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
   1. What software tool(s) did you use to import and prepare the HL-CRM geometry model for meshing? BOXER
2. Import & Preparation for meshing
   1. 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)

* 1. What problems, if any, did you identify immediately after importing the geometry model (eg, missing geometry, poorly translated geometry, other)? Some tiny gaps near TEs, and the parts were not well named
  2. 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 Renamed patches for ease of use, and patched some of the gaps near the TE's that were larger than the refinement required

* 1. What was required level of user expertise (novice, intermediate, expert) for this task? Intermediate
  2. How long did import take (both elapsed time and labor required --- in hours)? an hour of setup approximately

# Initial Meshing

1. What type of mesh family did you generate?

Structured multi-block

Unstructured tetrahedra

Unstructured hexahedra

Hybrid

Overset

Cartesian

other (please specify      )

1. Surface Meshing (Note: BOXER does not start by generating a “surface mesh” instead an Octree grid with local refinement is created to capture the geometry as a Digital image; this, together with the associated distance field, effectively acts as the geometry “surface”)
   1. What software tool(s) did you use to generate your initial surface mesh? BOXER
   2. How long did it take (elapsed time and labor – in hours)? ~ 1 hour to generate Octree - Setup for first intial run took ~ 3 hours of labour
   3. Provide a brief description of how mesh resolution was specified (explicit user inputs, sources, curvature based sizing, background distribution function, …) From the grid guide lines, the key details on the expected mesh size around key areas (LE's TE's etc) was found and then setting refinements to match these were chosen. Layer height was set to the estimed size of mesh being generated (medium)
   4. When/how did you judge surface mesh generation to be complete? Once all the features were being captured appropriately and the refinements look good the octree was deemed suitable
2. Volume Meshing
   1. What software tool(s) did you use to generate your initial volume mesh? BOXER
   2. How long did it take (elapsed time and labor – in hours)? No labour as this is a direct continuation from the Octree stage above; time to create the body-fitted mesh ~ 3 hours; then ~7 hours to insert the layer mesh (all run on a 96 core cpu)
   3. Provide a brief description of how mesh resolution was specified (explicit user inputs, sources, curvature based sizing, background distribution function, …) Resolution was determined from the Octree stage from the refinements set.
   4. For resolving surface boundary layers, what cell size growth rate did you use? Was it constant or variable? If variable, describe. Layer growth was chosen to be a inital fixed distance (from guide lines) and the growth rate chosen from the guide lines (for medium ~ 1.17)
   5. When/how did you judge volume mesh generation to be complete? When features captured were well resolved and the size generated was around a similar level to the guide lines.
3. Adherence to HiLift-PW3 meshing guidelines
   1. To what extent did your mesh(es) adhere to the HiLift-PW3 meshing guidelines? The y+ and initial layer height and growth were followed, the number of cells on the TE was a fairly close match as well.
   2. Was it possible to adhere to the guidelines on the first attempt, or were there iterations involved? Some iterations were carried out however the guide lines were fairly closely matched on the intial run. The additional runs were mainly to refine various areas to be a more suitable mesh.
   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 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): The guide lines do not directly correspond to the type of meshing being carried out using BOXER, although it was possible to translate the requirements into terms BOXER matches.

1. A priori metrics (such as skew, or maximum stretching ratio, maximum deviation of mesh nodes from OML or …)…BOXER aims to automatically achieve mesh quality suitable for a typical COTS solver – the mesh metrics & quality targets are built-in – the user cannot intervene.
   1. What a priori metrics did you apply on the initial mesh?
   2. What was the average and range of the metrics?
   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?
2. Were there any additional best practices that you used in generating the meshes?
3. 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)?
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? Process was initially following the gudelines for the coarse mesh, as things progressed, it became a case of get a mesh of best quality then figure out the steps to match it to the right family
2. In your opinion, what was the most time-consuming or tricky aspect of generating a family of meshes? optimising the cell count
3. How did the times (labor, CPU, etc.) needed to generate them compare? It took longer to generate each mesh as you go from coarse to fine, as the cell count increases.
4. Were there any problems that you encountered in one mesh resolution that you did not encounter in another resolution? No unexpected problems.

# 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, …): .msh
2. What are the names of the files you uploaded to the GMGW-1 server? CFS\_HL-CRM\_gapped\_config\_MEDIUM\_20170429.msh

# Miscellaneous

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

There’s a couple that I thought would be useful.

* The lack of a viewable online image of the model, this would be useful as then I would have a better expectation of the quality of CAD I’m meant to be importing. Knowing this I know when I import the model this is the level of detail that was uploaded, and not mistake it for missing something from the import
* The specification for coarse, medium fine, etc are good but some more general information could be useful such as example cell count. This information wasn’t obvious at first but was possible to find out through looking at the pre generated grids. It could do with being told explicitly if it’s available.
* The information on “how” to upload is vague and could include some more detail.