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).


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
   a. What software tool(s) did you use to import and prepare the HL-CRM geometry model for meshing?

2. Import
   a. 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:
   b. Did you use one of the alternate formats? If so, which one?
   c. What is your preferred geometry model format, and why?
   d. If you used neither the primary nor alternate CAD format, how did you convert the geometry model to something you could read?
   e. What problems, if any, did you identify immediately after import of the CAD file (eg, missing geometry, poorly translated geometry, other)?
   f. What was required level of user expertise (novice, intermediate, expert) for this task?
   g. How long did import take (both elapsed time and labor required --- in hours)?

3. Preparation for meshing
   a. 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

b. How much effort (in elapsed time and labor hours) was required in preparing the geometry?
c. What kind of computer resources were required (eg, RAM, disk)?
d. Did you have to re-work the geometry model after you started meshing? Why and how?

Initial Meshing
1. What type of mesh did you generate?
   - Structured multi-block
   - Unstructured tetrahedra
   - Unstructured hexahedra
   - Hybrid
   - Overset
   - Cartesian
   - Other (please specify)

2. Surface Meshing
   a. How (i.e. type of technique, name of software)
   b. How long did it take (elapsed time and labor – in hours)?
   c. What computer resources were required (kind of computer, RAM usage, # cores, CPU, disk, …)
   d. Provide a general description of how mesh resolution was specified (explicit user inputs, sources, curvature-based sizing, background distribution function, etc.)
   e. 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?
   f. How many cells and of what types? (Provide data for each mesh in the family)
   g. How many nodes? (Provide data for each mesh in the family)

3. Volume Meshing
   a. How (i.e. type of technique, name of software)
   b. How long did it take (elapsed time and labor – in hours)?
   c. What computer resources were required (RAM usage, # cores, CPU, disk, …)
d. Provide a general description of how mesh resolution was specified (explicit user inputs, sources, curvature-based sizing, background distribution function, etc.)

e. 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?

f. How many cells and of what types? (Provide data for each mesh in the family)

g. How many nodes? (Provide data for each mesh in the family)

4. Adherence to HiLift-PW3 meshing guidelines
   a. To what extent did your mesh(es) adhere to the HiLift-PW3 meshing guidelines?

   b. Was it possible to adhere to the guidelines on the first attempt, or were there iterations involved?

   c. 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
      - The guidelines were not appropriate for the CFD solver being used
      - Other

5. A priori metrics (such as skew, or maximum stretching ratio, or ...)
   a. What a priori metrics did you apply on the initial mesh?

   b. What was the average and range of the metrics?

   c. 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?

6. Were there any additional best practices that you used in generating the meshes?

7. What was the required level of user expertise (novice, intermediate, expert) for this task?

Adaptive Meshes and/or Customized Meshes

1. What adaptive meshing strategy did you use (technique and software)?

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?

2. To what format? Solver native? Or CGNS?

3. How big is the final volume mesh file (MB)? (Provide data for each mesh in the family)
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?
2. In your opinion, what was the most time-consuming or tricky aspect of generating a family of meshes?
3. How did the times (labor, CPU, etc.) needed to generate them compare?
4. Were there any problems that you encountered in one mesh resolution that you did not encounter in another resolution?
5. Did you make any further modifications to the mesh(es) before first trying to generate a flow solution?

Post Solution Mesh Modifications

1. After generating an initial flow solution, were additional mesh modifications made to improve solver convergence or solution accuracy?
2. Describe any post solution mesh modifications that were made.
3. How long did these modifications take (elapsed time and labor – in hours)?