Summary Questionnaire for Participants

1st AIAA Geometry and Mesh Generation Workshop (GMGW-1)

The purpose of this document is to collect data for an assessment of the current state of the art in mesh (grid) generation for a variety of mesh types and a variety of software tools. The comparisons will be made in terms of the quality of the mesh (either from a priori metrics or from the quality of the CFD solutions that were produced using the mesh) as well as the resources (human and computer) required to generate the meshes.

For GMGW-1, the geometry and meshes referred to below are for the NASA High Lift Common Research Model (HL-CRM).

* Geometry: <http://hiliftpw.larc.nasa.gov/Workshop3/geometries.html>
* Meshing Guidelines: http://hiliftpw.larc.nasa.gov/Workshop3/GriddingGuidelines-HiLiftPW3-v10.pdf

Completion of this questionnaire is required of all participants in GMGW-1 and participants in the 3rd High Lift Prediction Workshop (HiLift-PW3) who generate their own meshes (versus using the supplied baseline meshes). A separate copy of this Questionnaire should be completed for each family of meshes.

# Geometry

1. Software
   1. What software tool(s) did you use to import and prepare the HL-CRM geometry model for meshing? ANSA for conversion from STEP to surface triangulation, Chimera Grid Tools for overset structured grid generation.
2. Import
   1. Did you use the geometry in its primary (i.e. native CAD) format? If not, why?

I don’t have a reader for this file format

It didn’t import at all/fully/properly

Other: I wanted to test the fidelity of the geometry files in STEP format

* 1. Did you use one of the alternate formats? If so, which one? STEP
  2. What is your preferred geometry model format, and why?

No Preference

* 1. If you used neither the primary nor alternate CAD format, how did you convert the geometry model to something you could read?
  2. What problems, if any, did you identify immediately after import of the CAD file (eg, missing geometry, poorly translated geometry, other)? None
  3. What was required level of user expertise (novice, intermediate, expert) for this task? intermediate
  4. How long did import take (both elapsed time and labor required --- in hours)? 0.1

1. Preparation for meshing
   1. What steps did you take after import to make the geometry model ready for meshing? (Choose all that apply.)

Layering (hiding components)

Simplification/defeaturing (removing components)

Repair (fixing/recreating components that didn’t import properly)

Modification (changing components)

Shrink-wrapping

Other Create fine surface triangulation representation of geometry and a set of discretized surface curves that captures all surface features as starting point for structured overset surface grid generation

* 1. How much effort (in elapsed time and labor hours) was required in preparing the geometry? 3.75 hrs.
  2. What kind of computer resources were required (eg, RAM, disk)? Not easily measureable but very light on RAM and disk.
  3. Did you have to re-work the geometry model after you started meshing? Why and how? No, but I did explore different surface triangulation generation algorithms in ANSA.

# Initial Meshing

1. What type of mesh did you generate?

Structured multi-block

Unstructured tetraherda

Unstructured hexahedra

Hybrid

X Overset Structured

Cartesian

other (please specify      )

1. Surface Meshing
   1. How (i.e. type of technique, name of software) Algebraic (transfinite interpolation) and hyperbolic (pde) methods, Chimera Grid Tools
   2. How long did it take (elapsed time and labor – in hours)? 56 hrs. (medium mesh)
   3. What computer resources were required (kind of computer, RAM usage, # cores, CPU, disk, …) Desktop Linux workstation, less than 1GB RAM, single CPU, negligible disk space.
   4. Provide a general description of how mesh resolution was specified (explicit user inputs, sources, curvature-based sizing, background distribution function, etc.) User input as global parameters to a grid generation script, e.g. maximum stretching ratio, maximum interior grid spacing, curve end point spacing based on surface feature angle, hyperbolic tangent stretching function.
   5. Was the size of the surface mesh dictated by the CFD solver’s requirements, limited by time available, limited by available computer resources, or something else? No limitations, only dictated by workshop meshing guidelines.
   6. How many cells and of what types? (Provide data for each mesh in the family) 0.26 x 10^6 (coarse), 0.50 x 10^6 (medium), 1.01 x 10^6 (fine), 2.05 x 10^6 (extra fine), all are structured quadrilateral cells.
   7. How many nodes? (Provide data for each mesh in the family) 0.27 x 10^6 (coarse), 0.51 x 10^6 (medium), 1.03 x 10^6 (fine), 2.08 x 10^6 (extra fine).
2. Volume Meshing
   1. How (i.e. type of technique, name of software) Hyperbolic for near-body volume meshes and Cartesian for off-body volume meshes, both using Chimera Grid Tools. For domain connectivity, a distance-based hole cut approach was employed using Chimera Components Connectivity Program (C3P).
   2. How long did it take (elapsed time and labor – in hours)? 4.5 hrs. for volume mesh labor time, 1.2 hrs. for connectivity labor time, about 4 minutes for volume mesh generation computing time, under 3 minutes for connectivity computing time (medium mesh)
   3. What computer resources were required (RAM usage, # cores, CPU, disk, …) Less than 2GB RAM and single CPU for volume mesh; 13GB RAM and 24 OpenMP threads for connectivity, less than 2.4GB disk storage total for medium mesh file and connectivity data file.
   4. Provide a general description of how mesh resolution was specified (explicit user inputs, sources, curvature-based sizing, background distribution function, etc.) User input as global parameters to a grid generation script, e.g. maximum wall normal stretching ratio, wall normal grid spacing, hyperbolic tangent stretching function in normal direction to surface. Off-body Cartesian mesh spacing automatically matched to near-body curvilinear mesh outer boundary spacing. Stretched layers are added in all directions with prescribed stretching ratio.
   5. Was the size of the volume mesh dictated by the CFD solver’s requirements, limited by time available, limited by available computer resources, or something else? No limitations, only dictated by workshop meshing guidelines.
   6. How many cells and of what types? (Provide data for each mesh in the family) 22.99 x 10^6 (coarse), 63.17 x 10^6 (medium), 184.52 x 10^6 (fine), 553.91 x 10^6 (extra fine), all hexahedral cells.
   7. How many nodes? (Provide data for each mesh in the family) 23.95 x 10^6 (coarse), 65.05 x 10^6 (medium), 188.59 x 10^6 (fine), 564.73 x 10^6 (extra fine).
3. Adherence to HiLift-PW3 meshing guidelines
   1. To what extent did your mesh(es) adhere to the HiLift-PW3 meshing guidelines? All except for trailing edge grid spacing and multi-griddable number of points. Since an O-mesh topology is used here, the trailing edge spacing is chosen to ensure grid spacing continuity based on the blunt trailing edge face grid spacing (dictated by the number of points to be used on this face by the meshing guidelines). A multi-griddable number of points is not used since the OVERFLOW flow solver does not have this restriction.
   2. Was it possible to adhere to the guidelines on the first attempt, or were there iterations involved? It took a couple of iterations to ensure all relevant guidelines are satisfied.
   3. What were the reasons that you did not adhere to the guidelines? (chose all that apply)

The guideline does not pertain to the type of mesh generated

The guidelines were (locally) inconsistent and therefore could not all be satisfied

The tool that was used does not give enough control to adhere to the guideline

Adhering to the guideline would have required more resources than were available

x The guidelines were not appropriate for the CFD solver being used

x Other Guidelines do not follow best practice for O-mesh topology at trailing edge

1. A priori metrics (such as skew, or maximum stretching ratio, or …)
   1. What a priori metrics did you apply on the initial mesh? Prescribed max stretching ratio (local stretching ratio at concave corners may exceed max prescribed in order to march grid out of tight corners). Positive cell Jacobian as computed by the target flow solver OVERFLOW.
   2. What was the average and range of the metrics? The range of the local stretching ratio is about 1.05 to 1.3. I do not have a utility to measure the average currently but my estimate is that it is very close to 1.2.
   3. Did the a priori metrics point out any problems that needed to be fixed? If so, which metric and how many times did you need to re-mesh? It tooks a few tries to eliminate negative cell Jacobians on a few difficult meshes with tight concave corners and/or saddle points. Fortunately, CPU time for each attempt was small ( 0.1 - 0.25 minute).
2. Were there any additional best practices that you used in generating the meshes? I enforced continuity of grid spacing around all sharp surface features. Attempts were made to ensure grid spacings of neighboring grids are comparable in overlapped regions.
3. What was the required level of user expertise (novice, intermediate, expert) for this task? An expert in overset grid generation created these grids, but an expert is not required for this task. However, lower levels of expertise typically require longer labor time.

# Adaptive Meshes and/or Customized Meshes

1. What adaptive meshing strategy did you use (technique and software)? Not attempted yet but is available in the OVERFLOW flow solver.
2. What criteria was used for mesh adaptation (e.g., pressure, vorticity, …)?
3. What were the relative sizes of the baseline and adapted meshes?
4. Do you have any quantitative results (from the CFD) as to the benefit of adaptation?
5. To what extent does the adapted mesh adhere to the meshing guidelines?

# I/O

1. How long did it take to export the mesh? A few seconds (coarse, medium), 0.5 – 1.0 minute (fine ,extra fine).
2. To what format? Solver native? Or CGNS? PLOT3D
3. How big is the final volume mesh file (MB)? (Provide data for each mesh in the family) coarse (549 MB), medium (1561 MB), fine (4526 MB), extra fine (13553 MB)

# Mesh Families

1. What strategy did you use to generate the family of meshes (coarse, medium, fine, extra-fine)? That is, did you generate the coarse mesh first and refine it, or did you start each mesh generation task essentially from the beginning? A script was developed to perform the grid generation process where the resolution level (coarse, medium, fine, extra fine) is an input parameter. Various grid generation parameters are automatically adjusted based on the resolution level. Each member of the family is generated independently of each other. The grid script was first developed for the medium mesh. Adjustments were then made to include other refinement levels.
2. In your opinion, what was the most time-consuming or tricky aspect of generating a family of meshes? Maintaining an optimal number of overlapping grid points between neighboring meshes and adjusting smoothing parameters for hyperbolic marching scheme for meshes with concave corners at the finer levels.
3. How did the times (labor, CPU, etc.) needed to generate them compare? The grid script for the medium level was developed first and took 68.6 hrs. Smaller additions/modifications were then made to the script to handle the other levels and hence the development times for these levels were not as long: coarse (10 hrs.), fine (17.75 hrs.), extra fine (12.5 hrs.). The wall clock computing time to generate one instance of each mesh level is given next: coarse (just under 5 minutes), medium (just under 10 minutes), fine (just over 20 minutes), extra fine (about 73 minutes). This time includes both grid generation and domain connectivity computing time.
4. Were there any problems that you encountered in one mesh resolution that you did not encounter in another resolution? The coarse level mesh did not have sufficient overlap between neighboring grids in some regions and adjustments had to be made to the marching distances. The finer levels do not have this problem if we let the meshes have more overlap than is needed. Some adjustments were made to reduce excessive overlap. Also, smoothing parameters had to be adjusted for meshes with deep concave corners at the fine and extra fine levels.
5. Did you make any further modifications to the mesh(es) before first trying to generate a flow solution? No.

# Post Solution Mesh Modifications

1. After generating an initial flow solution, were additional mesh modifications made to improve solver convergence or solution accuracy? After the initial flow solution was computed, it was discovered that part of the hole boundary for the far field box grid came quite close to the geometry surface in a couple of local regions. This resulted in slow convergence of the far field box grid but did not appear to affect the surface loads. The error in the auto box grid blanking scheme was then fixed and the mesh regenerated.
2. Describe any post solution mesh modifications that were made. Same as 1.
3. How long did these modifications take (elapsed time and labor – in hours)? 6 hrs.