... within Epsilon

Twin Cities
ANSYS® User Meeting

May 2015

Workbench v16 Update

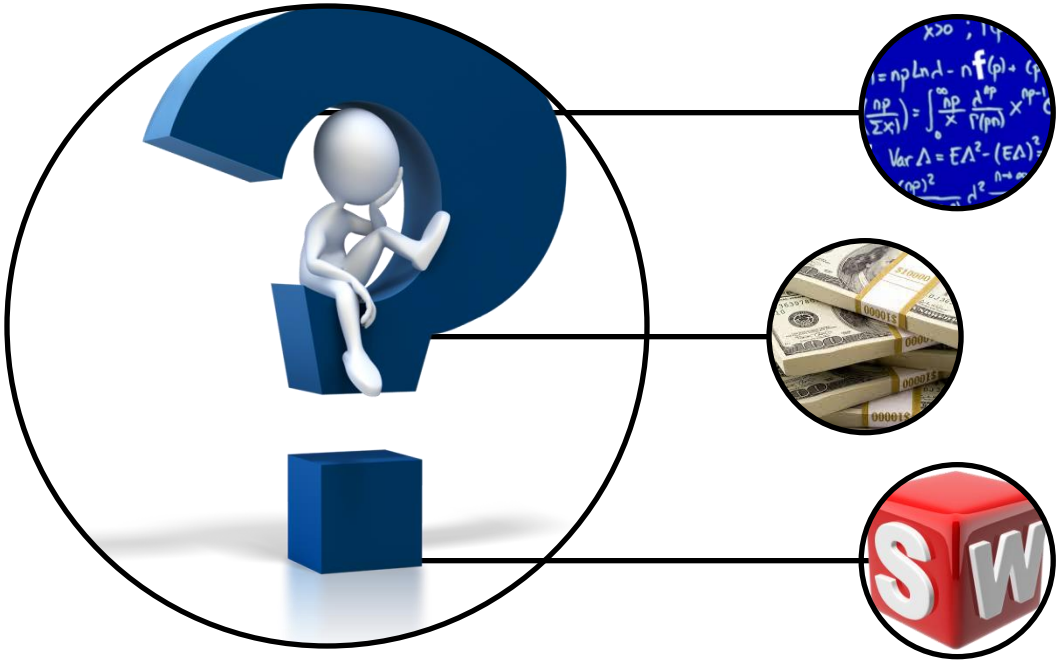
... within Epsilon

Agenda

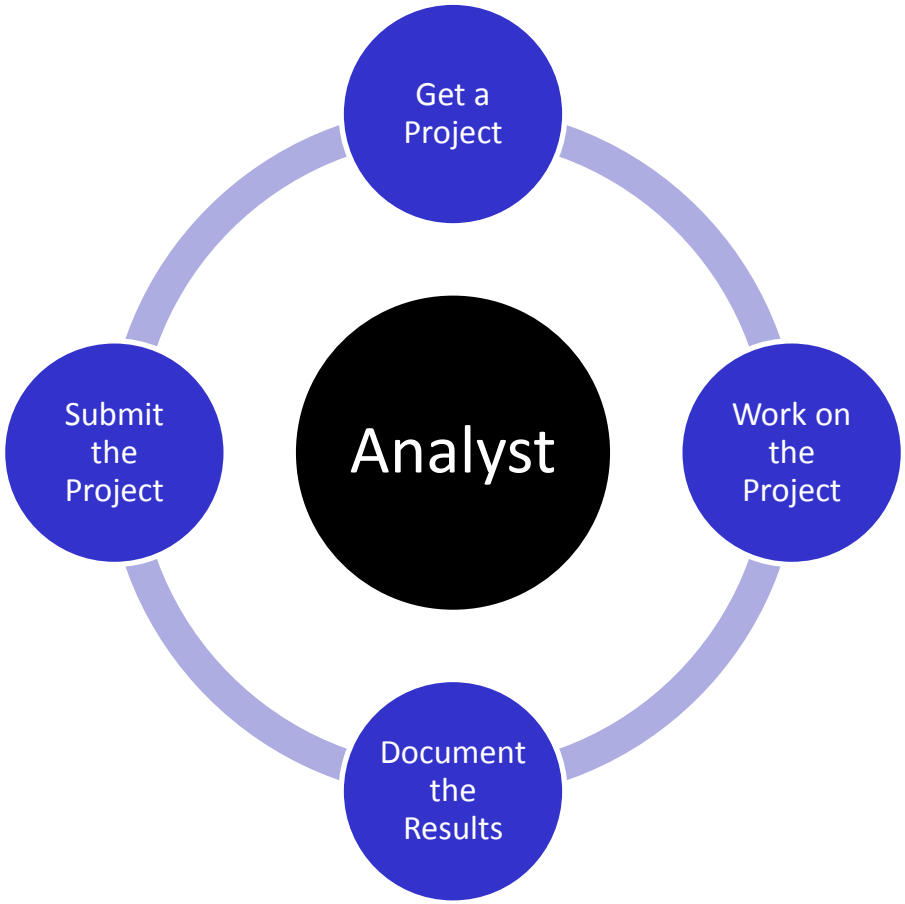
1. Usability Enhancements
2. Adaptive Meshing
3. Weld Connections
4. Spaceclaim
5. Material Modeling Improvements
6. V15 reminders

Moore's Law

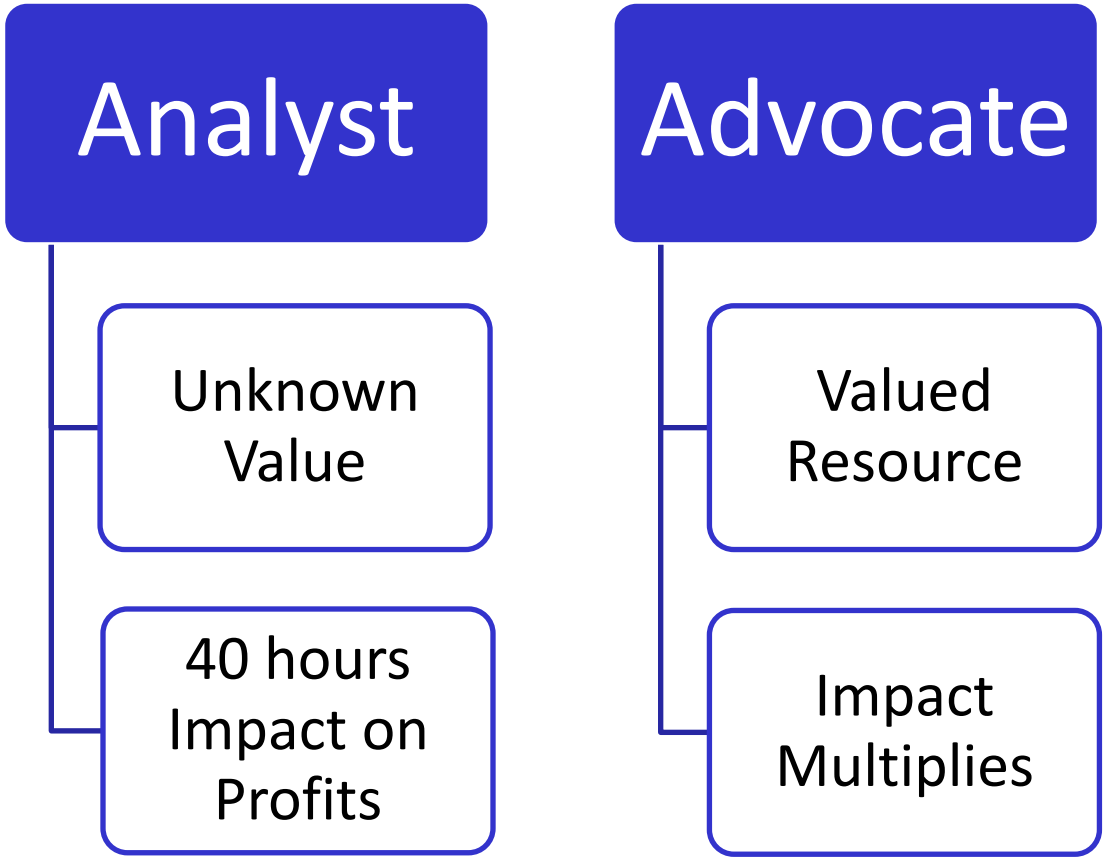
Simulation is increasingly underutilized!



Analyst vs Advocate



So what's the difference?



With more projects, whose expertise can you leverage?

Use Epsilon as a Resource!

- **Design**
  - Initial Analysis
  - Parametric Studies
- **Prototype**
  - Redesign
  - Failure Modes
- **Sustain**
  - Design Improvements
  - Reduce Costs

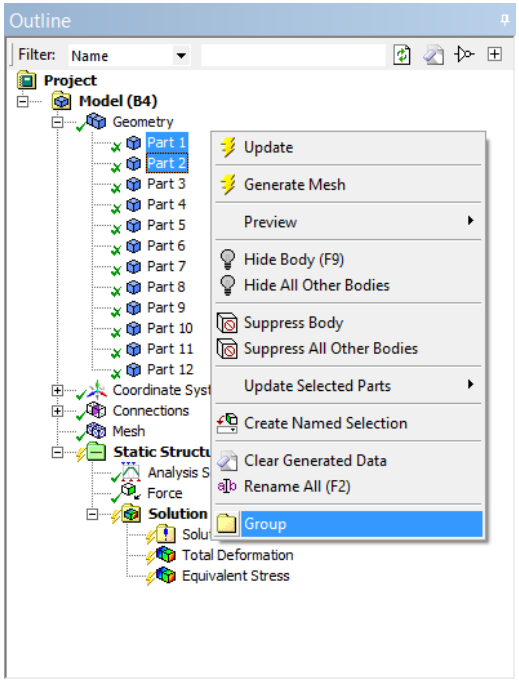
What's the Next Step?

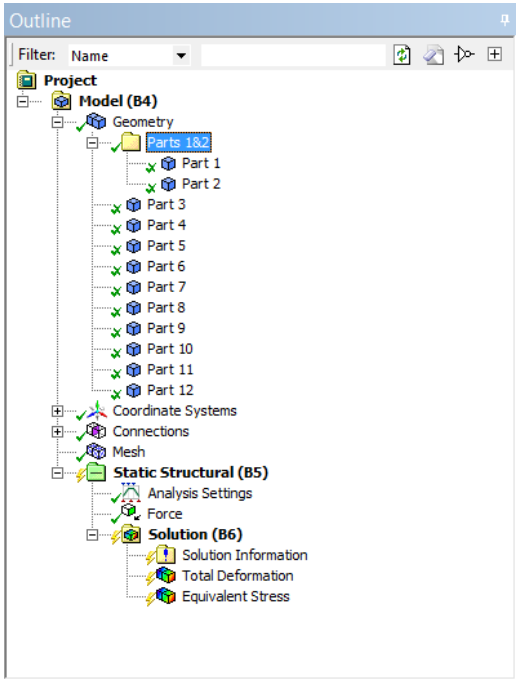
- Let's get an NDA in place
  - Just a phone call away
- Discover Opportunities for Simulation
  - **Epsilon has a proven approach**
- Become a Champion of FEA
  - Promote FEA as a necessity

Usability Enhancements

Folders

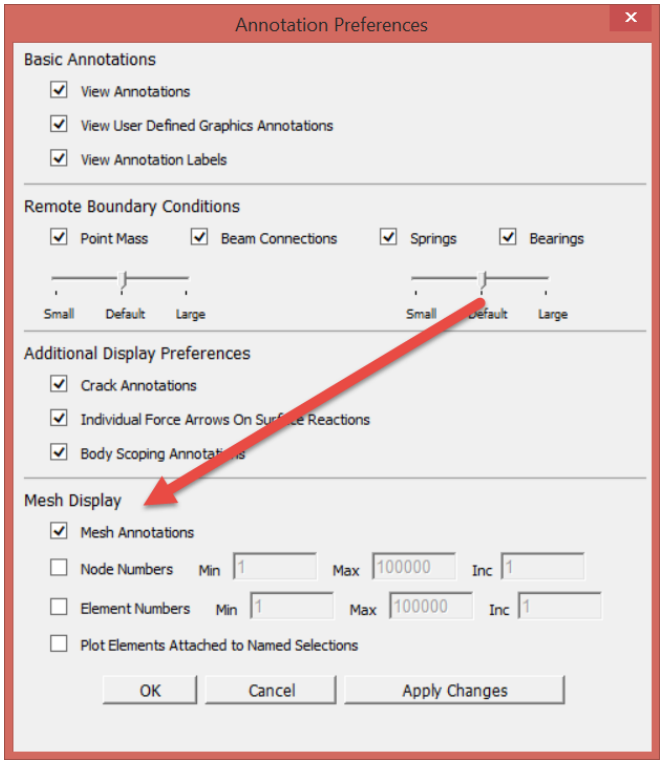
- Parts, connections, boundary conditions, etc. can be organized into groups
- Easily suppress whole groups without hunting for each item





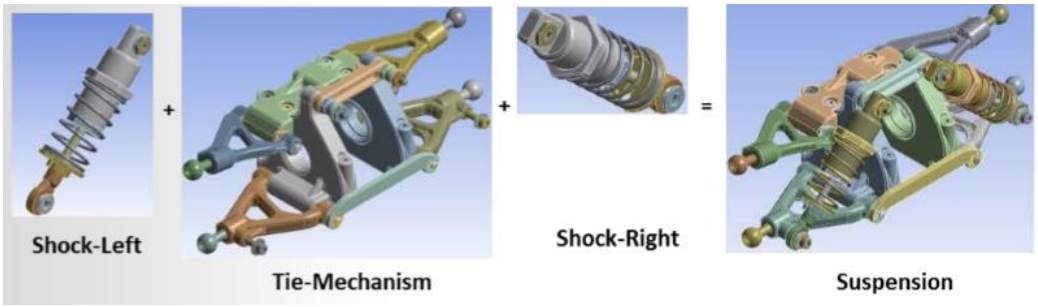
Node/Element Numbers

- Display Node/Element numbers through “Annotation Preferences”
- Show specific range of node/elements on the model if needed.

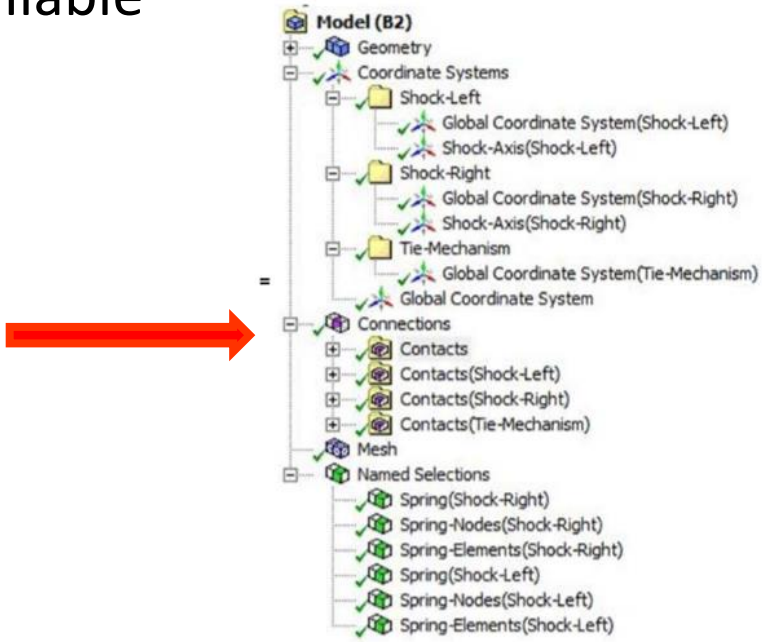


Sub-Assemblies

- Improved assembly management
  - Can import more details from sub-models
  - Improved detail management from folders
  - “Preview Assembled Geometry” available



From ANSYS V16 Release Notes



From ANSYS Release Notes

Face

- Now you can split faces using planes and surfaces!

B: Static Structural - DesignModeler

File Create Concept Tools Units View Help

Undo Redo Select:

Plane5 None Generate Share Topology Parameters

Extrude Revolve Sweep Skin/Loft Thin/Surface Blend Chamfer Slice

Point Conversion

Tree Outline

- B: Static Structural
  - XYPlane
  - ZXPlane
  - YZPlane
  - Extrude1
    - Sketch1
  - Plane5
  - 1 Part, 1 Body

Sketching Modeling

Details View

Details of Plane5	
Plane	Plane5
Sketches	0
Type	From Plane
Base Plane	XYPlane
Transform 1 (RMB)	Offset Z
FD1, Value 1	5 m
Transform 2 (RMB)	None
Reverse Normal/Z-Axis?	No
Flip XY-Axes?	No
Export Coordinate System?	No

Graphics

ANSYS R16.1

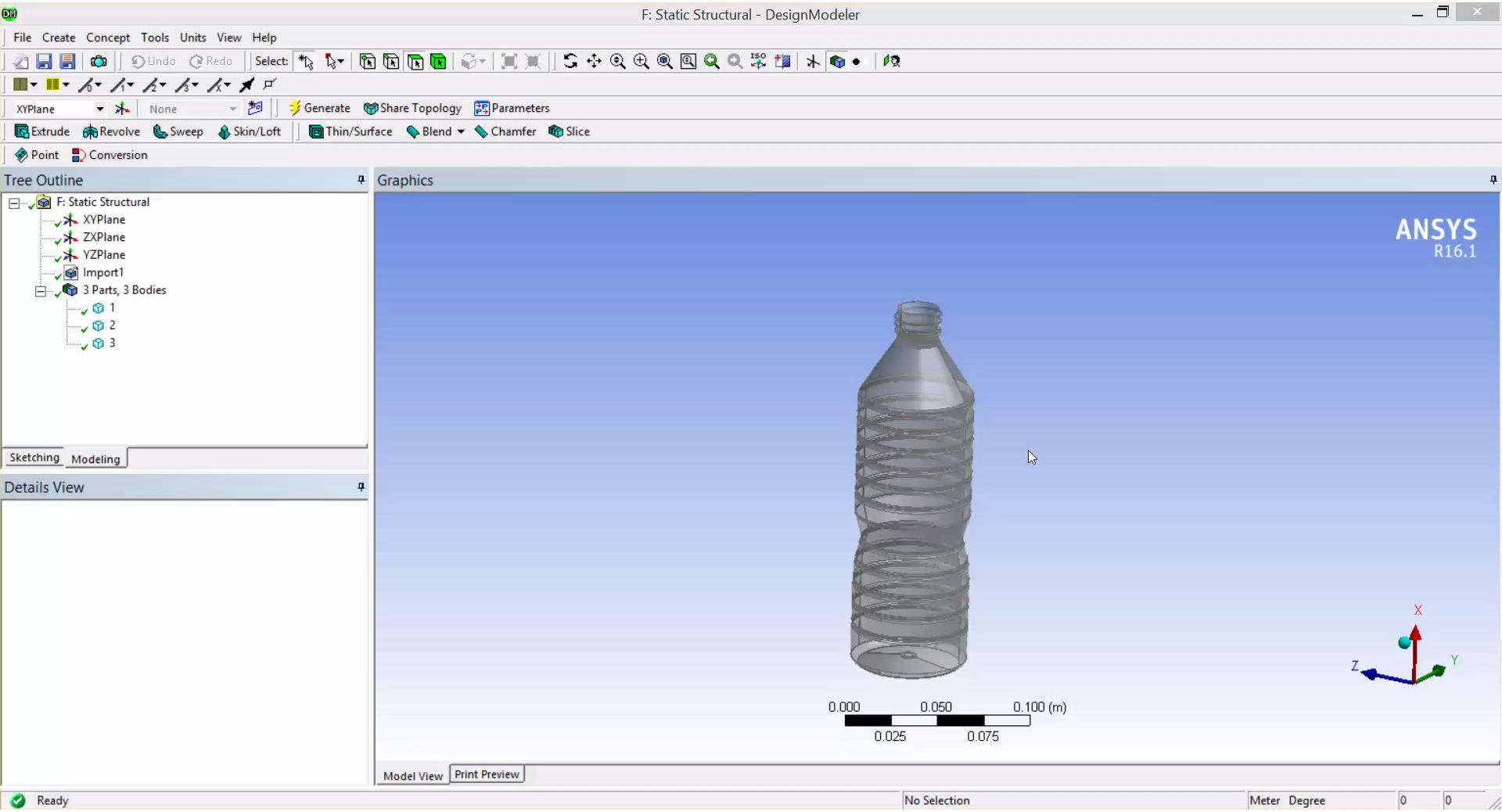
0.000 15.000 30.000 (m)

7.500 22.500

Model View Print Preview

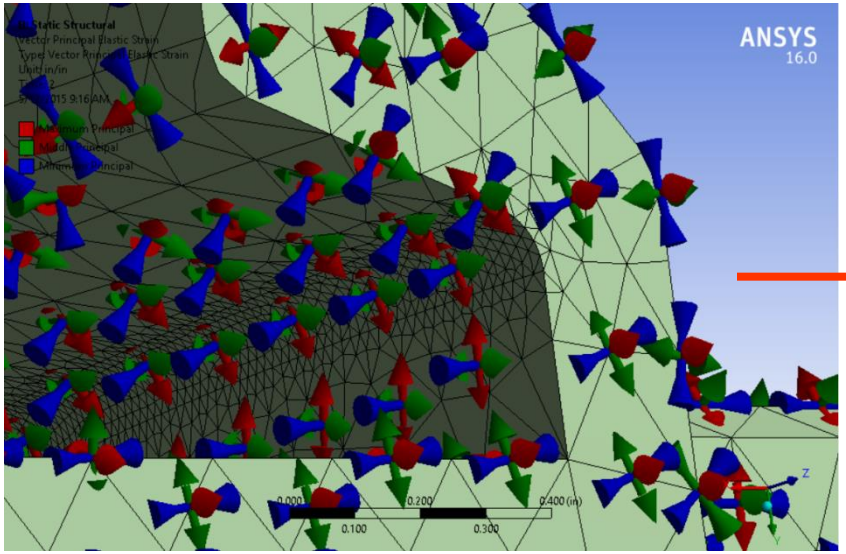
Ready No Selection Meter Degree 0 0

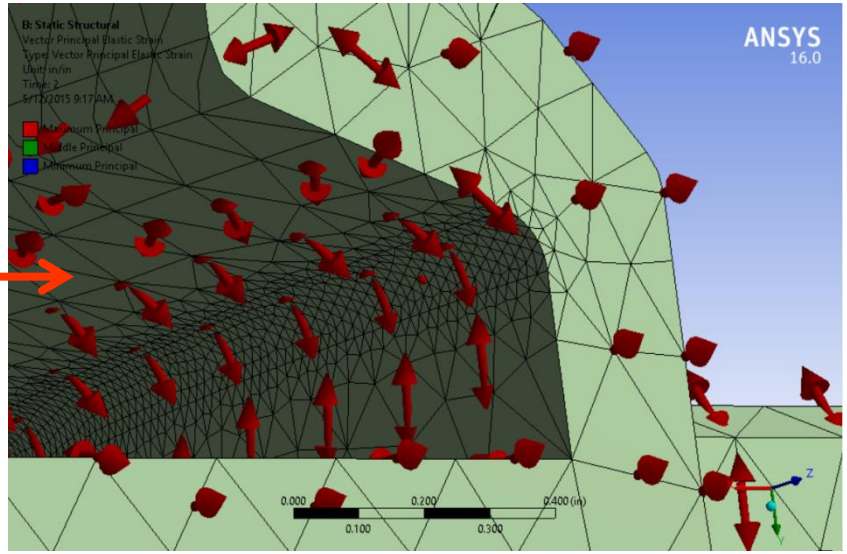
Detach and Body Repair



Vector Plotting controls

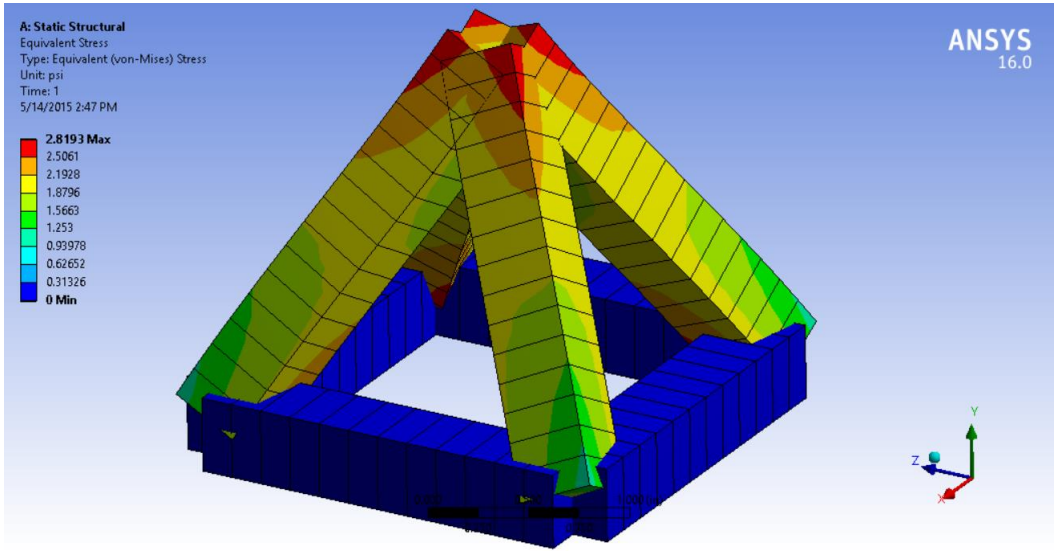
- Vector Plotting allows suppression of principals





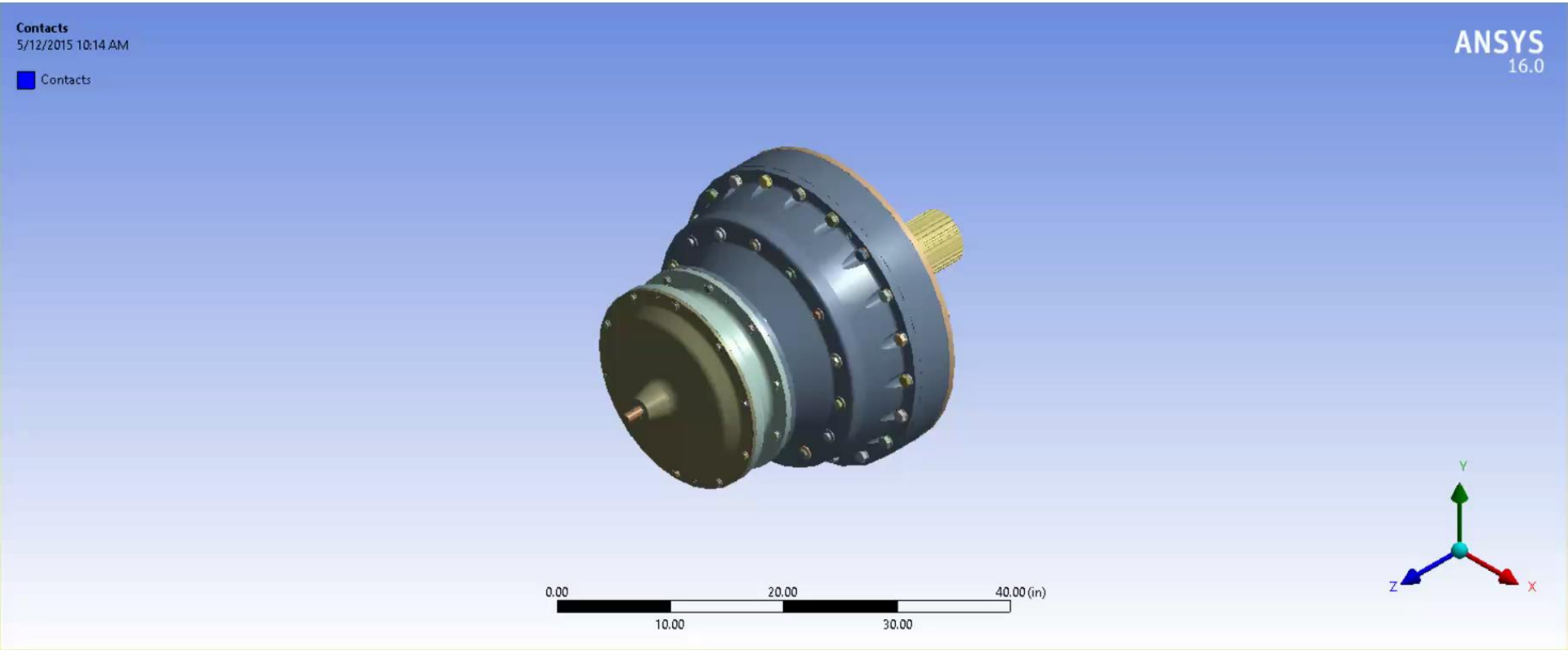
Analysis Options

- Tabular participation factor data for modal analyses
- Results Option: set default for calculating time history
- Can now plot stresses and strains for line bodies



Misc.

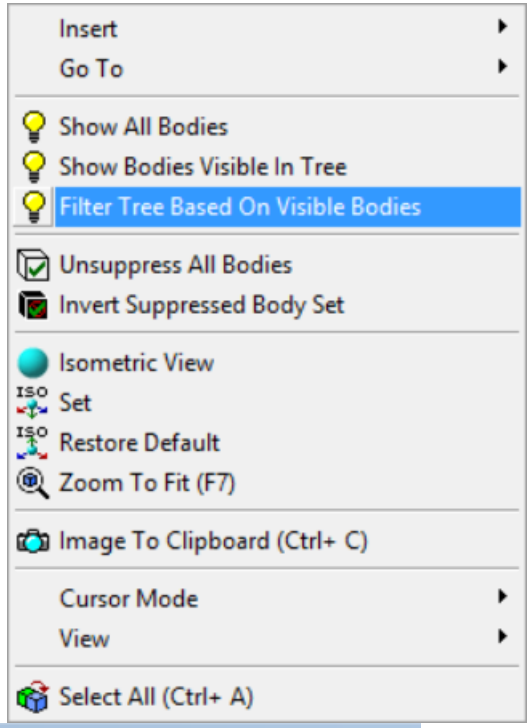
- Explode!



From ANSYS Release Notes

Misc.

- Can open Mechanical in read-only mode (avoids tying up license)
- Filter Tree Based on Visible Bodies
- Selection Information tells you distance between selected items



Selection Information

Coordinate System: Global Coordinate System | Show Individual and Summary

Entity	Distance Between Centroids (in)	Length (in)	Centroid X(in)	Centroid Y(in)	Centroid Z(in)	Body	Type	Radius (in)
Distance	0.25		0.	0.25	7.8552e-017			
2 Edges, Summary		10.996	3.0655e-017	0.125	0.			
Edge 1		5.4978	3.0655e-017	0.	-3.9276e-017	skinnyplate\Solid	Circle	0.875
Edge 2		5.4978	3.0655e-017	0.25	3.9276e-017	skinnyplate\Solid	Circle	0.875

From ANSYS V16 Release Notes

Misc.

- Thermal Fluid Line Bodies: Import pipe volumes as line bodies to model thermal fluid flow

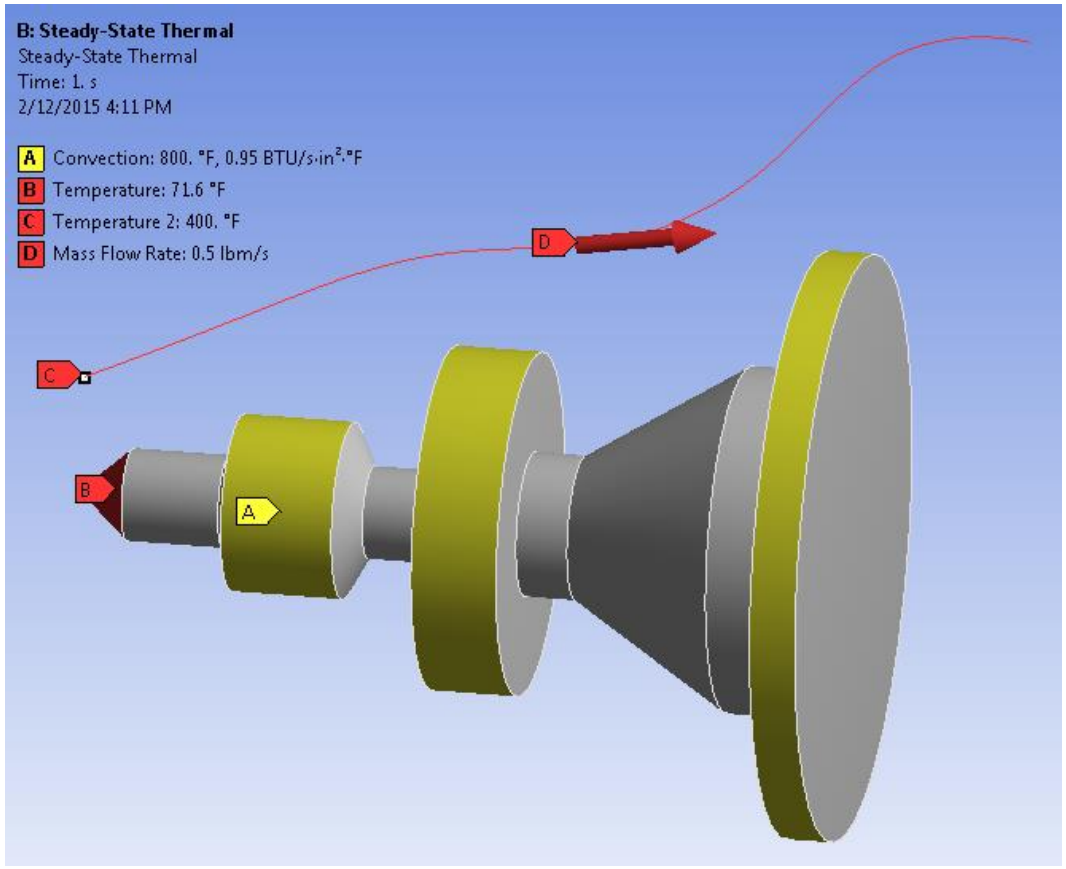


Image courtesy of PADT

Misc.

- Buckling: Now includes nonlinear effects, renamed Eigenvalue Buckling
  - Incremental loads may now be added after static pre-stress
  - Joints now supported in linear buckling

Ultimate buckling load of the nonlinear buckling is calculated as follows:

$$F_{buckling} = F_{restart} + \lambda F_{perturbed}$$

$F_{buckling}$	ultimate buckling load
$F_{restart}$	load applied at start point
$F_{perturbed}$	incremental load applied in buckling given prestressed effect resulted from $F_{restart}$
λ	load multiplier

From ANSYS Release Notes

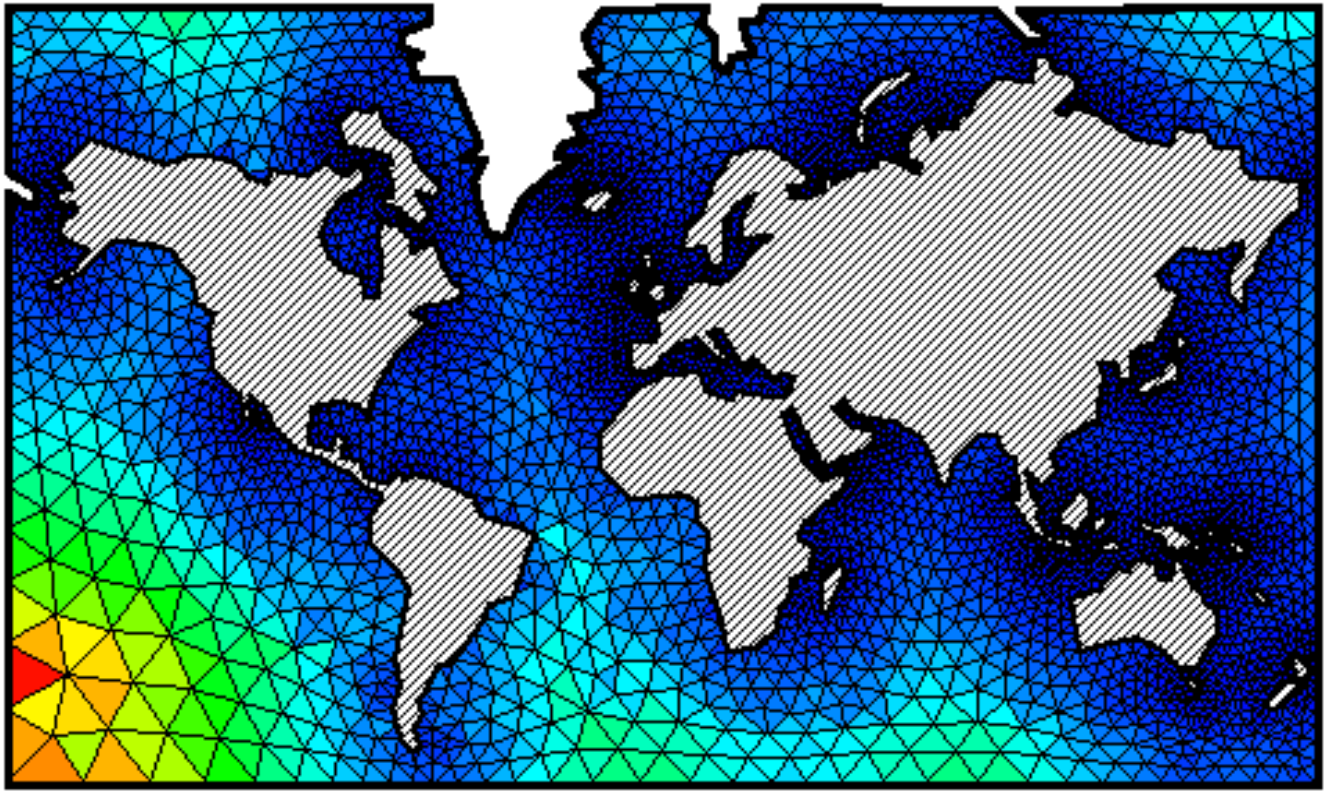
Misc.

- Distributed Mass
  - Paint bodies, or surfaces with specified mass
- Element Orientation—Use surface/edge guides
  - Can adjust normals within Mechanical – but operates on entire body
- Joint Element Control – define elements used in joints
  - Contact or MPC184's
- Solver Pivot Checking – continue solving when underconstraint detected
  - V15 became more sensitive to pivot errors
  - V16 let's us better ignore these errors
  - See PIVCHECK MAPDL command)

Misc.

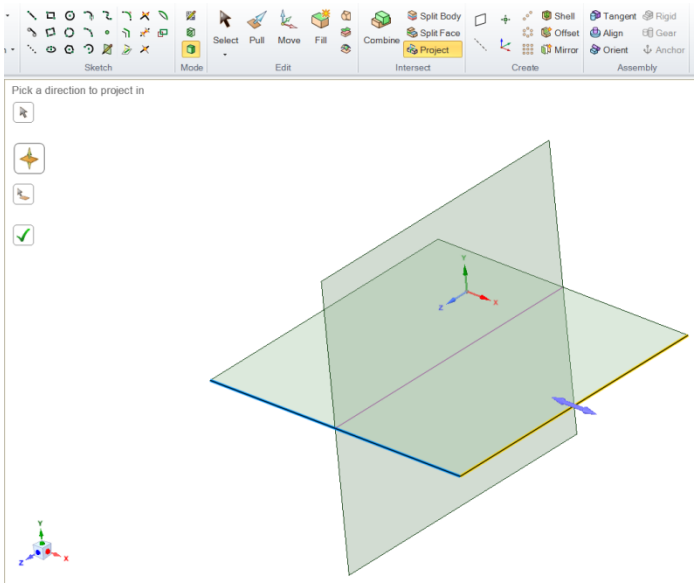
- IGES Import Instancing
  - Can reuse meshing for instanced bodies, for example
- DM – Upgrade Feature Version
  - Can choose whether to update legacy .agdb's
- DM – sort parts alphabetically
- Geometry attachment up to 2x faster
- Multi-threaded (parallel CPU) contact detection

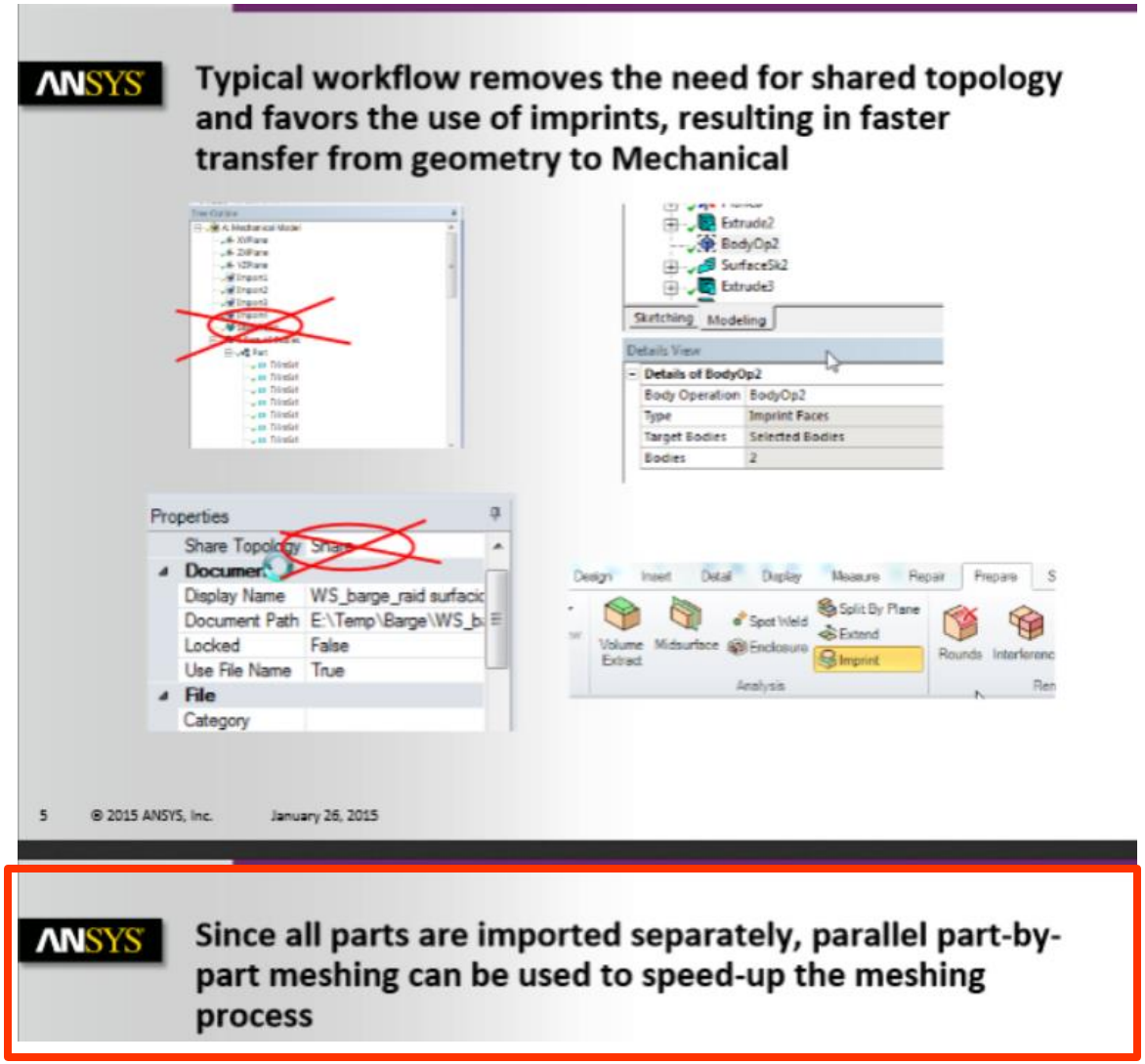
Meshing



Mesh Connections

- ANSYS inc. continues trend avoiding shared topology.
- This trend exists from WB inception.
- Also note Spaceclaim alongside DM info.

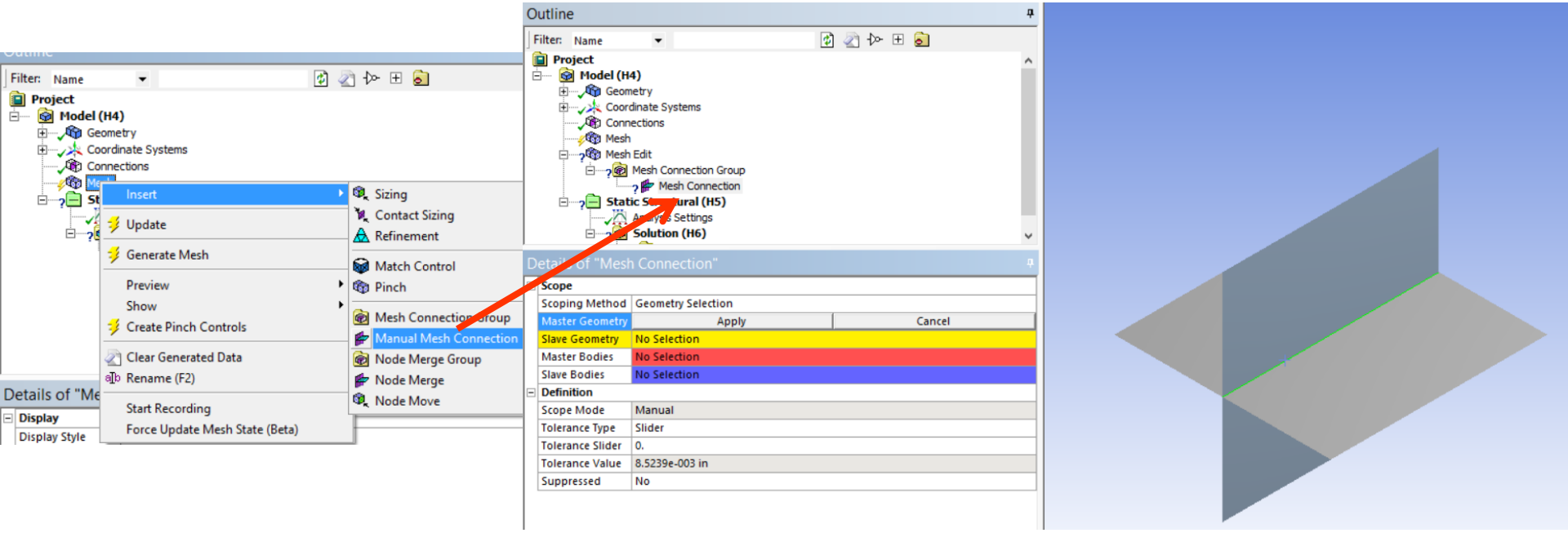




From ANSYS Release Notes

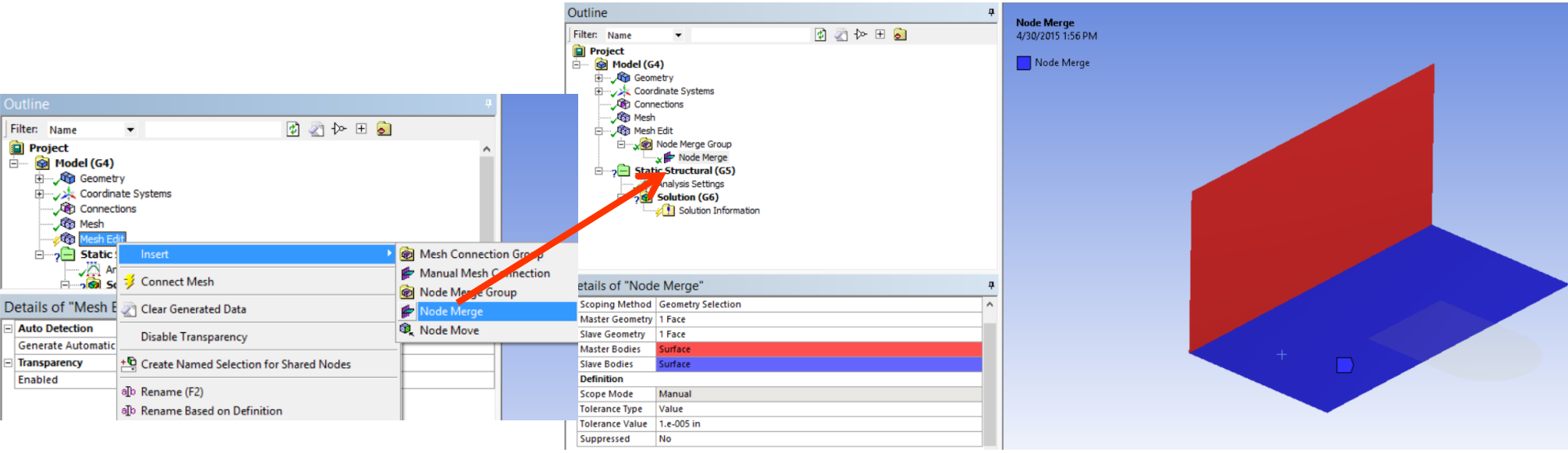
Mesh Connections

- Connect parts/bodies in ANSYS by selecting the common point/edge/face between the two.
- Allows for greater control of what is/isn't connected
- Connections can be "automated" similar to contacts
- <u>Large number of connections fail in our test cases</u>



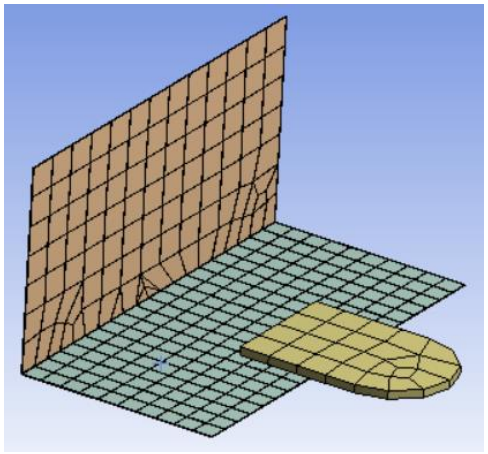
Node Merging

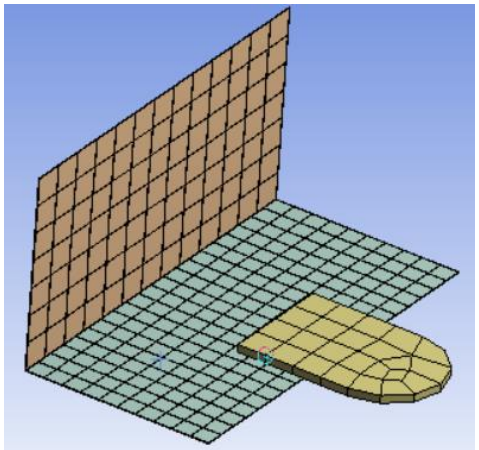
- Connect nodes between parts/bodies in ANSYS by selecting the common points/edges/faces.
  - Like NUMMRG command in MAPDL
- Allows for tolerance control between present node connections within the model



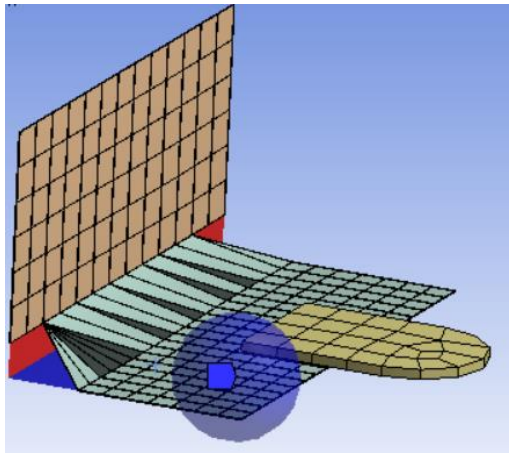
What's the Difference?!?

- Mesh Connection:
  - Mesh connection will move the nodes of the slave geometry to match the master geometry. Both bodies will share common nodes between the two.
- Node Merge:
  - Node merges will merge any nodes that are within the set tolerance. Small tolerances merge nodes on top of each other—large tolerances forces nodes to merge over a larger area.



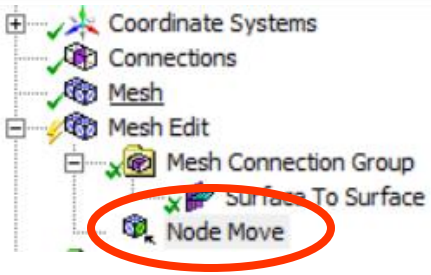


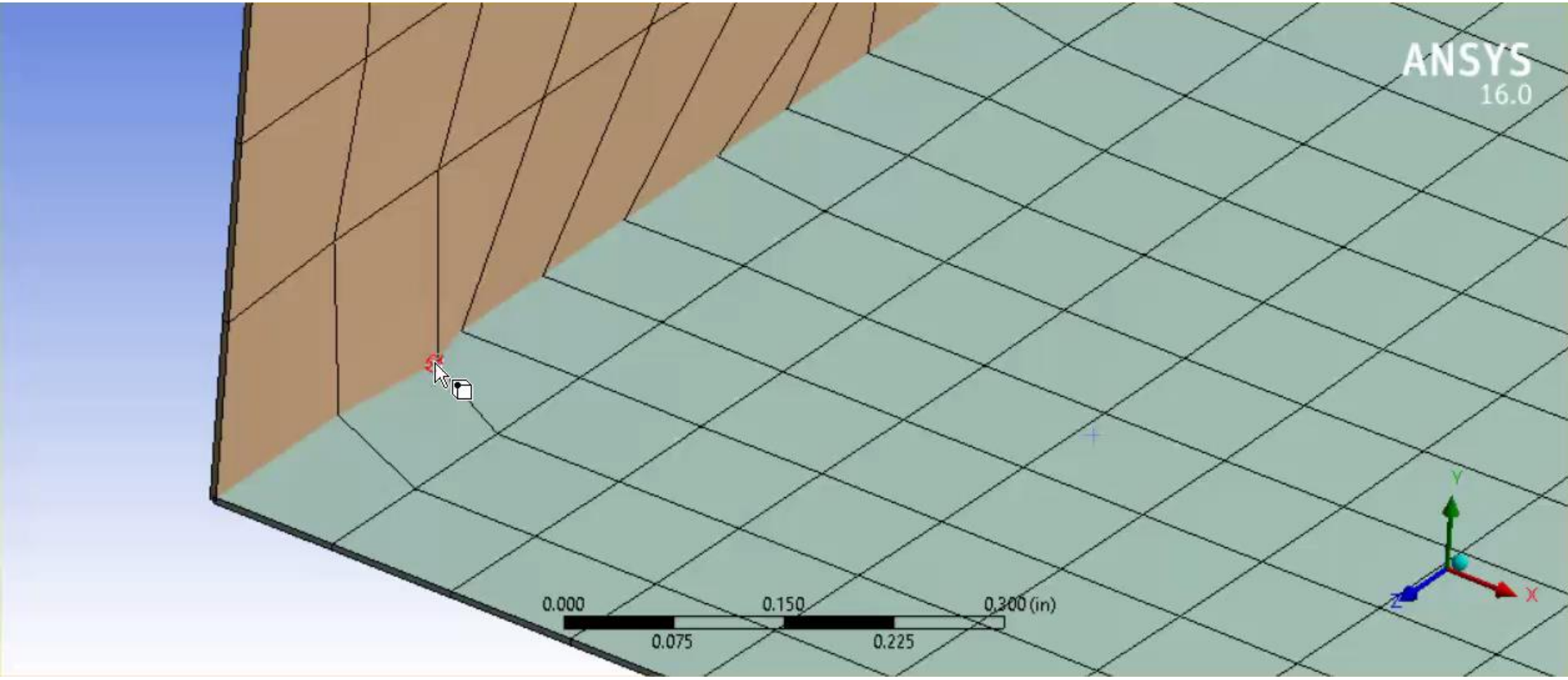
Tolerance: 1e-5, Nodes Merged: 2



Tolerance: .25, Nodes Merged: 25

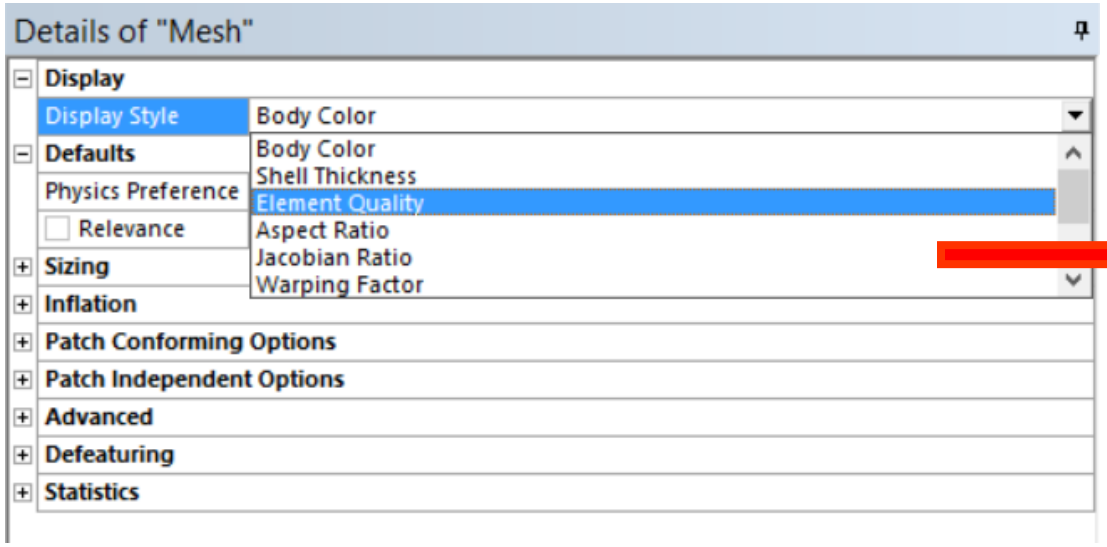
Node Moving

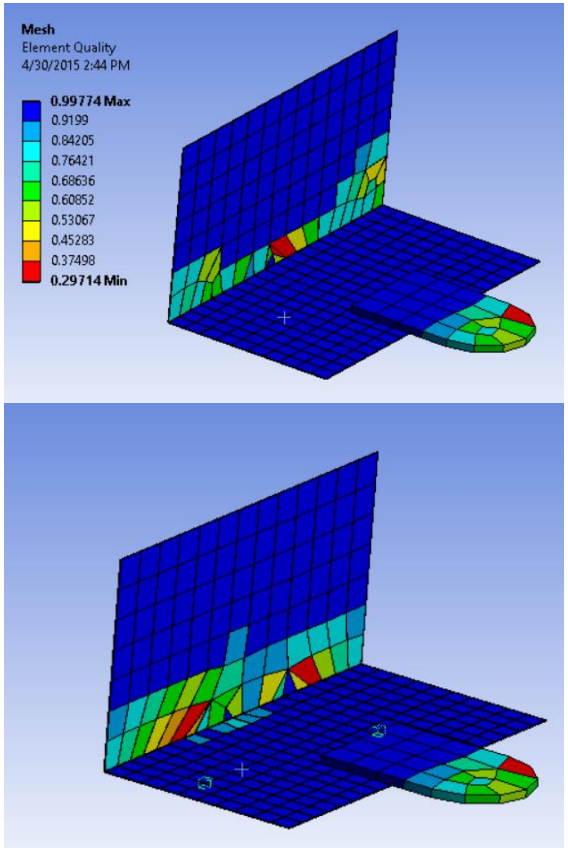

Coordinate Systems
Connections
Mesh
Mesh Edit
Mesh Connection Group
Surface To Surface
Node Move


ANSYS
16.0
0.000
0.150
0.300 (in)
0.075
0.225
Y
X
Z

Display Element Quality

- Display the mesh quality under “Mesh Details”
- Quick and easy way to look at the effects of mesh connection/node merging/node moving
- Uses combination of previous element quality viewing options





Adaptive Remeshing

- New feature for nonlinear, large deflection analyses: “Nonlinear Adaptive Region”
- Intended to ressolve convergence issues with high deformation load cases, esp. elastomers
- Extending trend from earlier “rezoning”
  - previously only in MAPDL
- Still has lots of caveats
  - Commercial application is still narrow/costly

Adaptive Meshing

- Implement during Solution options
  - Define subset of timesteps to check: less computation time
  - Multiple criteria options: skewness, strain energy, bounding box
  - Only remeshes when criteria is met and only in problem areas

Adaptive Mesh: Limitations

- Must use linear tetrahedral elements for 3D bodies
  - SOLID285's
- Scope to solid bodies or elements only – no shells
- Altered mesh cannot be shared between linked analyses
- No explicit indication where a remesh has occurred
- May introduce solution chatter that would otherwise not be present

Adaptive Mesh: Limitations

Cannot be used in combination with the following features/conditions on the same part:

- Cyclic Symmetry
- Contact Formulations: Normal Lagrange (3D), MPC, and Beam
- Contact Behaviors: Auto Asymmetric
- Point Mass, Beam Connection, Joints, Spring, and Bearing
- Remote Force, Remote Displacement, Moment, Thermal Condition, and Remote Point
- Spatially varying boundary conditions
- Coupling
- Constraint Equation

From ANSYS V16 Release Notes

Adaptive Meshing: Procedure

For 3D bodies, you must set the mesh to linear tetrahedral:

- Insert->Method->Definition->Method->Tetrahedrons
- Insert->Method->Definition->Method->Element Midside Nodes->“Dropped”

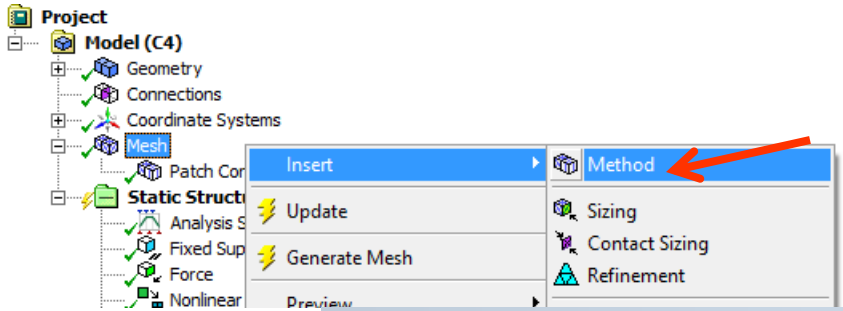
Large Deflection must be turned on:

- Analysis Settings->Solver Controls->Large Deflection-> On

Store Results At All Time Points:

- Analysis Settings->Output Controls->Store Results At-> All Time Points

It may also be helpful to define more loading substeps to check at more points before the elements distort too far



Details of "Patch Conforming Method" - Method

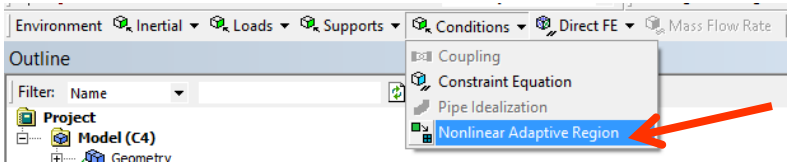
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Tetrahedrons
Algorithm	Patch Conforming
Element Midside Nodes	Dropped

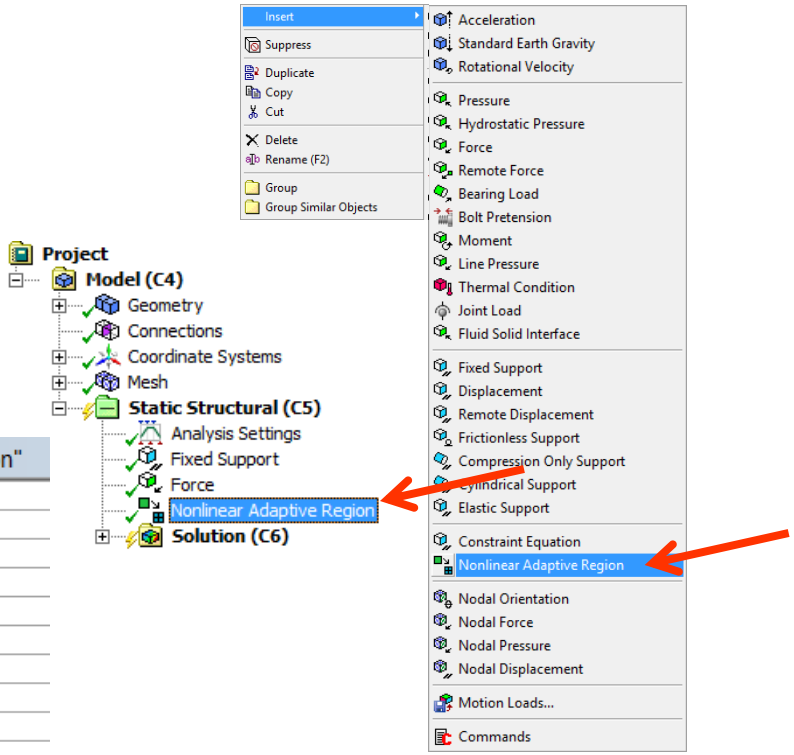
Details of "Analysis Settings"

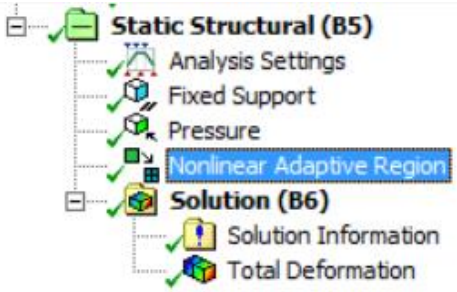
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Program Controlled
Solver Pivot Checking	Program Controlled
Large Deflection	On
Inertia Relief	Off
Restart Controls	
Nonlinear Adaptivity Controls	
Nonlinear Controls	
Output Controls	
Stress	Yes
Strain	Yes
Nodal Forces	No
Contact Miscellaneous	No
General Miscellaneous	No
Store Results At	All Time Points
Analysis Data Management	

Adaptive Meshing: Procedure

- With environment selected in the tree, insert Nonlinear Adaptive Region or select it from the Conditions toolbar
- Can only be scoped to a body or named selection of elements
- Select criterion and time range to check upon







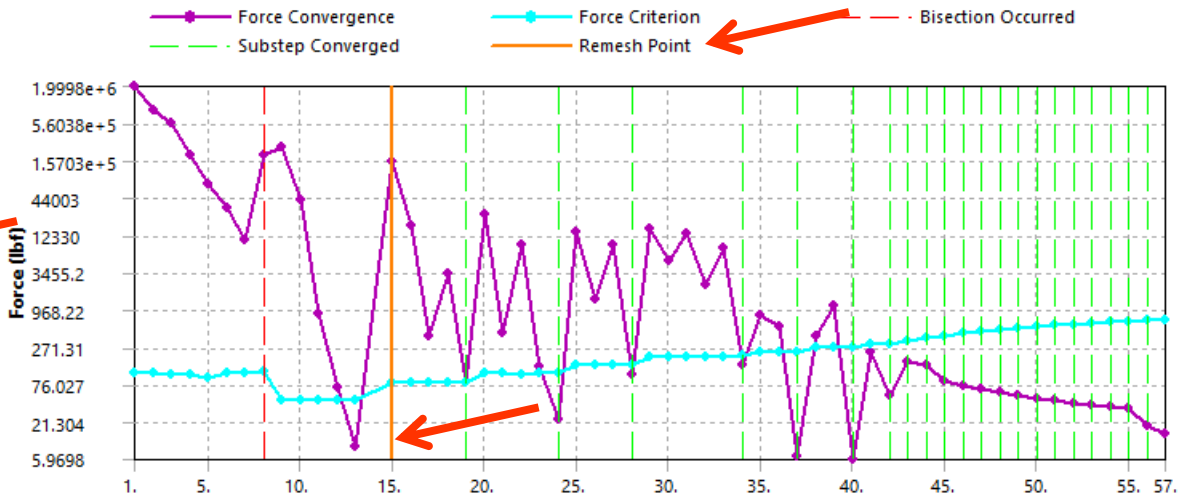
Details of "Nonlinear Adaptive Region"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Criterion	Skewness
Skewness Value	0.85
Check At	Specified Recurrence Rate
--- Value	1
Time Range	Entire Load Step
Suppressed	No

Adaptive Meshing: Results

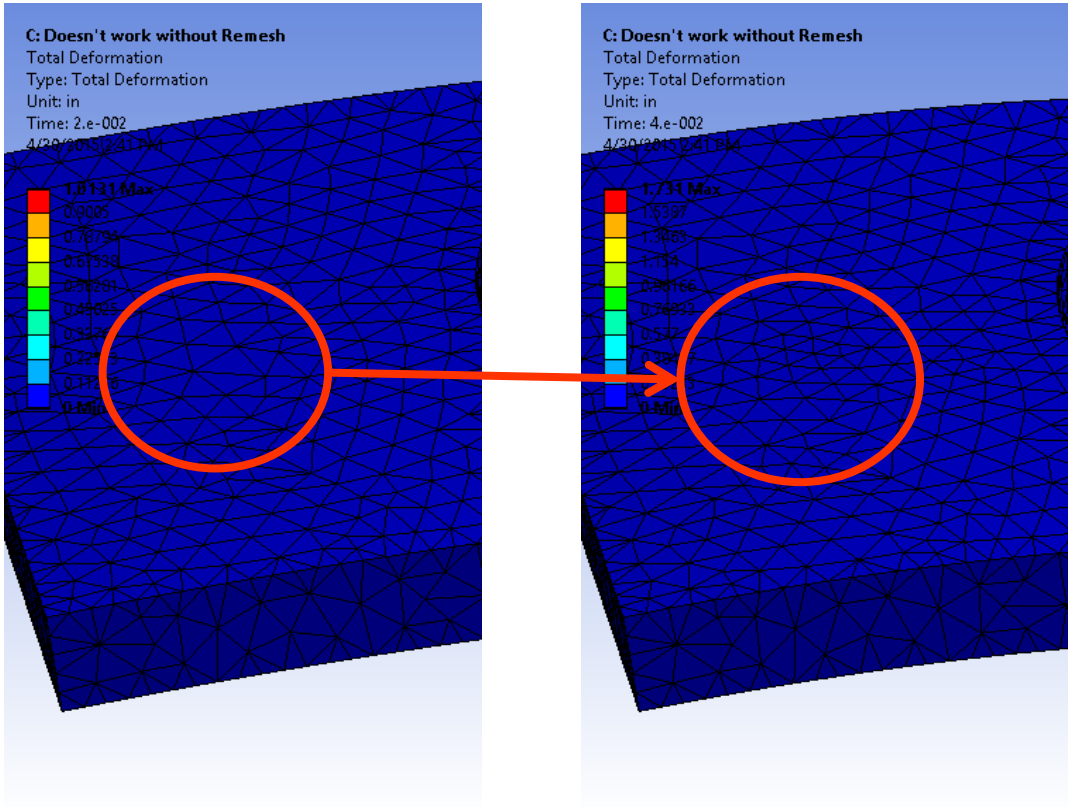
Check force/displacement convergence graphs or tabular data to find which time step/substep a remesh was applied

	Time [s]	Minimum [in]	Maximum [in]	Changed Mesh
1	2.e-002	0.	1.0131	
2	4.e-002	0.	1.731	Yes
3	6.e-002	0.	2.1757	
4	9.e-002	0.	2.6325	
5	0.135	0.	2.9689	



Adaptive Meshing: Results

Compare pre-mesh change and post-mesh change substeps to find regions of improvement



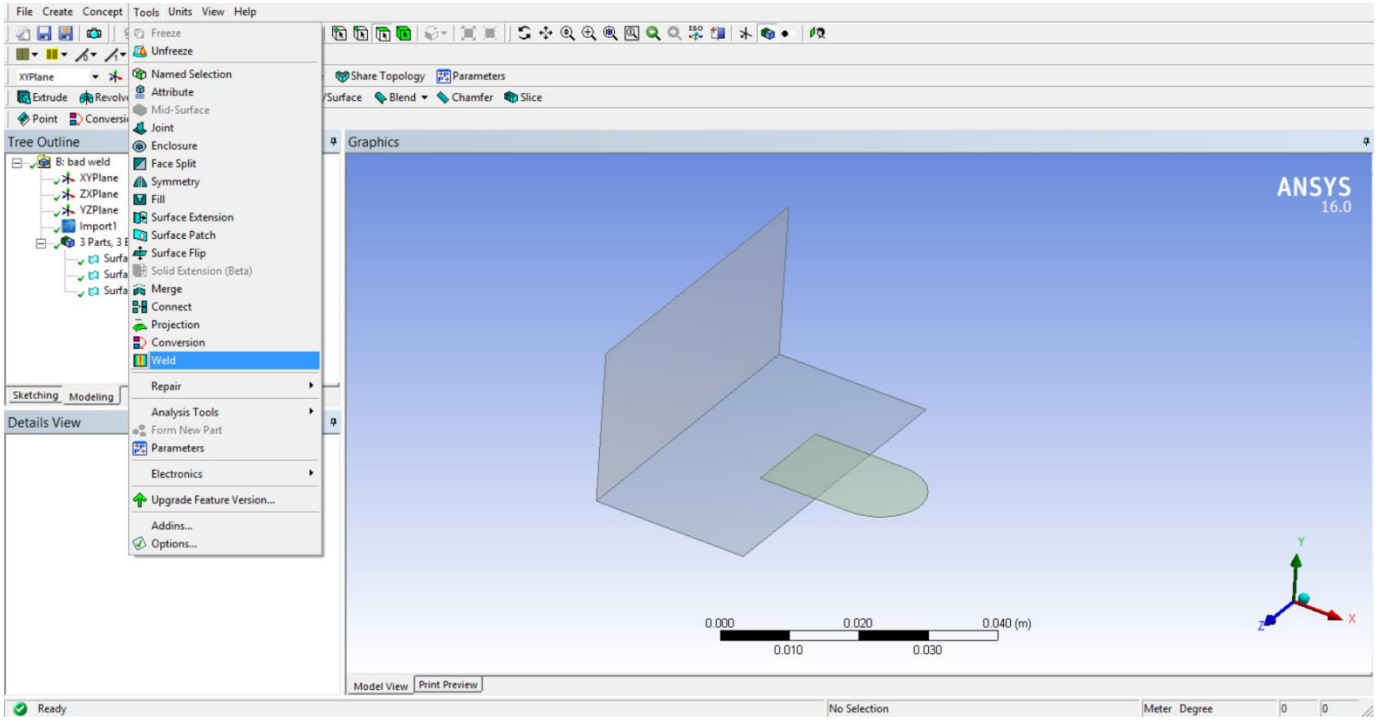
Meshing misc.

- Fixed Size Function For Sheets – set all surface bodies to same mesh density
- Face Meshing can be done without mapping– fewer elements in noncritical bodies attached to critical ones
- Tetra/Pyramid Free Mesh Type – give bodies a pyramidal surface, tetrahedral core mesh
- Vertex alignment options available in MultiZone meshing – previously only in face meshing
- Inflation allowed for 2D mapped surfaces
- Can now sweep across multiple parts or named selections, and define sweep element size with numerical argument or edge reference

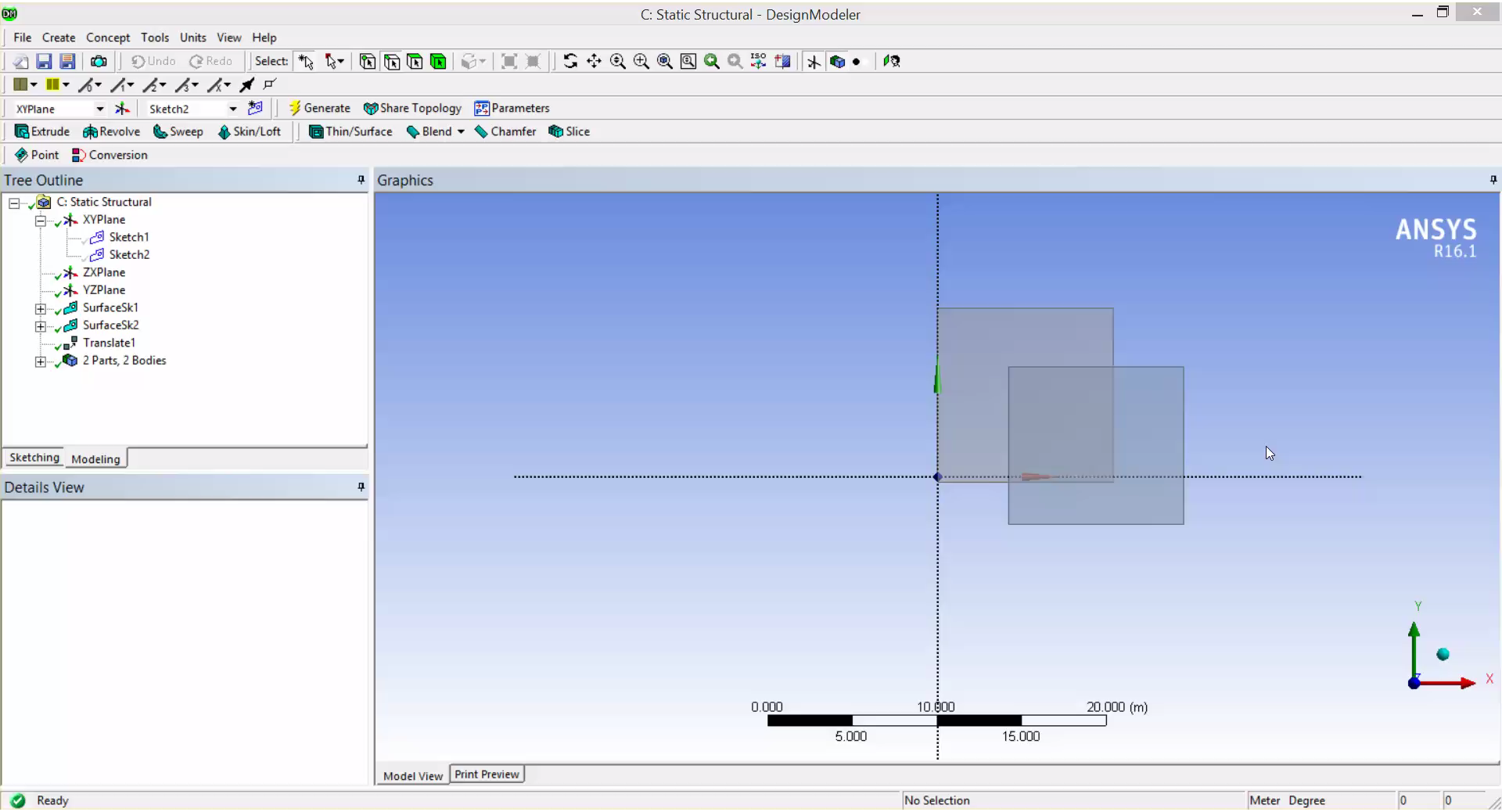
Weld Connections

Weld Connections

- ANSYS has started to implement welds in DM
- Two types of weld: Continuous and Skip (beta feature)
- Project->Surface Extension

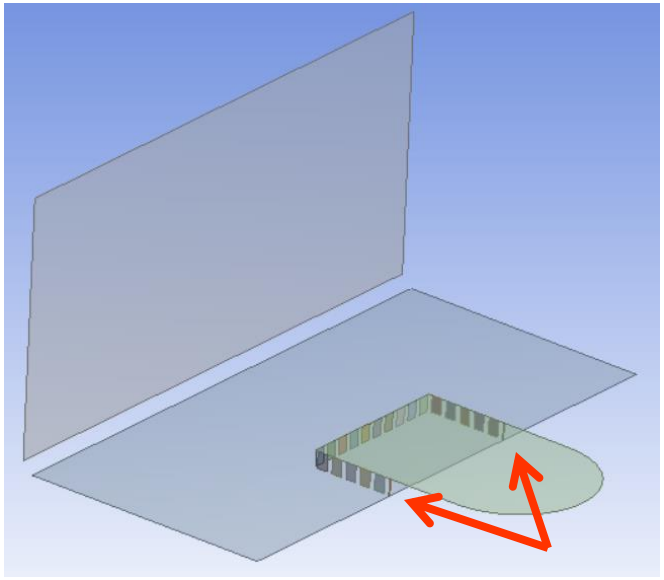


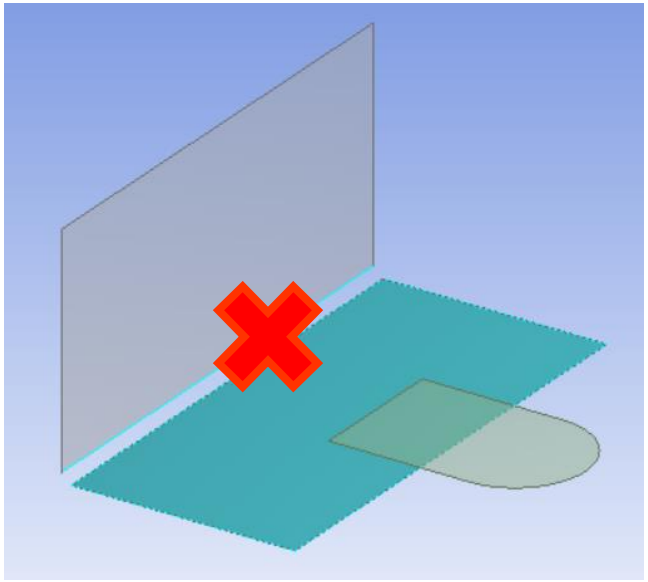
Weld Connections



Weld Connections

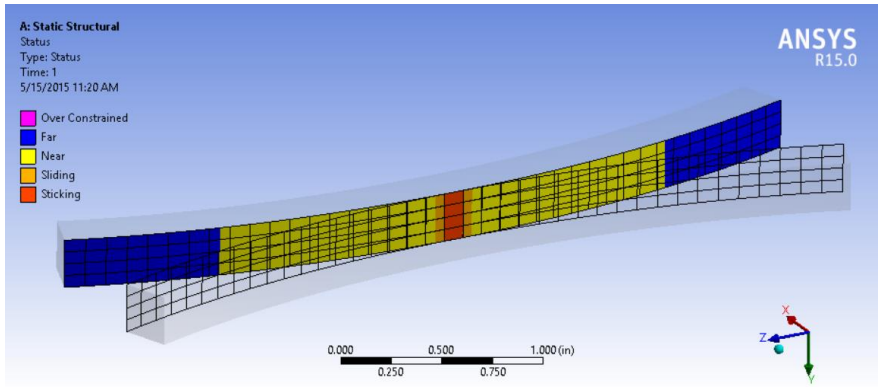
- Limitations
  - Only works on perpendicular surfaces
  - Only works in DM
  - Watch out for partial welds (see picture below)



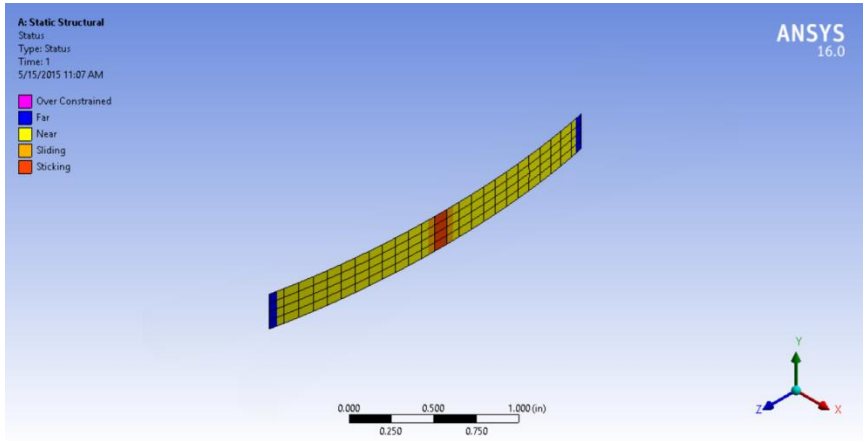


Contacts misc

- Contact smoothing for more accurate circular/cylindrical/spherical contact regions – contact is mapped based on geometry, not mesh
- Element orientation (bodies only)
- Bolt tool probes results for all pretensions
- Contact results report far contact fields as undefined rather than zero



V15



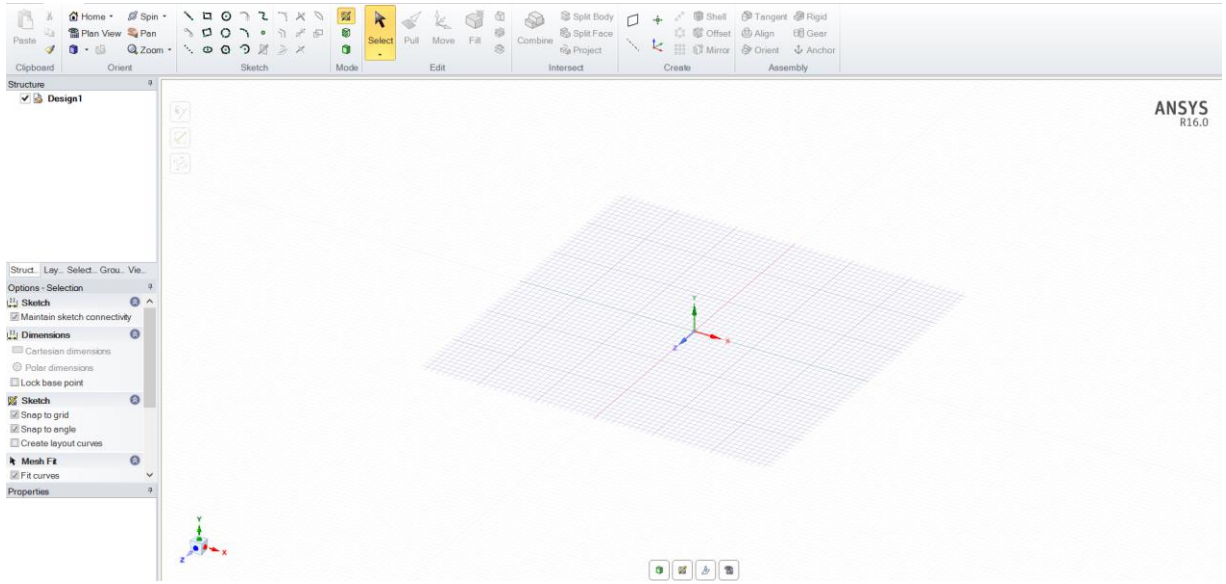
V16

Spaceclaim

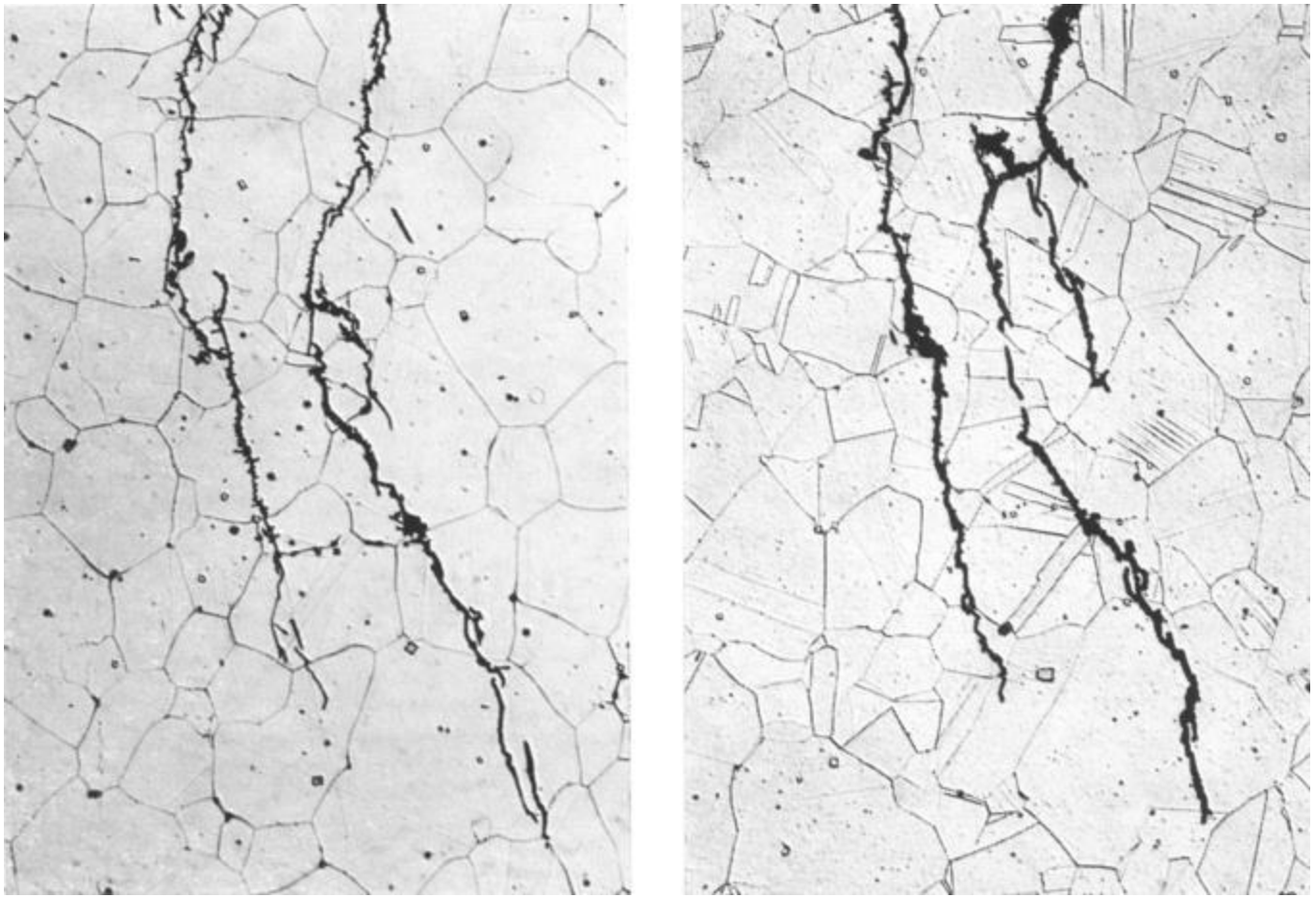
SPACECLAIM
CORPORATION

Spaceclaim

- Community usage?
- Can now read DM files
- Yes—you do need a separate license to run it
- User friendly, terrific user experience after learning
- Similar to Solidworks in feel
- No design tree! Able to make changes without corrupting the rest of the model!

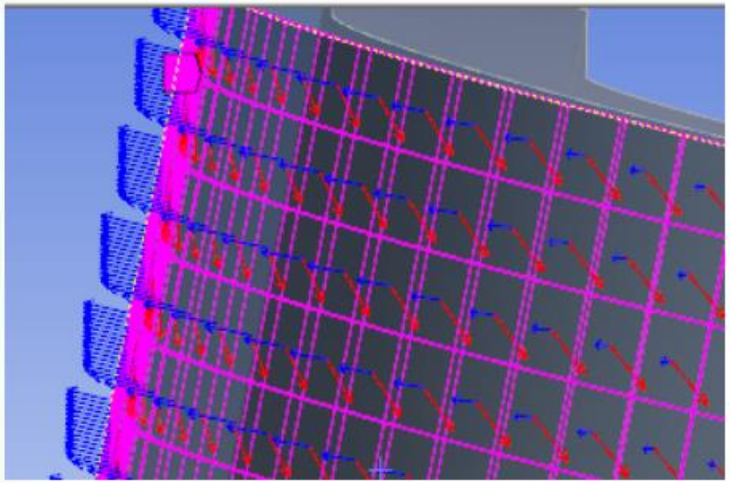


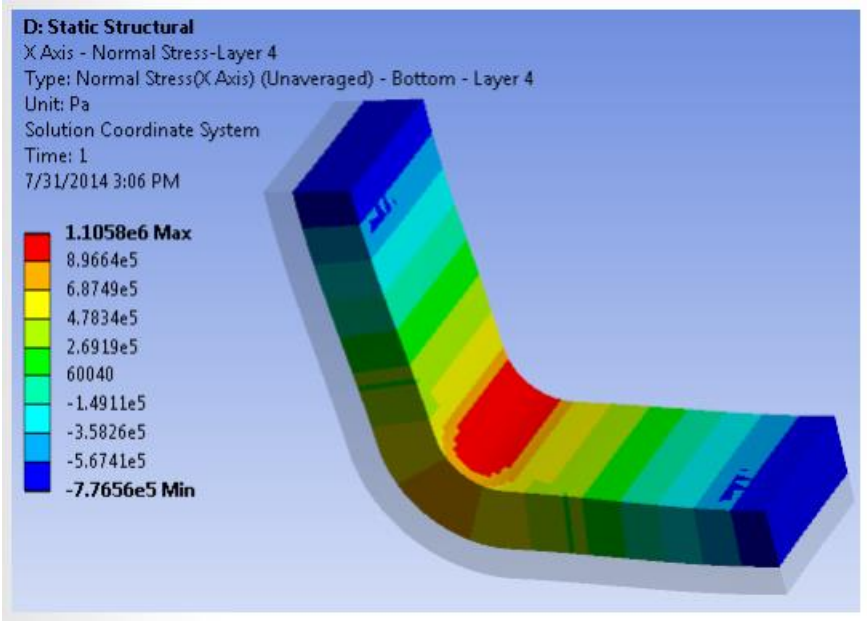
Material Modeling Improvements



Composites

- Better tree structure
- Imperfection modeling
- More individual ply control
  - Fiber direction
  - Ply stresses

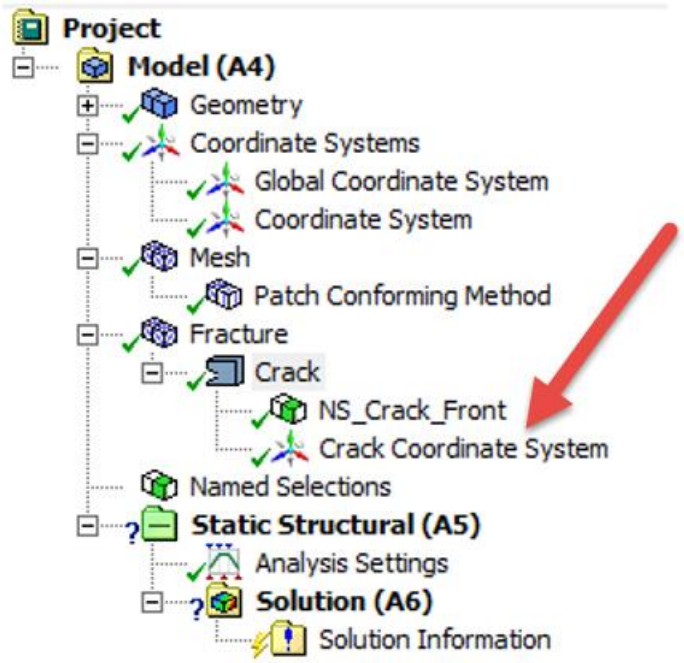




From ANSYS V16 Release Notes

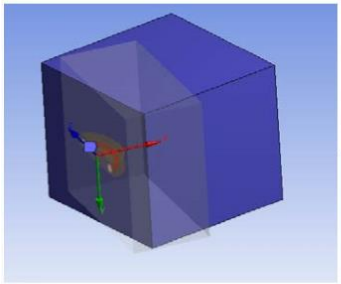
Inclined Crack Modeling

- Crack coordinate system created for each crack
- More crack definition features to work with new crack coordinate system

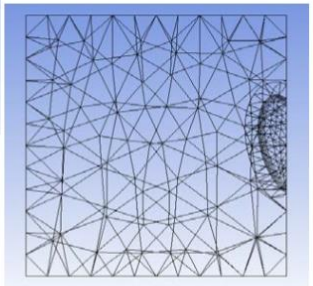


From ANSYS V16 Release Notes

Scope	
Source	Crack
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Coordinate System	Coordinate System
Align with Face Normal	Yes
Project to Nearest Surface	Yes
Crack Shape	Semi-Elliptical



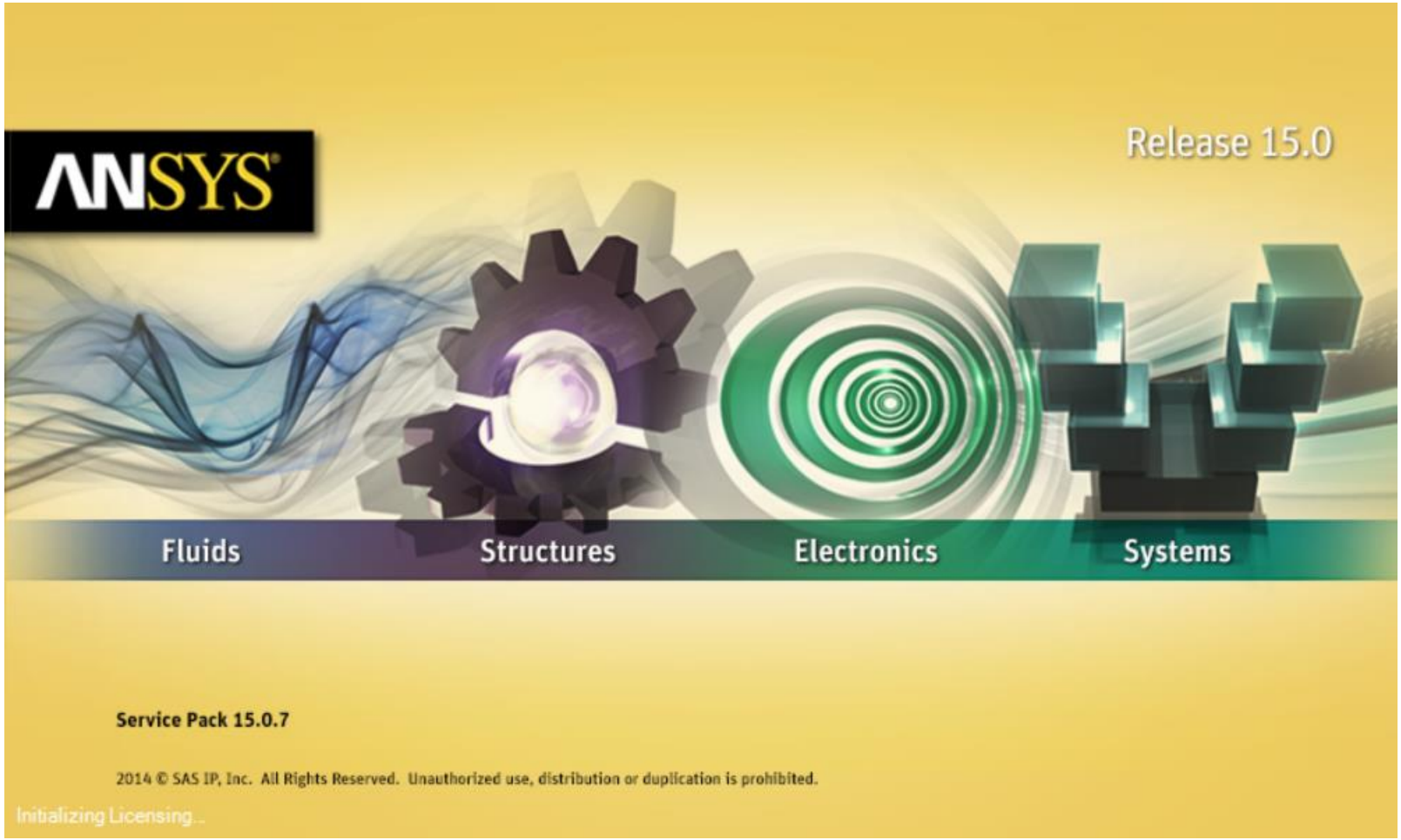
Crack annotations for inclined crack



Crack mesh for inclined crack in X-Z plane

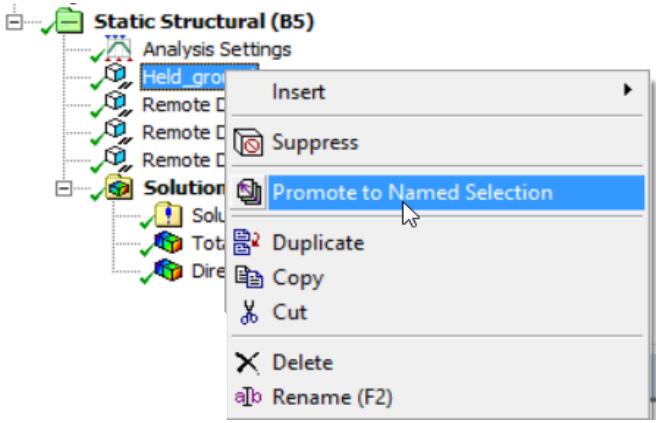
From ANSYS V16 Release Notes

V15 - Reminders



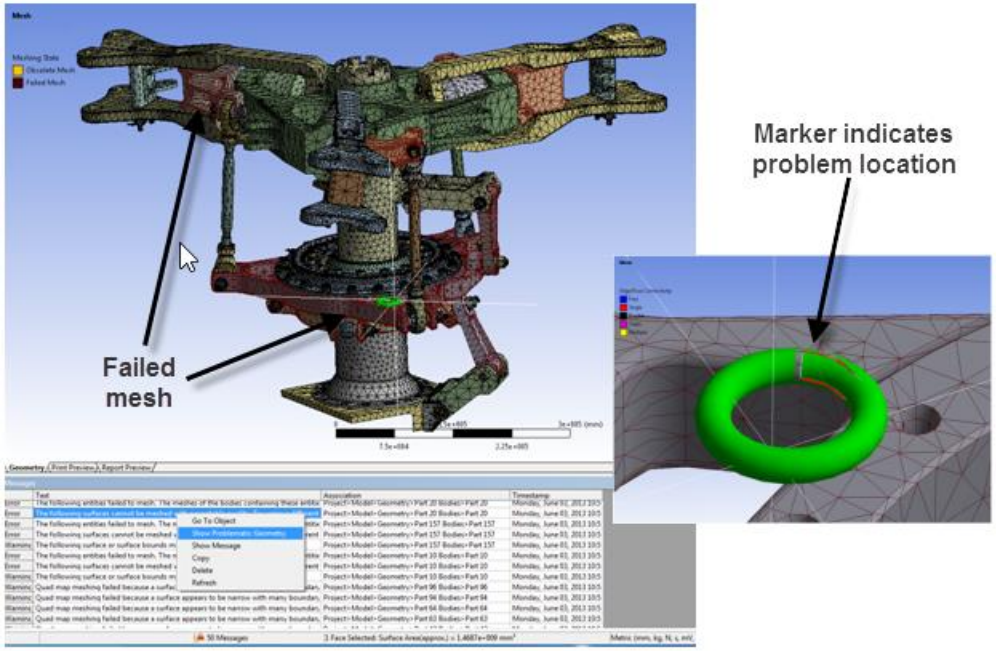
Preprocessing

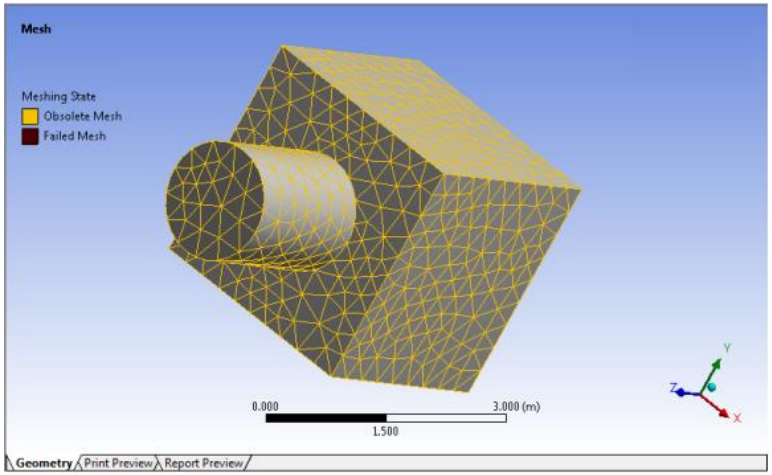
- Promote a scoping to a named selection!
  - Consider making named selections the norm /best practice
  - For those who don't plan well...
- New mapping processor
  - Can map stress/strain data from text file
  - UV mapping algorithm (surfaces don't ha
    - Projects data onto surface
    - Eases mapping from deformed to unreformed



Meshing

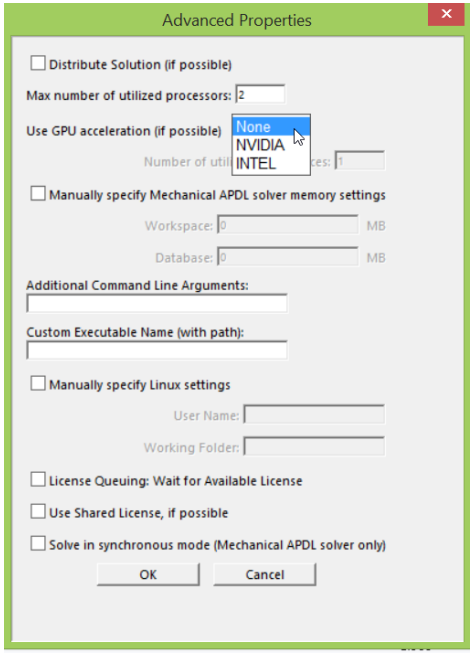
- Local Min-Size meshing
  - Overrides min-size in advance meshing (proximity and/or curvature)
- Mesh failures are color coded / highlighted by white axes





Images from help manual

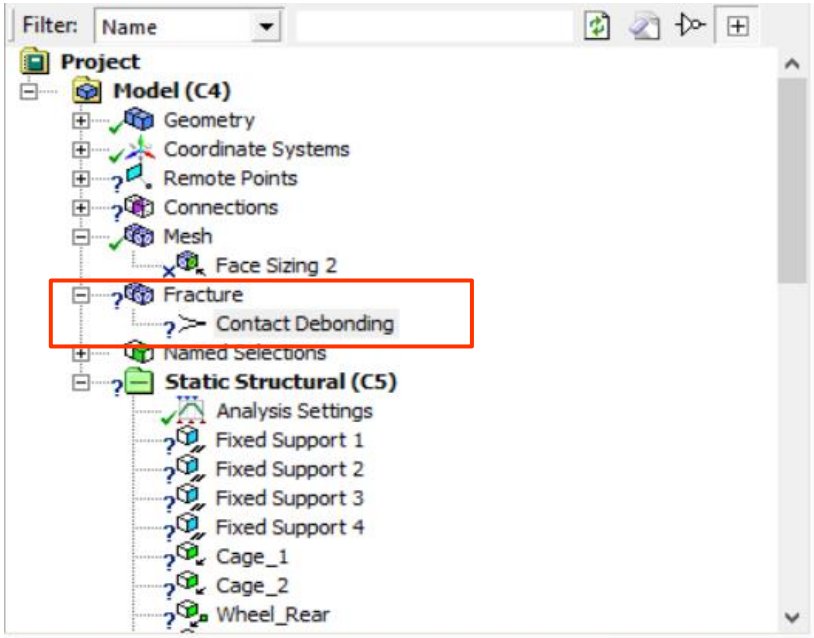
Solver Improvements

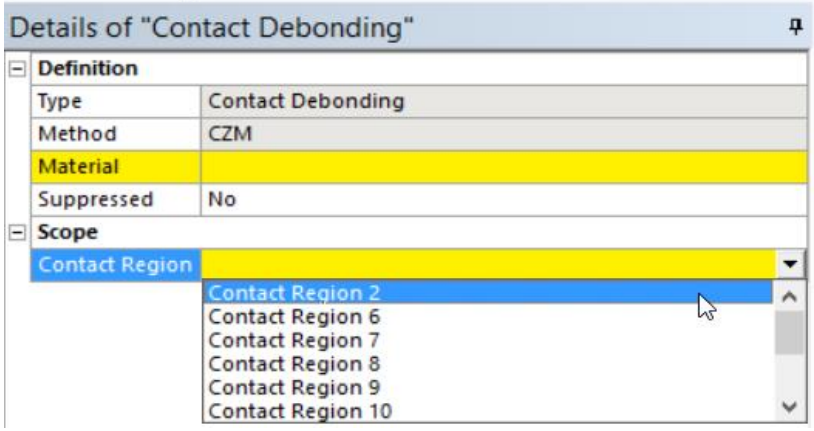


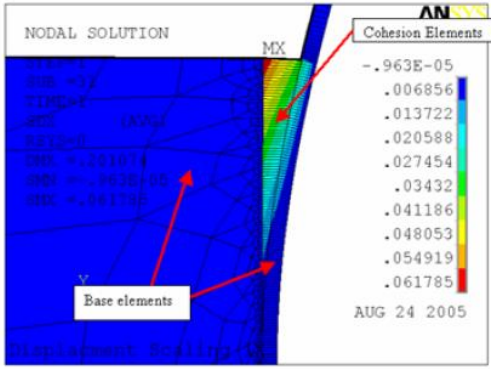
- GPU added to solve process settings / GUI
  - Can have 1 GPU on SMP using Sparse now
- New Arc-length methodology
  - Now based on “Crisfield“ theory
  - Less likely to retrace own steps backward
  - Supposed to do better with plasticity
- NROPT,UNSYM exposed in WB
  - Can aid help with non-convergence/unstable problems
  - More expensive computationally (30% on solve time?)
  - Can use the WB/Mechanical generated springs
- Sparse has better (um, different) detection/handling of singular matrices.
  - PIVCHECK command can be disabled.

Contact Debonding

- Contact Debonding!
  - CZM technology
  - Uses CONT17X
  - Define material model
- Interface Delamination
  - uses INT20X elements
  - more complex separation functions







More Contact

- MAPDL Contact -- user-programmable definitions
  - Stiffness based on frequency, nonlinear, etc.
- FTOLN now affects FKN!
- Contact surface wearing
  - “Archard” wear model (or user defined subroutine)
  - Moves contact node by depth of calculated wear
    - Controlled via material property, TB, WEAR
  - Total wear stored in NMISC data

Contact

- Bolt thread modeling
  - Need to have a refined mesh
  - Specify bolt axis/CS
  - Enter Pitch, Thread Angle, Diameter, etc.

Geometric Modification	
Interface Treatment	Add Offset, No Ramping
Offset	0. mm
Contact Geometry Correction	Bolt Thread
--Orientation	Program Controlled
--Mean Pitch Diameter	5. mm
--Pitch Distance	2. mm
--Thread Angle	60. °
--Thread Type	Single-Thread
--Handedness	Right-Handed

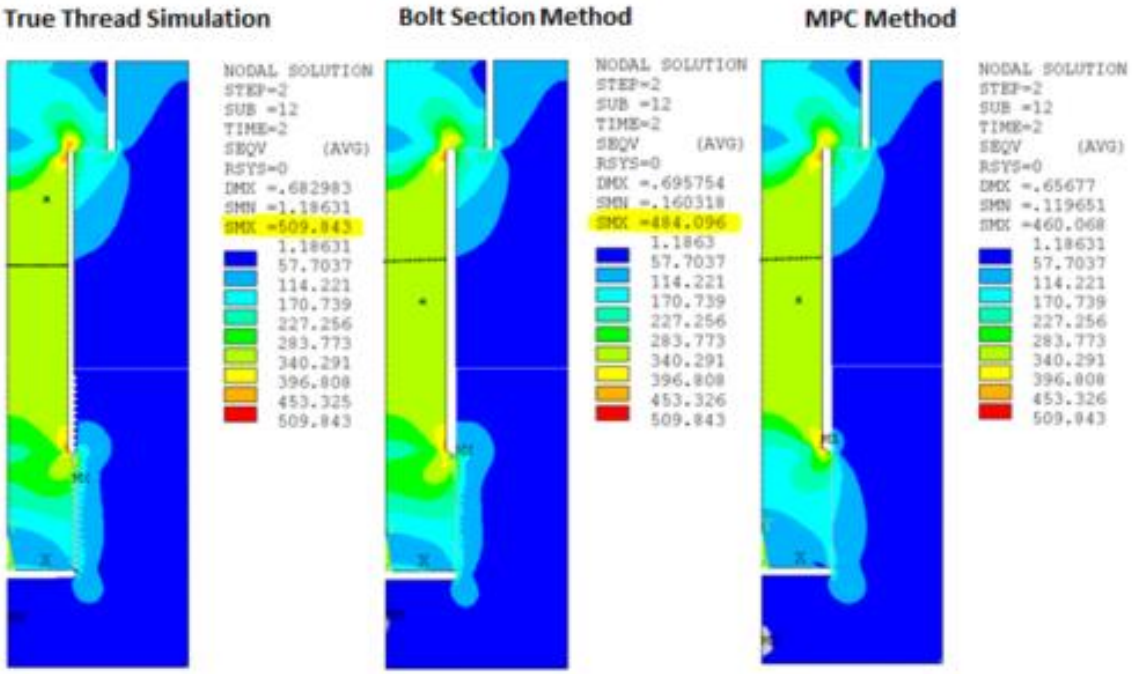


Image from help manual

Shell to Solid Submodels

- 3D shell model to 3D solid model
  - In “Transfer Key” specify Shell-Solid
    - Imported Loads detail window
  - As with most submodels, beware the rotating nodes CS’s
  - UY mapped for nodes within center region (20% the thickness)…
    - Beware over/under constraint here.
    - Ideally this would be a single set of nodes

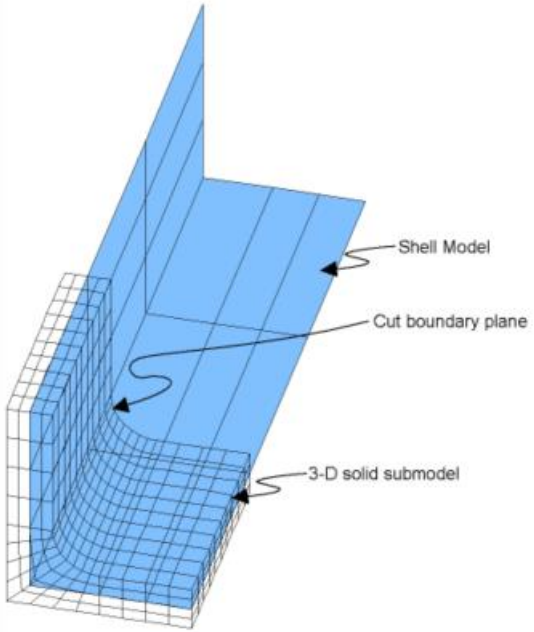


Image from help manual

Automatic Rezoning

- Manual Rezoning fully implemented in MAPDL
  - Writes deformed nodes
  - Makes facets/geometry in the deformed shape
  - Meshes new volume/area
  - Transfer displacements from previous analysis
  - Resolves -- Lather, rinse, repeat…
- Automatic Rezoning just splits the element edges
  - a.k.a. “Mesh Nonlinear Adaptivity”
  - An EREFINE essentially.
  - Manual is pretty clear this won’t help element shape distortion errors… yet.

Shared Topology

- Toggles on/off – if you don't know about it, it's a must read to avoid problems.
  - Default is to execute behind the scenes on way to Mechanical
  - Can be forced to earlier location in the tree
    - And then hidden – so you won't know that it is happening prior to other operations!
- Face coloring by shared topology specification
  - Not by success / actual connection
- Specify face joints manually…
  - Must be two parts within same body
  - Can be done after shared topology operation