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?
    Siemens PLM Software STAR-CCM+
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)? **The imported tessellation showed some curvature spikes in the CAD faces, which were cleaned up in STAR-CCM+'s CAD repair capabilities. Additionally, two self-intersected geometry faces were fixed manually.**

    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

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

e. How long did import take (both elapsed time and labor required --- in hours)? **0.5 hours**

Initial Meshing

1. What type of mesh family did you generate?

☐ Structured multi-block

☐ Unstructured tetrahedra

☐ Unstructured hexahedra

☐ Hybrid

☐ Overset

☐ Cartesian

☒ other (**1. Unstructured polyhedra core volume cells with prismatic hexahedra cells for the near-walls; 2. Unstructured Cartesian Hexahedra with prismatic hexahedra cells for near-walls**)

2. Surface Meshing

a. What software tool(s) did you use to generate your initial surface mesh? **Siemens PLM Software STAR-CCM+**

b. How long did it take (elapsed time and labor – in hours)? **2 hours**

c. Provide a brief description of how mesh resolution was specified (explicit user inputs, sources, curvature based sizing, background distribution function, …) **Curvature based sizing for the leading-edges of wings, flaps and slats; surface sizes (target and minimum) for the overall geometry; Edge-proximity sizes to space the number of faces between the two sharp edges of the blunt trailing-edges on the wings, flaps and slats; Proximity-based sizing for thin areas of the geometry (near the trailing-edges); CAD projection throughout the model for geometrical fidelity.**

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

The surface mesh generation was considered complete when the surface mesh reflected the gridding guidelines provided for this workshop. Note that the core volume meshing tools in STAR-CCM+ are generated off this refined surface mesh and are conformal to it. Therefore, the volume meshing guidelines also served to solidify the surface mesh requirements.

3. Volume Meshing

a. What software tool(s) did you use to generate your initial volume mesh? **Siemens PLM Software STAR-CCM+**

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

Mesh type	Coarse	Medium	Fine
Unstructured Cartesian Hexahedra (Trimmer) + Prismatic near-wall	**0.5**	**1**	**2**
Unstructured Polyhedra + Prismatic near-wall	**5.3**	**6.5**	**8.2**

c. Provide a brief description of how mesh resolution was specified (explicit user inputs, sources, curvature based sizing, background distribution function, …) **Automated wake refinement behind flaps(For Unstructured Polyhedra), slats and wing; Volume sizing boxes for the intermediate sizing between near-surface wake refinement and far-field cells; Automated near-wall prismatic meshing with near-wall thickness of layers specified; Adaptive prismatic cell layer thicknesses; Surface-based cell size specifications. Completely automated and repeatable setup.**

d. For resolving surface boundary layers, what cell size growth rate did you use? Was it constant or variable? If variable, describe. **We used a variable thickness distribution in the surface boundary layers, based on Geometric Progression from a specified near-wall thickness of the first layer of cells (based on gridding guidelines) to a specified overall thickness of the boundary layer cells. Also, the total number of layers were specified. Growth rate was auto-computed based on the above inputs.**

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

The volume mesh generation was considered complete when the desired level of local refinements (around the slats, behind flaps, etc.) in the volume mesh (based on generally accepted refinement principles from the solver) were satisfied.

4. Adherence to HiLift-PW3 meshing guidelines

a. To what extent did your mesh(es) adhere to the HiLift-PW3 meshing guidelines? **We satisfied all meshing guidelines, except for the refinement on the trailing-edges of the flaps/slats/wings. We reduced the TE refinement to 4-6-8 faces instead of 4-8-12, on account of our isotropic meshing tools.**

b. Was it possible to adhere to the guidelines on the first attempt, or were there iterations involved? **All implementations of the guidelines (with changes as described above) were implemented in the first attempt within STAR-CCM+ automated meshing pipeline.**

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): **Adhering to the guidelines would result in high cell count for the mesh. This is as a result of the isotropic surface mesh refinements**

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? **We applied metrics for minimum face quality (> 0.05), Maximum cell size [70 m], Volume Change [0-0.1], checks for negative volume cells, checks for zero-area faces, and Face Validity [0.5-1]. Cell quality was optimized using an automated optimizer for a range of [0.4-1]**
   b. What was the average and range of the metrics? **Minimum face quality was forced to be 0.05, with < 0.0001 % of mesh faces in this category on average. The Maximum cell size was 70 meters, which was never exceeded. Volume change was capped at 0.01 and < 4% of all volume mesh cells were found to be within this threshold. < 0.0001% of all mesh cells had Face Validity < 1. No negative volume or topologically invalid cells were generated by the meshing tools. The optimizer mentioned above was responsible to ensuring that the maximum amount of cells maintained good quality. This optimizer was run after the mesh generation and is an automated add-on tool for meshers in STAR-CCM+.**
   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? **No re-mesh was needed.**
6. Were there any additional best practices that you used in generating the meshes? **N/A**
7. What was the required level of user expertise (novice, intermediate, expert) for this task? **Intermediate**

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

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

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? **All mesh families in STAR-CCM+ were defined using a global size (called base size). All mesh settings for each family member of the mesh were set relative to this global size. This includes surface and volume meshing, and all meshing controls. This allowed us to generate the entire family of meshes simply by scaling this global size up or down from the Medium mesh, which was generated first. Note that there were som parameters, such as near-wall thickness of the boundary-layer cells, that were fixed for all family members of the mesh. Further, the TE refinement on the flaps/slats/wings were scaled directly by the requried number of mesh faces as specified in the gridding**

guidelines. The entire process is automated in STAR-CCM+, allowing for high degree of stand-off control from the user.

2. In your opinion, what was the most time-consuming or tricky aspect of generating a family of meshes? **The transition/growth of cells between refinement zones was the most time-consuming part of the meshing process. Considering the dissipative nature of the Polyhedra cells, this was an important setup step to ensure accuracy of the solver results.**
3. How did the times (labor, CPU, etc.) needed to generate them compare?

Mesh type	Coarse	Medium	Fine
Unstructured Cartesian Hexahedra (Trimmer) + Prismatic near-wall	**0.5 hours / 16 processors in parallel / 75 GB memory**	**1 hours / 16 processors in parallel / 100 GB memory**	**2 hours / 16 processors in parallel / 150 GB memory**
Unstructured Polyhedra + Prismatic near-wall	**5.3 hours / 1 processor / 62 GB memory**	**6.5 hours / 1 processor / 90 GB memory**	**8.2 hours / 1 processor / 145 GB memory**

4. Were there any problems that you encountered in one mesh resolution that you did not encounter in another resolution? **No.**

Post-Solution Mesh Modifications

1. After generating an initial flow solution, were additional mesh modifications made to improve solver convergence or solution accuracy? **No.**
2. Describe any post solution mesh modifications that were made? **None**
3. How long did these modifications take (elapsed time and labor – in hours)? **N/A**

I/O

1. In which format did you export your meshes? (CGNS, Solver-native, …): **CGNS**
2. What are the names of the files you uploaded to the GMGW-1 server? **The following files were uploaded to the GMGW-1 server:**

1. PLM_trimmer+prisms_coarse.cgns.zip

2. PLM_trimmer+prisms_medium.cgns.zip

3. PLM_trimmer+prisms_fine.cgns.zip

4. PLM_poly+prisms_coarse.cgns.zip

5. PLM_poly+prisms_medium.cgns.zip

6. PLM_poly+prisms_fine_2.cgns.zip

7. PLM_poly+ALM_coarse.cgns.zip

8. PLM_Participant_Questionnarie_completed.pdf

Miscellaneous

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

   N/A