Meshing Challenges for Applied Aerodynamics

Jeffrey Slotnick
6 January 2019
Overview

- RANS CFD is used to help **design the aerodynamic shape** of virtually every external component of a commercial airplane.

- **Accuracy of CFD simulations** drives the extent to which the methods are used, and is a function of several factors:
  - Appropriate geometric modeling
  - Choice of relevant physical models
  - Flow modeling

- Flow modeling includes choice of **mesh discretization**, numerical algorithms, and boundary conditions to solve the governing PDEs.

- The mesh must **properly resolve the relevant flow physics** for a **given geometric model** at a **specific flow/flight condition** at a **desired accuracy level**.
Current Meshing Practices

- Fairly well established procedures in place to build **fixed meshes** around complex geometry
  - Various types (unstructured, structured-overset, hybrid, etc.)
  - Various toolsets (with varying degrees of automation)
  - Resolution set by educated guess and best practices (developed from growing body of data from internal/external studies)
  - Typically one mesh used for range of flight conditions
  - Mesh refinement study seldom done in practice

- Emerging methods and procedures in development to build **meshes that adapt to the flow solution**
  - Considerable promise to improve accuracy for a given number of points, reduce engineer time, and lower level of user expertise needed
  - Several algorithms being evaluated for speed, robustness, effectiveness
  - Issues: surface geometry interface, robustness, mesh growth, strength of solver on irregular grids, etc.
  - Generally not in production use, but not many years away

**Fixed grids** continue to be the workhorse for CFD modeling of realistic geometry
Meshing Challenges

- Handling **geometric complexity** (large range of scales, components in close proximity, mesh control/quality in tight regions, etc.)
- Resolving **pertinent flow features** (unknown location and importance, varies with run flow conditions, etc.)
- Improving **speed** and **robustness**
- Better understanding of the **dependency of grid resolution on physical modeling**
- Better understanding of the **switch from RANS to turbulence-resolving simulations**
Specific Challenge – High Lift

- Accurate prediction of **low-speed maximum lift** enables stall speed determination:
  - Sets approach speed
  - Allows for more efficient high-lift systems with lower weight and cost
  - Allows for large potential reduction of flight test conditions during Certification

---

Aerodynamic and structural design and performance must consider the full flight envelope

Regions where CFD is typically best calibrated and most productive

![Diagram showing flight conditions and performance](image)
HiLiftPW-3 Experience

- From HiLiftPW-3, oil flow images from testing of the JAXA JSM configuration near stall conditions identified regions of separated flow behind several slat brackets.

- Many RANS CFD simulations predicted much larger “pizza slice” shaped separation patterns appearing at differing angles-of-attack, with high sensitivity to turbulence models and mesh resolution.

\[
\alpha = 21.57^\circ
\]

Test Problem

- Extracted segment from JAXA JSM model with constant chord high lift elements (slat, main, flap) with single slat bracket

Medium Mesh
Preliminary CFD Solutions on Model Problem

Grid dependence on Extrusion Grid – SA solutions

\[ \alpha = 15^\circ \]

**Medium Grid on Extruded Case**

\[ \alpha = 17^\circ \]

No “pizza slices” predicted.

**Coarse Grid on Extruded Case**

\[ \alpha = 15^\circ \]

Solutions look very similar at \( \alpha = 15^\circ \).

\[ \alpha = 16^\circ \]

Coarse grid solution separates behind the bracket at \( \alpha = 16^\circ \), generating the “pizza slice”.

\[ \alpha = 17^\circ \]

Courtesy: N. Moffitt (Boeing)
Preliminary CFD Solutions on Model Problem

Effect of turbulence model on Extrusion Grid – Medium mesh

**SA, \( \alpha=19.8^\circ \)  
SA-RC-QCR, \( \alpha=19.0^\circ \)**

No clear effect of curvature/rotation/nonlinear correction terms on flow field, even though the footprint of the bracket begins as a necklace vortex

Courtesy: N. Moffitt (Boeing)
Observations

- Current mesh resolution for high-lift CFD (as utilized in HiLiftPW-3) appears insufficient to (a) accurately assess physical modeling errors, and (b) to clearly identify the better turbulence model for RANS simulations for full configurations.

- Need a systematic study of mesh resolution/convergence (fixed grid, adaptive) using solvers that can utilize a wide range of turbulence modeling (RANS, hybrid RANS/LES, WMLES, etc.) on key test cases to clearly isolate numerical –vs- physical modeling errors.

- The extruded, single-slat-bracket periodic model problem appears relevant, but the question of whether the “pizza slices” are an artifact of insufficient numerical resolution has not been answered (as they seem to be a robust feature, across codes and models), and turbulence-resolving simulations have not begun.