R17.x Pre-processing Update

Maciej Ginalska
CFD Meshing
ANSYS CFD meshing solutions overview

- **ANSYS continues to develop three products for CFD meshing**
  - Fluent Meshing (formerly TGrid)
  - ICEM CFD (Hexa and Tetra/Prism)
  - ANSYS Workbench Meshing

- **These three products share underlying executables and have common functionality in many areas**

- **But on-going strengths and weaknesses remain**

- **Hence ANSYS have decided to simplify the licence arrangements**
ANSYS CFD consolidated meshing options

• In 2015 (R16.1), ANSYS streamlined licencing for its CFD meshing tools

• Our four meshing tools are now available through licences of:
  – ANSYS CFD
  – ANSYS CFD PrepPost, ANSYS Meshing
  – ANSYS CFX, ANSYS CFD-Flo
  – ANSYS Fluent
  – ANSYS Polyflow

• If you have legacy meshing licences (ICEM CFD, TGrid, CFX-CAD2Mesh, Fluent Solver, CFX Solver…) speak to your account manager about your migration options
ANSYS Tools Span the Breadth of Requirements

SpaceClaim can push models from Class 1 to Class 3 with low man-effort!

<table>
<thead>
<tr>
<th>Geometry Corruption</th>
<th>Class 1</th>
<th>Class 2</th>
<th>Class 3</th>
<th>Class 4</th>
<th>Class 5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Very High</td>
<td>High</td>
<td>Low</td>
<td>Medium</td>
<td>Medium</td>
<td></td>
</tr>
<tr>
<td>Geometry Complexity (# faces, parts or bodies)</td>
<td>&gt; 1,000</td>
<td>100 to 1,000</td>
<td>≈ 100</td>
<td>&gt; 100 parts</td>
<td>&lt; 1,000</td>
</tr>
<tr>
<td>Typical Applications</td>
<td>Automotive &amp; Aerospace UTM</td>
<td>Any</td>
<td>Any</td>
<td>Any</td>
<td>Aerospace, Turbo, Chemical &amp; Material Proc. Built Environment</td>
</tr>
</tbody>
</table>

Fluent Meshing

Workbench Meshing

ICEM CFD
ANSYS Meshing @ R17.x

**Fluent Meshing**
- Advanced unstructured meshing (Tet, HexCore, CutCell, Poly with prisms)
- Surface wrapper for complex/dirty CAD
- Scriptable in Workbench

**WB Meshing**
- Native hybrid meshing in Workbench
- Good for anisotropic geometries
- Persistent in Workbench without need of scripts

**ICEM CFD**
- Advanced hex meshing
- CAD tolerant OCTREE
- Scriptable in Workbench

**Shared Libraries**

- CFD
- Multiphysics
- CFD & Mechanical
WB Meshing Highlights in 17.0 for CFD Users

- Performance improvements
  - Linux & other improvements
  - Mesh transfer to Fluent
- Better meshing status
- Folders for meshing controls
- Contact match mesh
- Improved diagnostics
- Improved criterion based selections
- Hex meshing improvements
- Improved ICEM CFD Interactive blocking
Repair overlapping contact regions

• Repair Overlapping Contact Regions
  • Finds contact regions where an entity is in multiple contacts (when exporting to Fluent or CFX this is considered “overlapping”)
  • Fixes contact regions by suppressing original contact regions and creating new “Repaired” contact regions that do not overlap.
  • Overlapping cases that have cyclic redundancies are suppressed.

Typical overlapping case

Non-typical case

In cyclic redundant case: A // B, B // C, A // C
Since there are no unique pairs, Fluent/CFX don’t support this case, so all 3 contacts are suppressed.
Repair Overlapping Named Selections (16.2)

- Find & Repair Entities (Faces or Edges) that are in multiple named selections (thus causing overlapping zones in output to Fluent or CFX)
- Repair will place original named selections in Overlapping Named Selection folder and set “Send to Solver” to No. Repaired Named Selections will be at bottom of list. Only operates on Named Selections that are marked Send to Solver=Yes.
Performance Improvements

- Linux performance: Linux more comparable to Windows
- Other general and model specific performance improvements

![Average time for several mesh sizes and settings](image_url)
Performance Improvement: Fluent mesh transfer

- New option: Large Model Support
- Guidelines:
  - May want to use with models >10 Million
  - Models should be read into Fluent in parallel
  - Number of internal zones, and zone IDs may change
Better Meshing Status:

- Progress is now reported as parts are meshed in parallel
- As a part is meshed the topology (edges, faces, bodies) are highlighted to show what is being worked on
  - This can be turned off by unchecking “Highlight”
  - If user stops meshing, entity will stay highlighted, allowing user to find problematic geometry easier

Notes:
- If user stops meshing, parts that have been meshed are done
- Restarting meshing resumes with only unmeshed parts
Folders for Meshing Controls

- Mesh controls can now be grouped for easier organization
- Option: Group All Similar Children, will group all objects based on type
- Options to suppress, rename, nest groups, ungroup, delete objects in group
- Drag and drop capabilities to modify the grouping
Contact match mesh

- Connect mesh at mesh level
  - At R17 only works with tet mesh
  - Could avoid problems with shared topology

Notes:
- Mesh generated as separate parts
- Use Contact Match controls to match mesh
- Create Named Selection to inspect matched mesh
- Node merge can be added as second step to make mesh conformal
Capped section planes

- Shaded section planes
  - Shaded or hollow section plane
  - Plot by body colour or same colour for section plane
Improved diagnostics: Mesh display By Body Connection

- Default mesh display
- Turning wireframe mode on
- Turning on Mesh display by body connection.
Improved diagnostics: Close vertices

- Close Vertices: Checks for vertices that are very close together:
  - Scale can be adjusted depending on model sizes
Improved ICEM CFD Interactive Blocking

• Interactive ICEM CFD to create initial blocking:
• Often faster way of decomposing model
• Blocking topology stored w/model

If no blocking exists, it allows user to create one
Improved ICEM CFD Interactive Blocking

- Now user can adjust sizes, etc. and update mesh using the batch ICEM CFD to reuse the blocking topology
Coming in R17.2...

- WB meshing libraries updated to use Fluent Meshing Cutcell R17.2
  - Faster more accurate size functions
  - Quality improvements
  - Defect fixes

- Support for local proximity size function control

- Target Skewness (beta option in R17) will become a full feature
Main window layout

- Display/Selection controls
- Help/Hide Ribbon
- Common Icons with solver
- Movable Icon Sets
- Operation Icons
- Detachable Console
UI Enhancements @ R17
Updated Tree

Enclosed Region icons now give information
- Type (Fluid, Solid, Dead)
- Mesh condition (full, partial etc)
CAD import/management (R17.1)

• New CAD Assemblies Import Option
  – CAD tree retained
  – Fast re-import and re-tessellate of parts
  – Sort assembly geometry needed for analysis
  – A link is maintained for easy updates and design changes
  – Face-zone labels preserved for persistent mesh settings
  – Allows faster visualisation

• Overall time saving in part management of huge models
Demonstration on Youtube ANSYS Channel

From CAD-to-Mesh with Fluent Meshing: Part 1

Fluent Meshing has been going through some quite dramatic usability improvements during the last 2-3 years and as we continue to move in this direction, we also add and improve core capabilities. In these videos below we take you to a journey from extracting a few parts from a CAD assembly model to the volume meshing steps, all with tree and graphics driven operations within Fluent Meshing. In the first video we start by an introduction to our new light-weight CAD assembly module, or CAD management module. You will find that the CAD assembly tree maintains the hierarchy of your sub-assemblies, parts and bodies exactly as in your CAD package. It allows you extract out parts using suitable faceting for the engineering model that will actually be used for the simulation. The faceting can span from very coarse to high quality CFD surface meshes. We also show how easy it is to manage and replace parts from this module into the engineering module as part of design changes.
Interactive Join/Intersect operations

• **Finding right tolerance for Join/Intersect is sometimes tricky**
  – Interactive Join/Intersect has local Mark and Undo operations to avoid incorrect joins and improve efficiency

• **Icon for Gap closing, Join and Intersect**
  – Easier access to additional gap closing before join
Region Based Volume Meshing

• The ability to fill one or many regions with a volume mesh
  – Enables different mesh settings per region

• Partial region meshing (beta)
  – Ability to partly fill a region, or fill a remainder of a partly filled region
  – Scoped prism, Tet and HexCore

• Support for Quad and Hex meshes in Object based workflow
  – Allows mix-and-match with WB Meshing or ICEMCFD Hex meshing
Object based meshing – Quad/Hex support

• Hex and Quad cells are supported in the object tree
  – Import hex meshes from Workbench meshing
  – Very useful for thin regions where stretched hexes can save significant mesh cost

• Cell zone to Boundary zone -stitching
  – Connecting to boundary zones defined at the cell boundary
  – Use /boundary/remesh/stitch-with-preserve-boundary
Parallel Volume Meshing Capabilities

• The roadmap for CFD meshing is parallel!
  – Multi-release, staged development

• Two approaches are available today for parallel meshing with tet and prisms to speed up the task and utilise distributed memory
  1. “Region Based” prism/tet meshing (R16)
  2. Auto-decomposed prism meshing with serial tet (R17)
Parallel Prism+ Serial Tet Meshing schematic

Surface Mesh

Principal inertia axis method

Node 0 - Tet

Node 1 - Prism

Node 2 - Prism

Node 3 - Prism

Node 4 - Prism

Prism-Cap Surface mesh

HDF5 Write

Fluent Solver
Parallel Prism + Serial Tet Meshing efficiency

Scoped Parallel Prism + tet + lean data structure (beta)

- Combustor: 4.7 M Prism, 3.5 M Tet
- Landing Gear: 7.0 M Prism, 5.7 M Tet
- Oil platform: 3.6 M Prism, 5.7 M Tet
Parallel Prism Meshing I/O (beta)

• Writing out HDF5 format using DrivAer (prism only) mesh

Case write of 11.2 million prism

<table>
<thead>
<tr>
<th></th>
<th>Seconds</th>
</tr>
</thead>
<tbody>
<tr>
<td>Serial or -t1</td>
<td>99</td>
</tr>
<tr>
<td>4-node</td>
<td>18</td>
</tr>
<tr>
<td>8-node</td>
<td>12</td>
</tr>
<tr>
<td>16-node</td>
<td>4</td>
</tr>
</tbody>
</table>

- **Hdf5**
- **Mesh**
Native Polyhedral Meshing

• Efficiency (strong model dependency)
  – 2-3 times less memory and 2-3 times faster compared to serial tet generation and serial poly conversion in Fluent Solver

• Optimal creation
  – Supporting Baffles, Local refinements, Size Function and scoped prism settings
  – Selective concavity handling reduces the mesh size compared to converted polys with 15-30 % while reducing size change
  – Polyhedral driven quality criteria during initial tet generation and many other additional post quality enhancing operations

• Limitations
  – Only object driven meshing (tree)
  – Zone-specific prisms are not supported
  – Cannot mix with other mesh methods
    • Not included in region based meshing
  – Quad boundary is not supported
Native Poly Meshing

- **Landing gear with BOI**
  - 8.1 million poly cells
  - R16.1 (poly in BETA)
    - Worst initial OS = 0.99
      - 3 cells over 0.9 fixed by ANM
    - Time = 32 min
    - Peak memory 12 GB
  - R17.0
    - Worst OS = 0.9
    - Time = 20 min
    - Peak memory 12 GB
Several HexCore Enhancements (beta)

• Octree instead of Cartesian
  – Enabled via TUI command
    /mesh/hexcore/controls/octree
  – 10 – 50% speedup and memory reduction
    • Strong model dependency

• Size Field driven instead of Buffer layers
  – More accurate size distribution
  – Support of BOI

• Advanced peel layer control
  – More accurate gap creation leading to a reduction in the number of tets and consistently good quality
Scripted Automatic Meshing for Complex CHT

• Remove large gaps between solids
• Create missing geometry

Prepare:
• Naming objects and zones

Generate Main and Secondary Fluid Regions with Prism Layers

• Extract Contact Pairs
• Used for Mapping and NCIs

Coming Soon as an ACT!
Erling Eklund
Lead Product Manager FBU
Objectives

• The default Fluent Meshing Workbench functionality requires the user to write a journal to make the workflow parametric
  – Journal may be a few lines to e.g. mesh volume and smooth for better quality
  – But... scripting is always a barrier to entry
• ACT Solution ... To optionally or permanently access selective Fluent Meshing technologies within the Workbench environment and workflow
  – In particular Native Polyhedral meshing
• To deploy Fluent Meshing technology to non-expert users
• To allow Fluent Meshing users to benefit from the Robust Design process within Workbench
Available at the ACT App Store for V17.0 and V17.1

- Introduction Video
- Step-by-step instruction including any limitations
Comparison with Converted Poly Mesh

Native Poly mesh

Converted Poly mesh
ICEM CFD 17.x Highlights
17.x Highlights:

• Improved control of pre-inflation

• MultiZone improvements
  – 3D MultiZone
    – More automation in decomposition
    – Improved tools for editing free & swept blocks
  – 2D MultiZone from curves
  – Improvements in display

• Custom features
  – Improved toolkit exposure
  – Improved output interface for Exodus II for larger models
  – Improvements in aligning to existing mesh

• Improved robustness
Improved pre-inflation options:

- Additional pre-inflation settings
- Improved consistency:
  - Pre & post use similar controls
  - Local growth law and height limit factor
  - Ability to apply at part, face, or edge level
3D MultiZone

- New option for 3D MultiZone:
  - Easier than 2 step approach in past of 2D Blocking then 3D Fill
  - More similar to option in WB Meshing App
  - New option Mapped/Swept Decomposition to help control level of decomposition
3D MultiZone: Bottom-up approach to ICEM CFD Hexa extended to Swept/Free blocks

Top-Down Blocking: Historical approach to meshing in ICEM CFD

- All mapped (structured) blocks
- User is sculptor (need to think through blocking)
3D MultiZone: Bottom-up approach to ICEM CFD Hexa extended to Swept/Free blocks

Bottom-up Blocking: MultiZone automates blocking (decomposition)

- Any combination of blocks (Mapped, Swept, Free)
- Automation as much as possible
- Leverage block (topology) editing tools to finish (if needed)
# 3D MultiZone: Comparison of Approaches

<table>
<thead>
<tr>
<th>Top Down (Sculpting)</th>
<th>Bottom Up (Brick laying)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Often easier/faster to get an all hex mesh</td>
<td>Faster to get first mesh (but may not be all hex)</td>
</tr>
<tr>
<td>Global topology: avoids getting stuck in corners</td>
<td>More flexibility: Can stop at any time with a valid mesh (but may have free blocks, i.e. tets/pyramids)</td>
</tr>
<tr>
<td>Topology is less constrained to CAD, most work is in associating to CAD</td>
<td>Topology is associated (constrained) to CAD, most work is in freeing some of the original constraints.</td>
</tr>
</tbody>
</table>

User can also use a combination of both approaches!
3D MultiZone: Combination of Approaches

- MultiZone Blocking
- Convert Swept to Mapped
- Extrude Block
- Delete Free Block
- Block from faces for middle region
- Update associations
- Simplify Swept to Mapped
- Block from faces to connect to base
- Improve shape
- Mesh
Free/Swept/Mapped Block Editing Features

- Merge Vertices
- Merge Faces
- Split Blocks
- Convert free block face to mapped/free
- Imprint faces
Block editing: Merge vertices

- Merge vertices added for swept block (collapses outside faces)
- Merge vertices added for free blocks
Block editing: Merge faces

- New option for block faces:
  - Allows selection of free or mapped faces attached to free blocks
  - Limitations for other types or faces

- Face corners (traditional options):
  - Only works on mapped faces attached to mapped blocks
Block editing: Split Free Blocks

- Easier tools to split up free blocks to make swept or mapped regions:
Block editing: Convert Blocks

- **Tools to convert 3D blocks:**
  - Mapped
  - Swept
  - Free
  - Y-Block

- **Tools to convert faces:**
  - Convert free block face to free
  - Convert free block face to mapped

- **Tools to alter blocks:**
  - Reverse sweep direction
  - Merge sheet w/free blocks

- **Tools to edit mesh method for blocks:**
  - 3D free block mesh type
  - Free face mesh type

- Almost all of these operations have been improved in 17.0
- Convert free block face to mapped is new operation to help convert free blocks to swept or mapped.
Block editing: Imprint

1 Free Block

- Imprint Loops into free face
- Use the 2 faces to slice the free block forming a mapped region
2D Blocking from curves:

- 2D Blocking can be generated directly from curves
- Use combination of free and mapped blocks to help models w/large length scale differences
Improvements in projected edge shape

- Projected edge shape does a better job of showing mesh representation
Associate edge or face to reference mesh

Given a pre-meshed edge or face, associate edge->curve or face->surface and use Reference Mesh to use existing mesh.
Improved toolkit exposure:

- Removed product licensing restrictions, now toolkits have separate display options:
  - Settings->Products replaced by Settings->Tools
Thank you