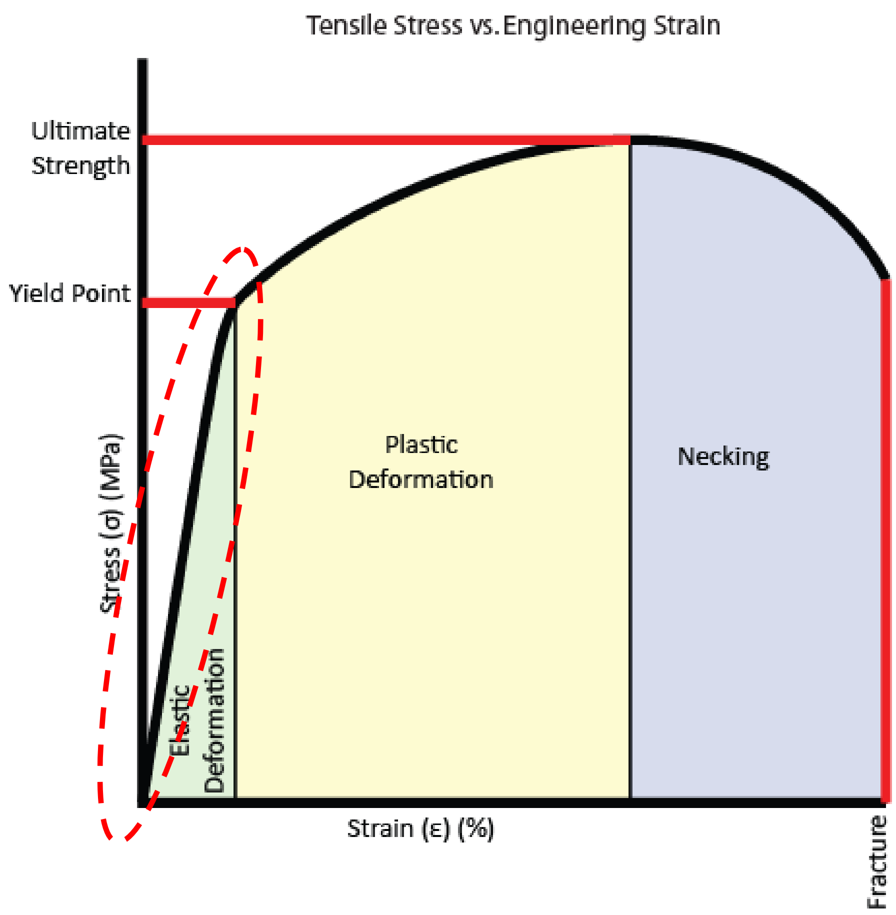
Appendix A

Introduction to SimWise FEA Basics and Fundamentals

FEA Assumptions (#1)

- The relationship between stress and strain is linear and elastic



FEA Assumptions (#2)

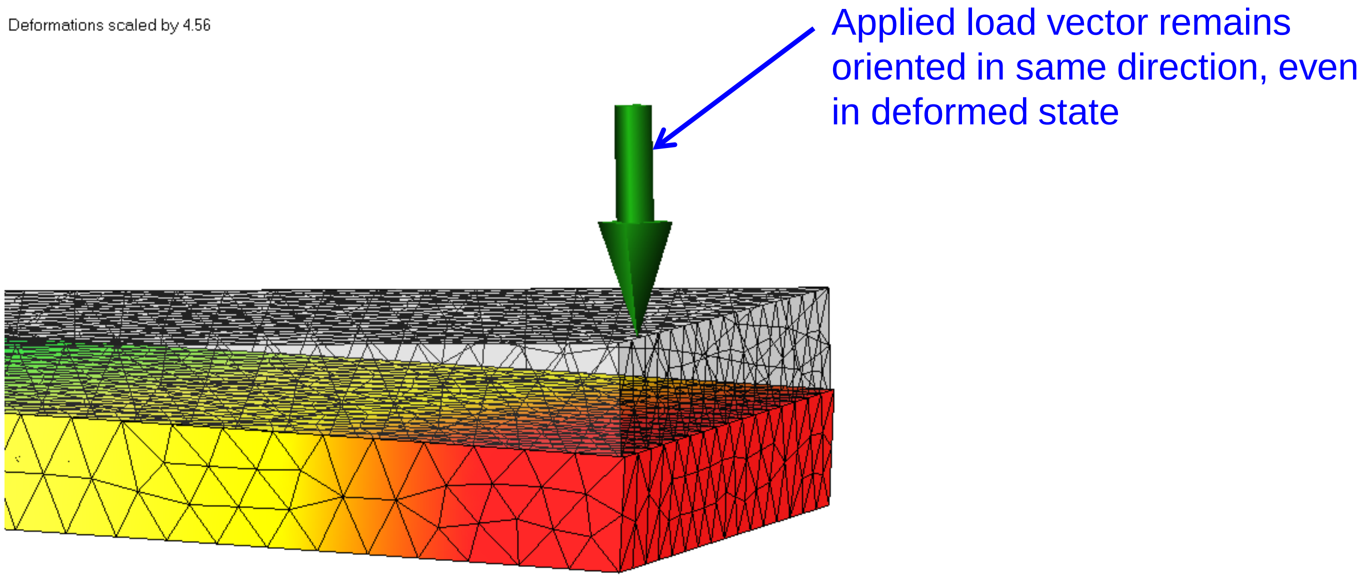
- Deformations are assumed small (strain <1)

$$\varepsilon = \frac{L' - L}{L} = \frac{\Delta L}{L}$$

$$\begin{aligned} \varepsilon &= normal\ strain \\ L &= original\ length \\ L' &= new\ length \\ \Delta L &= change\ in\ length \end{aligned}$$

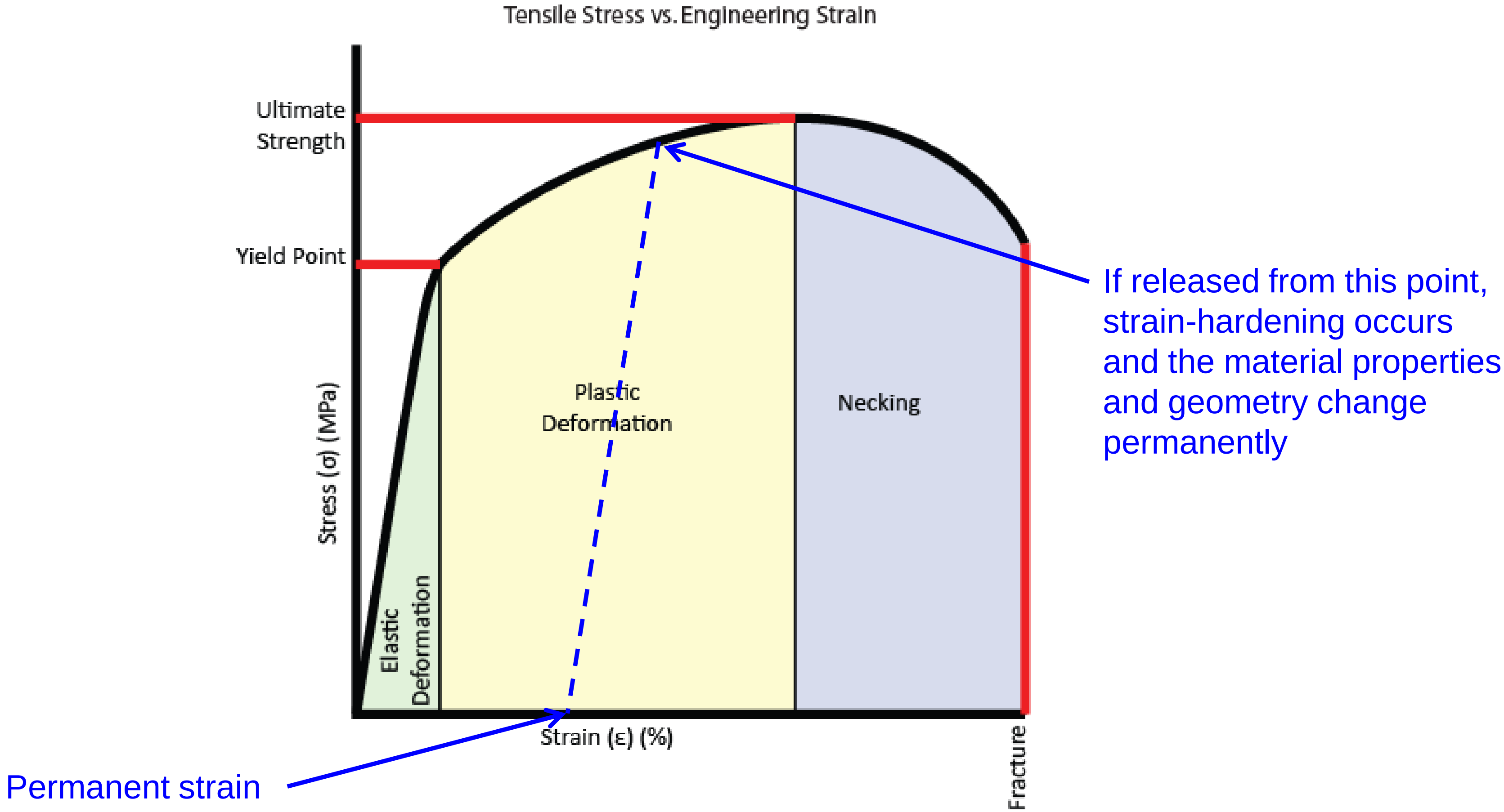
FEA Assumptions (#3)

- The magnitude, orientation and distribution of the loads does not change during deformation



FEA Assumptions (#4)

- Any strain-hardening of the material due to deformation can be neglected



Key Considerations in FEA modeling

Geometry

- *Unlike kinematic motion simulation where small geometric features such as fillets , rounds and sharp edges do not affect the results of the simulation, the opposite is true when it comes to structural analysis.*

- *Certain geometric features can introduce large and undesirable stress concentrations and others can alleviate them*

- *Fillets, grooves, sharp edges and holes, are some features that can have a significant impact on stress distribution in the model*

Key Considerations in FEA modeling

Material Properties

- *The amount a part deforms, the point at which it fails, and the relationship between longitudinal-to-lateral deformation is largely governed by its material properties*

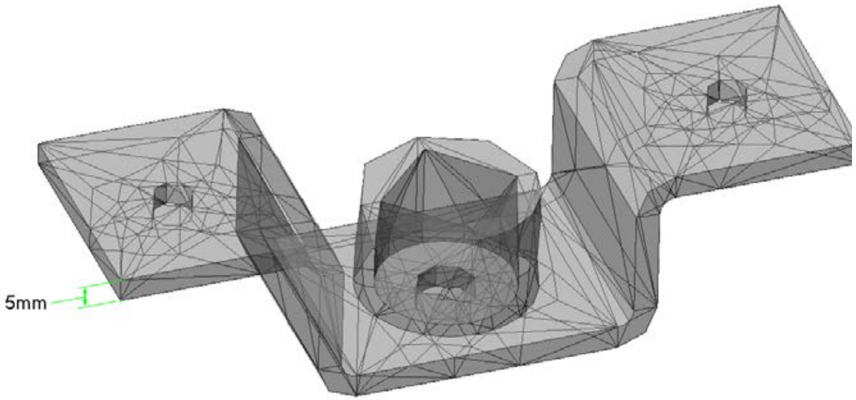
Mass Density	7.85e+3	kg/m^3
Elastic Modulus	2e+11	Pa
Poisson's Ratio	0.29	
Yield Stress	3.31e+8	Pa
Ultimate Tensile Stress	4.48e+8	Pa
Specific Heat	418	J / (kg K)
Thermal Conductivity	46.7	W / (m K)
Thermal Coeff. of Expansion	1.13e-5	m / (m K)

Key Considerations in FEA modeling

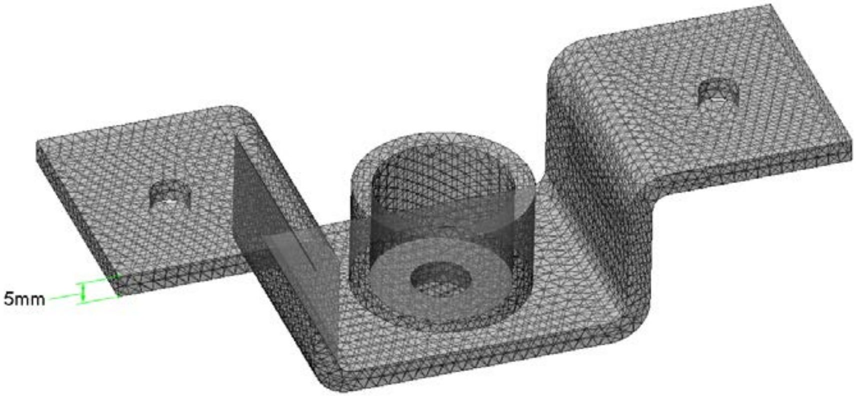
Mesh Quality

- *Mesh quality is affected by the size and shape of the part, as well as how the features transition between one another.*

- *A good mesh needs to sufficient to represent the geometry without introducing large stress errors*



Poor (final) Quality



Good (final)Quality

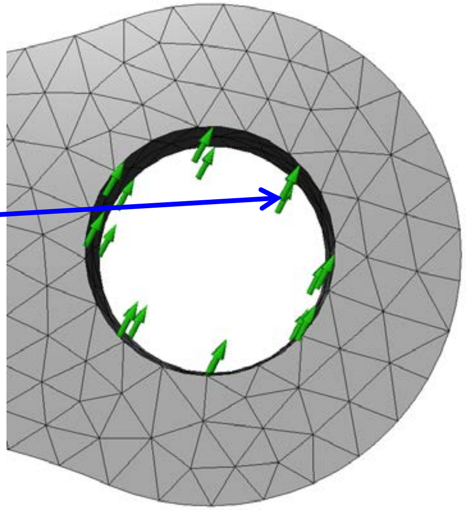
Key Considerations in FEA modeling

Boundary Conditions

- ***Boundary Conditions*** *include any feature added to help represent the conditions acting on the part or assembly, such as gravity, applied forces, torques or restraints.*

- *Boundary Conditions do not change value, position or direction during deformation*

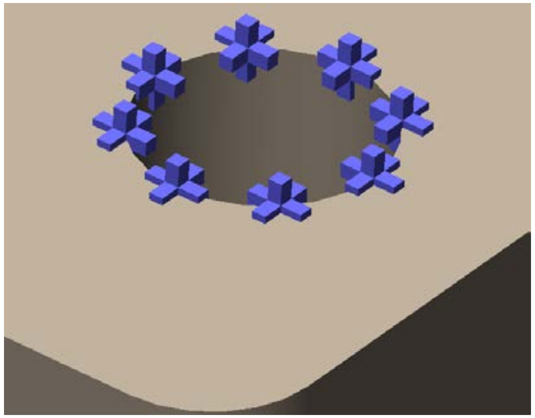
Load does not move with
the part as it deformations



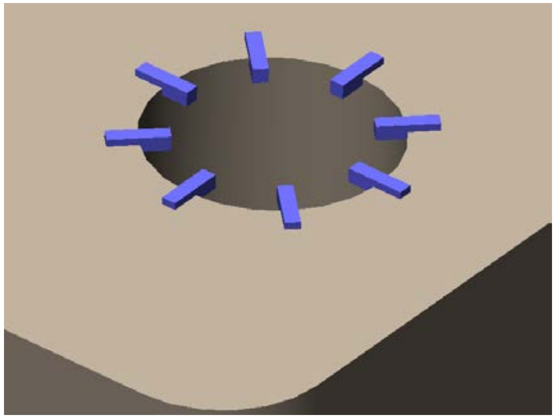
Key Considerations in FEA modeling

Restraints

- *The **restraints** should define the proper degrees of freedom of the physical connection they represent*



Bolt Hole



Avoid → Fixed restraint over defines actual connection

Desired →Normal to face restraint prevents radial movement

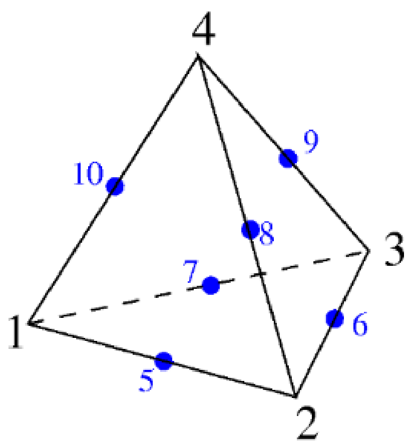
- Over-restraining a part or assembly can actually prohibit deformation and act to increase stress

Meshing

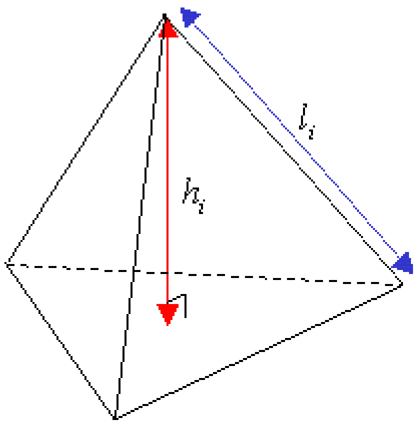
- *SimWise uses* ***10-node tetrahedral elements*** *(4 vertex nodes and 6 mid-side nodes)*

- *The quality of an element is commonly defined by the* ***aspect ratio.*** *The aspect ratio is defined as the ratio between the longest edge and the shortest normal dropped from a vertex to the opposite face.*

- *An ideal mesh element will have an aspect ratio of 1.*



10-node Tetrahedron



$$AR = \frac{L_i}{h_i}$$

Meshing

The larger the aspect ratio, the more distorted an element is. The more distorted an element is, the greater the stress differences between nodes and therefore the greater the overall stress error in the element.

Aspect Ratio = 1

Ideal

Aspect Ratio = >1

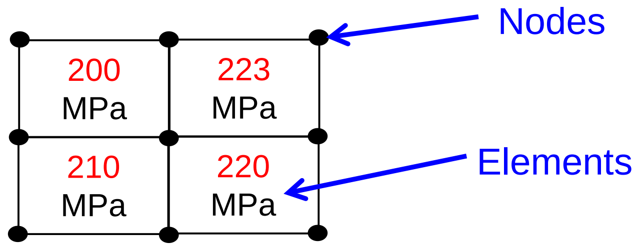
Distorted

How SimWise Solves An FEA

- *Using the loading information and material properties, displacements are calculated at each node*

- *Using the stress/strain curve, strains and stresses are calculated at each node*

- *Using the stresses for all nodes of an element, the overall average stress in that element is calculated.*



Simplified mesh representation of four adjoining elements

Von Mises Yield Criterion

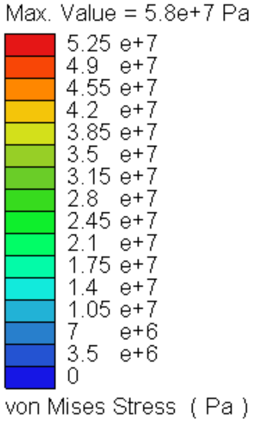
- ***Von Mises*** *stress are a combination of all principal stresses*

- *Used for* ***ductile materials***

- *Von Mises values are typically compared to the yield point (or yield stress) of the material.*

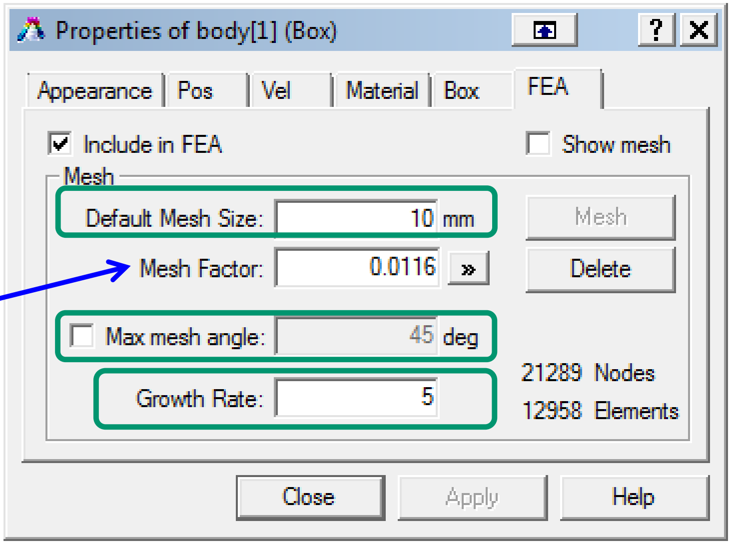
- *The ratio of the yield stress to the actual simulation stress is the* ***Factor of Safety*** *(FOS)*

$$\mathbf{FOS} = \frac{\textbf{Yield stress}}{\textbf{Von Mises Stress}}$$



Mesh Settings

➢ *There are automatic settings and there are User-driven settings for meshing*



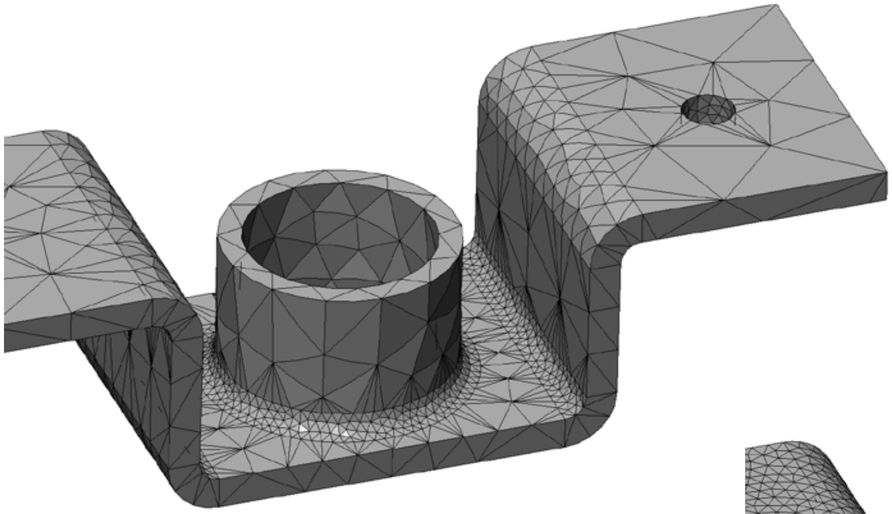
For automatic meshing, SimWise uses a Mesh Factor of 10
The Mesh Factor is an element size factor based on the overall size of the body

Mesh Settings (Mesh Size)

- ***Default Mesh Size*** *is the initial mesh size recommended by SimWise. This value is based on the overall length, width and height of the body.*

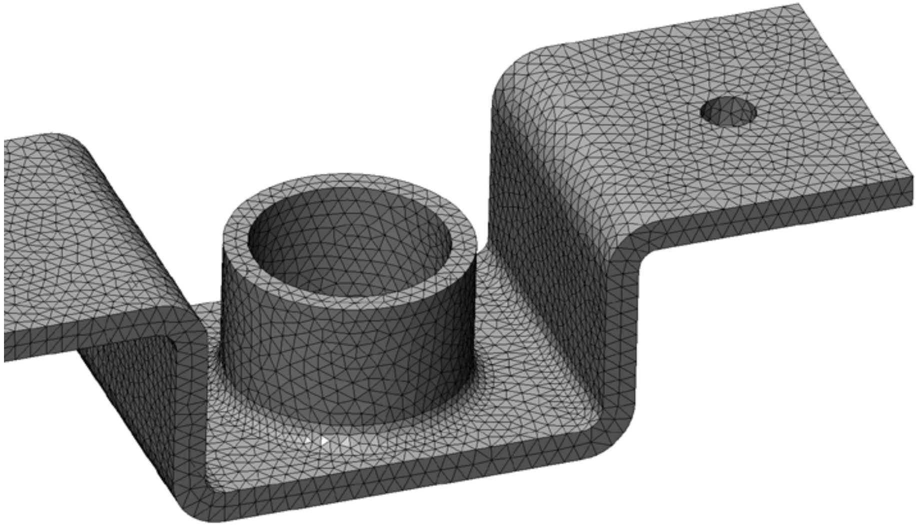
- *In most cases, the default mesh size is acceptable, so long as there is further mesh refinement, either using manual methods or by H-adaptive meshing, to be assured that results have converged*

Mesh Settings (Mesh Size)



Default mesh
(Element Size = 35.4)
(Mesh Factor = 10)

- Either mesh may be acceptable to begin with. Depending on loading and other boundary conditions, one mesh size may help achieve convergence faster than the other



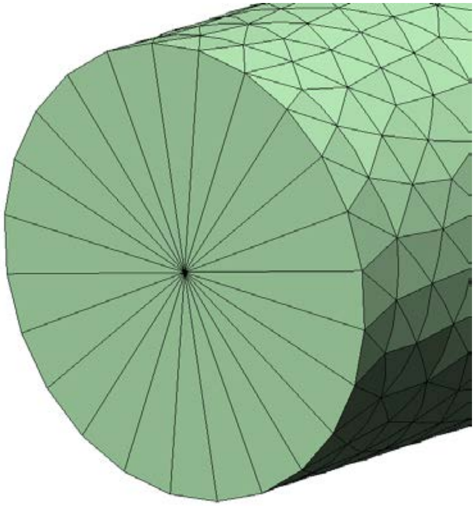
User-defined mesh
(Element Size = 2.4mm)
(Mesh Factor = .00312)

Mesh Settings (Max Mesh Angle)

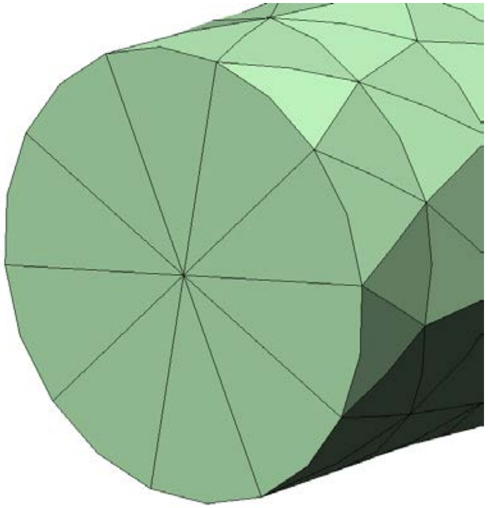
- ***Max Mesh Angle*** *Maximum spanning angle, across curved surfaces, for the generated elements. Default is 30 deg. A smaller angle will increase the number of elements across all curved surfaces in the model. A large angle will decrease the number of elements across curved surfaces, thus resulting in a coarser curved surface mesh.*

- *This feature can help improve solve times by intentionally defining larger angle values and creating a coarser mesh, to roughly represent curved surfaces. Not all curved surfaces in the model will necessarily exhibit high stress. Therefore, it may not be necessary to mesh all curved surfaces using the default angle. As with any FEA mesh input parameter, proper judgment must be used when determining when to use this feature.*

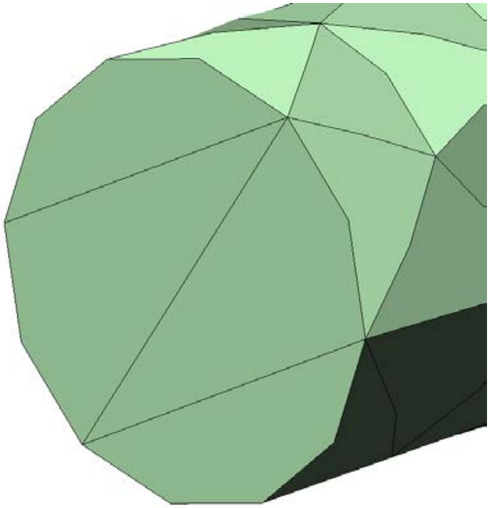
Mesh Settings (Max Mesh Angle examples)



Max Angle = 15^o



Max Angle = 35^o

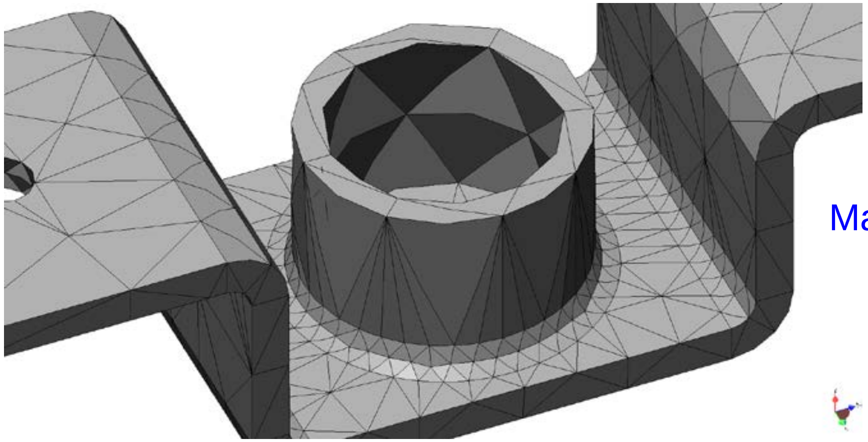


Max Angle = 60^o

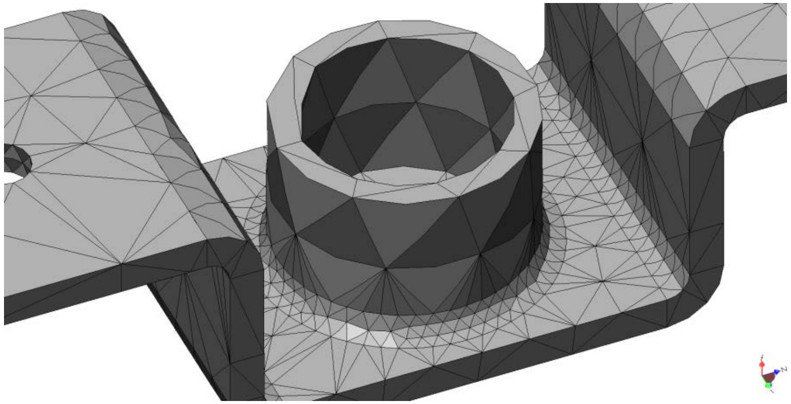
Additional increases of the angle beyond this point will not results in any difference. The reason is because, using 60 deg, the angle is already able to span across the entire curved surface

Mesh Settings (Max Mesh Angle examples)

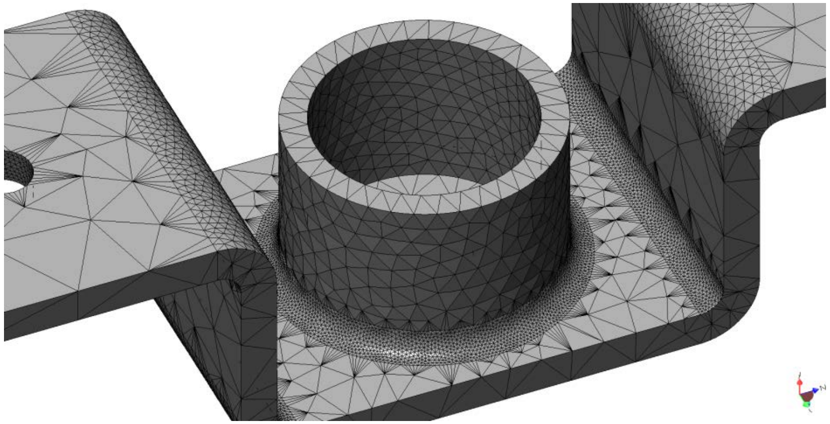
As the angle is changed, note the change in the mesh representations of the curved surfaces. The smaller the angle, the better the fit to the curve, but the greater the number of mesh elements in the model



Max Angle = **90** deg



Max Angle = **45** deg



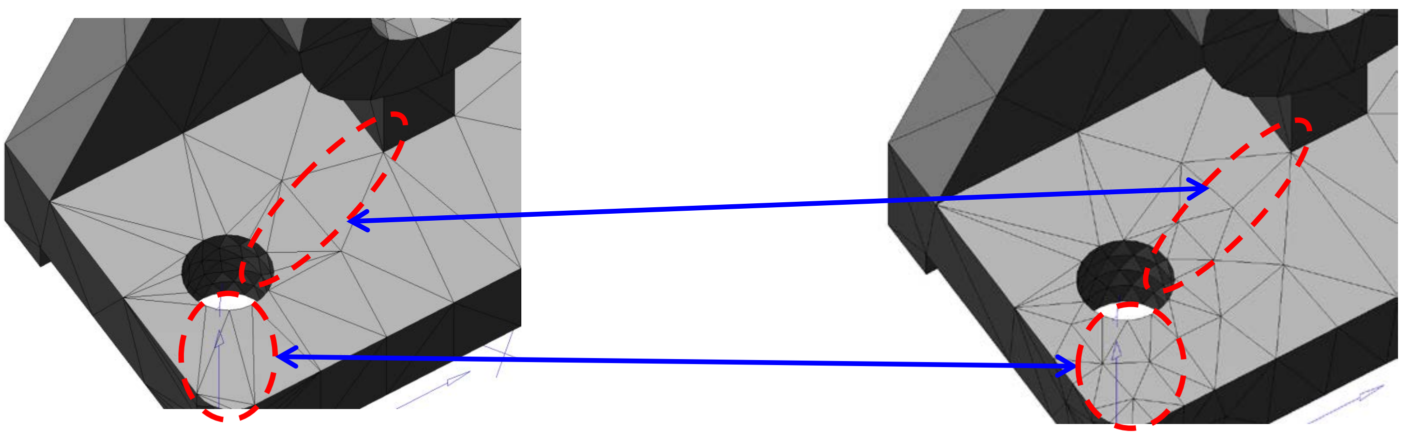
Max Angle = **10** deg

Mesh Settings (Growth Rate)

- *The **Growth Rate** governs the rate at which the size of adjacent elements can grow. The ratio of the edge lengths of adjacent elements will not exceed the specified growth rate.*

- *The larger the growth rate, the faster the mesh will transition from a small size to a large size*

Growth Rate = **5** | Growth Rate = **2**

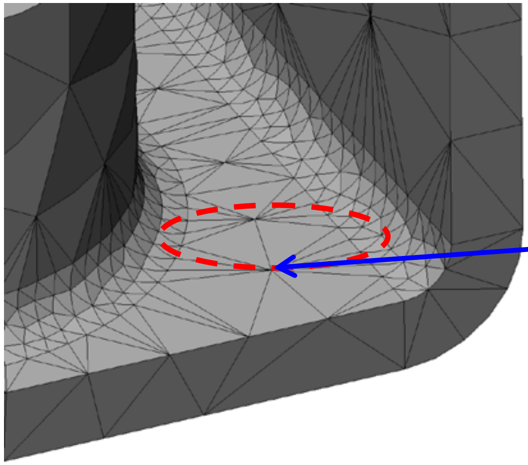


Nodes: 2,222
Elements: 1,019

Nodes: 2,653
Elements: 1,256

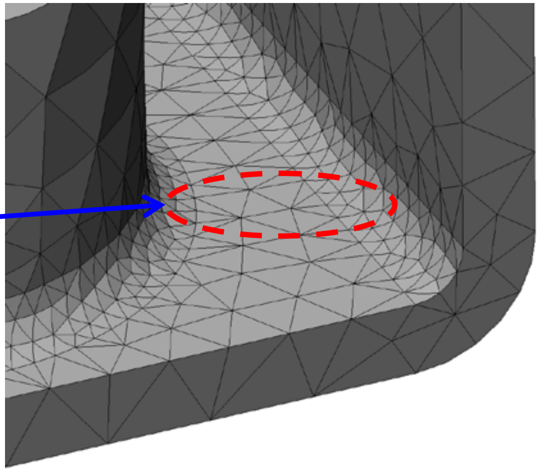
Mesh Settings (Growth Rate)

Growth Rate = **5**



Nodes: 9,180
Elements: 4,504

Growth Rate = **2**

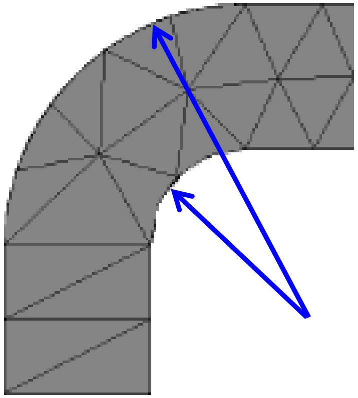


Nodes: 10,975
Elements: 5,398

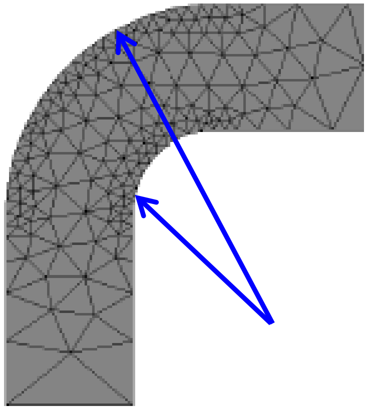
Mesh Control

- ***Mesh Control*** *is a feature that allows the User to specify the initial mesh size of a geometric feature, regardless of the body's default mesh size*

- *From the starting feature on which the mesh control is applied, the mesh size then gradually transitions (or grows) to a larger element size until it meets up with the default mesh size*



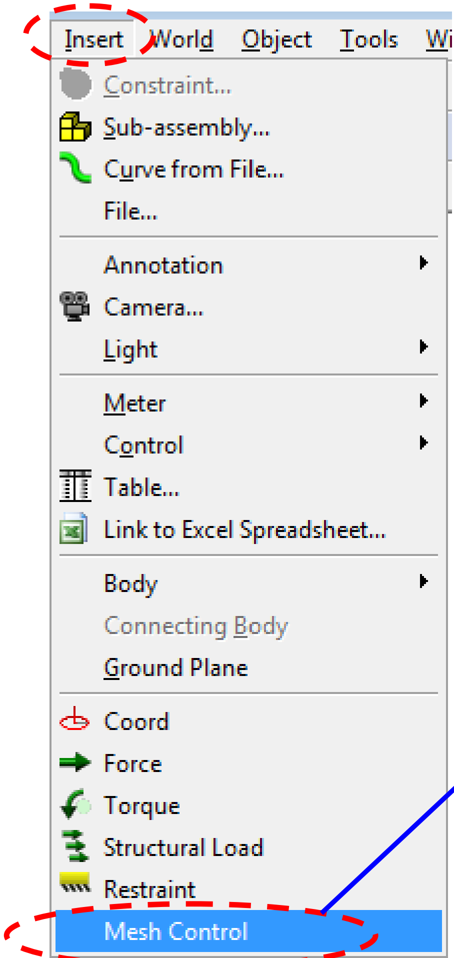
No Mesh Control

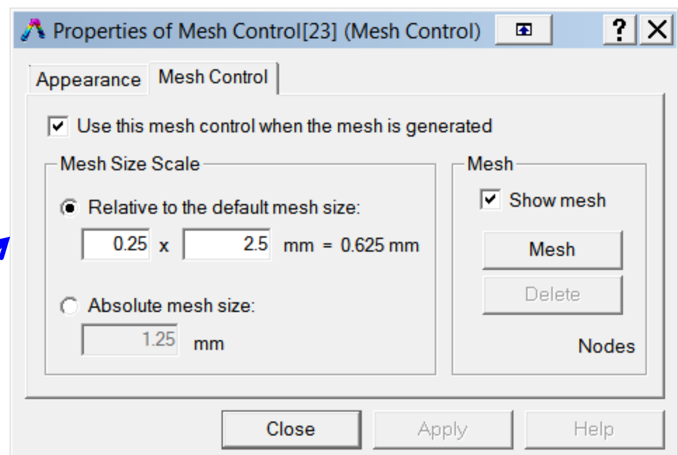


Mesh Control

Mesh Control

Accessing Mesh Control



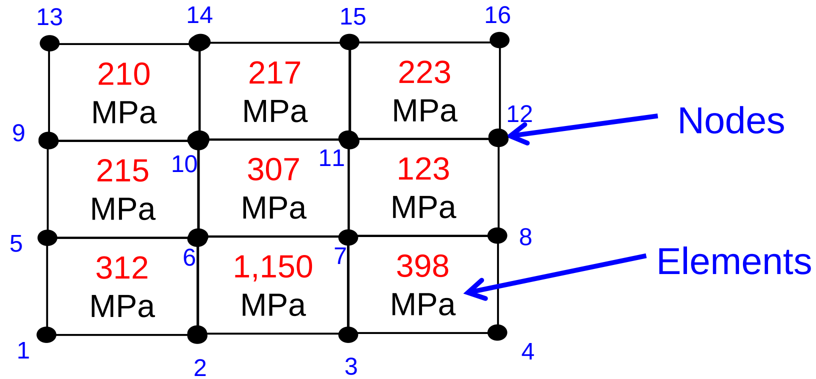


Error Estimates in FEA

➢ *Due to the nature of mesh quality in general FEA modeling, there exists some level of error in the stress values for elements. For some elements this may be large, for others it may be small*

The picture below shows a representation of a simple element error estimate. Using node 6, the maximum stress acting on an element is 1,150 MPa and the minimum is 215 MPa. Therefore, the reported error for node 5 might be (1,150-215)/1,150 = 81.3%. Similarly, taking the error on node 11 we get (307-123)/307 = 59.9%. This is a quite large difference in error across a single element (307 MPa element). H-adaptive tools in SimWise allow the user to control how well the program will automatically reduce this error by refining the mesh in this area.

Simplified mesh representation of nine adjoining elements



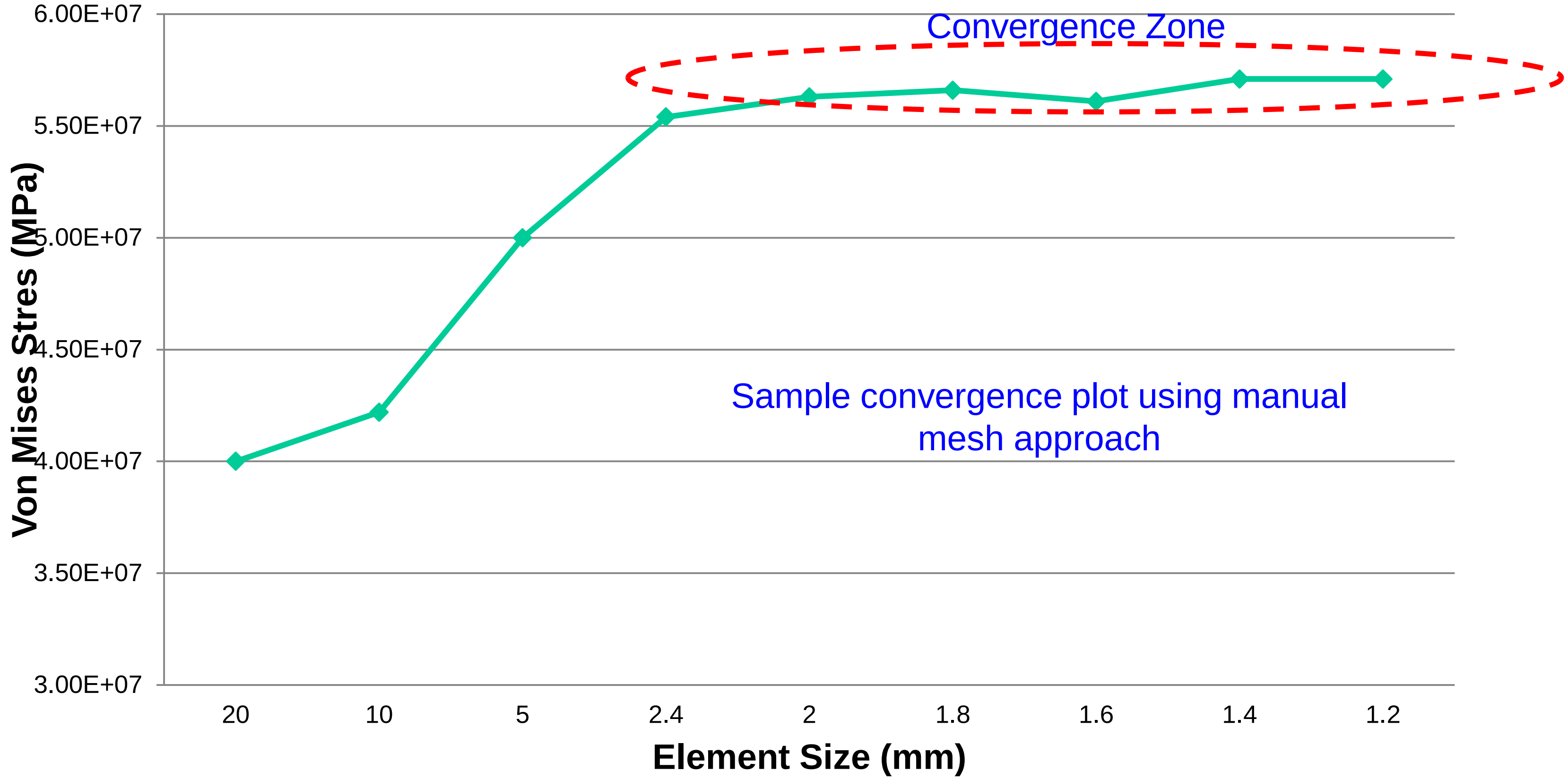
Design Simulation Technologies

Convergence

- *The process of successively refining a mesh to produce optimal results is called* ***convergence***

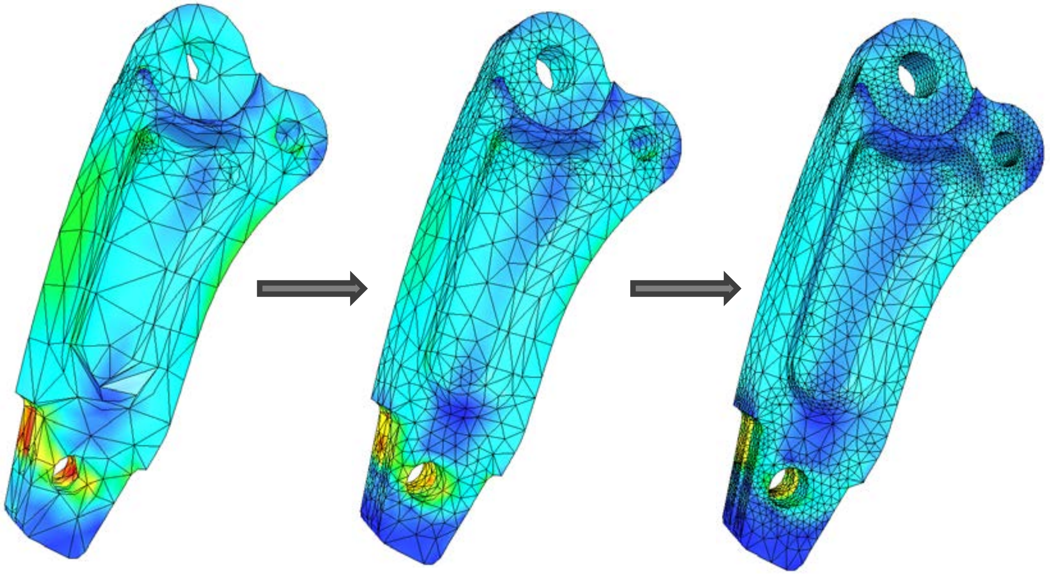
- *When the mesh is continually refined in areas of high stress error such that successive runs begin yielding negligible overall result differences, one can say the solution has converged within the desired tolerance*

Convergence



H-Adaptivity

- *SimWise uses a process called **H-Adaptivity** to automatically and selectively modify the <u>element size </u>and <u>element density </u>in areas that experience high stress error.*

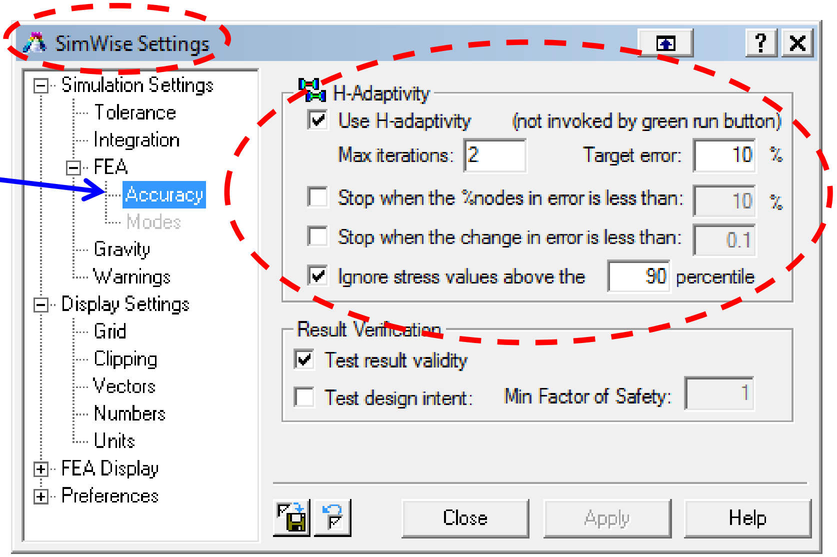


H-adaptive refinement

Convergence Criteria

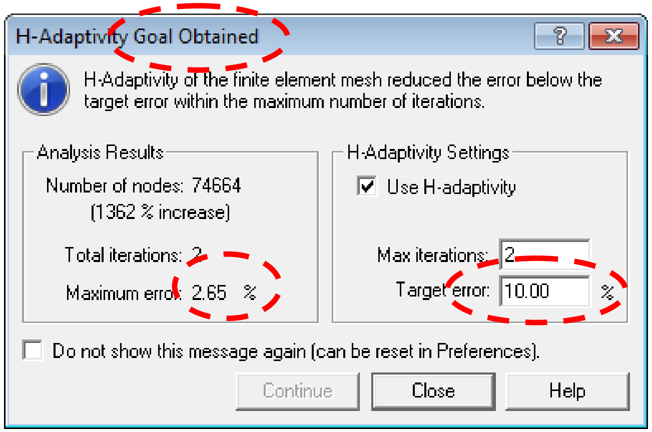
➢ *SimWise allows the User to specify error **convergence criteria** for the H-adaptive process. These criteria can be things such as the percentage error across elements, the percentage of nodes that are in error and the overall change in stress error between successive mesh refinements.*

Accessing H-adaptive settings

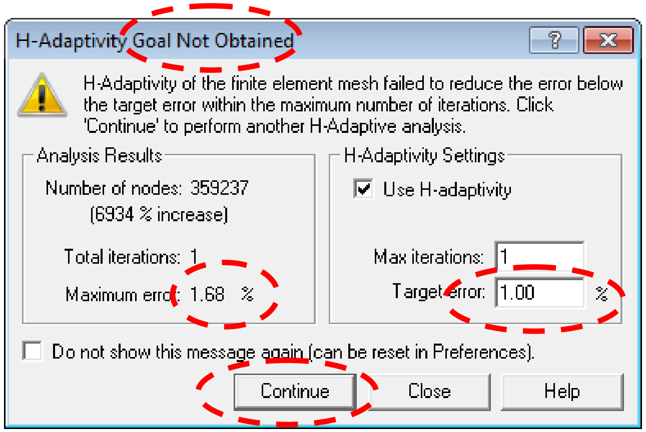


H-Adaptive Results

➢ *When SimWise completes an H-adaptive solve, a message window appears notifying the User of the results*



Successful convergence

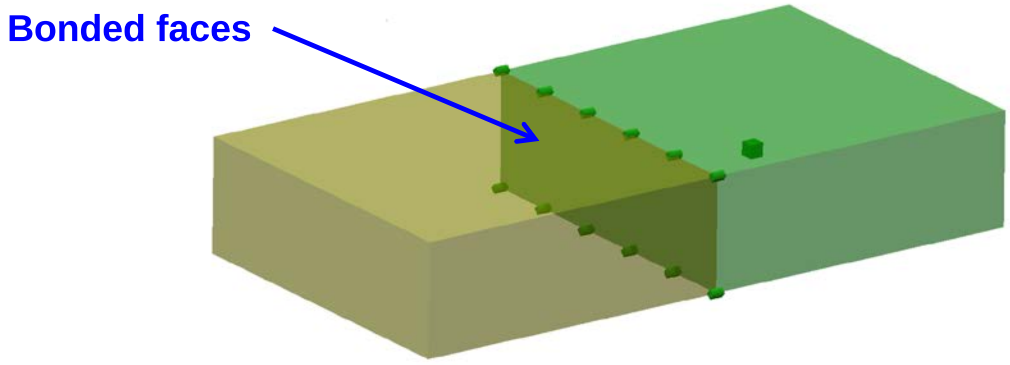


Failed convergence
(User is allowed to define new criteria and continue the solve)

Bonding

➢ ***Bonding*** *is the FEA process of joining two or more bodies together by joining the nodes on specified faces*

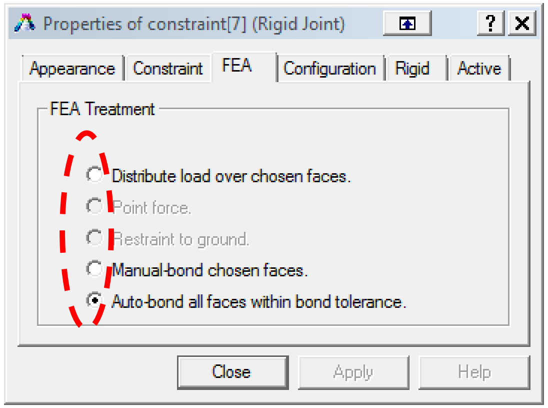
*Note: A **rigid joint** must exist between bodies for them to be bonded*



➢ *Bonds are not flexible on a large scale but instead treat the bodies as if they were welded at the specified faces. Bonded nodes are allowed to move a very small amount based on a percentage movement of the connected nodes.*

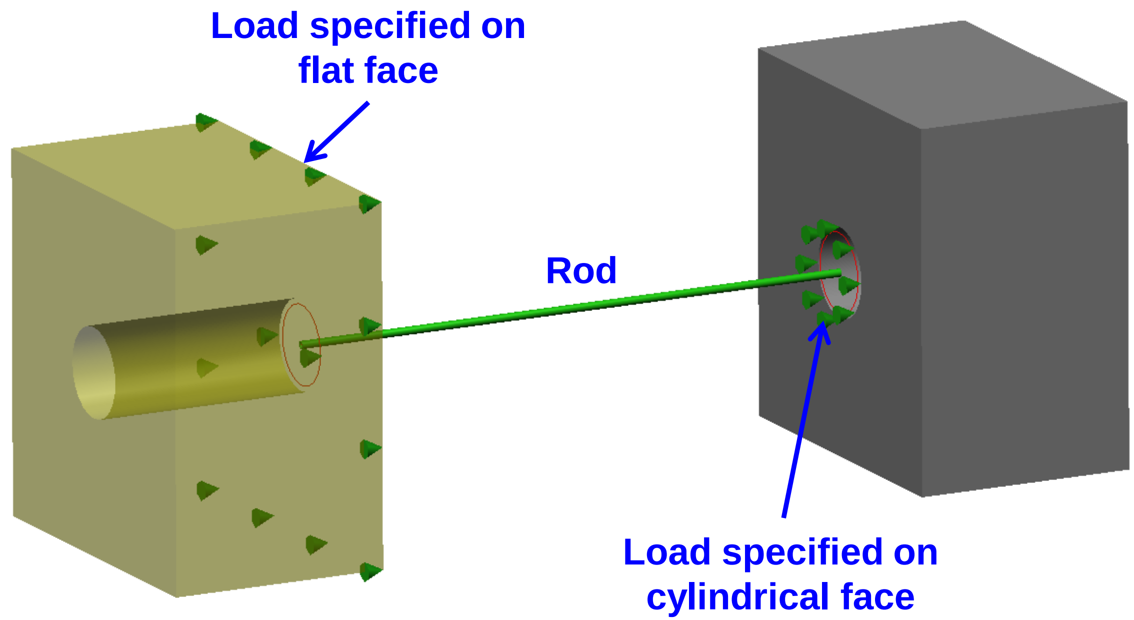
Bonding

- *There are different treatment types for bonding*
- *Different constraint types will have different treatment options available*



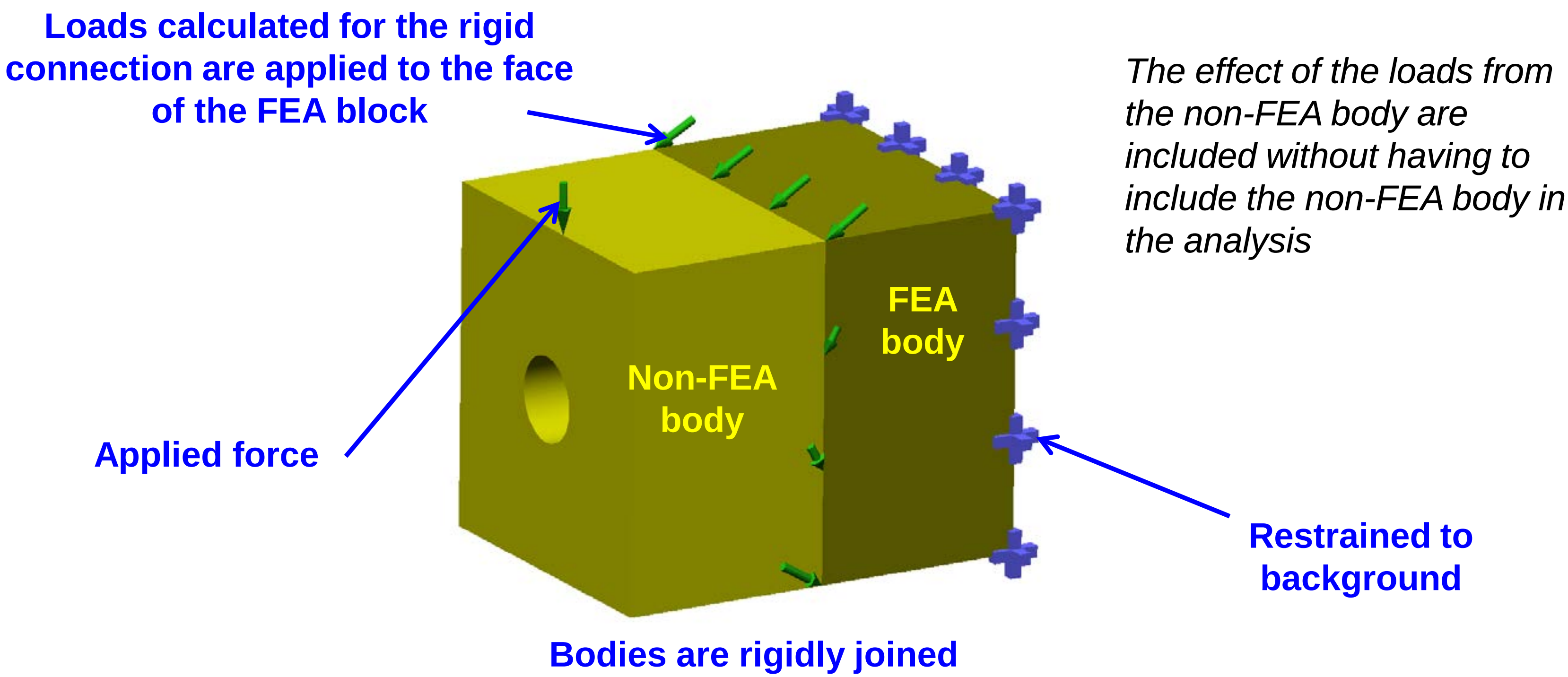
Bonding (Distribute loads over chosen faces)

- *The* ***Distribute loads over chosen faces*** *treatment option uniformly distributes the loads transmitted through the connection, onto specified faces*



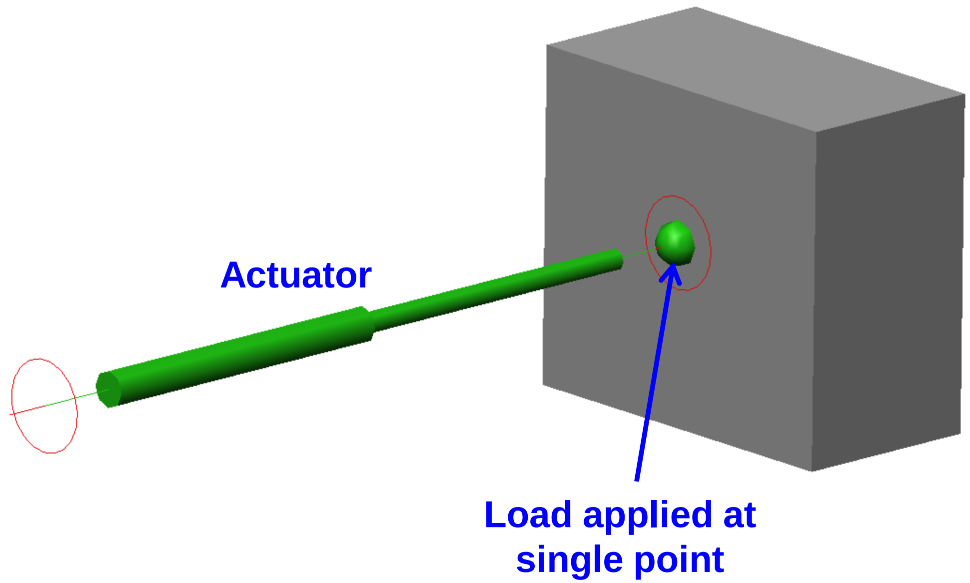
Bonding (Distribute loads over chosen faces)

- *The* ***Distribute loads over chosen faces*** *treatment option is also useful when it is desired to transmit loads between an FEA body and a non-FEA body.*



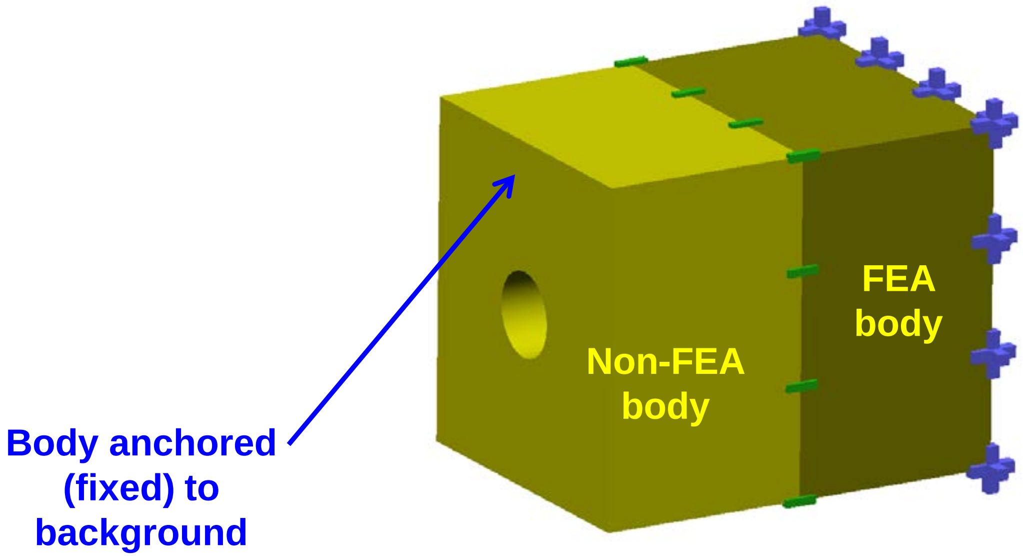
Bonding (Point Force)

➢ *The **Point force** treatment option is used for pt-pt constraints, such as rods, ropes and linear actuators. This option applies all transmitted load through single connection points (nodes) on each body*



Bonding (Restrain to ground)

➢*If one body, which is also a non-FEA body, is fixed in space using the anchor tool, and it is attached to another body using a rigid joint, the* ***Restrain to ground*** *option is available.*

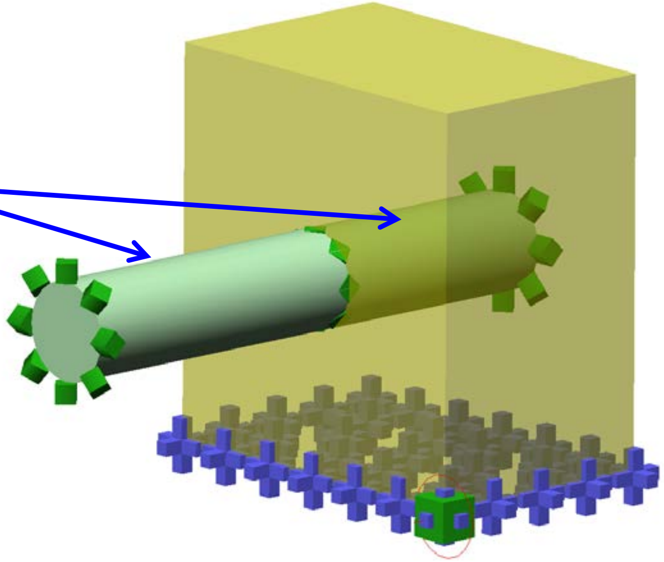


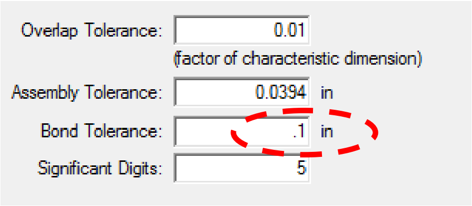
Bonding (Manual bond chosen faces)

- *The* ***Manual bond chosen faces*** *option allows the User to override the automatic bonding of faces and specify different faces to be joined.*

- *This option is particularly useful when, for example, a feature like a weld, which joins to adjacent faces, is intentionally left out of the model.*

The inside cylindrical face of the hole and the outside cylindrical face of the shaft have been designated as the chosen faces to bond together





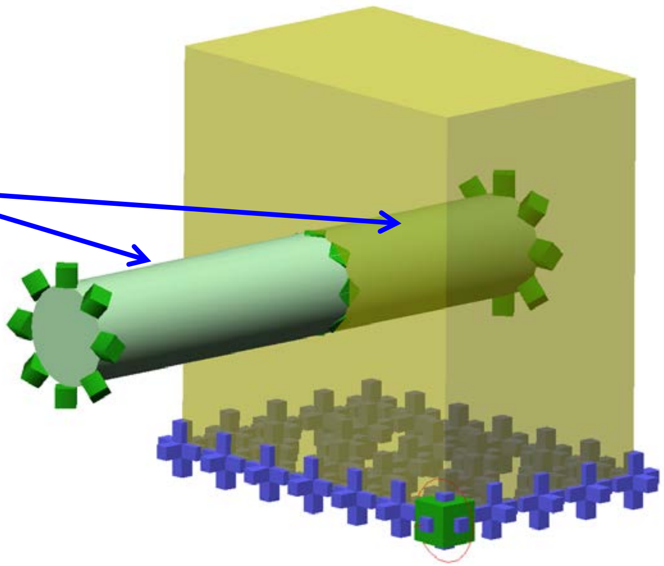
Note: The faces must be within the bond tolerance value in order to successfully bond

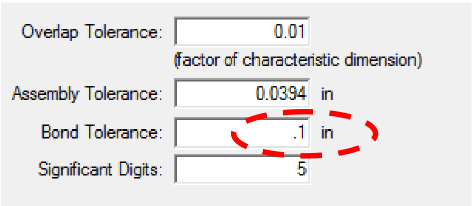
Bonding (Auto-bond all faces within bond tolerance)

- *The **Auto-bond all faces within the bond tolerance** option allows SimWise to automatically join any pair of faces that are within the specified bond tolerance*

- *This option is similar to the Manual bond chosen faces feature except that it automatically detects the faces to join.*

The inside cylindrical face of the hole and the outside cylindrical face of the shaft have been automatically bonded





Note: The faces must be within the bond tolerance value in order to successfully bond