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? ANSA
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)

X 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)

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

X None

Layering (hiding components)

Simplification/defeaturing (removing components)

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

Modification (changing components)

Shrink-wrapping

Other

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

# Initial Meshing

1. What type of mesh family did you generate?

Structured multi-block

Unstructured tetrahedra

Unstructured hexahedra

X Hybrid (tria/quad on surface) penta/hexa in layers and pyramid tetra in the rest of the domain

Overset

Cartesian

other (please specify      )

1. Surface Meshing
   1. What software tool(s) did you use to generate your initial surface mesh? ANSA
   2. How long did it take (elapsed time and labor – in hours)? 2-3 hours
   3. Provide a brief description of how mesh resolution was specified (explicit user inputs, sources, curvature based sizing, background distribution function, …) Automatic curvature dependent refinement, in combination with Size Boxes. Additional anisotropic mesh at leading and trailing edges as a manual step.
   4. When/how did you judge surface mesh generation to be complete?  
      All features well resolved and all mesh checks (quality, unmeshed, intersections, proximities etc) passed
2. Volume Meshing
   1. What software tool(s) did you use to generate your initial volume mesh? ANSA
   2. How long did it take (elapsed time and labor – in hours)? 3 hours
   3. Provide a brief description of how mesh resolution was specified (explicit user inputs, sources, curvature based sizing, background distribution function, …) Size Boxes controlling mesh refinement areas
   4. For resolving surface boundary layers, what cell size growth rate did you use? Was it constant or variable? If variable, describe. Constant growth rate from 1.1 to 1.25 depending on mesh refinement level
   5. When/how did you judge volume mesh generation to be complete?  
      When mesh quality checks (negative volume and flipped elements) were passed
3. Adherence to HiLift-PW3 meshing guidelines
   1. To what extent did your mesh(es) adhere to the HiLift-PW3 meshing guidelines? To a quite good extend. The only rule we did not follow is that we decided to keep the same first layer height for all mesh refinement levels.
   2. Was it possible to adhere to the guidelines on the first attempt, or were there iterations involved? Yes a couple of iterations were needed to establish a cell count close to the guidelines. Unstructured meshing does not always scale up and down in a predicted manner in a mesh refinement study.
   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):

1. A priori metrics (such as skew, or maximum stretching ratio, maximum deviation of mesh nodes from OML or …)
   1. What a priori metrics did you apply on the initial mesh? Fluent skewness on surface of 0.5 (except for anisotropic mesh regions), volume mesh was checked for negative volume and concavity. OpenFOAM criteria were monitored but could be satisfied throughout the model due to the very high aspect ratio layers elements.
   2. What was the average and range of the metrics? Average OpenFOAM orthogonality=16.6, Max=88.5, Max OpenFOAM Skewness= 9.5
   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? Negative layers were identified in initial meshing stages near high proximity areas that were further refined.
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? expert

# 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? The medium mesh was generated first and then using batch mesh tool in ANSA we automatically scaled up and down the mesh characteristic lengths. A couple of tests were needed to find the scale factor that would result to the expected increase/decrease in cell count. Finally a scale factor of around 1.25 was used to scale up and down the length values.
2. In your opinion, what was the most time-consuming or tricky aspect of generating a family of meshes? The generation of the anisotropic meshes at the leading and trailing edges
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? Finer meshes tended to have more negative elements that needed fix quality operations

# 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, …): CGNS and OpenFOAM
2. What are the names of the files you uploaded to the GMGW-1 server? Hl-crm-sealed-medium.cgns, hl-crm-gapped-coarse.cgns, hl-crm-gapped-medium.cgns, hl-crm-gapped-flaps-fine.cgns and similarly for OpenFOAM

# Miscellaneous

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