from Re in FUN3D defined *per unit grid length*

Example 1- Incompressible Turbulent Flow on a Tetrahedral Wing-Body Mesh

- Same case that was used for the compressible demo
- Only need to change &governing_equations namelist and remove mach_number from &reference_physical_properties

```
 ………
 &governing_equations
  viscous_terms = "turbulent"
  eqn_type = "cal_perf_incompress"
  artificial_compress = 5.0
/
 &reference_physical_properties
      gridlength_conversion = 1.00
      dim_input_type = “nondimensional”
      temperature_units = “Rankine”
      reynolds_number = 17410.0
      temperature = 580.0
      angle_of_attack = 0.0
      angle_of_yaw = 0.0
    /
……………………
```

Example 1- Incompressible Turbulent Flow on a Tetrahedral Wing-Body Mesh

- May need to make some modifications to solver parameters

```
&nonlinear_solver_parameters
      time_accuracy        = "steady"
            pseudo_time_stepping = "on"
            schedule_number      = 2
            schedule_iteration   = 1   50
            schedule_cfl         = 10.0  100.0
            schedule_cflturb     =  1.0   35.0
         /
```

- Note that for this very coarse test case the solution is sensitive to the choice of the artificial compressibility factor
- Default artificial compressibility factor is 15

Example 1- Incompressible Turbulent Flow on a Tetrahedral Wing-Body Mesh

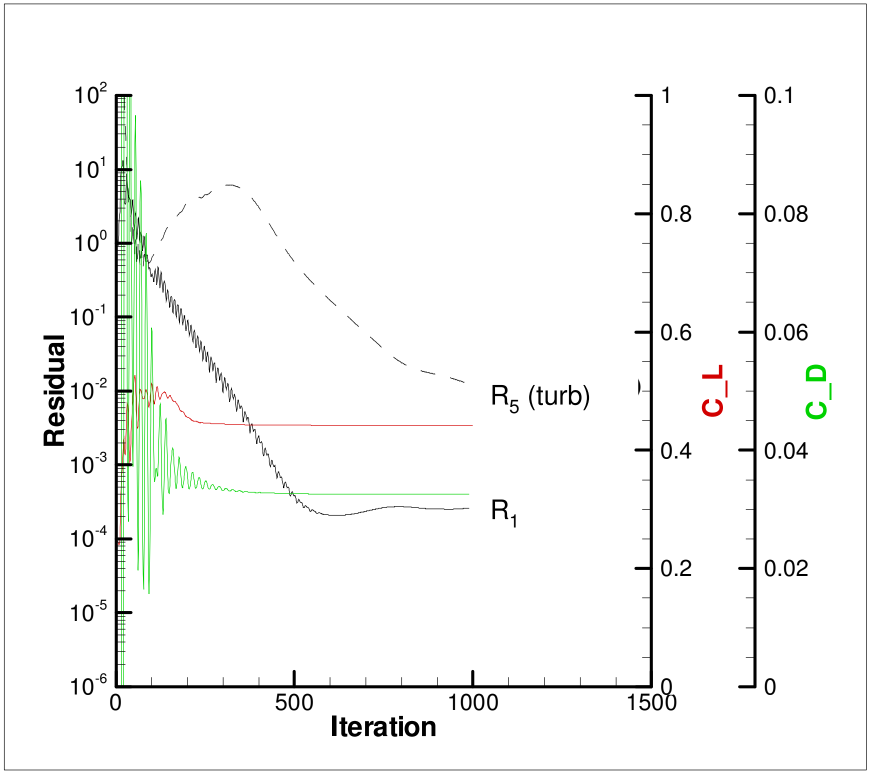
- Execution

```
(mpirun ./nodet_mpi --animation_freq -1 >screen_output)>& error_output
```

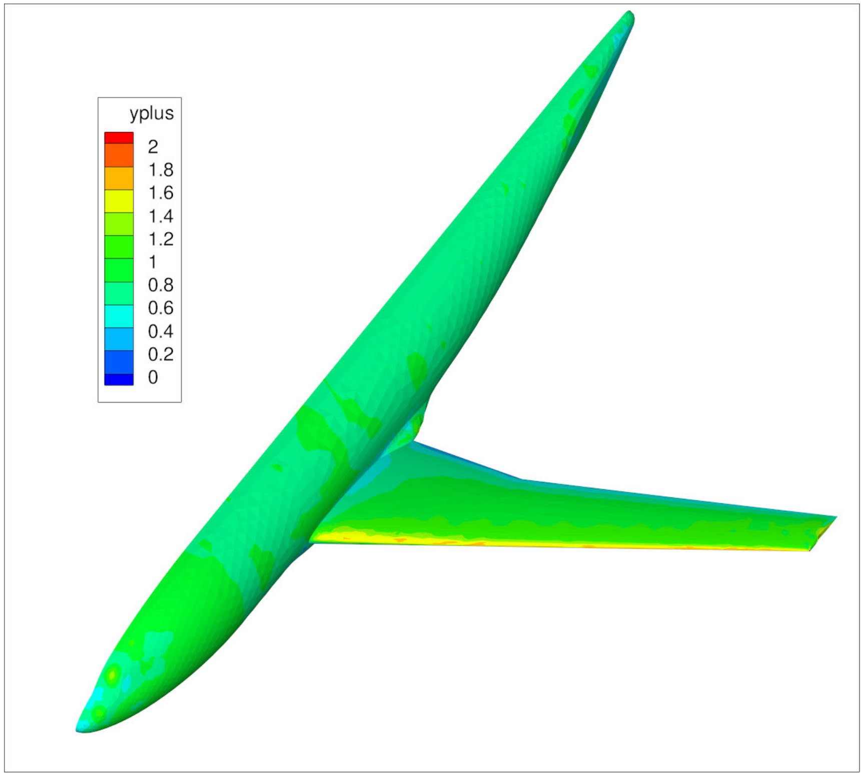
- Did it work?
  - As always, last line or screen output should be: Done.
  - Did the residuals converge (2-3 orders of magnitude reduction)?
  - Are the forces steady(non-oscillatory)? If not, is the variation less than significant in terms of engineering accuracy?
  - Is the yplus < 1.0 on surface?

Example 1 - Incompressible Turbulent Flow on a Tetrahedral Wing-Body Mesh

Residual/Force History
From f6fx2b_trn_hist.tec



Yplus Contours
From f6fx2b_tec_boundary.plt



Troubleshooting

- What if the residuals don't converge?
  - Techniques for the compressible equations should work
    - Increase linear sweeps (can be expensive)
    - Lower CFL numbers
    - Start with some first order iterations
  - Artificial compressibility factor may be too high/low- try altering it
- What if the forces/moments aren't steady?
  - Lower CFL numbers
  - Try restarting solution with a time-accurate computation (constant delta time)
- What if the solution is very sensitive to artificial compressibility factor?
  - Grid may be too coarse
  - Flow may not really be incompressible
  - Try comparing to low-Mach number compressible solution

What We Learned

- Overview of incompressible equations (artificial compressibility factor)
- Non-dimensionalization of incompressible equations
- Turbulent incompressible steady-state wing/body case
  - Input variation from the compressible case
  - Output variable to check
  - Troubleshooting