



Glenn Research Center

---

# CFD Analysis Process



# CFD Analysis Process

- 
1. Formulate the Flow Problem
  2. Model the Geometry
  3. Model the Flow (Computational) Domain
  4. Generate the Grid
  5. Specify the Boundary Conditions
  6. Specify the Initial Conditions
  7. Set up the CFD Simulation
  8. Conduct the CFD Simulation
  9. Examine and Process the CFD Results
  10. Further Analysis?
  11. Report the Findings

*The objective is confidence that the CFD results provide accurate, credible, and useful information.*

# 1) Formulate the Flow Problem

Determine the following:

1. What is the objective?

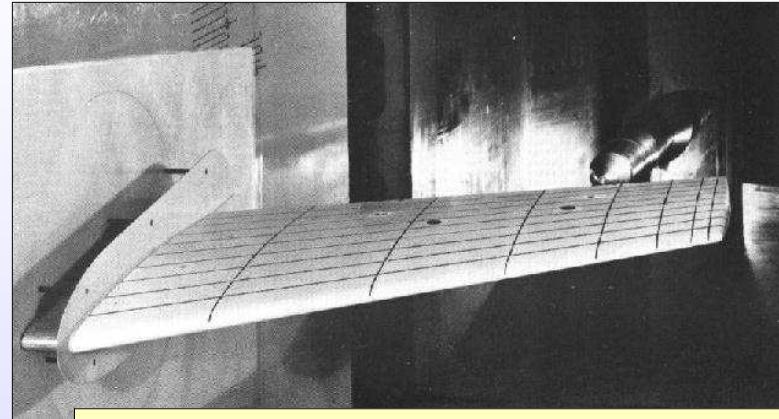
- Engineering quantities
- Performance
- Proof of concept

2. What is known?

- Freestream conditions
- Geometry
- Configuration

3. What analysis approach to use?

- Steady or unsteady flow?
- Are viscous forces important?
- Are shocks present?
- What equations to solve?
- What other flow models are needed?



## ONERA M6 Wing

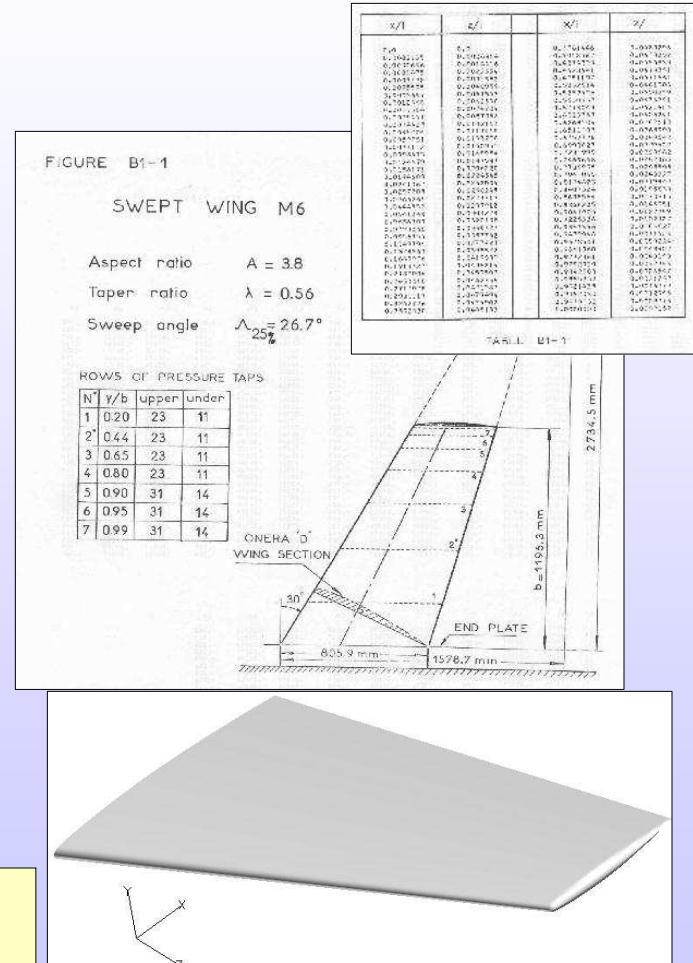
- Section coordinates and wing tip shape
- Tunnel test section:  $M$ ,  $Re$ ,  $T_t$
- Wing:  $\alpha$
- Aerodynamics quantities to determine:
  - Static pressure distributions (loading)
  - Lift and drag coefficients



## 2) Model the Geometry

Determine the following:

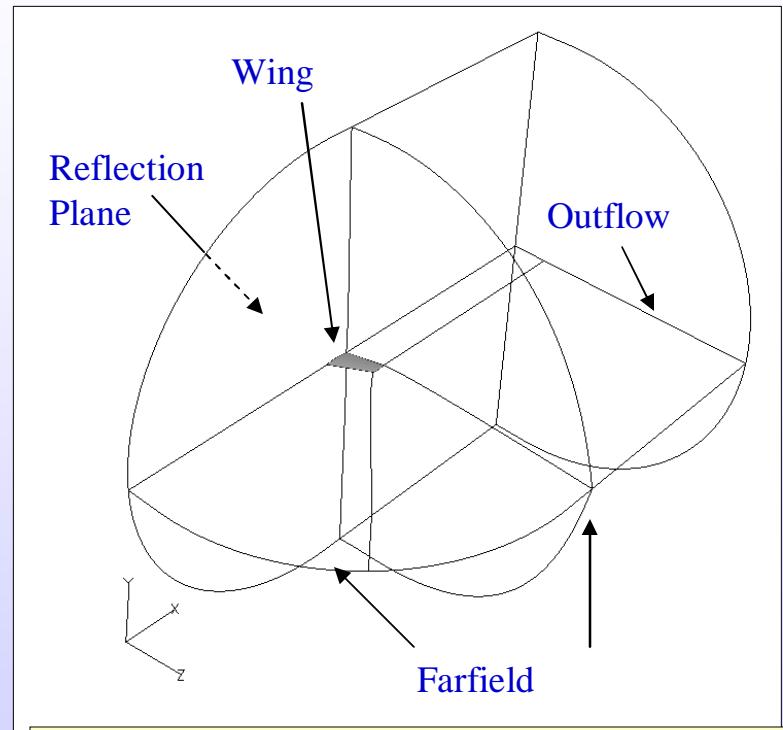
1. What geometric features are significant?  
Steps, leading edges, trailing edges
2. Any simplification of the geometry needed?  
Smooth over small steps
3. What computational format for geometry?  
Lines, curves, surfaces  
CAD file format (IGES, Step, ...)



## 3) Model the Flow Domain

The flow (computational) domain is the control volume (bounded by the control surface) in which the flow field is computed.

- Body is part of surface of flow domain.
- The geometry of the flow domain is also computationally modeled (CAD).
- Shape of the flow domain usually considers the grid topology (this is especially true of structured grids).
- Extent of flow domain depends on choice of boundary conditions.
  - Subsonic flow boundaries need to be farther out since waves travel in all directions.
  - Supersonic flow boundaries can be closer to body if one considers wave motion.
  - Reflection (flow symmetry) planes can effectively reduce the size of the flow domain.



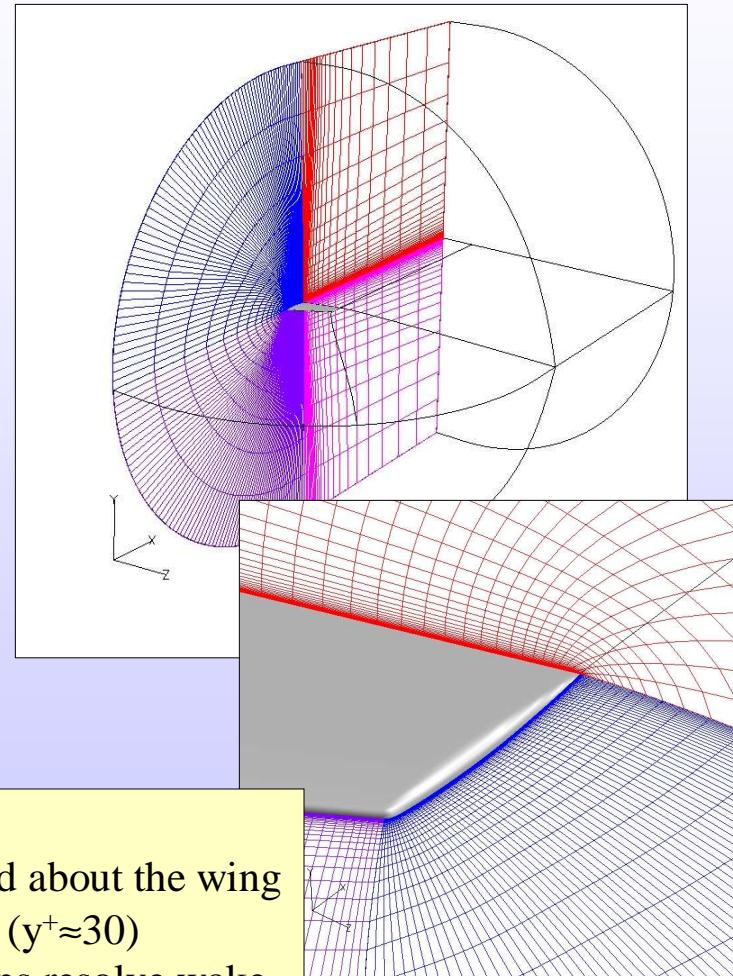
### ONERA M6 Wing

- Wing surface part of domain boundary
- Reflection plane at root of wing
- Farfield boundaries out 15 chord lengths
- Outflow boundary downstream

## 4) Generate the Grid

A grid is generated within the flow domain.

- The grid consists of finite-volume cells on which the CFD equations are approximated.
- The flow domain may be divided into zones for various reasons:
  - Simplify grid generation.
  - Reduce memory requirements.
  - Divide grid for parallel computation.
- Grids can be structured or unstructured.
- Software packages are available for grid generation: Gridgen, ICEM CFD, VGRID.



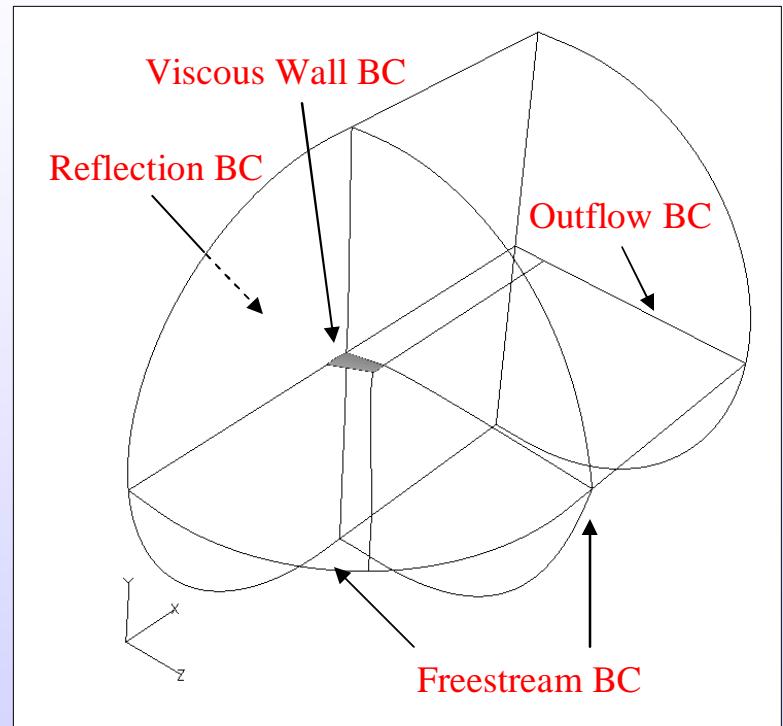
### ONERA M6 Wing

- Single-zone C-grid wrapped about the wing
- Cluster grid normal to wall ( $y^+ \approx 30$ )
- Clustering downstream helps resolve wake
- Stretch grid away from wing (15-20%)

## 5) Specify the Boundary Conditions

Numerical conditions need to be applied at the boundaries of the flow domain and zones.

- Specify types of boundary condition:
  - Viscous (no-slip) wall
  - Inflow / outflow
  - Reflection
  - Coupled
- Zone-to-zone boundaries and overlapped zones require topology and coupling specifications.
- Specification of boundary conditions may be part of grid generation package.
- Additional inputs may be required in the flow code input process (i.e. flow rates, pressures,...).



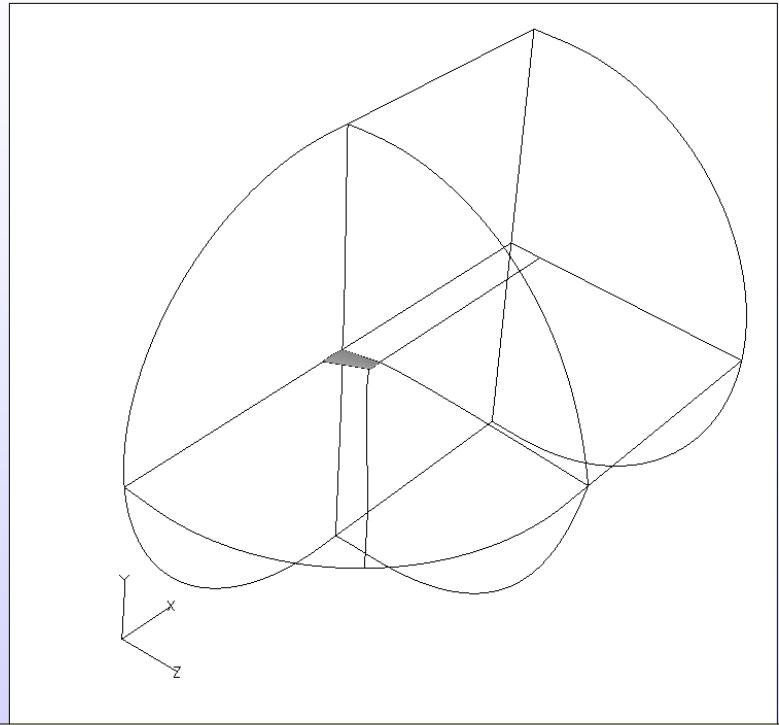
### ONERA M6 Wing

- **Freestream** BC at farfield boundary
- **Outflow** BC at outflow boundary
- **No-slip, adiabatic wall** BC at wing surface
- **Reflection** BC at reflection plane

## 6) Specify the Initial Conditions

Marching numerical methods require a flow field from which to start.

- One choice is to start with a uniform flow field with conditions of the freestream or inflow conditions.
- Initial transients in the flow at the start of the marching may inhibit convergence and perhaps cause the simulation to fail.
- An auxiliary program can be developed to approximate the final solution to enhance convergence.
- An initial solution that satisfies mass conservation helps simulations of internal flows.



**ONERA M6 Wing**

- Initialize flow with tunnel flow conditions



# 7) Set Up the CFD Simulation

The simulation requires several input files:

- Grid file
- Initial solution file
- Input data file
- Auxiliary files (i.e. multi-processor, local boundary conditions, chemistry,...)

## Input Data File

### Reference state

- Freestream flow conditions (Mach, p, T, Re)
- Configuration (angle-of-attack & sideslip)

### Physical model inputs

- Dimensionality (3D, 2D, axisymmetric)
- Flow equations (RANS, PNS, Euler)
- Turbulence model
- Gas model / chemistry

### Numerical algorithm inputs

- Time-marching / space-marching
- Explicit / implicit operators
- Damping schemes
- Convergence acceleration
- Convergence monitoring

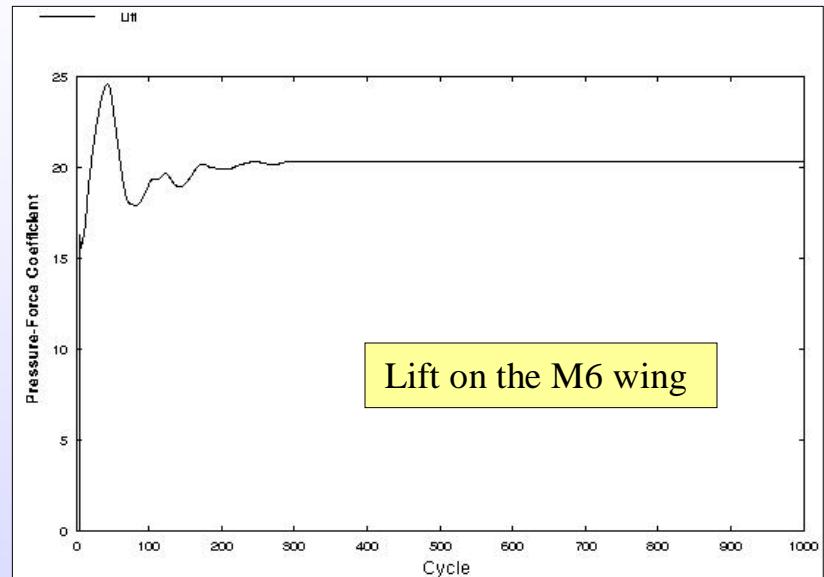
### ONERA M6 Wing

- Common Grid File (`m6wing.cgd`)
- Common Solution File (`m6wing.cfl`)
- Input Data File (`m6wing.dat`)
- Multi-Processor Control File (`m6wing.mpc`)



## 8) Conduct the CFD Simulation

- Simulations typically require CPU times on the order of hours and days.
- Marching methods are monitored to determine iterative convergence.
- Residuals of conservation equations should approach zero as the number of iterations increases.
- Satisfaction of conservation statements (mass, momentum, energy, ...) are also useful for monitoring iterative convergence.
- Convergence to design / performance quantity (lift, drag, recovery) is often a critical test for iterative convergence.



### ONERA M6 Wing

Monitored the residuals, as well as, the lift and drag on the wing with number of iterations.

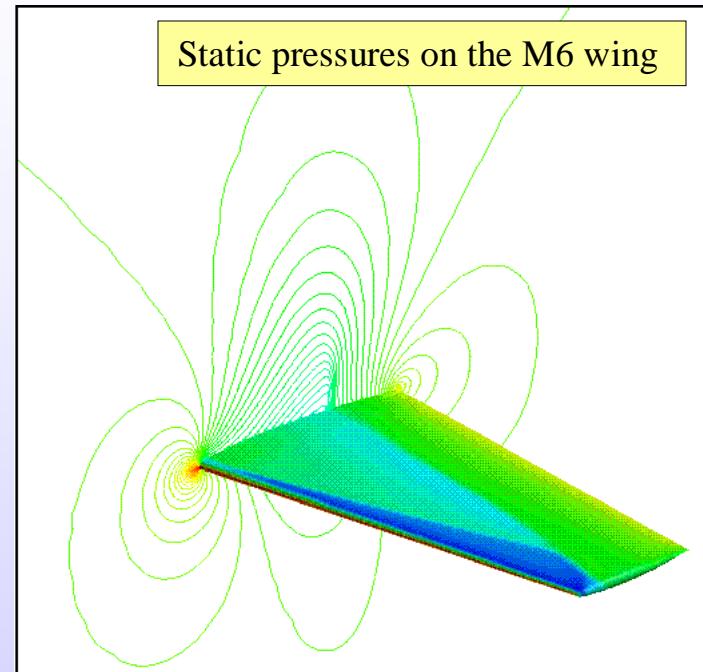
## 9) Examine and Process the CFD Results

### Visualization:

- View flow properties (Mach, pressure, vectors) to get overall view of flow ([CFD: Colorful Fluid Dynamics](#)).
- Various packages: Fieldview, Ensight, TecPlot, CFPOST.

### Solution Processing:

- Extract data (lift, drag, recovery, spillage, etc...) useful for iterative convergence monitoring and engineering design (CFPOST).





# 10) Further Analysis?

Once a simulation has reached iterative convergence and the results examined and processed, there may be various reasons to make changes and continue with another simulation:

- Change of physical model parameters to examine sensitivity.
  - Turbulence model / parameters
  - Gas / chemistry model
- Change of numerical algorithm parameters to examine sensitivity.
  - Implicit or explicit method
  - Time step parameters
  - Numerical flux parameters
- Change or refine the grid to examine grid sensitivity.
- Change the geometry as part of design parametric study.
- Change initial solution to examine iterative convergence.



## 11) Report the Findings

CFD results, like any other data, should be reported along with some idea of the level of error that it contains and indications of how much confidence one has in the data.

- What are engineering results and uncertainty of those results?
- How much error is there in the iterative convergence?
- How much error is there in the grid convergence?
- How sensitive are the results to model parameters (turbulence, etc...)
- How sensitive are the results to algorithm parameters (CFL, etc...)
- How do the results compare to similar experimental or theoretical data?
- Statistical analysis may be useful for reporting such information.