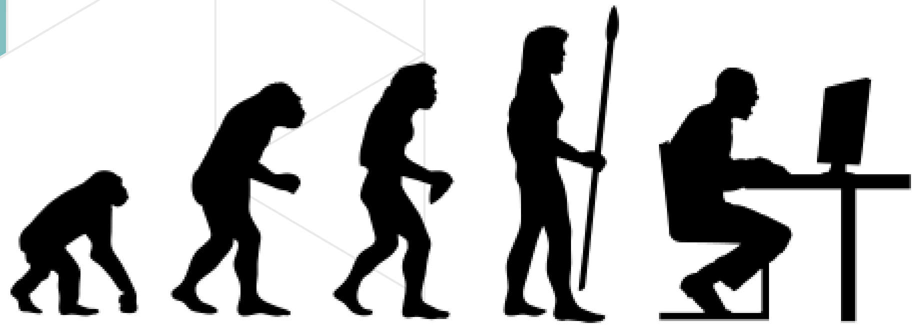
June 8, 2017

Autodesk Moldflow Summit

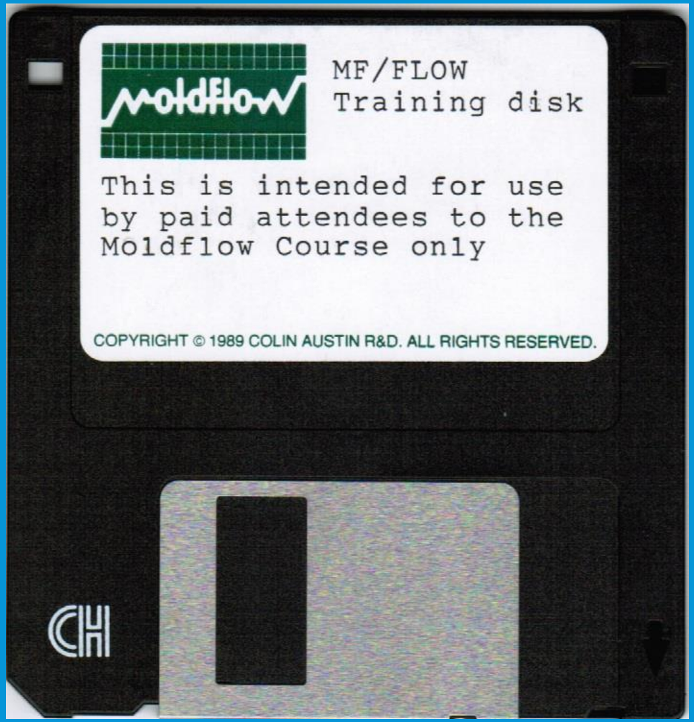
The Evolution of Meshing



Matt Jaworski
Senior Moldflow Technical Specialist

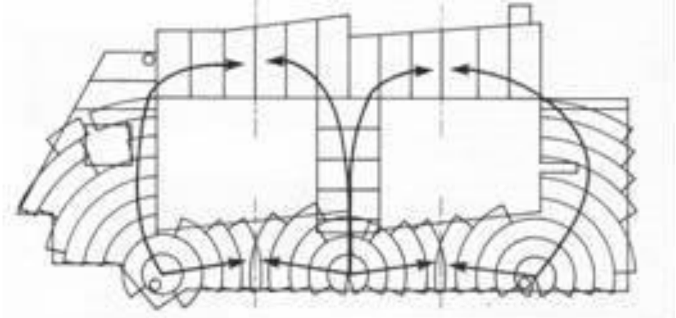
Tim VanAst
Senior Engineer/Technical Consultant

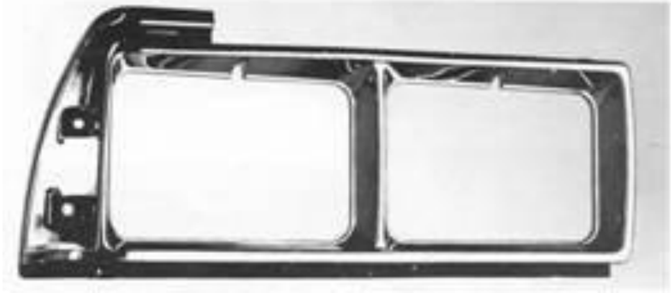
A Long Time Ago…

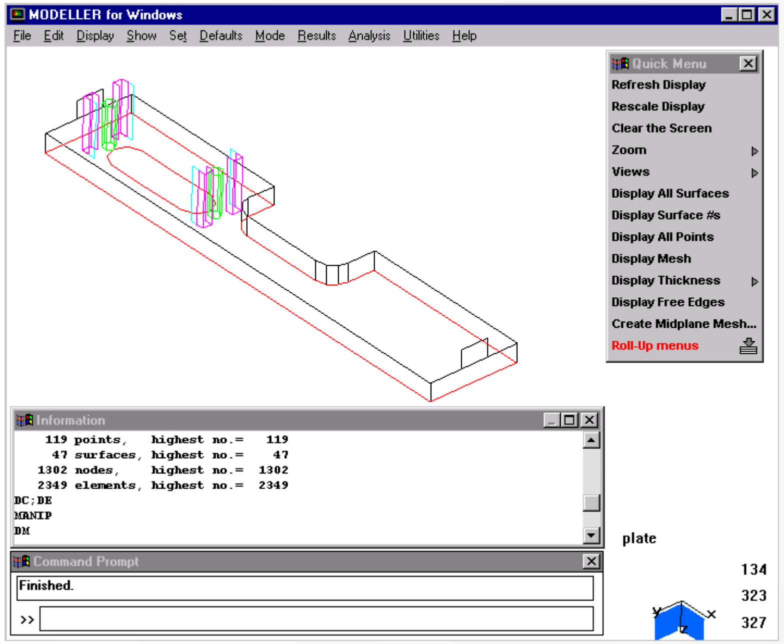


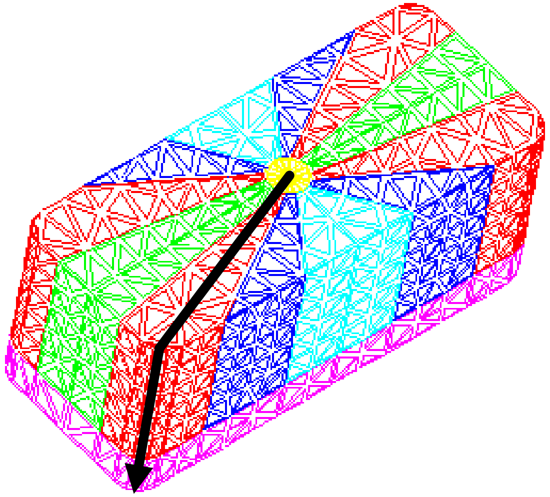
Meshing: Remember Where We Came From!

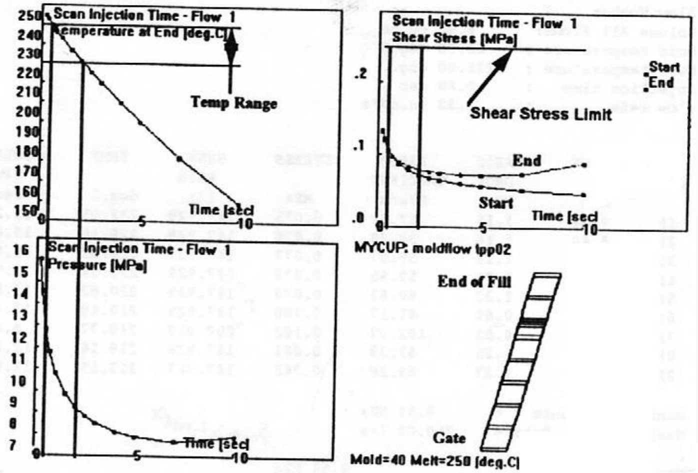
- Text Only UI
- Layflat
- Dominant Flow Path
- 2D Strip Files
- UNIX
- MFG
- MFVIEW
- MODELLER
- PO X,Y,Z

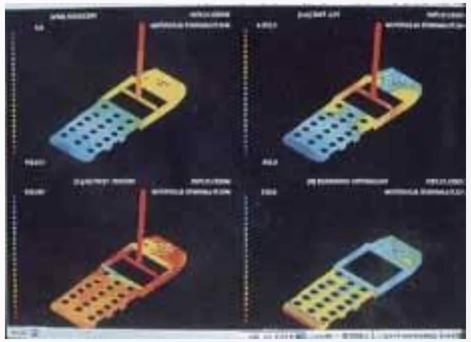










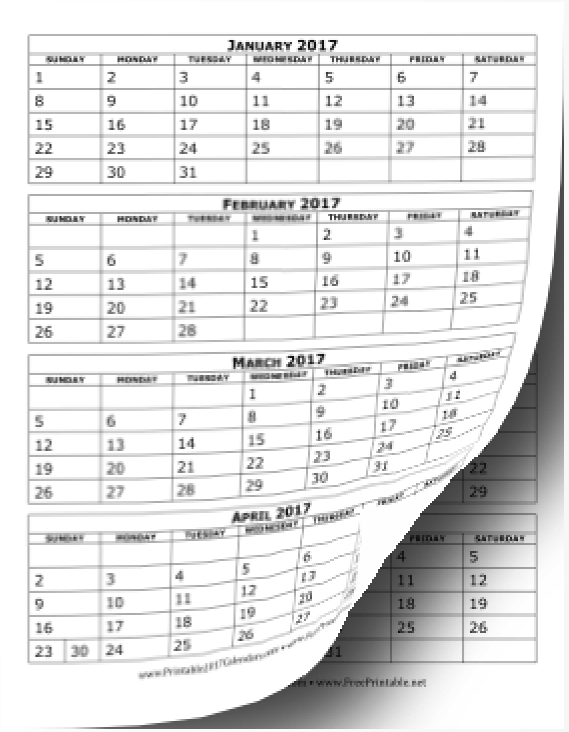


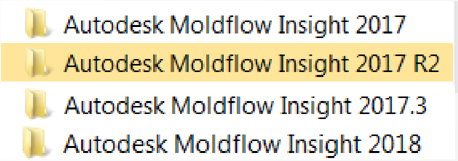
Meshing: It’s what we do

- Meshing used to take 80% of the project time, but it’s what we did to make it work
- The recommended aspect ratio of 6 was onerous at best, but it’s what we did, or tried to do, to make it work
- Putting 3 elements across a thickness change took extra time, but it’s…wait, you should still do that!
- Advances in meshing, and solver, technology has changed how we should look at meshing, and how we should be meshing

New Development and Release Schedule Changes

- Moved to a new “agile” development schedule
  - More frequent releases (Goal is Quarterly)
- 2017 R2 release was the start of this new schedule (Released at K-Show 2016) with 2017.3 following in Jan 2017
- Installs as a complete standalone install not an update or Service Pack that “patches”
  - Material DB updates, new features, fixes
- Can have multiple versions coexist (i.e. 2017 SP2 & 2017 R2)





2017

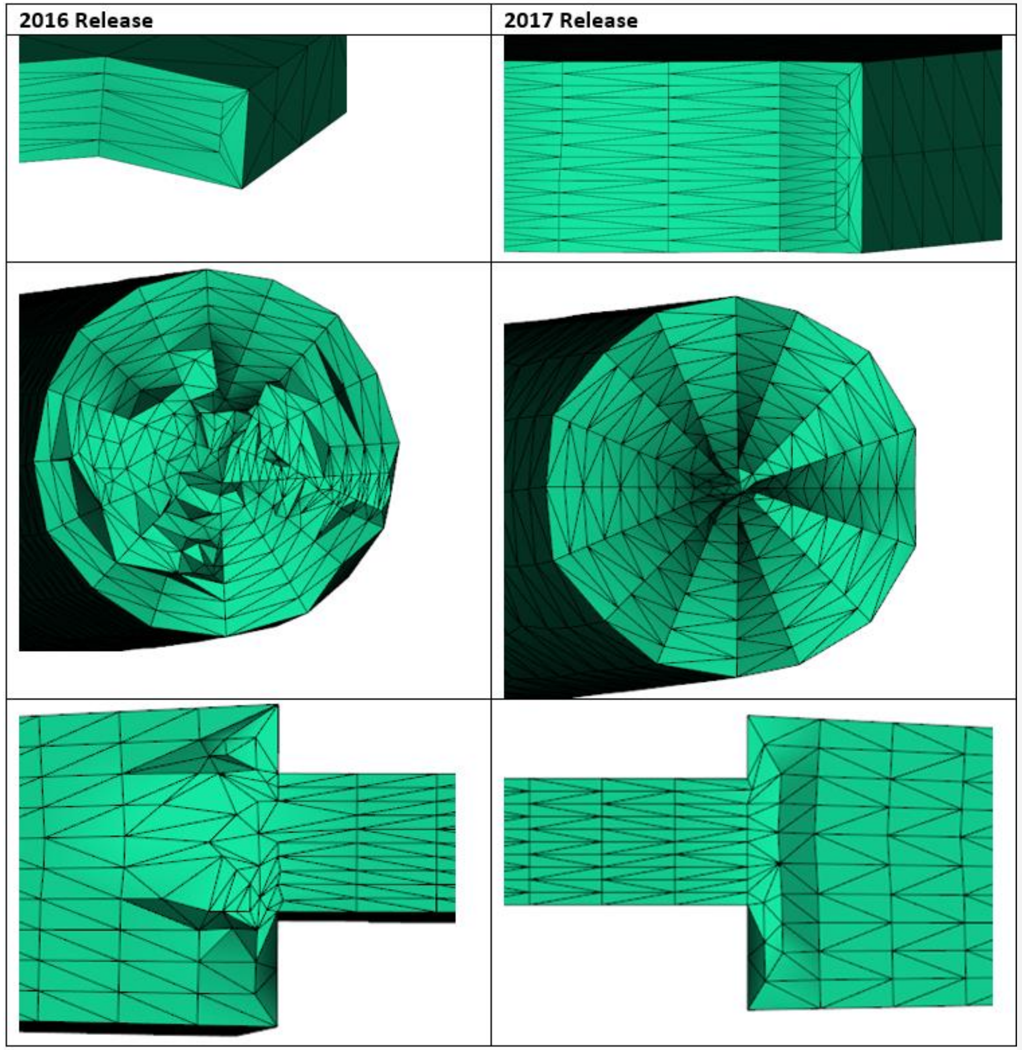
2017 3D Meshing Algorithms by Advancing Layers

- Improved mesh quality for thick models
  - The new algorithm does not rely on surface mesh match, it can generate tetras with regular shape even in unmatched chunky regions

- Significantly reduced percentage of flat tetra elements

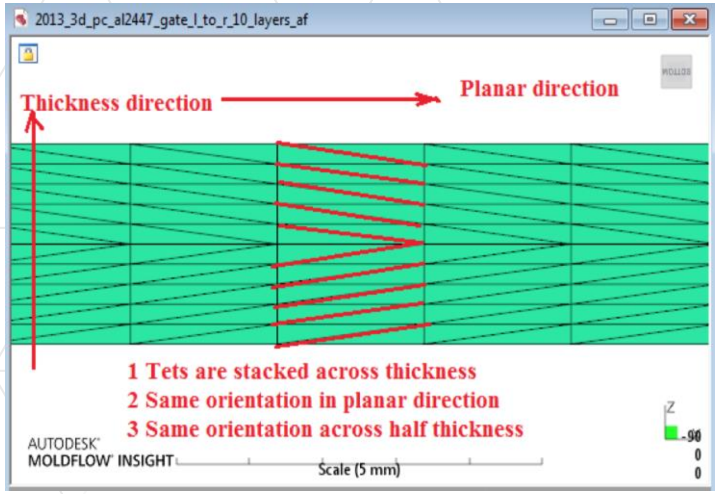
- Avoided dramatic increase in element counts when number of layers is high

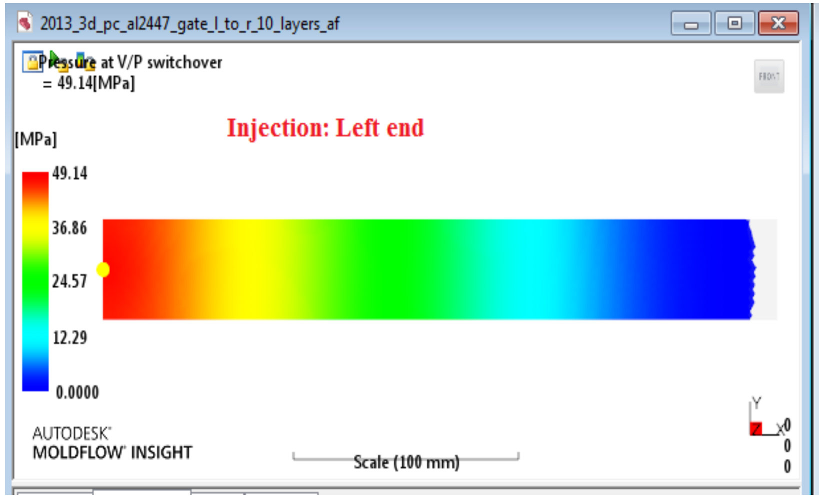
Releases	Number of tetra elements (in thousands)			Differences between 10 and 20 layers
	6 layers	10 layers	20 layers	
2016	267K	455K	1083K	138%
2017	263K	424K	818K	93%

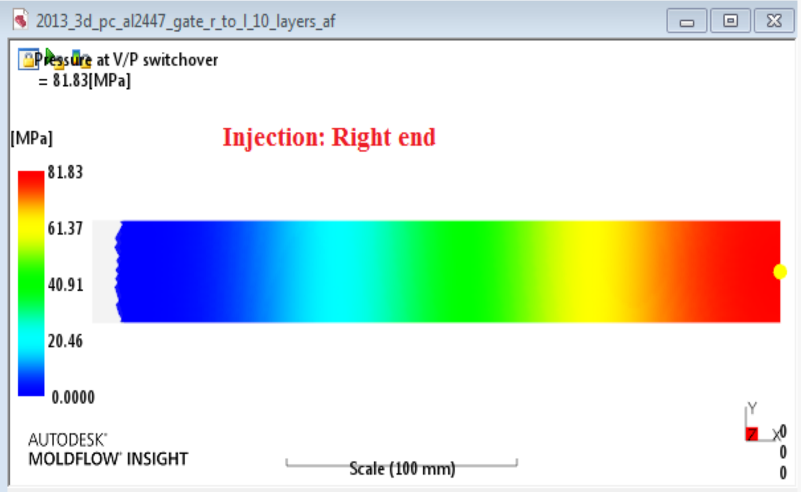


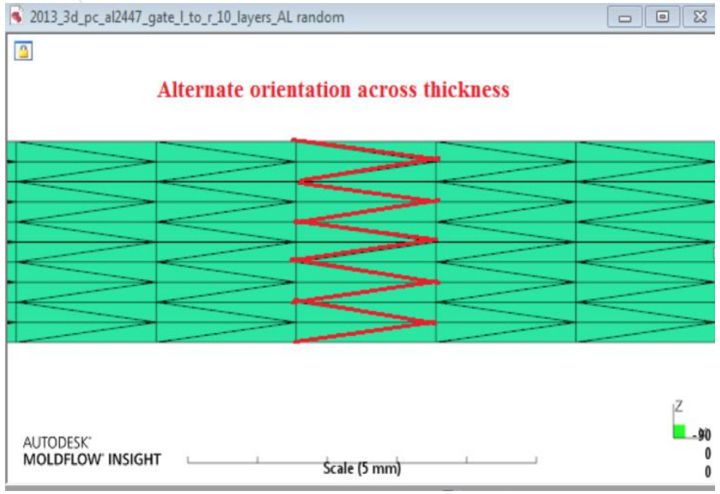
2017 New 3D Meshing Algorithms by Advancing Layers

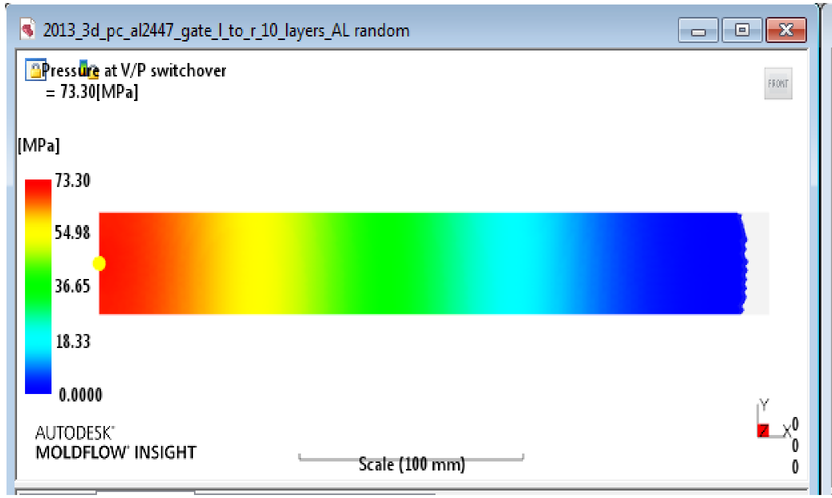
- Alternate orientation eliminates certain sensitivity!

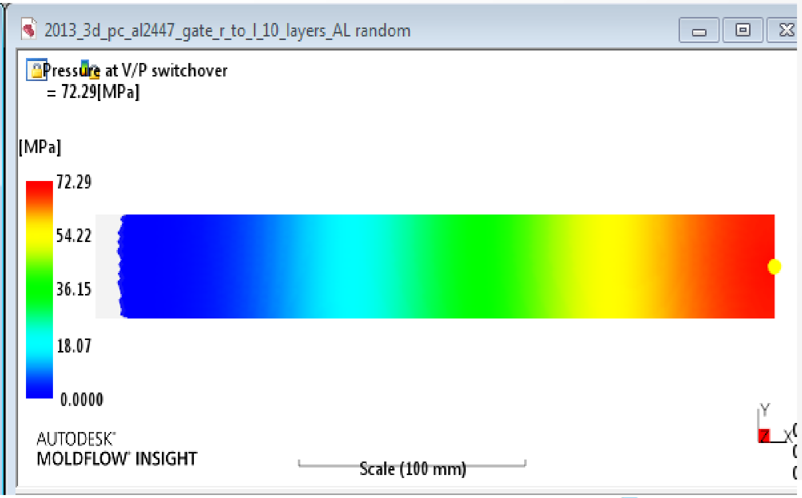






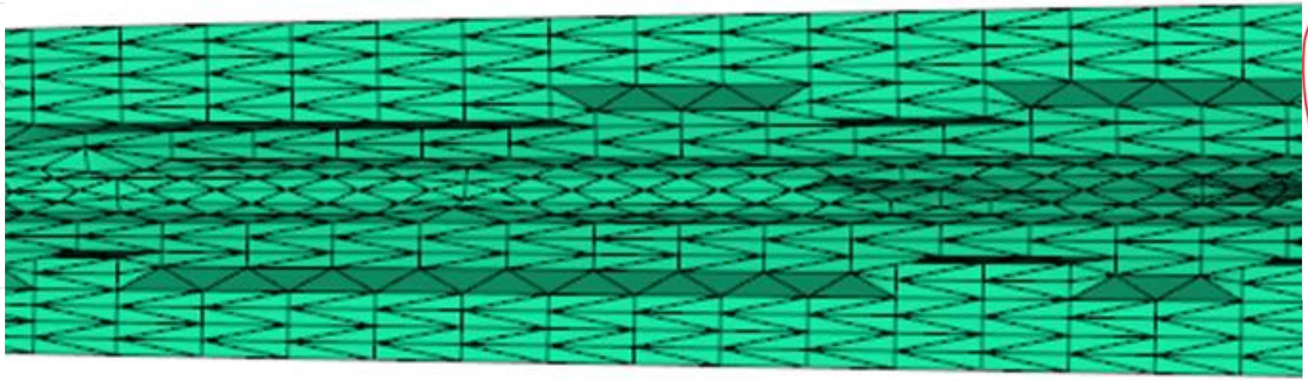


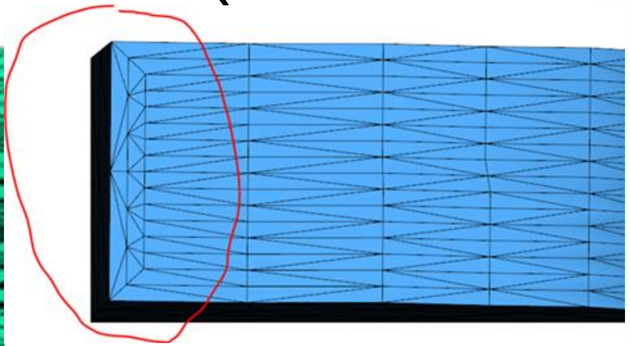


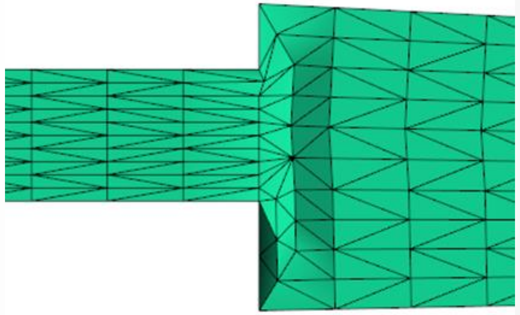


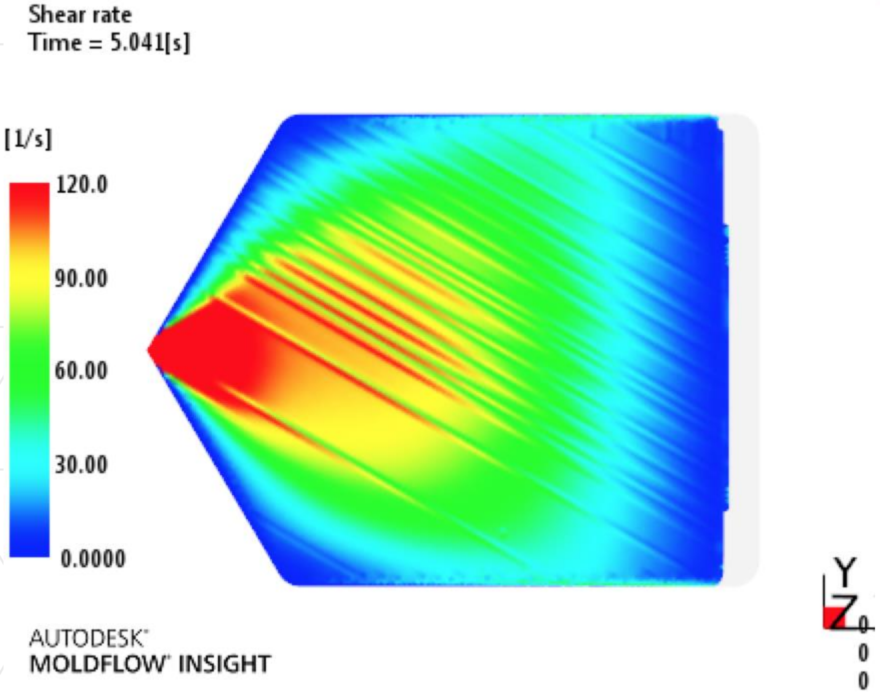
2017 New 3D Meshing Algorithms by Advancing Layers

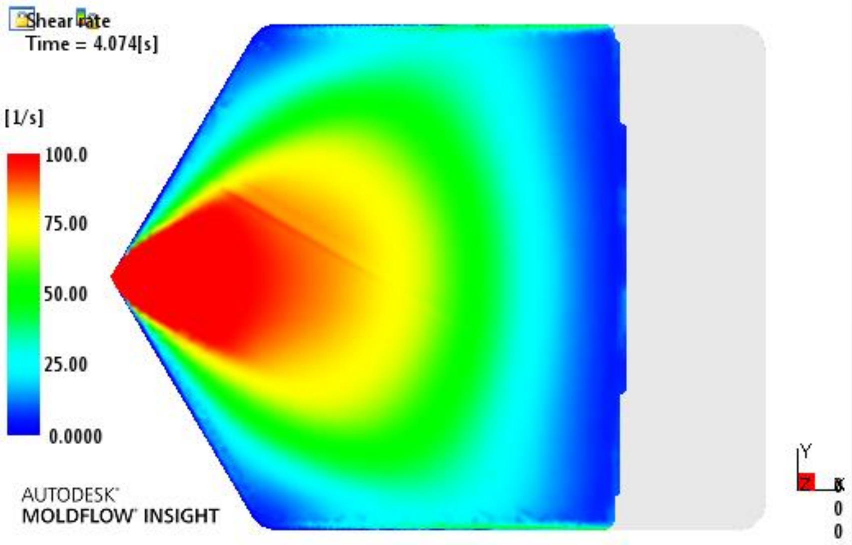
- More smooth near edges and corners (no bitter aftertaste)







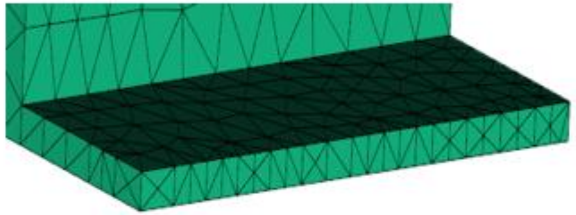


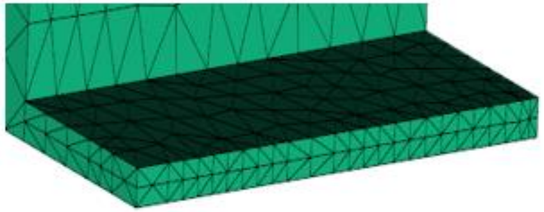


2017 New 3D Meshing Algorithms by Advancing Layers

AUTODESK.

- Extra refinement on edges





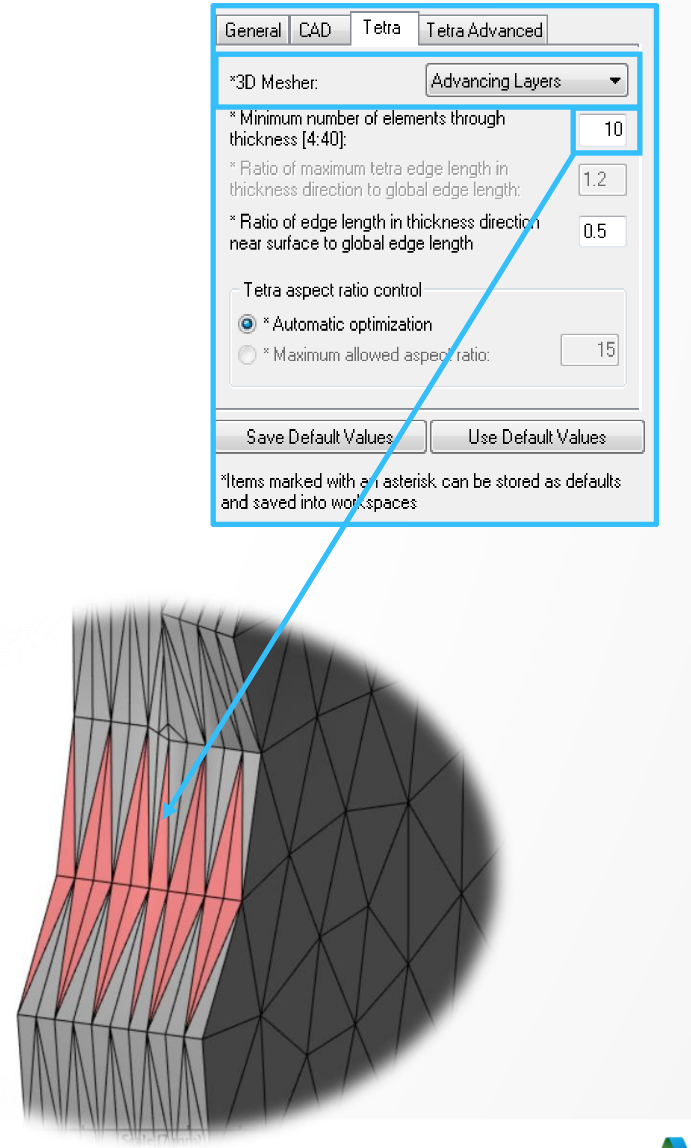
- Faster
  - For chunky models, the new 3D mesher is about 2 times faster than Moldflow 2016 for same layers. It can generate 1 million tetras in 3-4 minutes.

- Improved analysis accuracy – better mesh=better pizza

- The chance for users to repair 3D meshes is much lower.
  - In fact, it is not recommended to repair 3D meshes since 2017 release

2017 – Other Changes

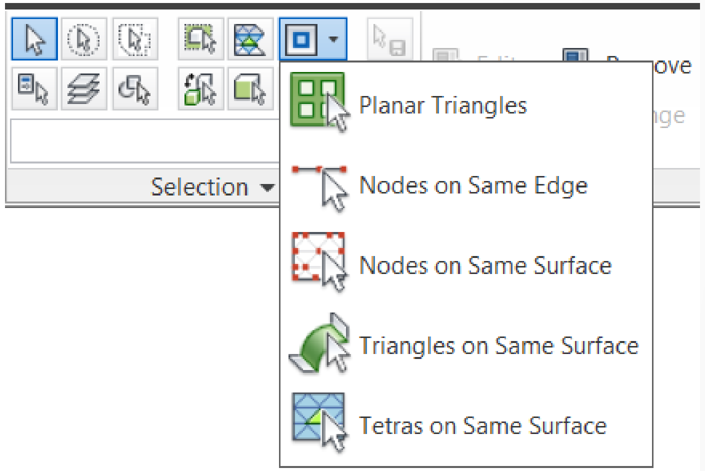
- 3D mesh default is now 10 layers through the thickness
- Can use Query tool to highlight CAD faces by labels

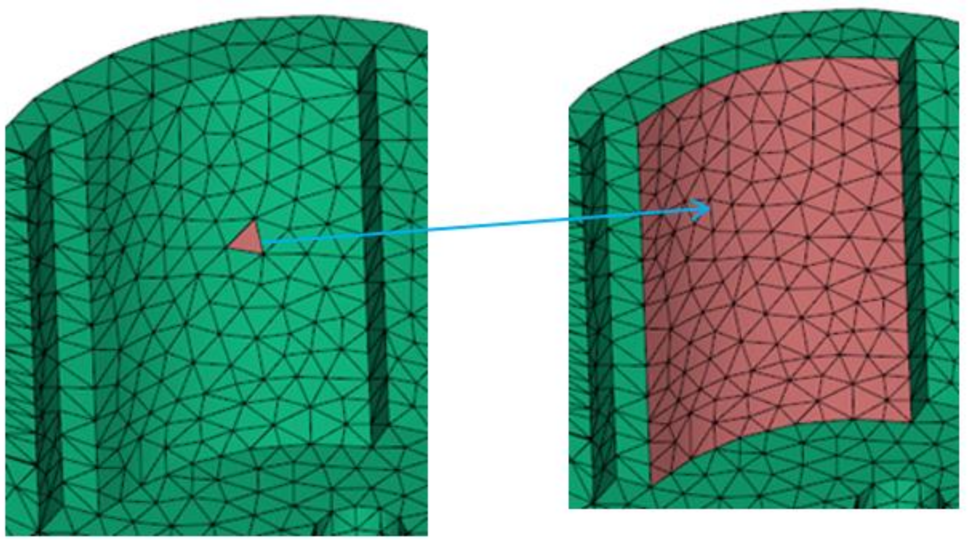


2017 – Mesh Selection Tools

- Select Nodes by Feature Edge
- Select Nodes by Surface
- Select Triangles by Surface
- Select Tetras by Surface

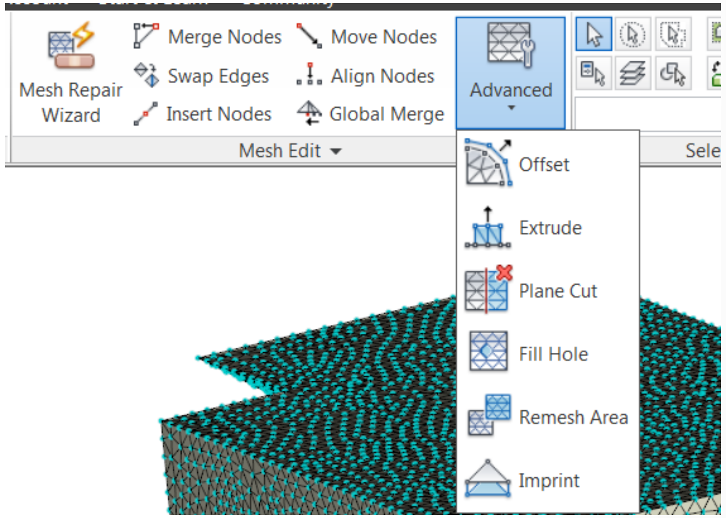
- All the 4 selection tools can work on curved surfaces. They can support multiple/incremental selections by holding "Ctrl" key

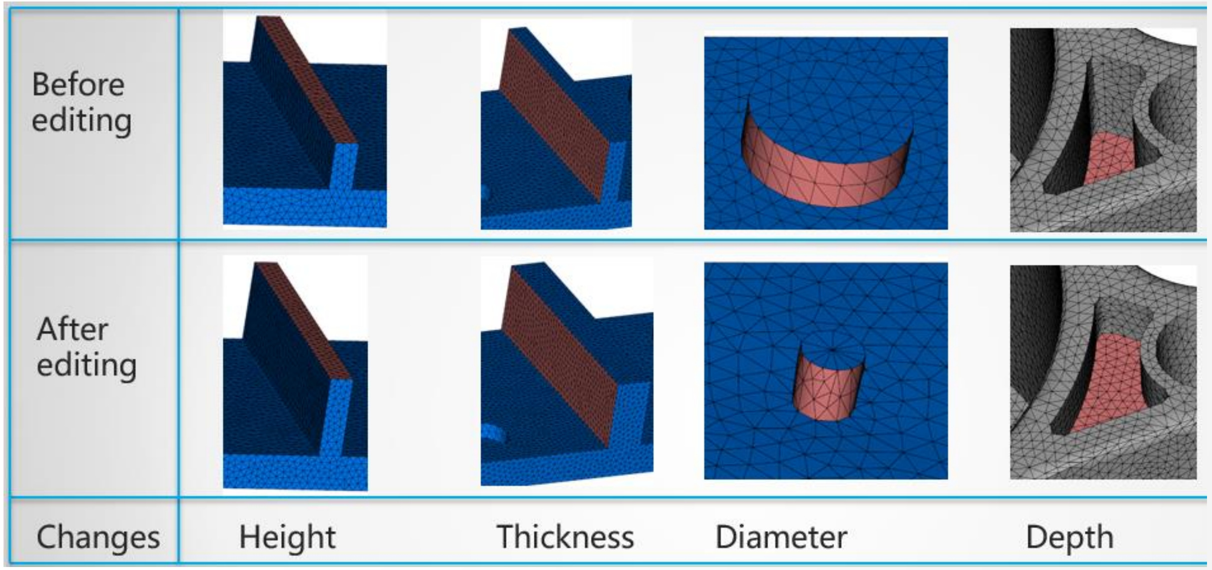




2017 - Advanced Mesh Editing Tools for Design Changes

- Offset
  - Select triangles and move them in surface normal direction.
- Extrude
  - Select triangles as base and create new features or new bodies.
- Plane Cut
  - Cut triangular meshes by XY, XZ or YZ planes
- Fill Hole
  - Fill holes by smooth patches based on surrounding elements.
- Imprint
  - Imprint existing surface meshes to increase match ratio.

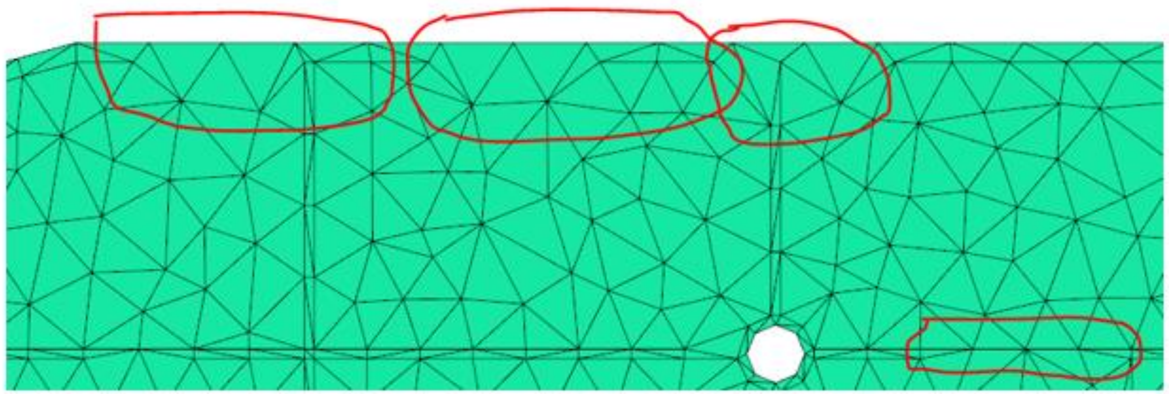




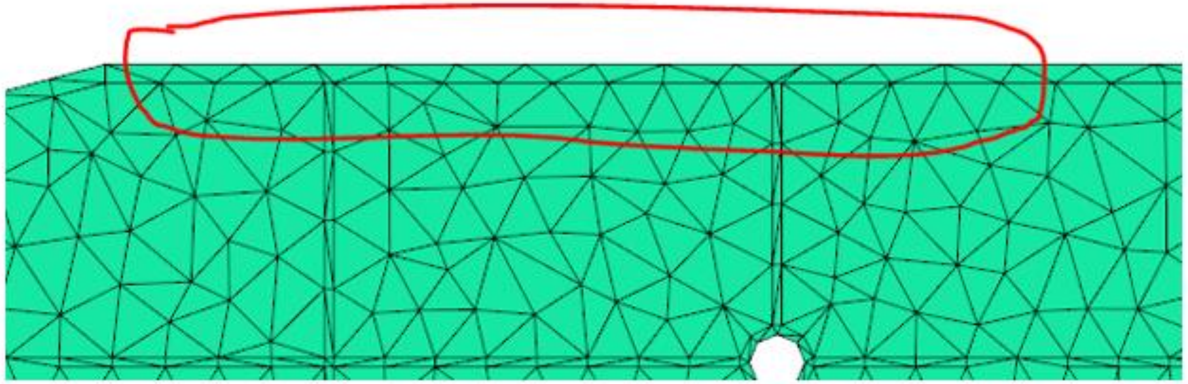
2017 - Mesh match near thin ribs

- Mesh matching has been improved for models with thin ribs

This is the result from Moldflow 2016 release:



This is the result from Moldflow 2017 release:



2017R2

2017R2 - CAD meshing on Linux

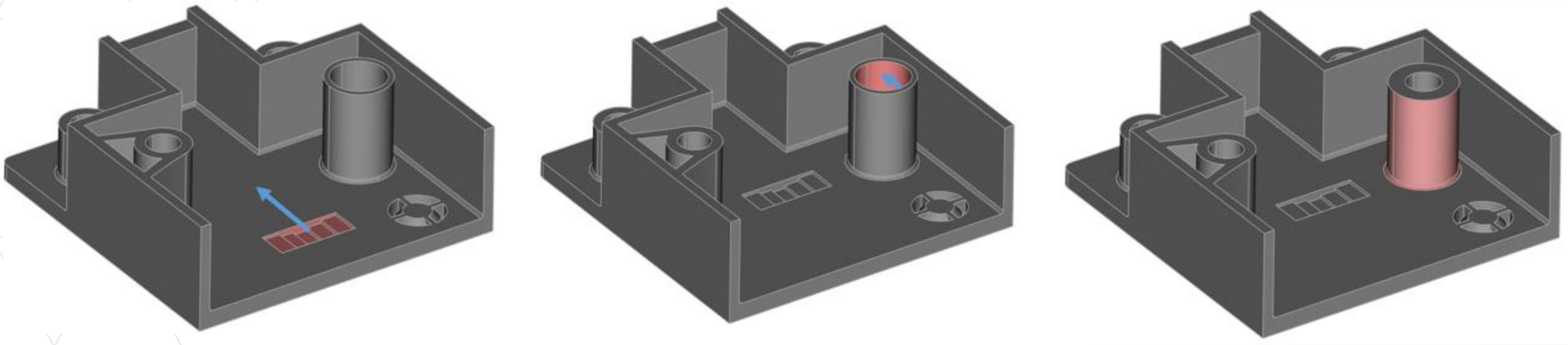
AUTODESK.

- Meshing on Linux
  - Supported by the following mesh types:
    - Dual Domain
    - 3D
  - Supported on the following Linux systems:
    - RedHat 6.5/CentOS 6.5 or higher
  - Benefits
    - Provides access to the same functionality as on Windows
  - Limitations
    - No support for 3D channels

AUTODESK.

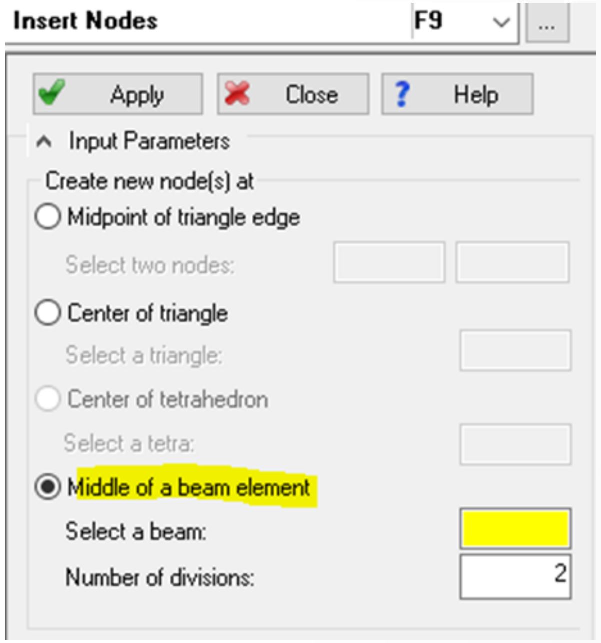
2017R2 – Direct Geometry Modification

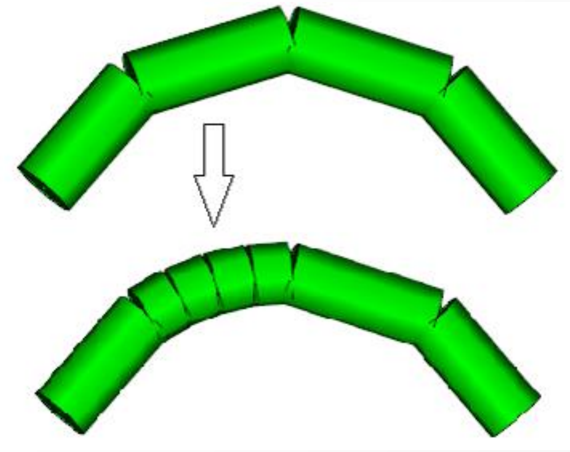
- Users can select CAD faces and modify geometry directly
- With DOE, Parametric Geometry Optimization is supported



2017R2 – Insert Nodes on Beam

- Users can split beam elements
- When the beam has underlying curve, new nodes will be created on the curve
- When the beam has NO underlying curve, new nodes will be created on the axis of the beam
- “Remesh Area” tool can split beams only when they have underlying curves

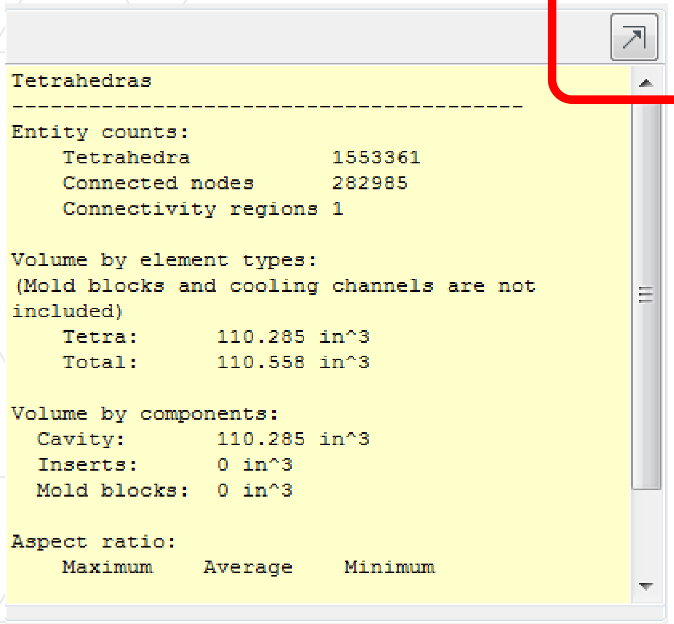


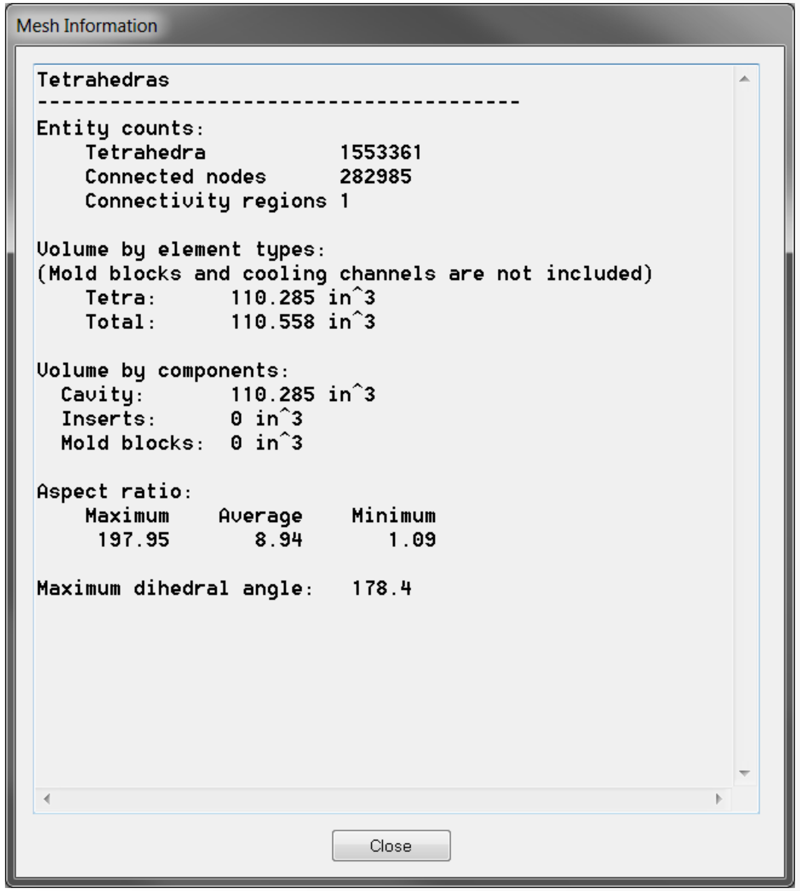


2017R2 – Mesh Statistics Pop-up

AUTODESK.

- Mesh statistics can be displayed as a larger popup window





2017R2 – Other Changes

- Users can specify the number of element layers for cores/inserts separately. By default, the number of element layers for cores/inserts is 6, while 10 layers for parts.
- Boundary conditions on nodes are protected, including trigger nodes of valve gates. They will not be lost when using Global Merge or Remesh Area tools.

Meshing: It's what we do

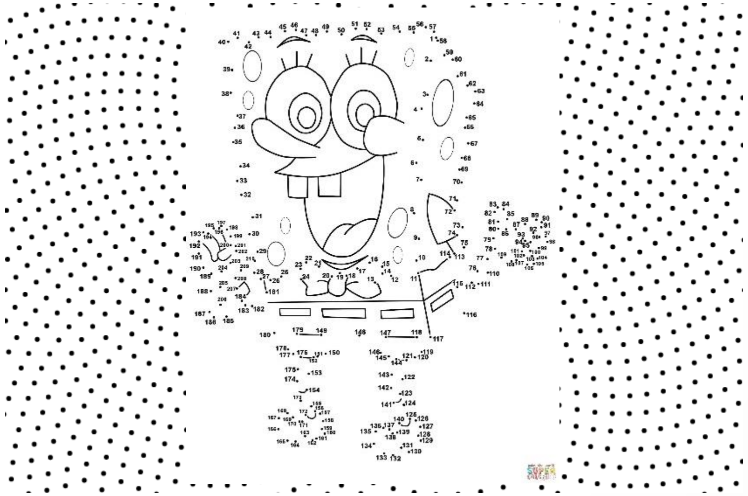
AUTODESK

What my friends think I do

What my mom thinks I do

What my daughter thinks I do

What my coworkers think I do



What I think I do

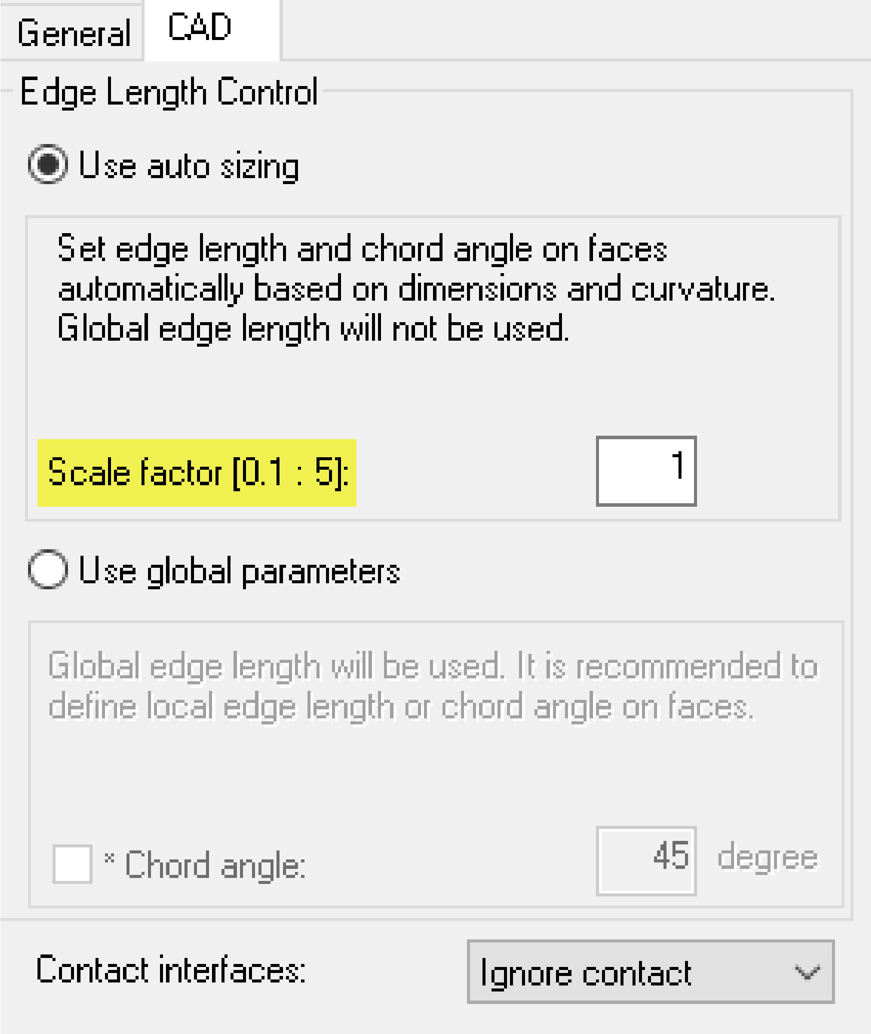
What I do with the **2017.3+ mesher**

2017.3

It‘s not R3, it’s .3

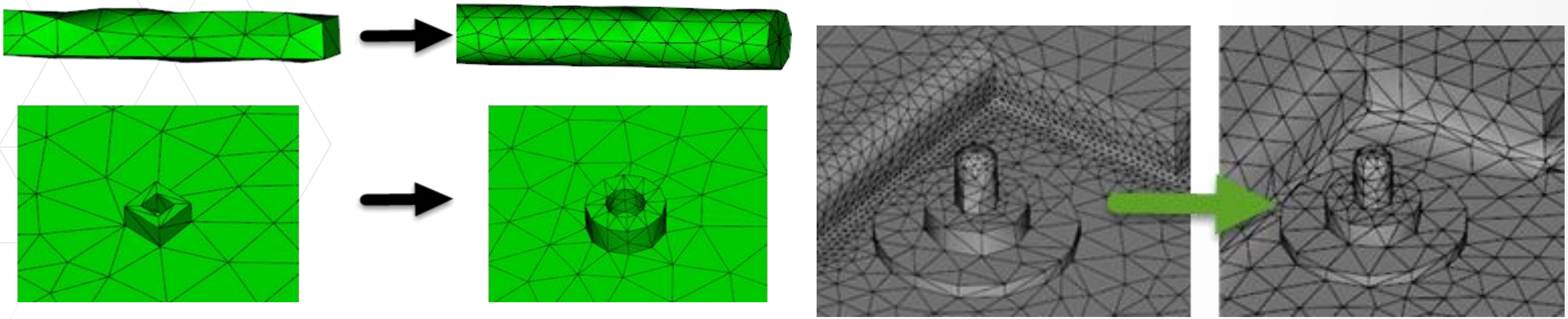
2017.3 - Auto-sizing for CAD Surface Mesh Generation

- On by default (Global edge length or global chord angle will not be used)
- Set local edge length and chord angle on CAD faces automatically without dramatic increase in element counts
- Users can adjust edge length by a “scale factor”
- No need to define local mesh density manually
- Auto-sizing does not overwrite user specifications



Auto-sizing for CAD Surface Mesh Generation

- For parts: smoother mesh, especially around high curvature area



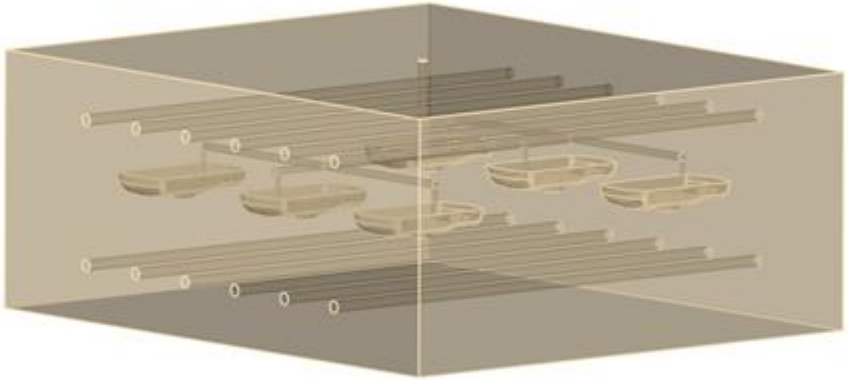
- No chord angle is applied on fillets in order to prevent an excessive fillet mesh density

Auto-sizing for CAD Surface Mesh Generation

- For mold blocks: large elements on external boundaries vs. smaller elements on internal boundaries
- Same edge length on surface meshes of assembly contact interfaces

Create CAD bodies for mold blocks

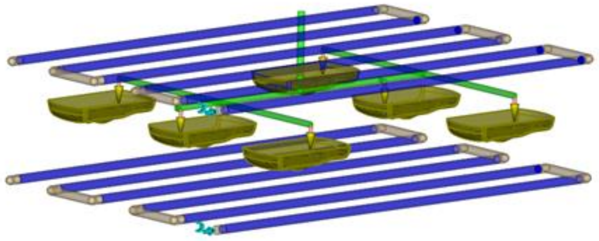
- Use the Mold Block Wizard to create a mold block around your CAD parts
  - <u>Non-default option, because it relies on a clean CAD model with no modeling errors</u>
  - Part and inserts **MUST** be in CAD
  - The cooling channels, and feed system can be CAD or beams with curves
  - Supported for Dual Domain and 3D meshes
  - Supported for analysis sequences that include Cool (FEM)
  - The CAD mold is constructed by subtracting the internal CAD components from a cuboid mold

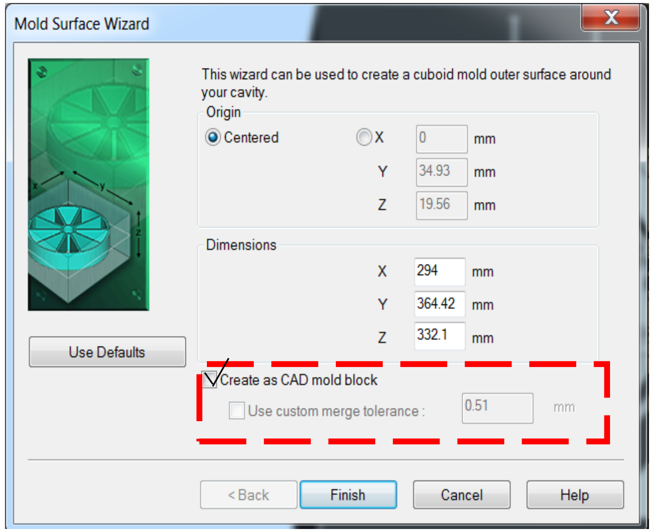


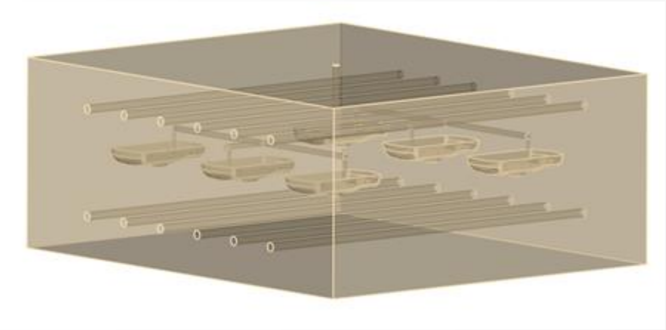
cuboid is a convex polyhedron bounded by six quadrilateral faces, whose polyhedral graph is the same as that of a cube.

Create CAD bodies for mold blocks

- Workflow
  - Ensure that the layers hosting the CAD components, runners and cooling channels are visible
  - Click (Geometry tab > Create panel > Mold block)
  - For clean models, select **Create as CAD mold block**. Otherwise generate the mold as regions
  - If you prefer, first create the mold block as regions, inspect and adjust the dimensions if necessary, and then create the CAD mold block. The mold block regions are then deleted.

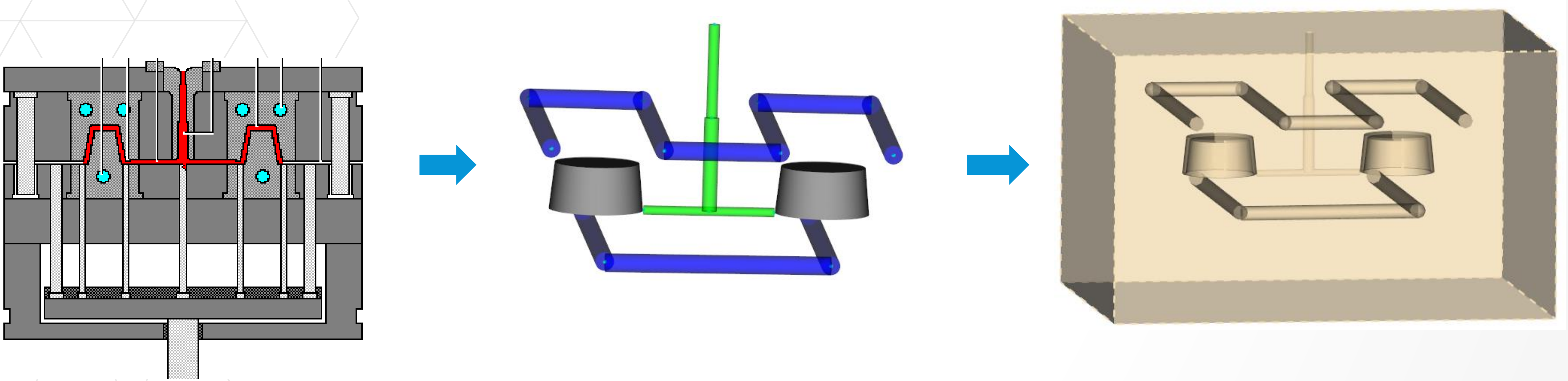






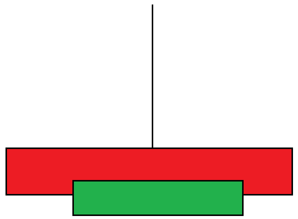
Create CAD bodies for mold blocks

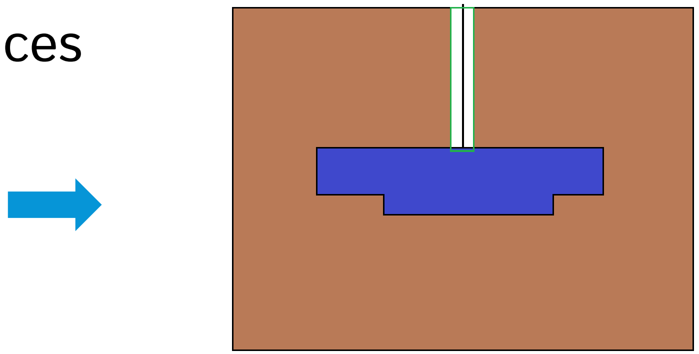
- Not recommended that you run simulations with real mold blocks
  - If you import a mold, delete all mold components, or make them invisible
  - Leave thermally significant components only



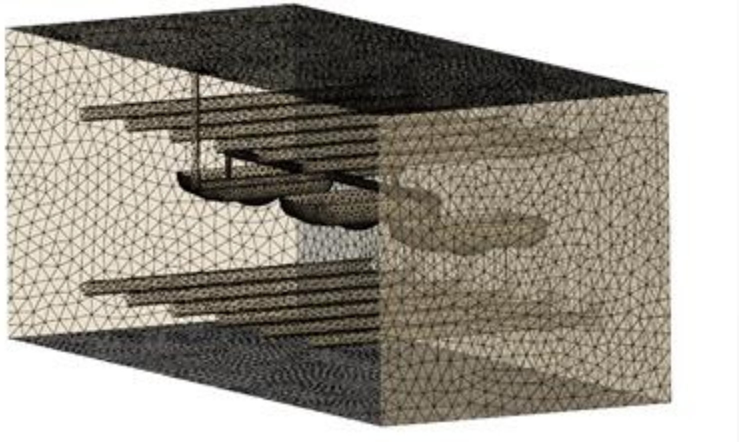
Create CAD bodies for mold blocks

- Advantages
  - No need to stitch contact interfaces



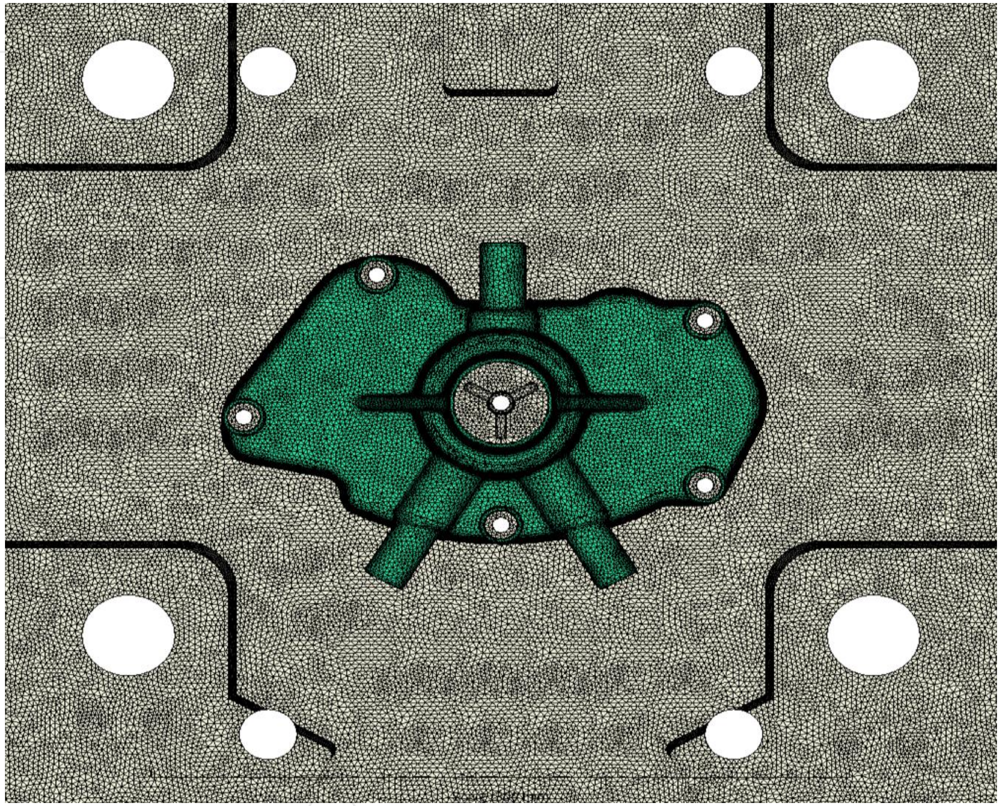


  - Meshing is fast because only those components relevant to the analysis are included
  - When you mesh with the mold mesher, large edge lengths are assigned to the mold external boundary, while the edge lengths on the mold internal boundary match the part edge lengths more closely

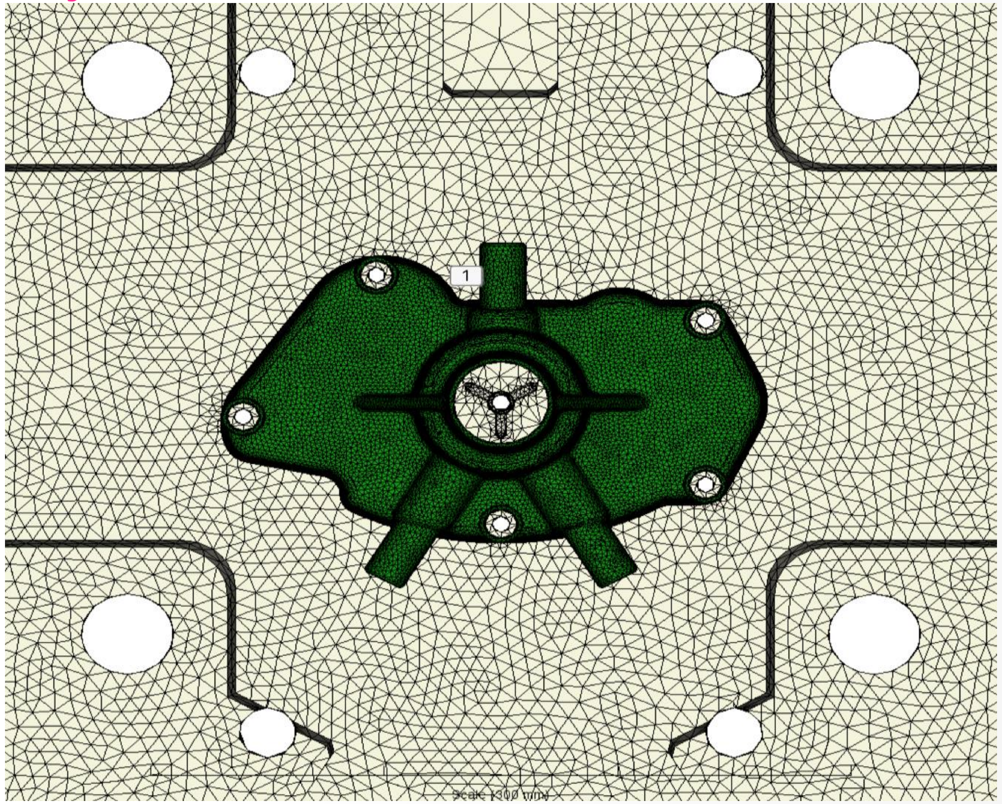


Auto Sizing for CAD

AMI 2017: One global edge length, no chord angle.
10 M Tets.



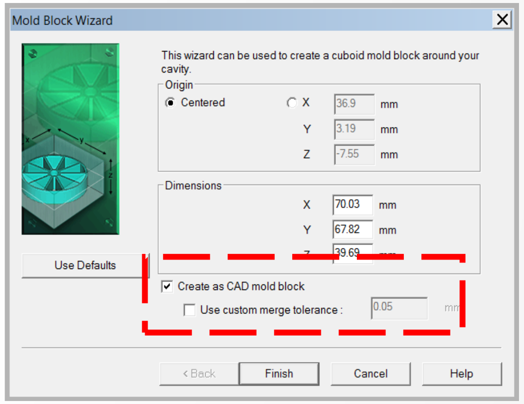
New: Separate global edge length for each body.
Chord angle on selected faces.
1.7 M Tets.
7 days to 4min

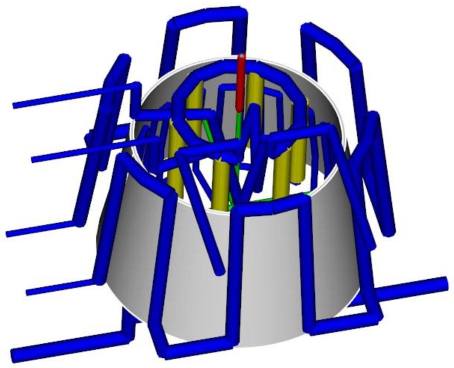


Create CAD Bodies for Mold Blocks

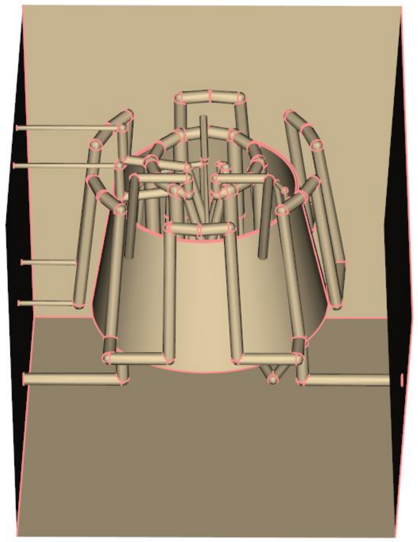
Previously, mold blocks are represented by regions.
Users need to stitch contact interfaces to form mold internal boundary

Now we can create CAD bodies for mold blocks.
Users do NOT need to stitch contact interfaces manually

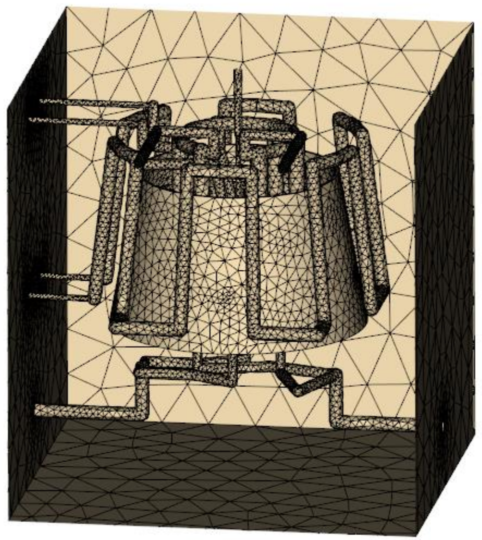




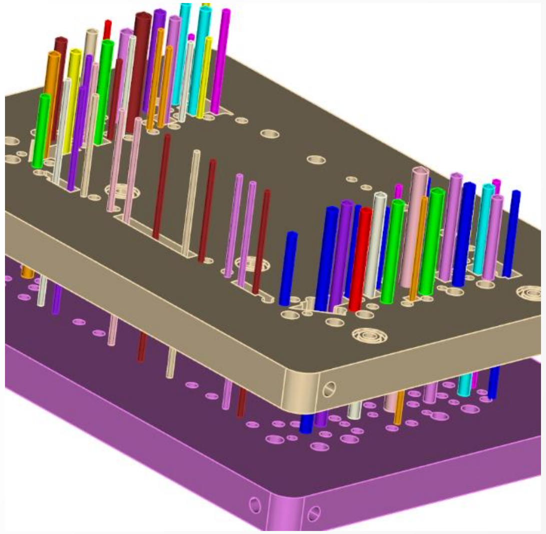
Internal components



CAD body for mold block

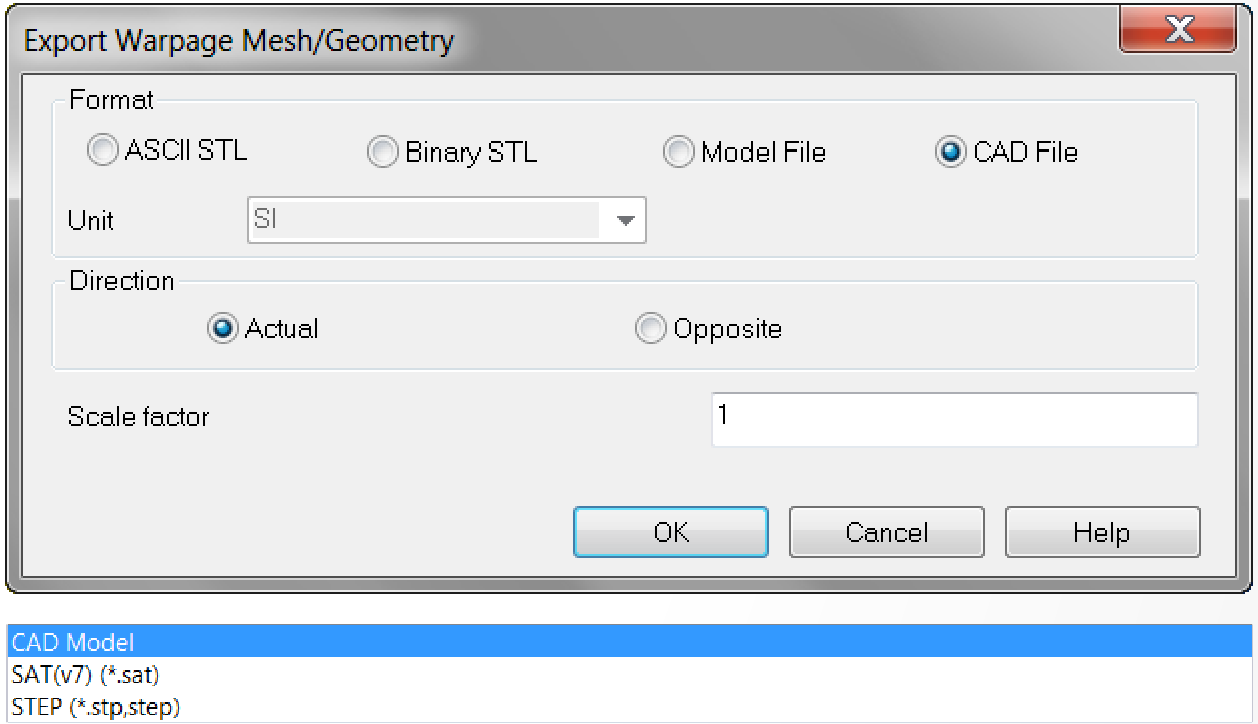


Surface mesh for mold block



2017.3 – Other Changes

- Warped shapes can now be exported as STEP or SAT

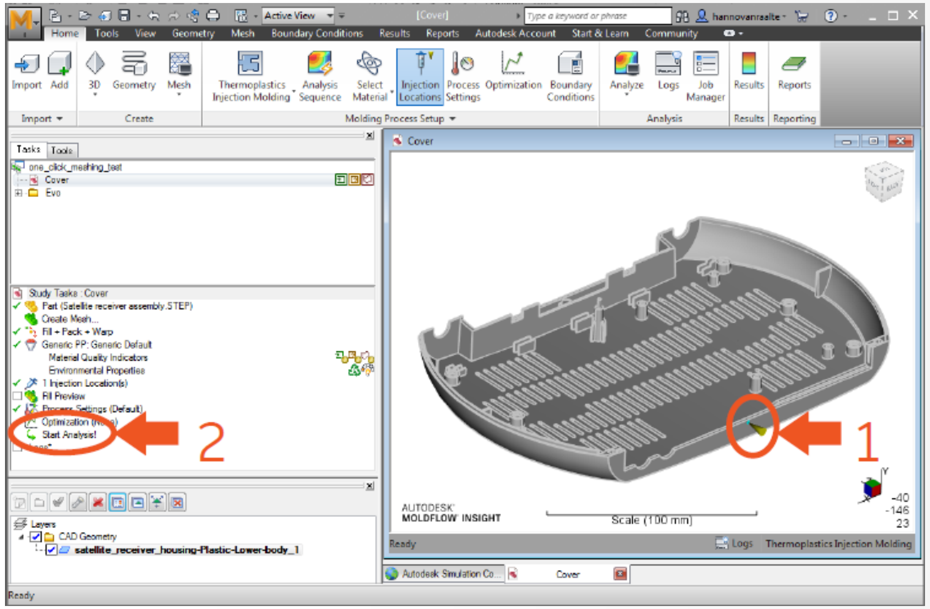


- When meshing curves, the minimum number of beams on baffles/bubblers is default 3. This can be changed in Curves tab on mesh UI.

2018.0

2018 - One Click Meshing and Analysis

- With one click you can now automatically mesh the model and launch the specified analysis together, saving time
- Boundary conditions are now applied to the imported CAD model
- New meshing algorithms apply an appropriate global mesh density and also refine the mesh around small features automatically



<u>Conventional workflow</u>

1. Import model without mesh
2. Mesh the model
3. Set required boundary conditions
4. Start Analysis

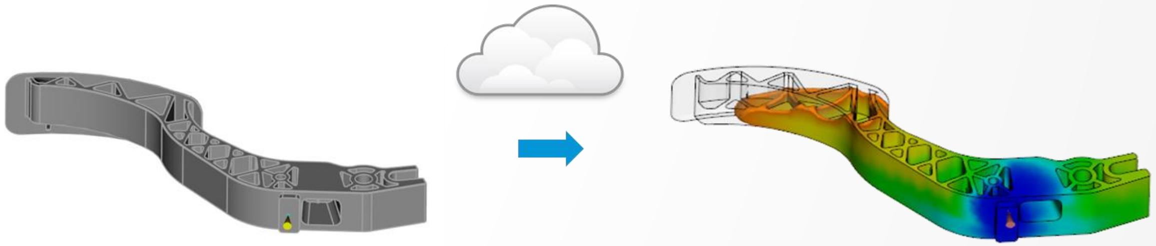
<u>One Click Meshing and Analysis</u>

1. Import model without mesh
2. Set required boundary conditions
3. Start analysis

2018 - One Click Meshing and Analysis Benefits

AUTODESK.

- Designers or engineers who are new to Moldflow: Run and check results quickly with less learning, preprocessing
- Experienced users: Launch an analysis (mesh + solve) conveniently by end of his work day and check result tomorrow
- Seamless workflow with Geometry Optimization
- Works on any model: CAD, IGES, STL
- Supports all mesh types:
  - Midplane (*except CAD body), Dual Domain, 3D
- Supports most conventional analysis sequences
- Close studies or client during mesh / solver running
- Cloud, API and batch queue support



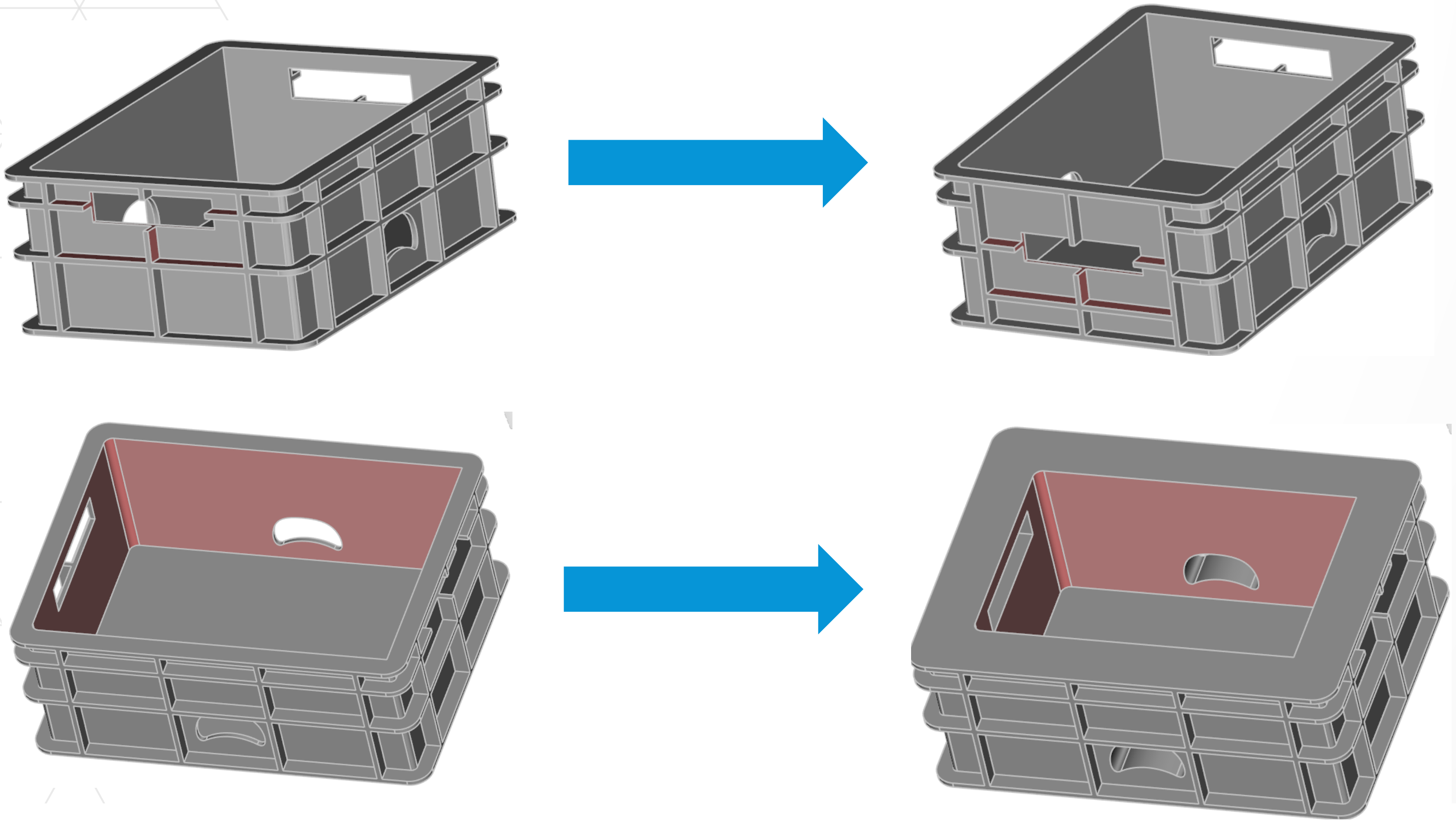
2018 - One Click Meshing and Analysis Limitations

- Accuracy:
  - Default mesh settings may not guarantee required accuracy
  - Review mesh generated & results for proper accuracy
- Does not support:
  - Boundary Conditions :
    - Prohibited gate locations, surface loads, dispensing controller, valve gates controllers
  - Analysis sequences:
    - Cool (FEM), Coreshift, Thermoset Dispensing
  - 3D runner + Part CAD Geometry
  - Mixed CAD and Mesh
  - “Runstudy” can’t launch Mesh (both Windows/Linux)
  - Some Mesh settings are not recorded: such as Auto Sizing Scale Factor

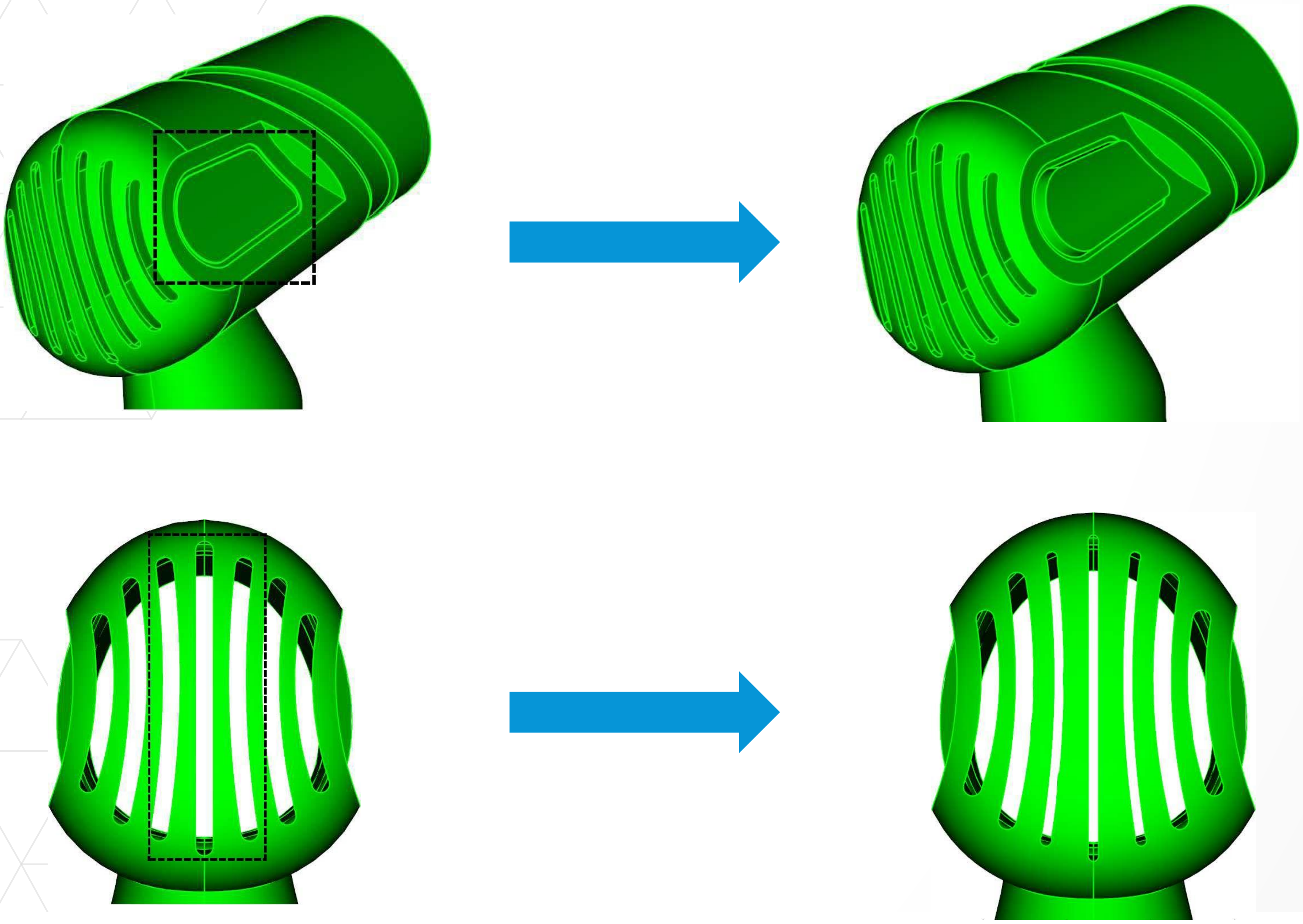
Note: These are just limitations for One click meshing and Analysis, not for conventional mesh and analysis.

Geometry Deformation Enhancements

- Large deformation – Multistep improvements
- Allows large deformations that were not possible before

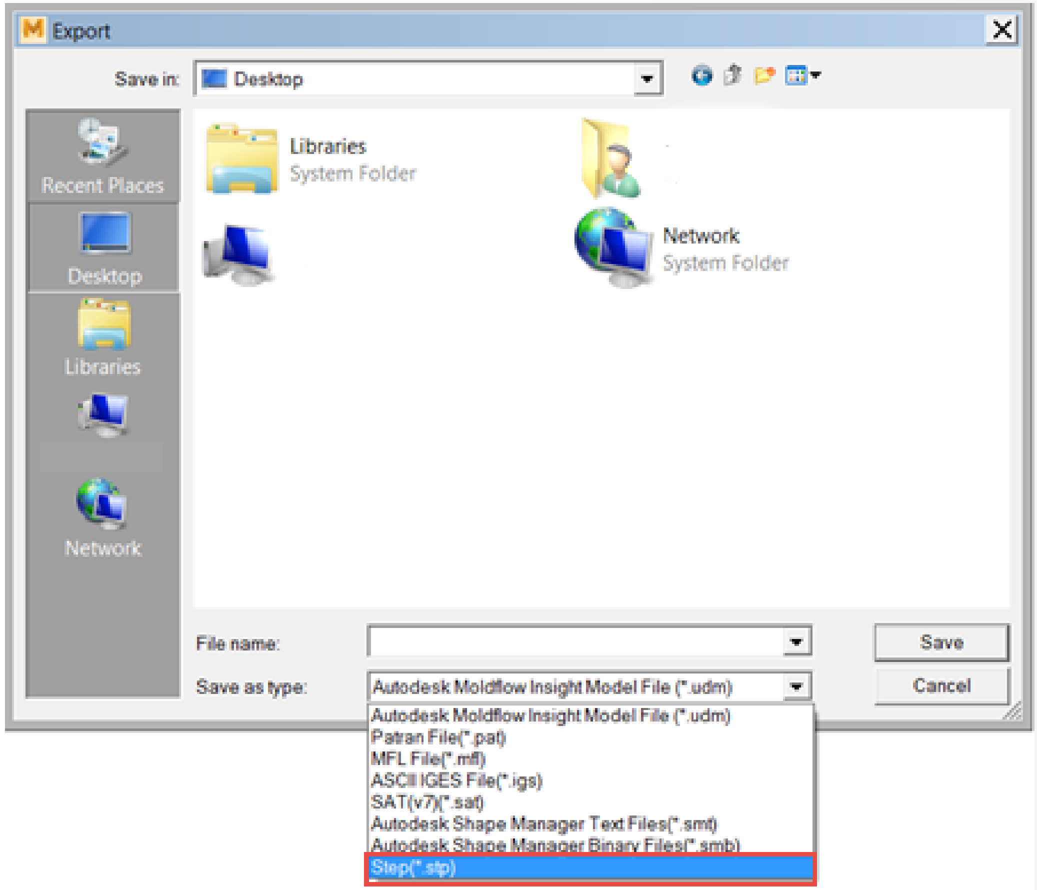


Geometry Deformation Enhancements



CAD Export As STEP

- CAD models can now be exported in STEP (*stp) format
- The CAD bodies must be visible
- Multiple CAD components in an assembly are grouped together and save as one STEP file
- Geometry transformation performed in Synergy, is preserved when you open the STEP file the CAD package
- Limitations:
  - The layer name, and any custom colors assigned in the layers, are not preserved



AUTODESK®

Make anything.