GMGW-1 Participant Questionnaire

1st AIAA Geometry and Mesh Generation Workshop

The purpose of this document is to collect data for an assessment of the current state of the art in mesh generation for a variety of mesh types and a variety of software tools. The comparisons will be made in terms of the quality of each mesh submitted (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?

    SimModeler by Simmetrix Inc.

2. Import & Preparation for meshing
    a. Which of the supplied geometry files did you use:
        - ☐ Native: NX (prt) file (HL-CRM gapped config)
        - ☐ CREO file (HL-CRM gapped config)
        - ☐ IGS file (HL-CRM gapped config)
        - ☐ STP file (HL-CRM gapped config)
        - ☒ Parasolid (x_t) (HL-CRM gapped config)
        - ☐ Native: NX (prt) file (HL-CRM partially-sealed config)
        - ☐ CREO file (HL-CRM partially-sealed config)
        - ☐ IGS file (HL-CRM partially-sealed config)
        - ☐ STP file (HL-CRM partially-sealed config)
        - ☐ Parasolid (x_t) (HL-CRM partially-sealed config)

    b. What problems, if any, did you identify immediately after importing the geometry model (eg, missing geometry, poorly translated geometry, other)?

    None. We selected the Parasolid model based on the fact that the original model was presumably created in NX and Parasolid is the kernel in NX, thus avoiding translation issues.

    c. What steps did you take after import to make the geometry model ready for meshing? (Choose all that apply)
        - ☐ None
        - ☐ Layering (hiding components)

☒ Simplification/defeaturing (removing components)
☐ Repair (fixing/recreating components that didn't import properly)
☐ Modification (changing components)
☐ Shrink-wrapping
☒ Other

Added a box around the model. Combined some edges to simply topology where possible. Scribed model edge along leading edges to make it easier to specify mesh grading in these areas.

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

e. How long did import take (both elapsed time and labor required --- in hours)?

2

Initial Meshing

1. What type of mesh family did you generate?

☐ Structured multi-block

☒ Unstructured tetrahedra

☐ Unstructured hexahedra

☐ Hybrid

☐ Overset

☐ Cartesian

☐ other (please specify)

2. Surface Meshing

a. What software tool(s) did you use to generate your initial surface mesh?

SimModeler by Simmetrix Inc.

b. How long did it take (elapsed time and labor – in hours)?

Approximately 20-25 hours

c. Provide a brief description of how mesh resolution was specified (explicit user inputs, sources, curvature based sizing, background distribution function, …)

Most of the mesh resolution was specified using user inputs. The surface mesh feature a combination of isotropic elements for which the mesh size was manually specified, 2-D boundary layers extending from edges onto faces for which the size of the first layer and the gradation factor were manually specified, and 1-D boundary layers with the same specifications. On some faces with significant curvature (slat and flap coves for example), curvature based refinement was applied.

d. When/how did you judge surface mesh generation to be complete?

No particular metrics were used to determine the state of the surface mesh. Visual inspection and comparisons to the committee provided grids were performed to determine when the mesh was complete.

3. Volume Meshing
    a. What software tool(s) did you use to generate your initial volume mesh?

       SimModeler by Simmetrix Inc.

    b. How long did it take (elapsed time and labor – in hours)?

       4 hours

    c. Provide a brief description of how mesh resolution was specified (explicit user inputs, sources, curvature based sizing, background distribution function, …)

       The volume mesh was again specified with user inputs. Refinement boxes were also manually specified to refine the mesh elements closer to the wing. The 3-D boundary layers were manually given a first layer height and a gradation factor and allowed to grow until the elements were isotropic (unless they ran into another boundary layer)

    d. For resolving surface boundary layers, what cell size growth rate did you use? Was it constant or variable? If variable, describe.

       A constant growth rate of 1.25 was used.

    e. When/how did you judge volume mesh generation to be complete?

       Have not yet run a simulation. Likely will modify some settings at that point in time.

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

       All the meshing guidelines were adhered to.

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

       Guidelines adhered to on first attempt.

    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 tools used do 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 (describe):

5. A priori metrics (such as skew, or maximum stretching ratio, maximum deviation of mesh nodes from OML 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?

    Other than the guidelines provided by the committee, experience regarding high-lift flows was used to guide the design of some aspects of the mesh. The flap and slat coves contain regions of separated, recirculating flow, thus the elements in those regions were kept fairly isotropic, giving a fine resolution in the spanwise direction. The end caps of the slats and flaps were also refined and were represented with isotropic elements with curvature refinement towards the leading edge to better represent the vortices created at those corners. The wing-root juncture was refined and given isotropic elements to better capture the root vortex.

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

    Intermediate to expert. I would say expertise in aerodynamics and CFD might be equally as important as expertise in mesh generation, and the quality of the mesh that can be generated is heavily dependent on both.

Adaptive Meshes (Only answer if you generated an adapted mesh)

1. What adaptive meshing strategy did you use (technique and software)?
2. What criteria were used for mesh adaptation (e.g., pressure, vorticity, …)?
3. What, if any, further treatments (e.g. smoothing) were applied? (Please describe)

Mesh Families

1. What strategy did you use to generate the family of meshes (coarse, medium, fine, extra fine)? For example, did you generate the coarse mesh first and refine it, or did you start each mesh generation task essentially from the beginning?

    We first developed the settings for the medium mesh and then copied and modified those for the coarse mesh.

2. In your opinion, what was the most time-consuming or tricky aspect of generating a family of meshes?

    Nothing in particular.

3. How did the times (labor, CPU, etc.) needed to generate them compare?

    Once the settings were developed for one mesh, creating settings for the second mesh was just a matter of changing some numbers, so was much quicker (5-10x faster).

4. Were there any problems that you encountered in one mesh resolution that you did not encounter in another resolution?

Post-Solution Mesh Modifications

1. After generating an initial flow solution, where 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)?

I/O

1. In which format did you export your meshes? (CGNS, Solver-native, …):

   Ugrid.

2. What are the names of the files you uploaded to the GMGW-1 server?

   UCBoulder-Simmetrix-*.ugrid.gz

Miscellaneous

1. Are there any other aspects of your HL-CRM mesh generation experience that you would like to draw our attention to?