Geometric Singularities in Finite Element Problems

Alex Grishin, PhD
1/9/2024

Background

- A subtle, yet very important practical issue in building, solving, and interpreting finite element models is the occurance of singular points –locations at which some solution quantity (usually stress or strain) fails to converge to a finite value upon mesh refinement
- Historically, this is an important problem in the field of differential equations –especially numerical solutions to these equations.
- Though well understood mathematically, engineers –especially novices, struggle with the idea that certain locations in a finite element model produce results which may not be reliable –even in models which may be perfect in every other way.
- This issue is so pervasive that users will have no problem finding literature and tutorials on the subject (see, for example here and here)
- However, while most of the literature does a fine job of providing guidelines and remedies for handling such situations, we've found very little which helps users identify where and why they occur (and will provide the links we *have* found where appropriate)
- In this article, we'd like to explore this in more detail (where and why geometric singularities occur)

Definition: Mesh Convergence

- To begin, we'd like to define exactly what we mean by a *geometric* singularity in a finite element model. We'll do so in terms of the mesh convergence rate
- In the absence of singularities or other pathologies, finite element solutions should exhibit the following *error** in the computed stress, σ_h[1]:

$$e_\infty = \|\sigma - \sigma_h\|_{L_\infty} \leq C h^{p+1} \left\|\sigma^{(p+1)}\right\|_{L_\infty} \quad (1)$$

- This term is independent of the FE model

- $\|\cdot\|_\infty$ is the 'infinitey norm'. It just measures the maximum absolute value over a set
- h is the element size. The last term ($\sigma^{(p+1)}$) means the 'pth+1' deriviativie of σ
- If we just assume this value is bounded *for a given problem*, we can replace (1) with (2):

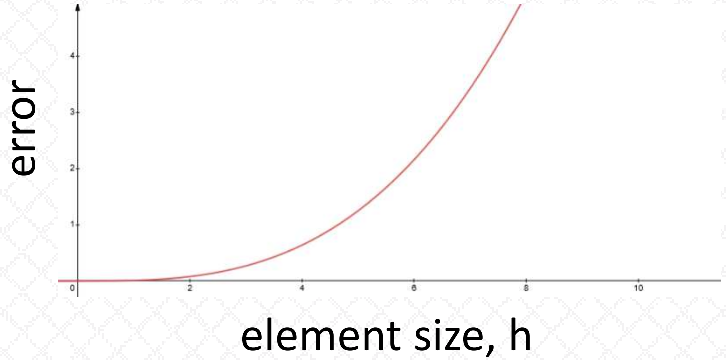
$$e_\infty = \|\sigma - \sigma_h\|_{L_\infty} \leq D h^{p+1} \quad (2)$$

- Since p is the degree of the underlying FE shape functions being used, (2) tells us that the expected error in the maximum absolute value of stress shold grow as a monomial function of the element size raised to a power of underlying shape functions plus one

*This is a classical result in the theory of finite elements. This monomial depence on rate of convergence and the degree of the basis functions used is a major reason for the widespread adoption and success of the method historically

Definition: Mesh Convergence

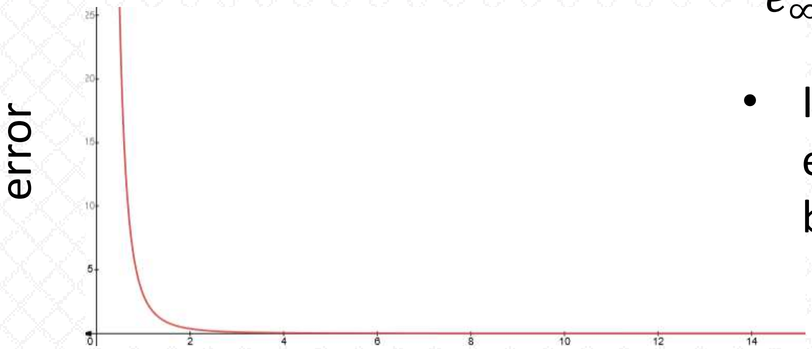
- If the underlying element shape functions are of 2nd order (quadratic), then we could expect a (cubic monomial) curve that looks like the one below



- Instead of plotting the error as a function of element size, we could also plot the error vs. number of elements (or number of degrees of freedom) by noting that, in general the number of elements, n is inversely proportional to element size, h:

$$n \propto 1/h \approx c/h$$

$$e_{\infty,n} = \|\sigma - \sigma_n\|_{L_\infty} \leq E n^{p+1} \quad (3)$$



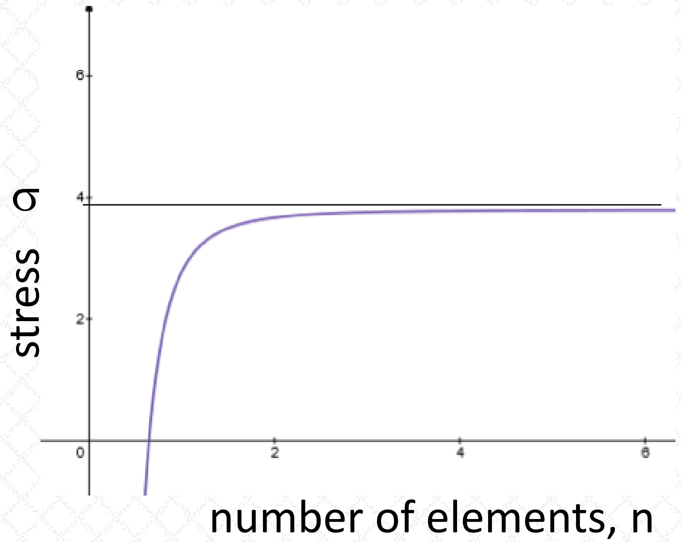
- It's also helpful to note that the number of elements, n is bounded from below ($n \geq 1$), because $h \leq |\Omega|$

Definition: Mesh Convergence

- where Ω is the model domain. In other words, the maximum element size, h is bounded by the model size (can't have less than 1 element in the model)
- Finally, we can subtract the exact solution, σ from both sides to obtain:

$$\|\sigma_n\|_{L_\infty} \leq \left\|En^{p+1} - \sigma\right\|_{L_\infty} \qquad (4)$$

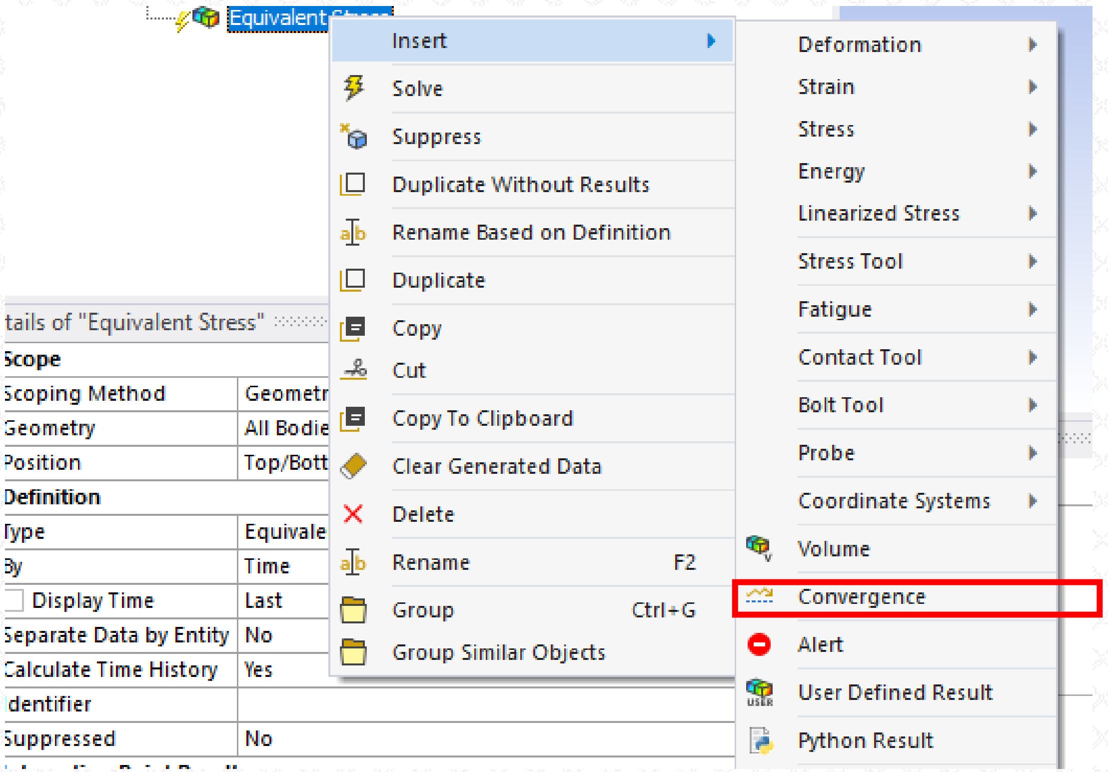
- We expect a stress 'convergence' graph to look like a plot of (4) (note that there is some sign ambiguity in (4). We chose the form which users will encounter most often when performing convergence studies)



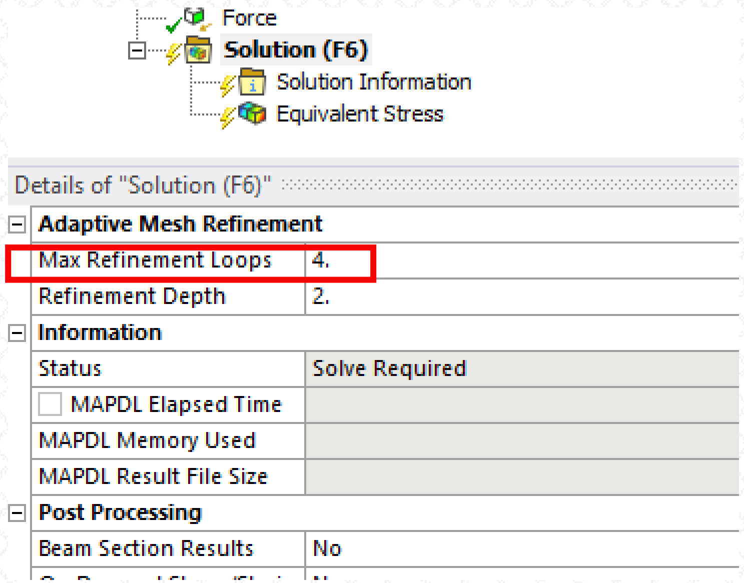
- Now, we won't know what σ is in most cases, but we should still see a curve converging toward it
- When this is not the case, we'll know we have a problem

Definition: Mesh Convergence

- Of course, we can produce a graph like (4) in Ansys manually –by graphing the maximum stress in a model vs. number of elements for successive values of grid refinement
- But Ansys offers a convenient tool which generates curves almost like (4) automatically
- This is called the ‘convergence’ tool (simply right-click on any result object of interest in the Mechanical tree outline and insert->Convergence)

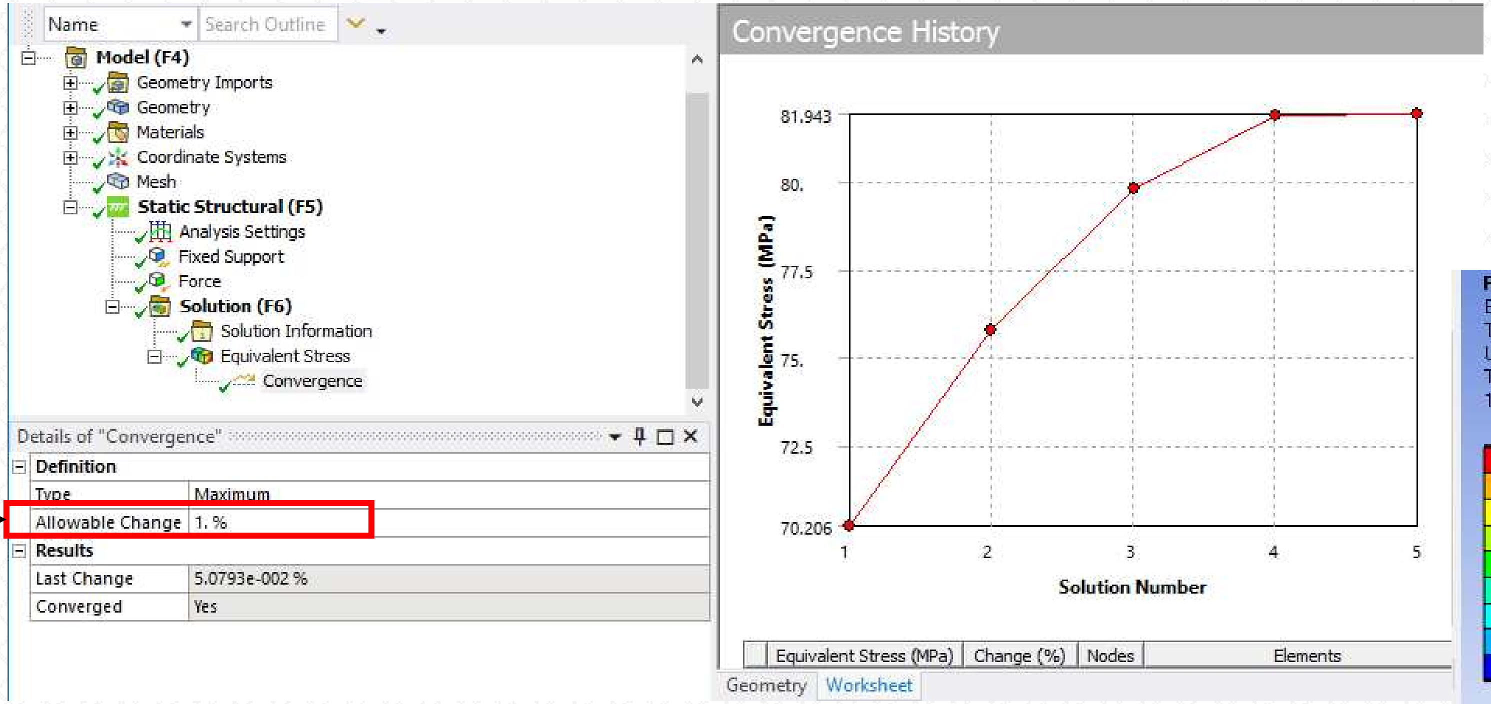


- You set the number of points on every ‘convergence’ plot by setting the ‘Max Refinement Loops as shown below

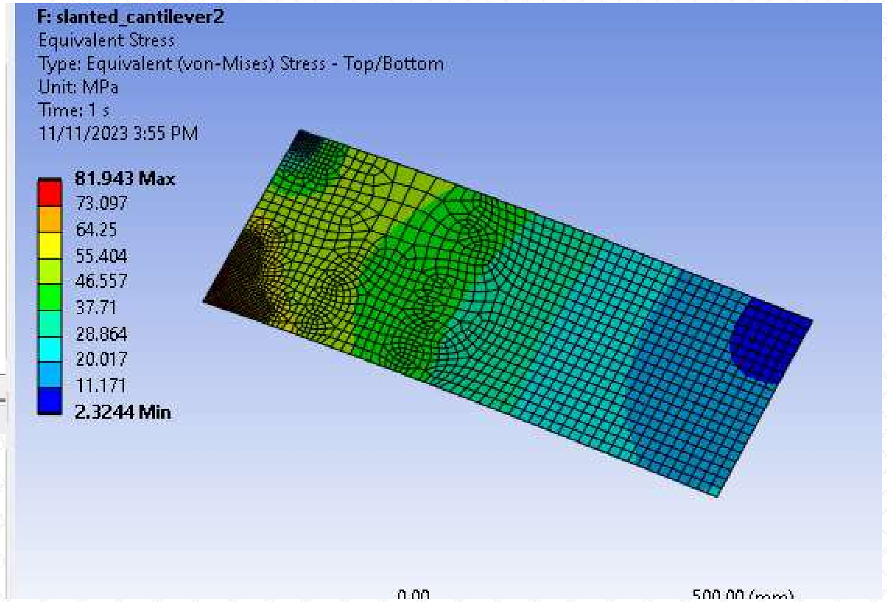


Definition: Mesh Convergence

- Below is a graph of the convergence of von Mises stress for a particular model in Ansys (the details aren't important for now) for four refinement 'loops' (levels of refinement)



- refinement is carried out according to the stress *a posteriori* error*
- Most refinement therefore occurs locally around the highest stress

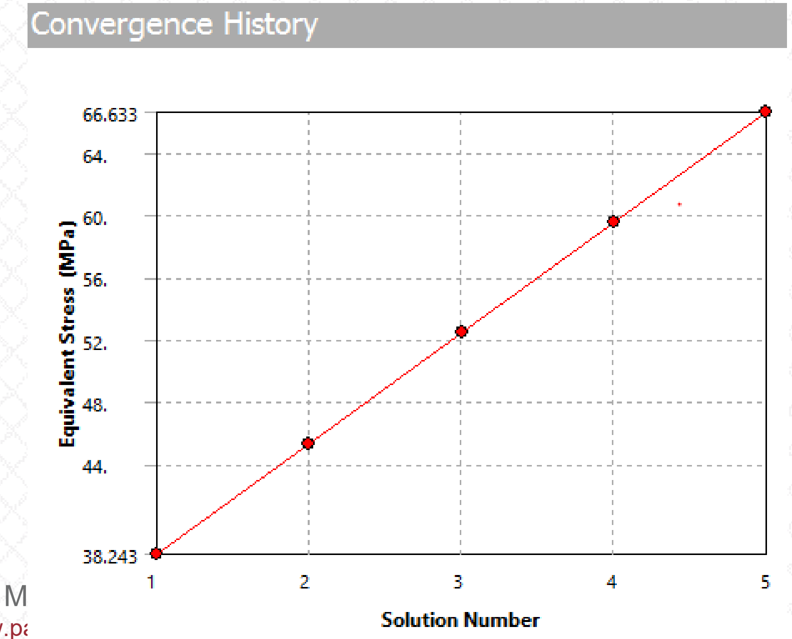


- users specify the maximum 'Absolute Change' allowed during a refinement loop to determine whether convergence is achieved

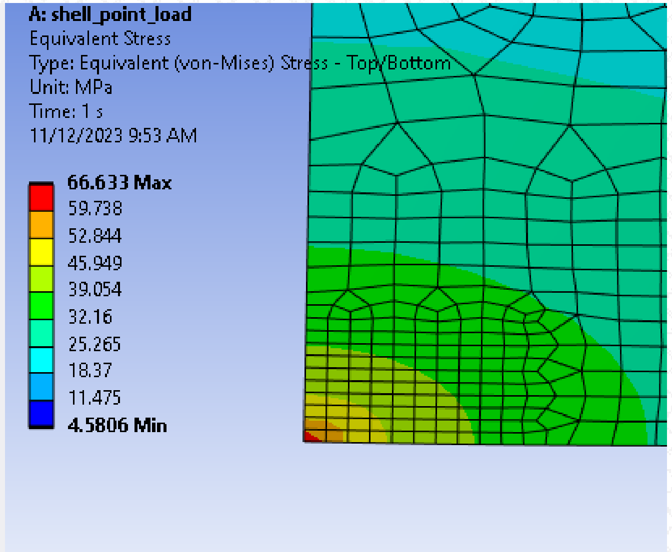
*an important algorithm beyond the scope of this article. See here for details

Definition: Geometric Singularity

- Now we're in a position to define what we mean by a geometric stress singularity
- A geometric singularity is the source of any mesh-refinement convergence plot in which the quantity of interest fails to converge (it violates equation (4). See below)
- The word 'geometric' (we use that to distinguish this phenomenon from other sorts of numerical or theoretical convergence issues) may refer to the loading or boundary condition pattern as well as the geometric domain*
- This singularity happens to be caused by a point (nodal) force load, which should beg some questions:
  1. Are point loads always singularities?
  2. If the answer to 1. is 'no', when can we expect them?
  3. Can other sorts of boundary conditions or geometry lead to this behavior?



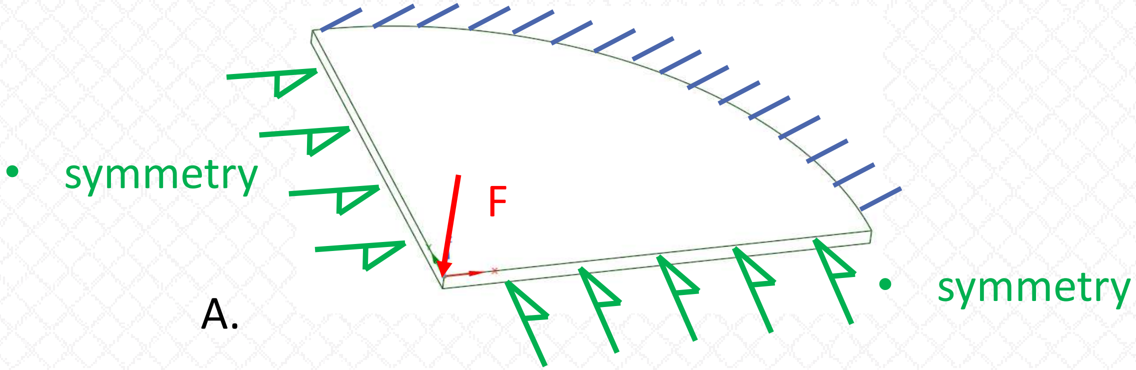
- An actual stress singularity!

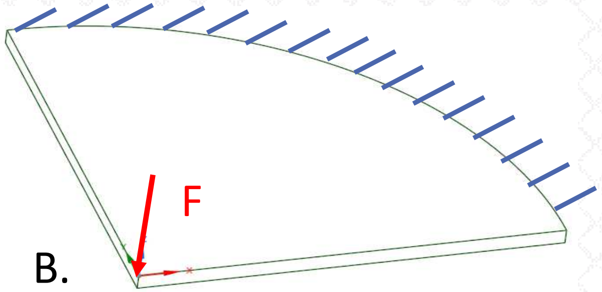


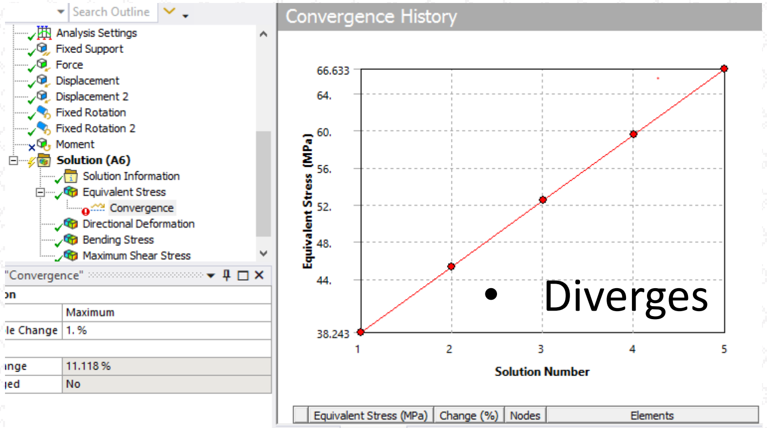
*unfortunately, there are other possiblities –such as a material discontinuity – which may cause convergence failure, but those are beyond the current scope

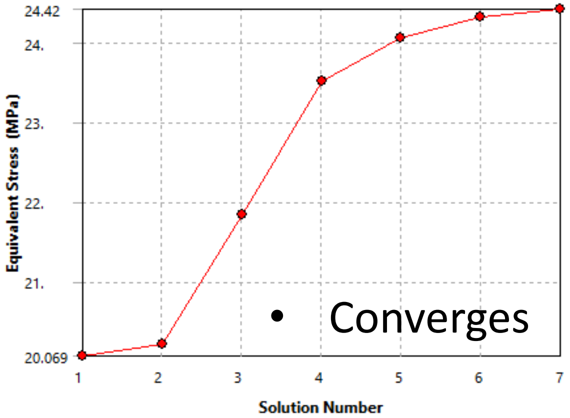
Motivation: A Mystery To Be Solved

- To help further motivate this article, consider a thin, quarter-symmetric circular plate (using plate elements having 6 DOFS: ux,uy,uz,rotx,roty,rotz) loaded and constrained as shown in the two cases below
  A. Transverse load at center. Symmetry on cut faces*. Fixed on circular boundary
  B. Transverse load at center. Fixed on circular boundary (no symmetry)
- The mystery: Case A has a geometric singularity. Case B doesn't. Why?









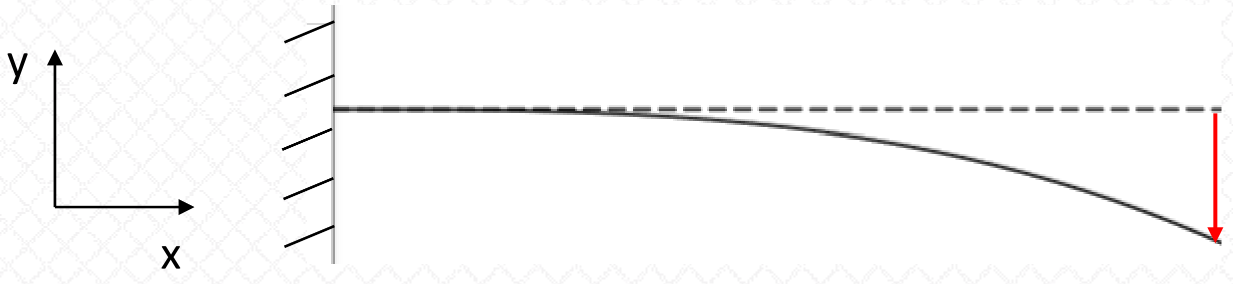
*Note that the mesh convergence tool will not work on regions with symmetry boundary conditions.

- We get around this by applying normal 0 displacements along cut faces and parallel 0 rotations along the same faces

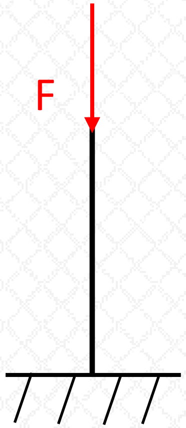
- The odd 's' shape of this curve has to do with the fact that element refinement is not occuring uniformly (a downside to using the automated convergence tool)

Introduction and Preliminaries

- We have some work to do before answering the questions on the previous slide
- To start the discussion, it’s important to point out that we need at least two spatial dimensions for a geometric singularity to exist (this is also partly the reason for naming it a ‘geometric’ singularity)
- Thus, a solution field has to be a function of the form f(x,y) or f(x,y,z) in order to have a gometric singularity
- All readers doubtless already have an intuition that these singularities have something to do with abrupt, or ‘step’ changes in geometry or loading –and so they do. But for most domains with fewer than two canonical dimensions, no load or constraint can be so abrupt as to cause a stress singularity*
- This is because all result quantities and their derivatives are a function of a single spatial direction (f(x) or f(y)) and how abruptly a solution can change between elements is strictly governed by the continuity of the result (in the case of finite elements: chape functions with C(1) or higher continuity)
- In other words, all loads and boundary conditions in one spatial direction act to either define or sub-divide the associated boundary value problem for which bounded solutions exist
- Another way of putting this: Point loads and Dirichlet boundary conditions all produce the same types of well-behaved, bounded results in a single spatial dimension



- No singularities here

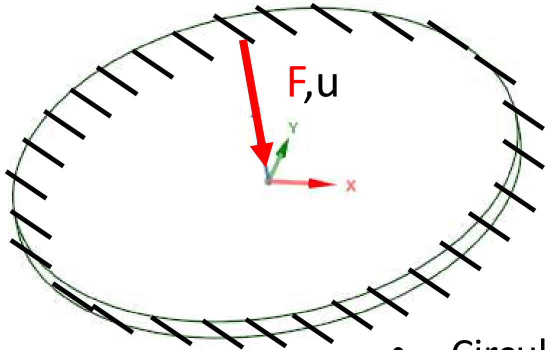


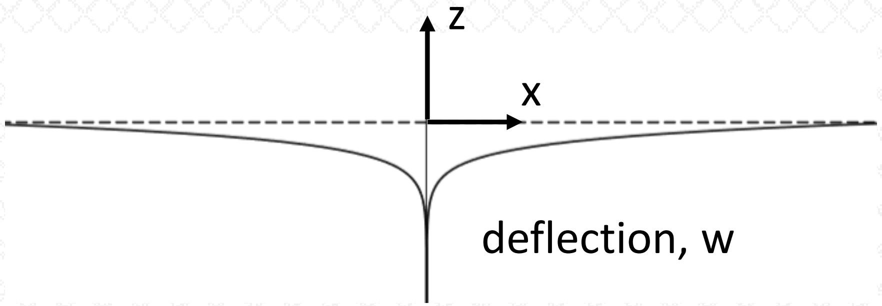
*An exception would be 2D membrane elements (shell209). For that case, everything we’ll say about membranes in 3 dimensions applies in 2

Geometric Singularities: Type I
Divergence of Primary Solution Variable
Reduced-Order Elements

- By far the most severe type of geometric singularity would be one which either invalidated the results, or (worse) prevented one from obtaining a result in the first place (let alone a verifiable one)
- We'll name these 'Type I' singularities, and they arise from a special sub-class of differential equations ('elliptic' equations). One such equation is Poisson's Equation in the primary variable of interest (the one we solve for in the resulting algebraic system). They are characterized by mesh convergence failure of this variable (and when this fails to converge, so will all its derivatives)
- Among Ansys' structural element types, only a few solve Poisson's Equation for deflection*. These fall in to a category we'll call 'Reduced-Order' elements in recognition of the fact that, at least for problems in elasticity (in three dimensions), these element-types do not carry the full elastic strain tensor corresponding to the spatial dimensions of the problem
- Poisson's Equation for elastic deflection is used to model membrane problems. So, the first problem we'll look at is a circular membrane subject to transverse central point load

*Equations of the form:
$\nabla^2 u = f(r,\theta)$





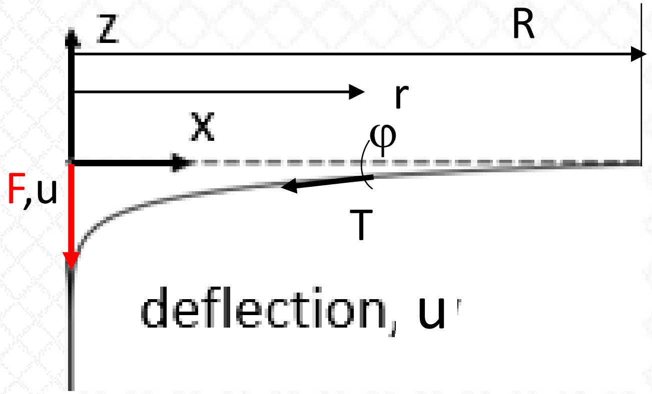
- Circular menbrane wth a central point load, F and transverse deflection response, u

Geometric Singularities: Type I
Divergence of Primary Solution Variable
Reduced-Order Elements

- The differential equation for this problem [1] can be simply derived by applying Newton's third law to an arbitrary radius, r:

$$-F + 2\pi r T sin\varphi = 0 \qquad (5)$$

$$sin\varphi \approx tan\varphi = -\frac{\partial u}{\partial r} \qquad (6)$$



- Now, substitute (6) into (5) and solve for the slope:

$$\frac{\partial u}{\partial r} = -\left(\frac{F}{2\pi T}\right)\frac{1}{r} \qquad (7)$$

*Leissa, A.W. (2001), Singularity considerations in membrane, plate and shell behaviors. International Journal of Solids and Structures, 3341-3353

Geometric Singularities: Type I
Divergence of Primary Solution Variable

Reduced-Order Elements

- Differentiating (7) produces Poisson's Equation:

$$\frac{\partial^2 u}{\partial r^2} = \left(\frac{F}{2\pi T}\right)\frac{1}{r^2}$$

- But in this case, we can solve (7) directly to obtain (making use of the fact that u@(r=R)=0:

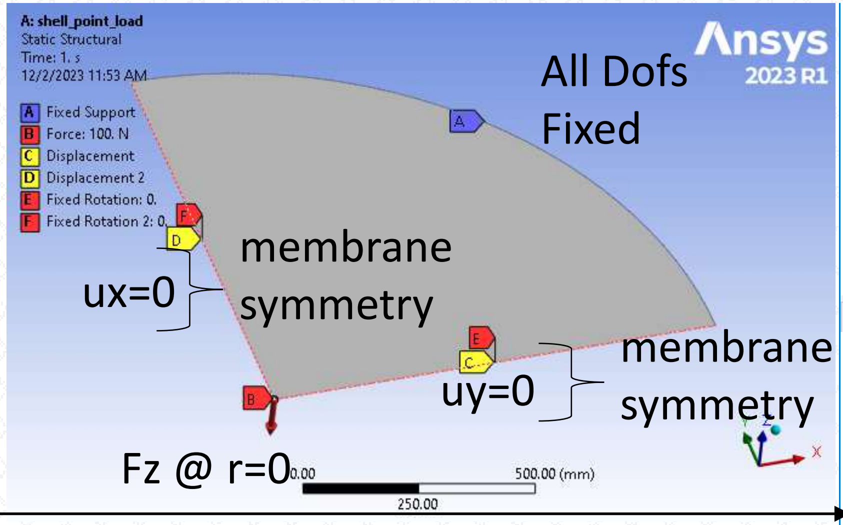
$$u = \frac{F}{2\pi T}\ln\left(\frac{r}{R}\right) \qquad (8)$$

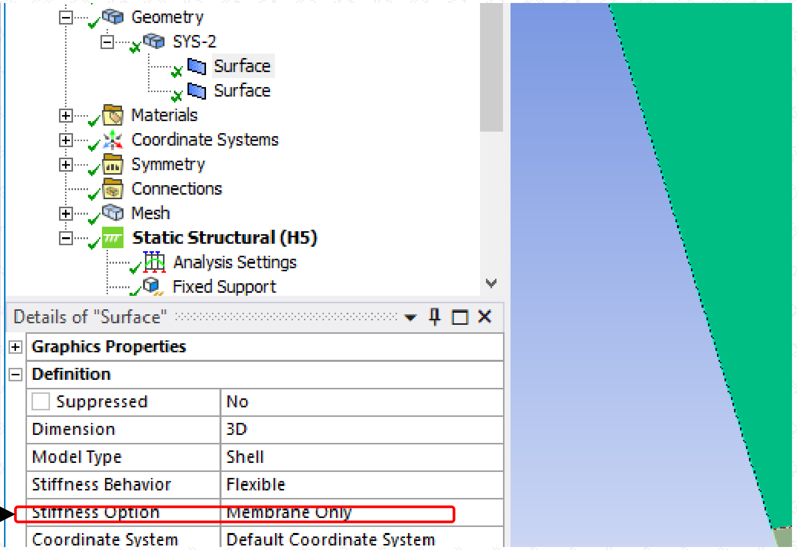
  - These equations tell us that u(0) = ∞ (Equation (8)), and $\frac{\partial u}{\partial r}(0) = \infty$(Equation (7))
  - In other words, we should expect any finite element approximation of this problem to result in a geometric (or some type of) singularity, as both the primary solution variable (deflection in this case) and the strain are unbounded at r=0

Geometric Singularities: Type I
Divergence of Primary Solution Variable

Reduced-Order Elements

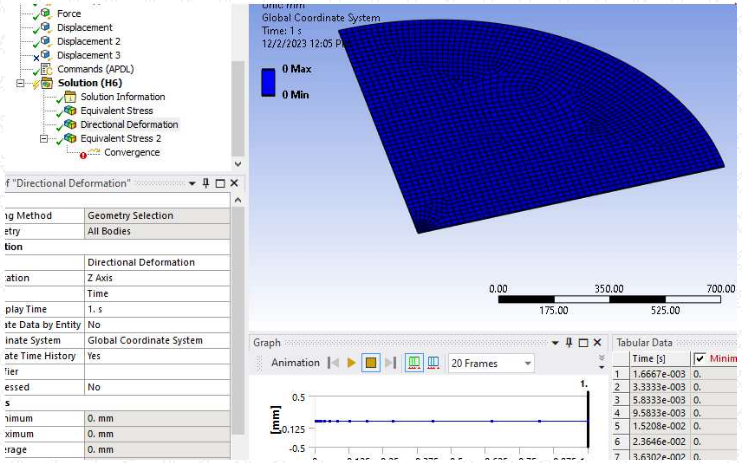
- Trying to simulate the circular membrane (by setting the 'membrane only' option on surface shell elements) using quarter symmetry, we discover a peculiar fact about this problem





¼ Symmetry Membrane Model

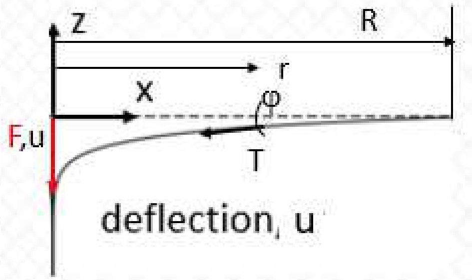
- Attempting to solve this problem as a purely linear one (small deflections) results in a null solution!

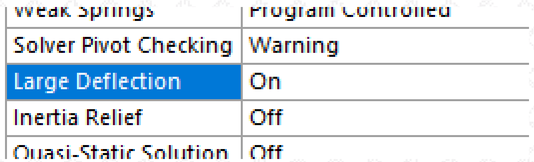


Geometric Singularities: Type I
Divergence of Primary Solution Variable

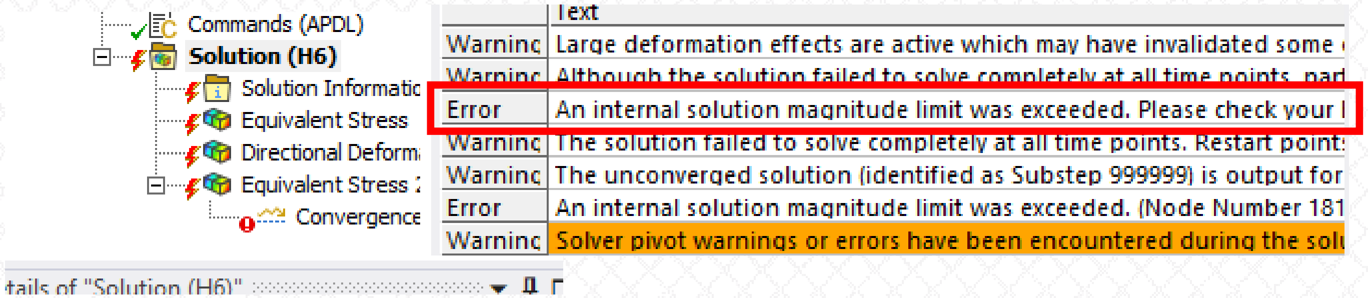
Reduced-Order Elements

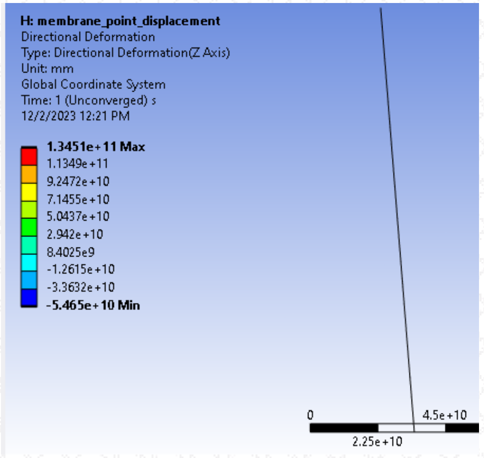
- The reason for the null solution is that shell/membrane elements in Ansys cannot solve equations like (7) or (8) with the small deflection assumption
- With this assumption, the transverse load, F acts at a right angle to the undeformed plane of the membrane and hence cannot act to distort it
- The only way to simulate the evolving strain reaction to the transverse load (the membrane has to strectch to accomodate a vertical displacement at r = 0) is turn on “Large Deflection” effects (under Analysis Settings)





- When we do this, we get a the following result





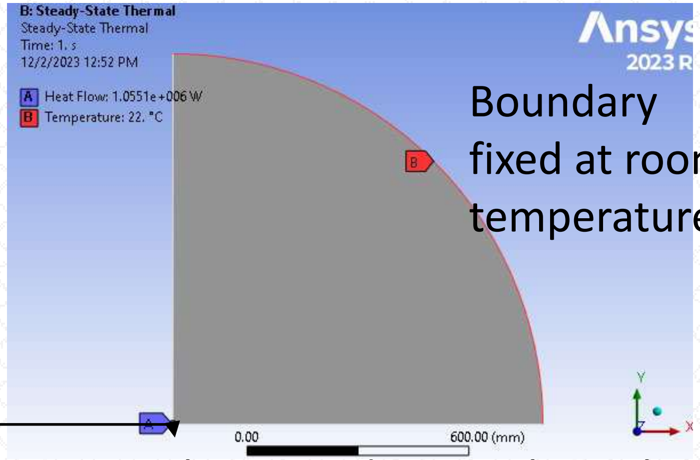
- Unconverged maximum deflection =1.35 x 10^{11} mm

Geometric Singularities: Type I
Divergence of Primary Solution Variable
Reduced-Order Elements

- We can understand this as Ansys trying to approximate the unbounded solution of Equation (8)
- If a linear-strain (small deflection) solution existed, we would expect a large (but still 'bounded') deflection. And this deflection would grow in an unbounded way with mesh refinement
- But nonlinear Newton-Raphson iterations of the membrane problem attempt to reduce the error between successively perturbed linear solutions –in other words, for the given mesh in this case, it attempts to reduce the error in approximating ∞ -- which isn't possible with finite precision
- To see what a *linear* solution of the Poisson Equation would look like, we can attempt to solve an analogous problem* in conductive heat transfer
    - Governing Equation:

$$-k\nabla^2 T = 0$$

Heat applied to single point at r=0



Boundary fixed at room temperature

2D ¼ symmetry heat conduction through cylinder

*not quite analogous: This example results in a Laplace Equation. But it still retains the qualities we wish to exploit

Geometric Singularities: Type I
Divergence of Primary Solution Variable
Reduced-Order Elements

- It's worth dwelling for a moment on the solution to this equation for point sources in 2 dimensions
- The governing equation (from the previous slide –obtained by omitting the transient term of the heat equation) leads directly to Fourier's Law (upon integration)

$$-k\nabla T = q$$

  - but the heat flux q (heat per unit time per unit area) at any location r from the origin is simply the point load Q (heat per unit time) divided by the area

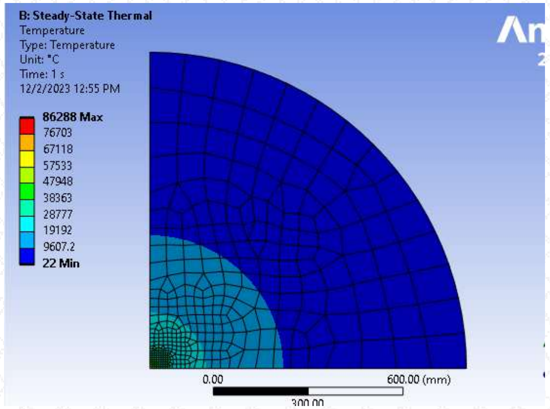
$$-k\frac{\partial T}{\partial r} = \frac{Q}{2\pi r t} \qquad (9)$$

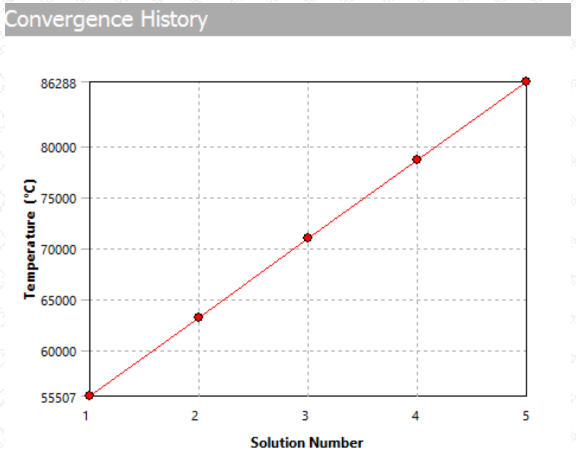
  - for a disk of unit thickness t, then we can rewrite Equation (9) as Equation (10)

$$-k\frac{\partial T}{\partial r} = \frac{Q}{2\pi r} \qquad (10)$$

  - Finally, the solution to transient heat conduction problem in 2D with a point source is:

$$T(r) = T_0 + \frac{Q}{2\pi k} ln\left(\frac{R}{r}\right) \qquad (11)$$



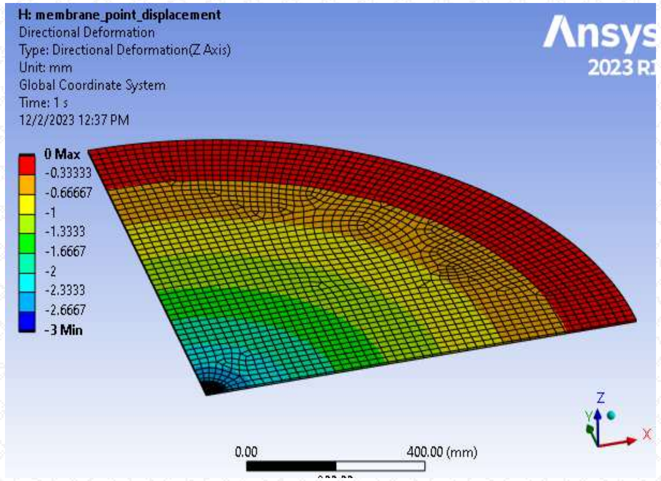


- And once again, T(0)=∞ and $\frac{dT}{dr}(0) = \infty$
- This is indeed another Type I singularity

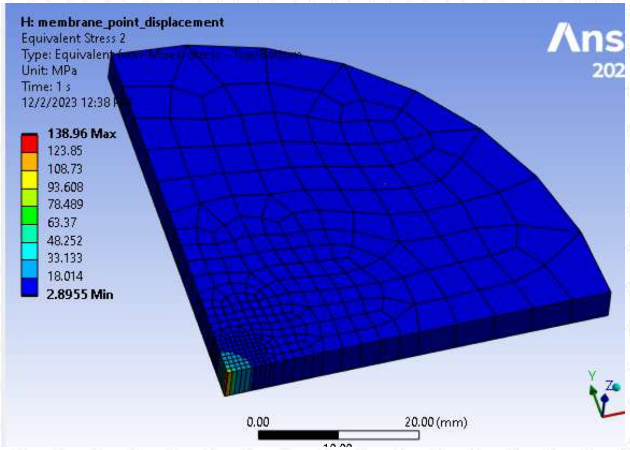
Geometric Singularities: Type I
Divergence of Primary Solution Variable

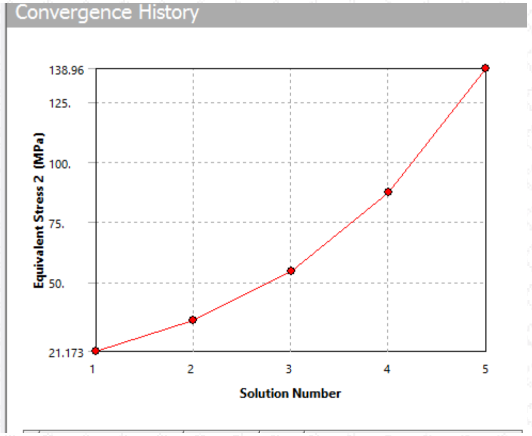
Reduced-Order Elements

- Returning to the ¼ -symmetry membrane problem, let's explore further
- What happens if we replace the transverse applied force with a displacement?
- We do this to enforce 'boundedness' of the primary solution variable. What about the derivatives?



- Replacing the load with a constant displacement (to force the primary variable to be bounded)...





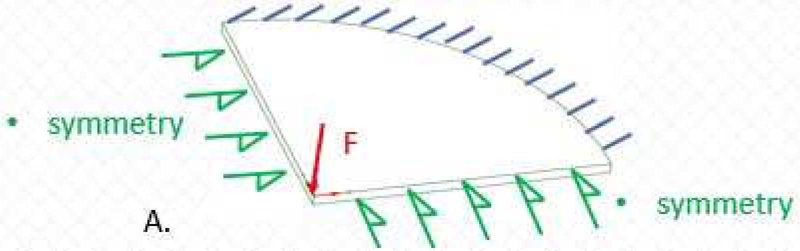
- ...the stress and strain are unbounded, even while the displacement converges. This is new type of singularity
- So, singularities in which the primary variable converges while it's deriviates do not DO exist, and we'll look at them next...

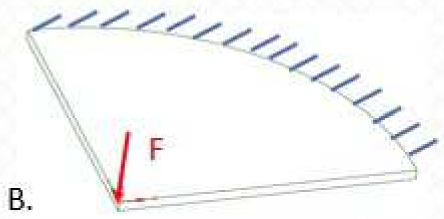
Geometric Singularities: Type II

Convergence of Primary Solution Variable –Divergence of Derivatives

Reduced-Order Elements

- There are certain types of finite elements which may experience geometric singularities in which the primary soluton variable converges, but its derivatives do not. This type of behavior is typically found in plate/shell elements, so we'll take a look at those now
- We'll call this a 'Type II' singularity. As we'll see, elastic shell elements tend to exhibit this type of singularity –but not always where one might expect them!
- A major theme we want to stress here is that zero-dimensional point loads are not, in and of themselves, the source of such singularities –their relationship to the associated differential equation is
- Let's return to the ¼-symmetry plate/shell on slide 9. We're now ready to tackle the mystery presented in that slide: The symmetry model below left generates a Type II singularity, while the model without symmetry below right converges absolutely (no singularities!)





Geometric Singularities: Type II
Convergence of Primary Solution Variable –Divergence of Derivatives
Reduced-Order Elements

- Solving the differential equation for plate problems is significantly more involved than solving the simple axisymmetric Poisson problems we've dealt with so far
- Even with axisymmetry, the resulting equations have more terms to deal with. Instead of providing a crash course in plate theory (well beyond the scope of this article), we'll simply refer readers to our earlier reference from slide 12 ([1]. For a more comprehensive treatment, see [2])
- From the references above, the analytic deflection, u in problem A. of the previous slide is:

$$u(r) = \frac{F}{8\pi D} r^2 ln r + \frac{F}{16\pi D}\left(R^2 - r^2\right) \qquad (12)$$

- where, D (the "plate stiffness" or "flexural rigidity") is given by:

$$D = \frac{Et^3}{12(1-\nu)}$$

- In spite of the appearance of the troublesome logarithmic term, this solution is bounded at r=0. Specifically:

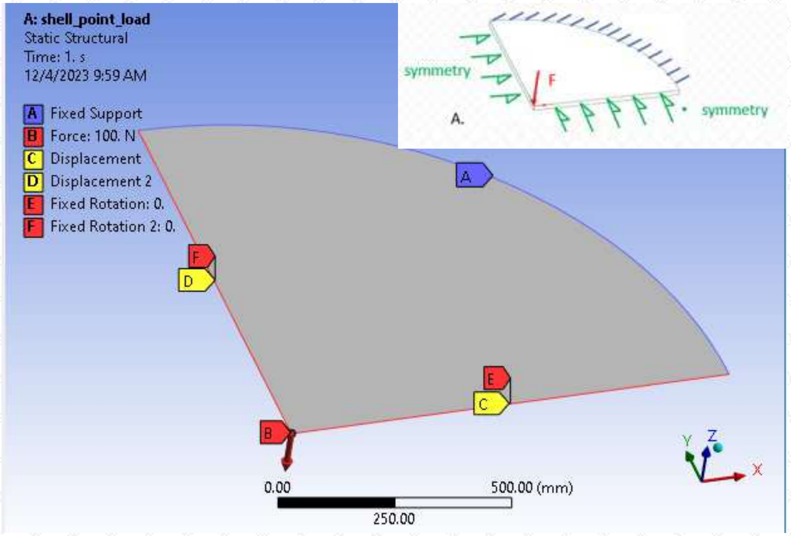
$$u(0) = u_{max} = \frac{FR^2}{16\pi D} \qquad (13)$$

Geometric Singularities: Type II

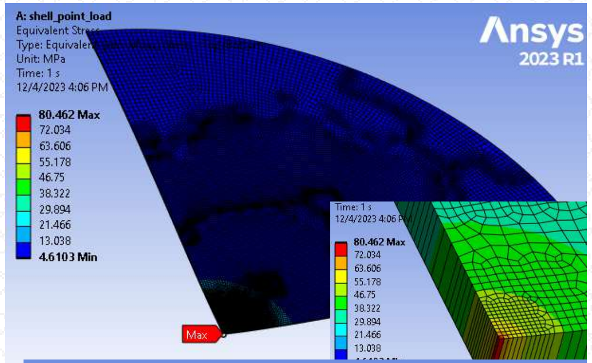
Convergence of Primary Solution Variable –Divergence of Derivatives

Reduced-Order Elements

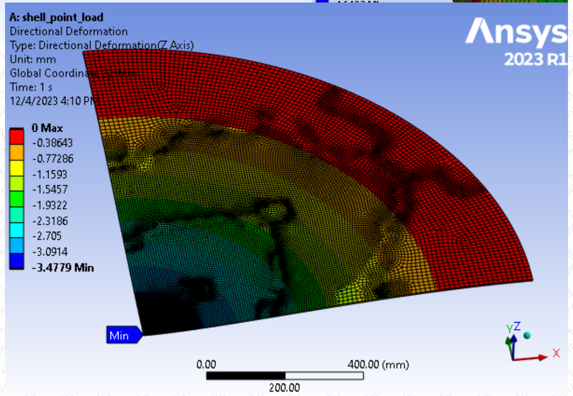
- but the bending moments (and corresponding stresses) involve $\frac{\partial^2 u}{\partial r^2}$ and $\left(\frac{1}{r}\right)\frac{\partial u}{\partial r}$ (we'll come back to this later), and these expressions include isolated terms in lnr and 1/r
- So, while the deflection u(0) is bounded, the stresses aren't
- This generates a Type II singularity for this problem as we can easily verify in Ansys

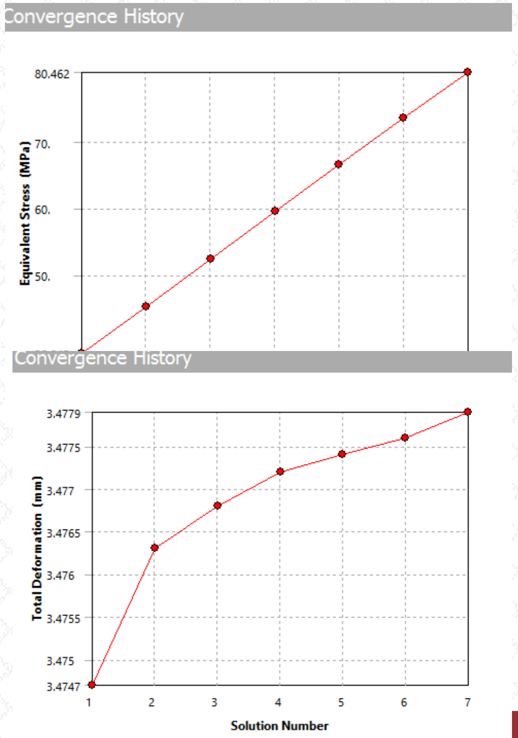


- stress diverges



- displacement converges



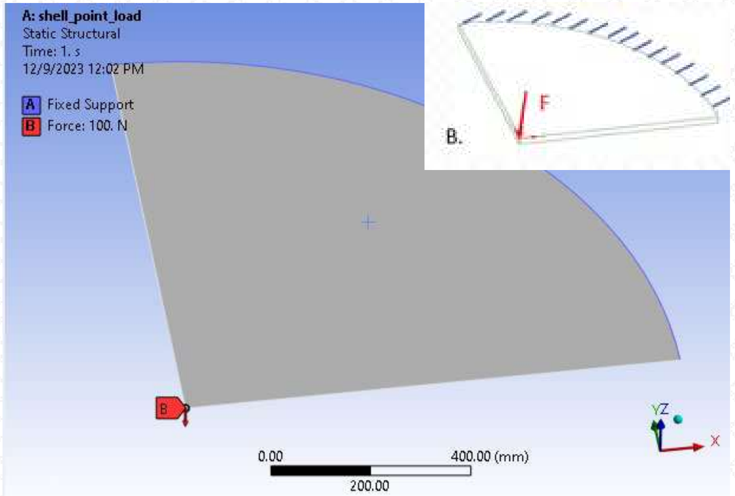


Geometric Singularities: Type II

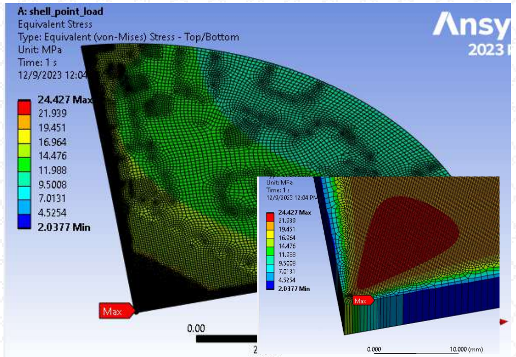
Convergence of Primary Solution Variable –Divergence of Derivatives

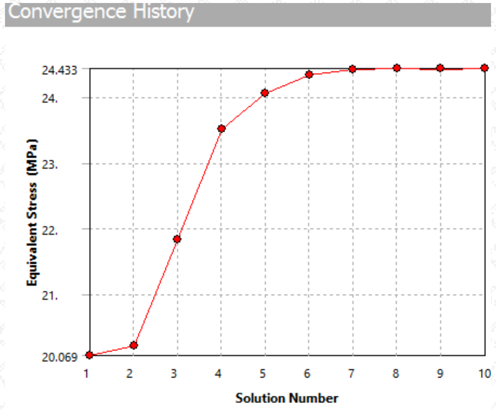
Reduced-Order Elements

- but look what happens when we simply delete (or suppress) the boundary conditions on the symmetry edges
- Both the stress and deflection converge(!)

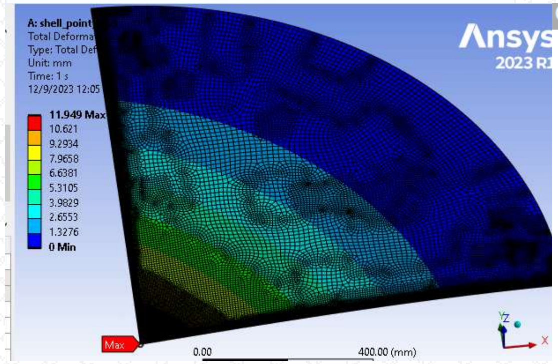


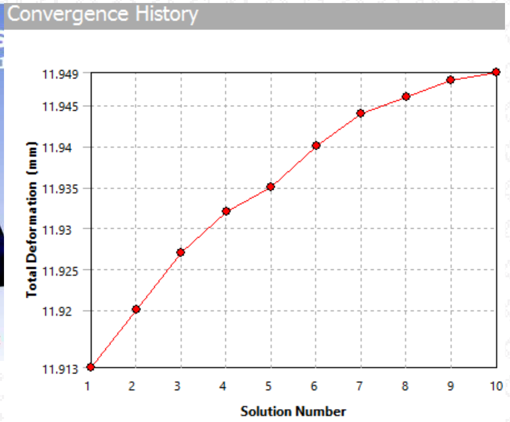
- stress converges





- displacement converges





Geometric Singularities: Type II

Convergence of Primary Solution Variable –Divergence of Derivatives

Reduced-Order Elements

- Since diving into plate theory is beyond the scope of this article, we'll try to pick out the relevant characteristics of the governing equations to help explain why this happens (while pointing to online references)
- First, note that Ansys allows users to split the shell behavior into "Membrane Only"(which we utilized for the model on slides 14 and 15) and "Membrane and Bending"
- We'll focus on the "Bending" portion. To keep things simple, we'll use a "thin plate" formulation (this is NOT Ansys' formulation, but the points we make about strain and curvature still hold for the most part. We'll draw from references here and here)

- Importantly, notice there's no z-component of stress or strain

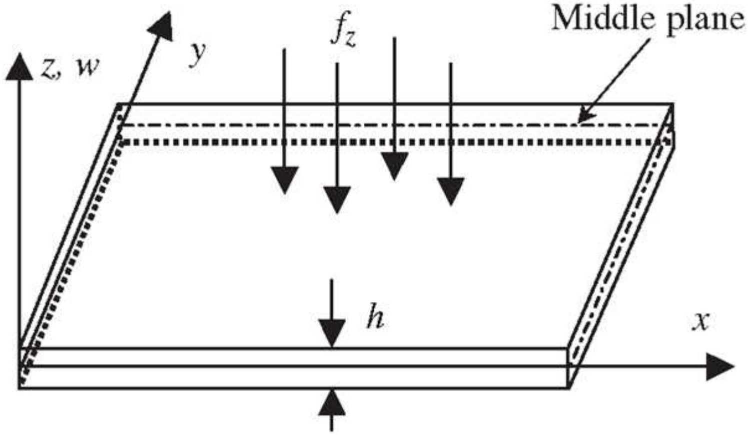
$$w = w(x, y) \qquad u = -z\frac{\partial w}{\partial x} \qquad v = -z\frac{\partial w}{\partial y}$$

$$\varepsilon_x = -z\frac{\partial^2 w}{\partial x^2} \qquad \varepsilon_y = -z\frac{\partial^2 w}{\partial y^2} \qquad \gamma_{xy} = -2z\frac{\partial^2 w}{\partial x \partial y}$$

$$\gamma_{yz} = \gamma_{zx} = 0$$

- Assume $\sigma_z = 0$. Therefore:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \end{Bmatrix} = -z\frac{E}{1-\nu^2}\begin{bmatrix} 1 & \nu \\ \nu & 1 \end{bmatrix}\begin{Bmatrix} \partial^2 w/\partial x^2 \\ \partial^2 w/\partial y^2 \end{Bmatrix} \qquad \tau_{xy} = -2zG\frac{\partial^2 w}{\partial x \partial y}$$



Geometric Singularities: Type II

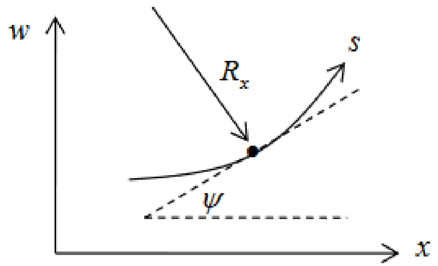
Convergence of Primary Solution Variable –Divergence of Derivatives

Reduced-Order Elements

- In pure bending, all stresses and strains are proportional to a planar gradient of the slopes $\theta_x = \frac{\partial w}{\partial x}$ and $\theta_y = \frac{\partial w}{\partial y}$
- But "gradient of slope" has another name: Curvature
- From this reference:

$$\kappa_x = \frac{\partial^2 w / \partial x^2}{\left[1 + (\partial w / \partial x)^2\right]^{3/2}} \qquad (6.2.2)$$

Also, the radius of curvature R_x, Fig. 6.2.2, is the reciprocal of the curvature, $R_x = 1/\kappa_x$.



- Users may think of plate elements as beams which bend in two directions
- This results in a (potentially) bi-directional curvature field. This bi-directionality is the culprit behind stress singularities in plates/shells

As with the beam, *when the slope is small*, one can take $\psi \approx \tan\psi = \partial w / \partial x$ and $d\psi / ds \approx \partial \psi / \partial x$ and Eqn. 6.2.2 reduces to (and similarly for the curvature in the y direction)

$$\kappa_x = \frac{1}{R_x} = \frac{\partial^2 w}{\partial x^2}, \qquad \kappa_y = \frac{1}{R_y} = \frac{\partial^2 w}{\partial y^2} \qquad (6.2.3)$$

This important assumption of small slope, $\partial w / \partial x, \partial w / \partial y << 1$, means that the theory to be developed will be valid when the deflections are small compared to the overall dimensions of the plate.

The curvatures 6.2.3 can be interpreted as in Fig. 6.2.3, as the unit increase in slope along the x and y directions.

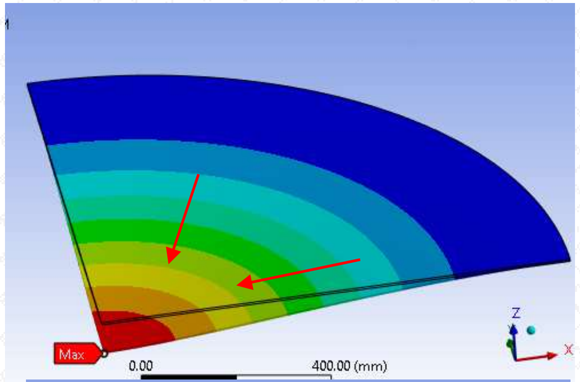
Geometric Singularities: Type II

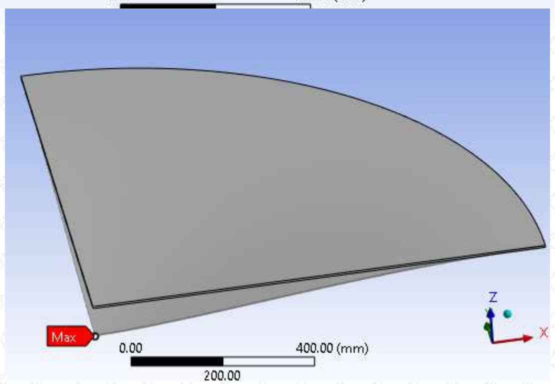
Convergence of Primary Solution Variable –Divergence of Derivatives

Reduced-Order Elements

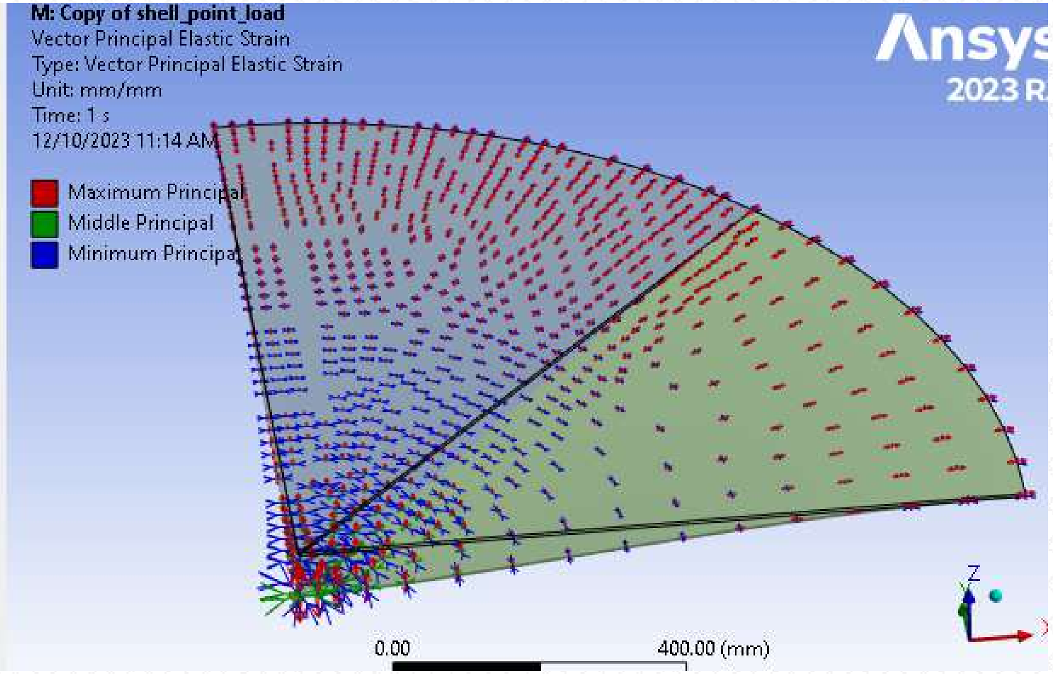
- We can understand what's going on in slides 21 and 22 by relating the strain field to their eigenvalues
- Since the curvature and strain are proportional to one another, the displaced shape can tell us what's happening

- The model with the singularity (case A) has a strain field whose distinct eigenvalue directions 'point' toward the singularity
- While case B's strain eigenvalues decompose to parallel orthogonal values as shown in next slide...





Case A Displacement



Case A Vector Principal Elastic Strain

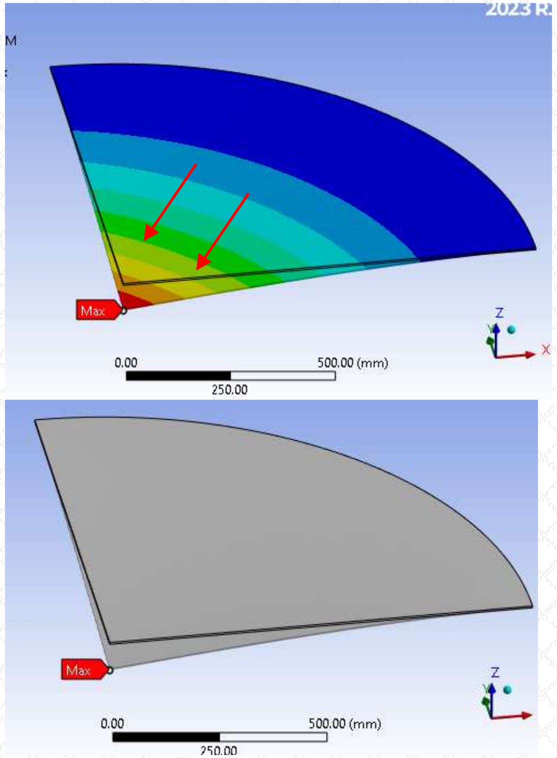
Geometric Singularities: Type II

Convergence of Primary Solution Variable –Divergence of Derivatives

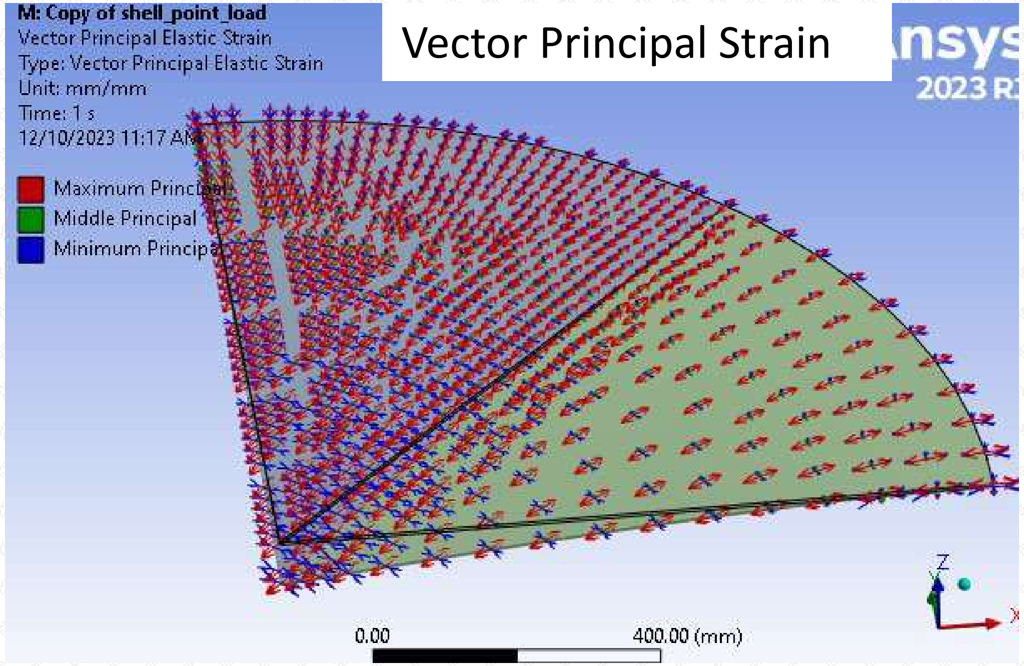
Reduced-Order Elements

- Note that around the external force, the principal elastic strain field is parallel
- A parallel eigenvector field implies zero Gauss Curvature (for shell elements)

- This gives us a new ‘quick’ check of whether or not we have a type II singularity
- If the Gauss Curvature around a candidate location is zero, we *cannot* have a type II singularity(!)
- The only way to check this directly in Ansys is to calculate the principal curvatures from the k11, k22, k12 (smisc items 12 thru 14)
- But we can indirectly check it the way we’re doing here (via the connection between principal curvature and principal strain



Case B Displacement



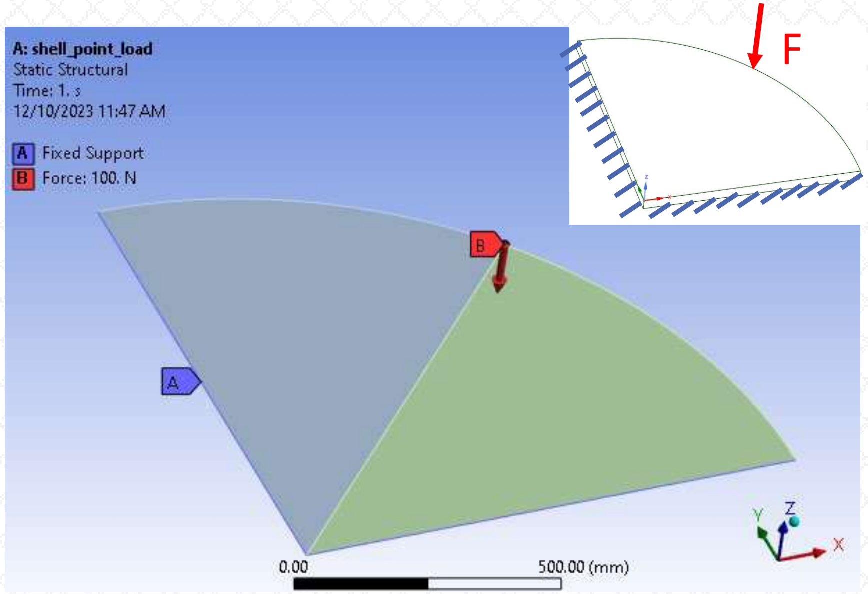
Case B Vector Principal Elastic Strain

Geometric Singularities: Type II

Convergence of Primary Solution Variable –Divergence of Derivatives

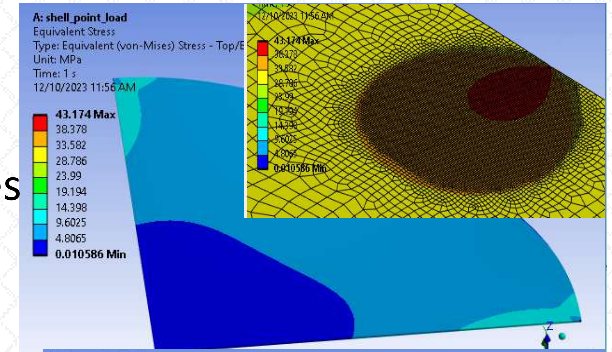
Reduced-Order Elements

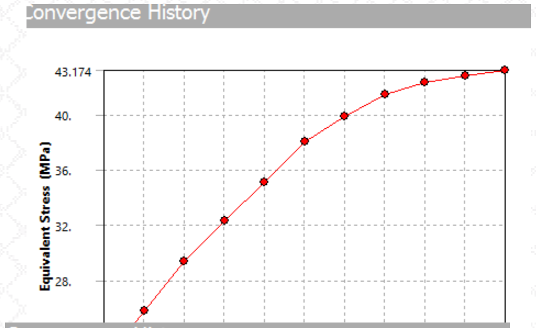
- Some users may still be unsure why the case B displaced shape results in zero Gauss Curvature around the applied force
- The reason is that on unconstrained, convex boundaries, the lowest internal energy state has no Gauss Curvature (i.e. only these configurations allow the material to bend in only a single direction)
- We can quickly check this by redefining the boundary loads and boundary conditions as below



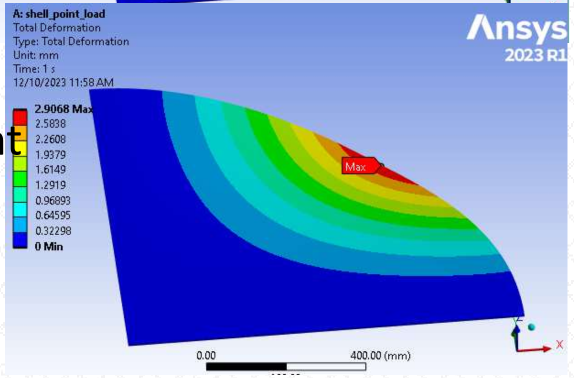
Case C

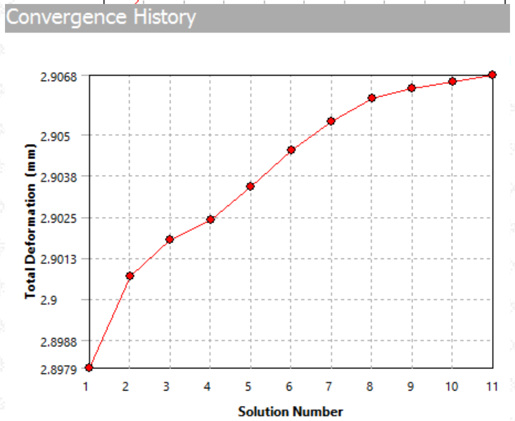
- stress converges





- displacement converges



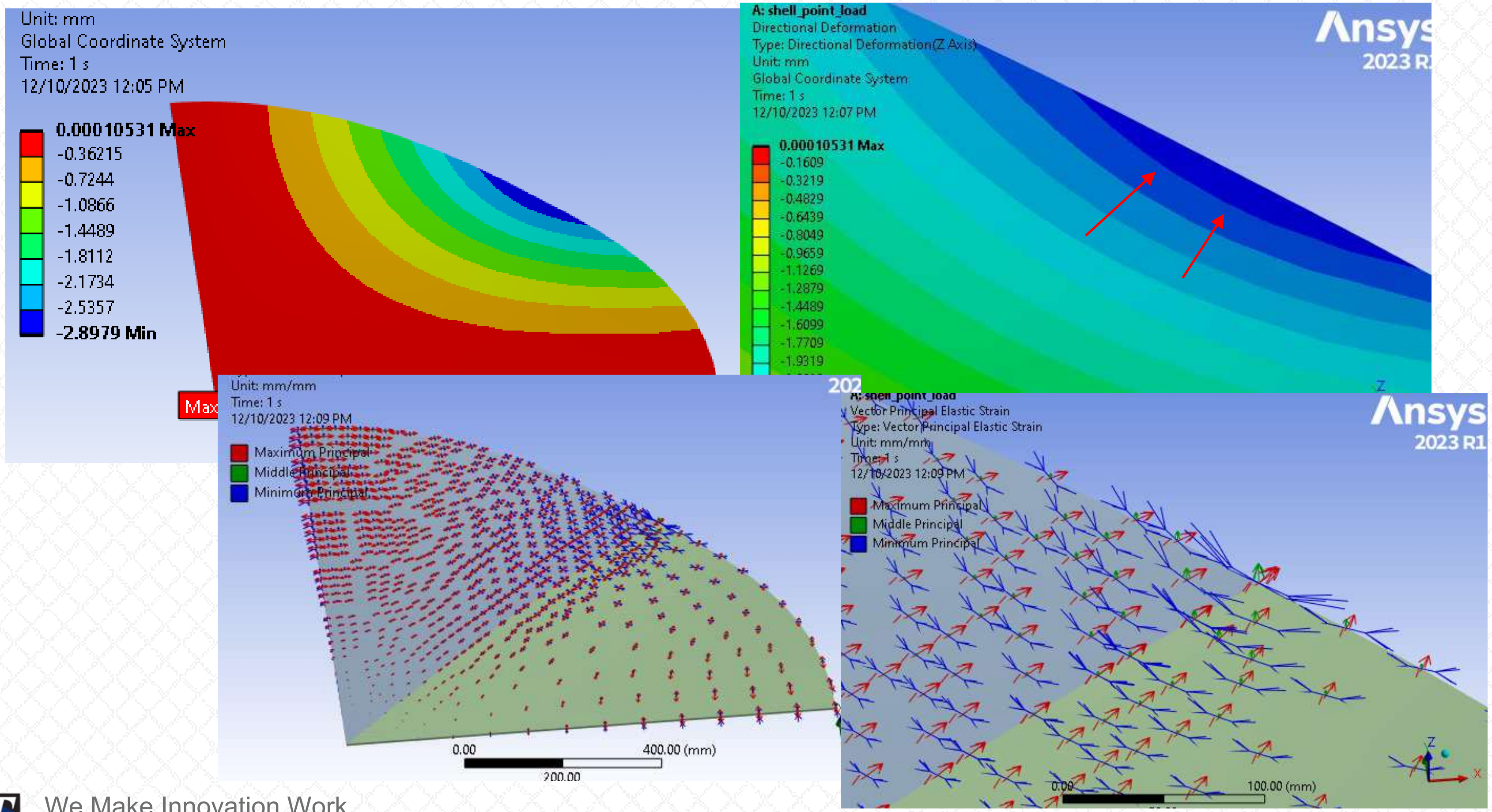


Geometric Singularities: Type II

Convergence of Primary Solution Variable –Divergence of Derivatives

Reduced-Order Elements

- A further check of the displacement and vector strain results confirm our conclusions



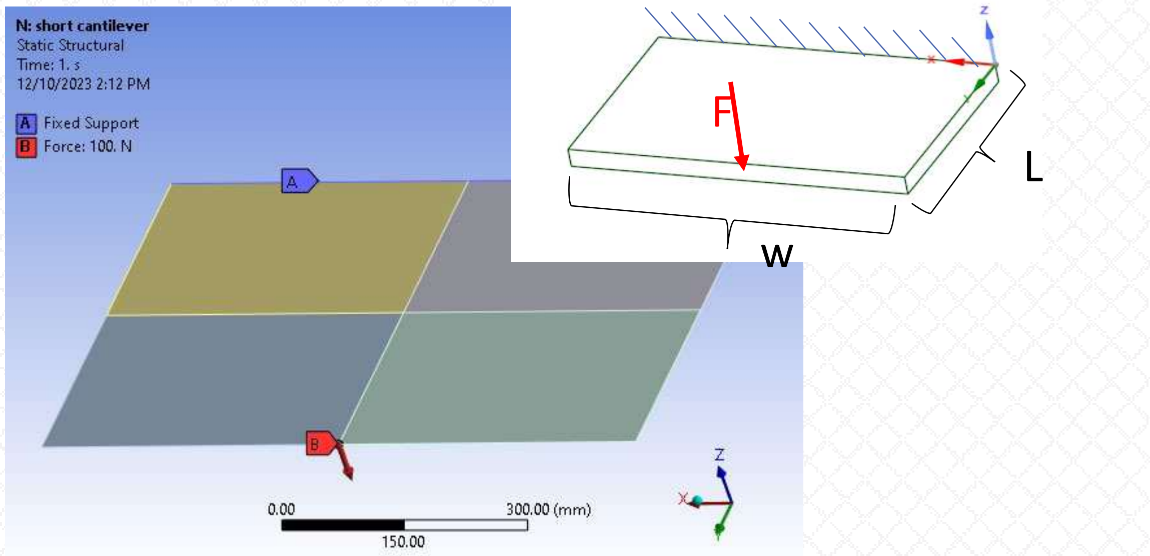
- Zoom in and increase contour levels to see that the displacement gradient is becoming parallel around the load application location

- Case C Vector Elastic Principal Strain

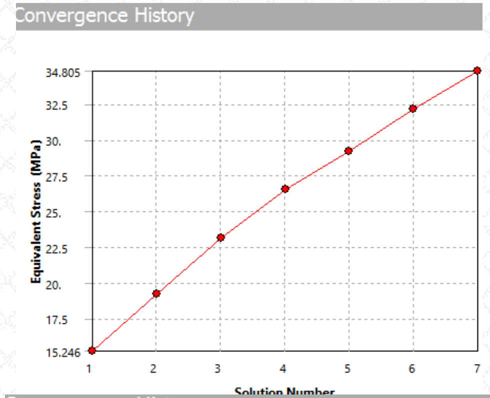
Geometric Singularities: Type II
Convergence of Primary Solution Variable –Divergence of Derivatives
Reduced-Order Elements

- When a point load is applied to ANY location other than a convex edge or corner, we can expect the curvature (and hence the bending strain) near the load to have two distinct components –and thus expect a type II singularity

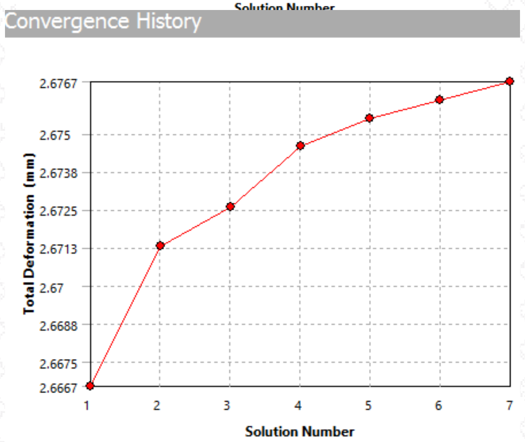
W/L = 2



- stress diverges



- displacement converges

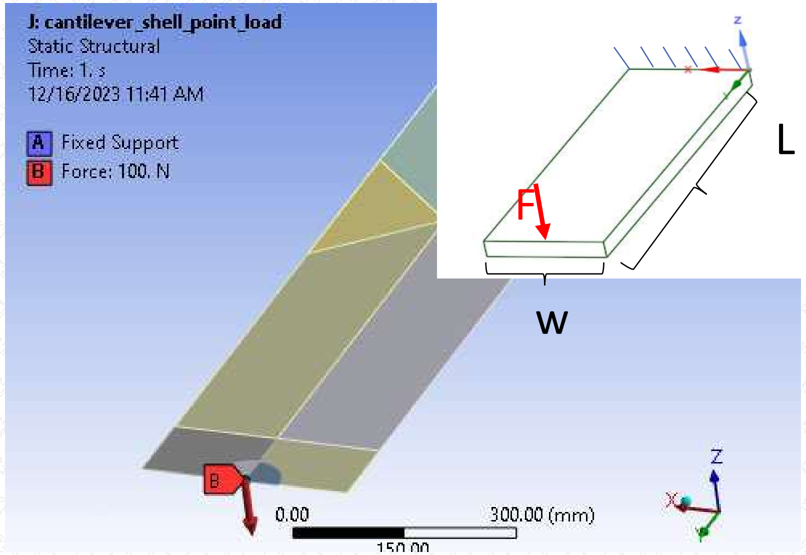


Geometric Singularities: Type II

Convergence of Primary Solution Variable –Divergence of Derivatives

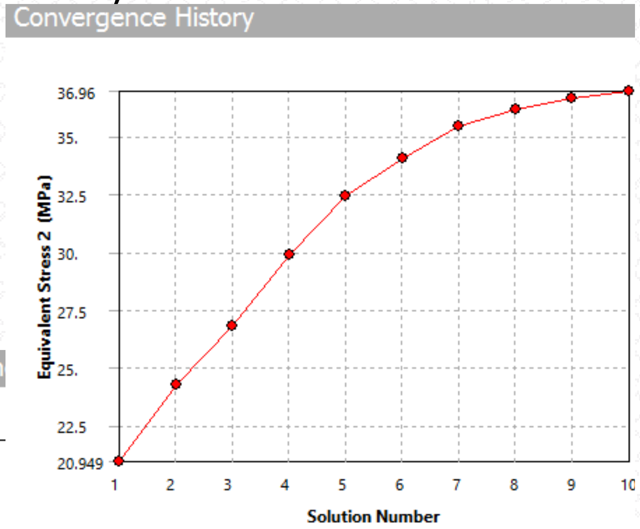
Reduced-Order Elements

- ...Unless it doesn't! Simply decreasing the width-to-length ratio of the cantilever leads to a situation which DOES converge
- This is why we chose the definition of 'geometric singularity' the way we did (slide 8)

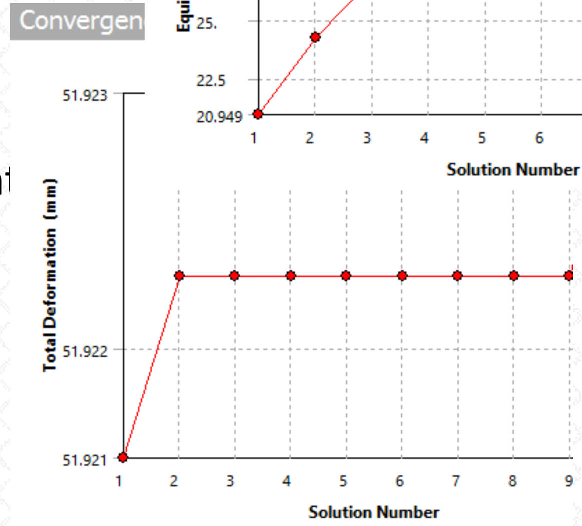


W/L = 1/3

- stress converges



- displacement converges

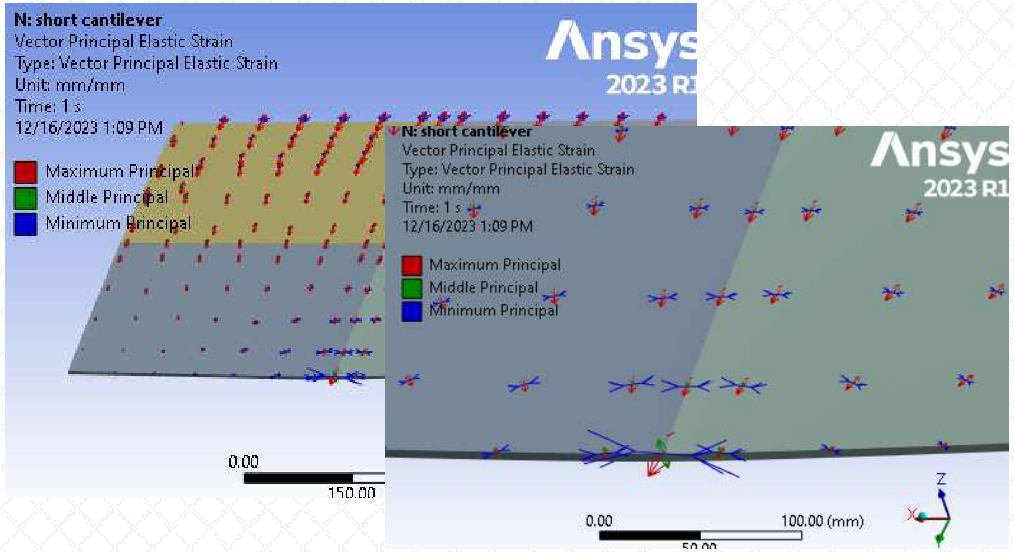


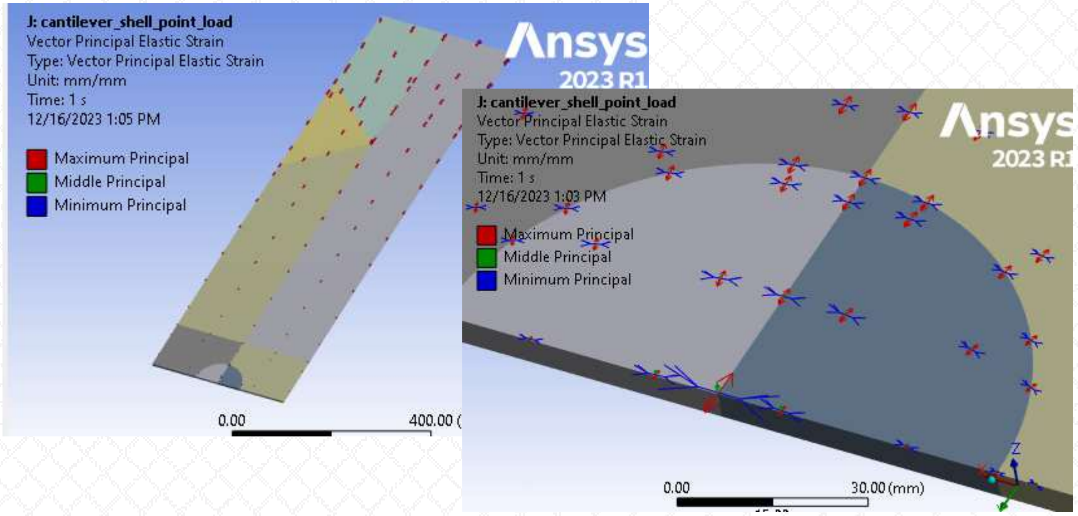
Geometric Singularities: Type II

Convergence of Primary Solution Variable –Divergence of Derivatives

Reduced-Order Elements

- Thus, the convergence of stress/strain in the vicinity of an applied point load at a non-convex boundary* of a rectangular plate of dimension w x L x h (width x length x thickness) depends on the ratio of overall width to length w/L
- The reason for this is that, although the strain/curvature field exhibits the same fundamental pattern as the case A vector field of slide 25, the component along the long side is so much greater than the short side that the plate behavior approaches that of a beam



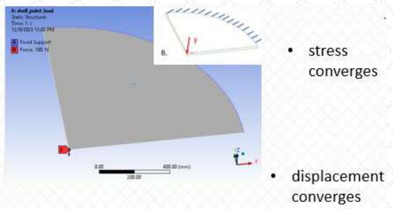


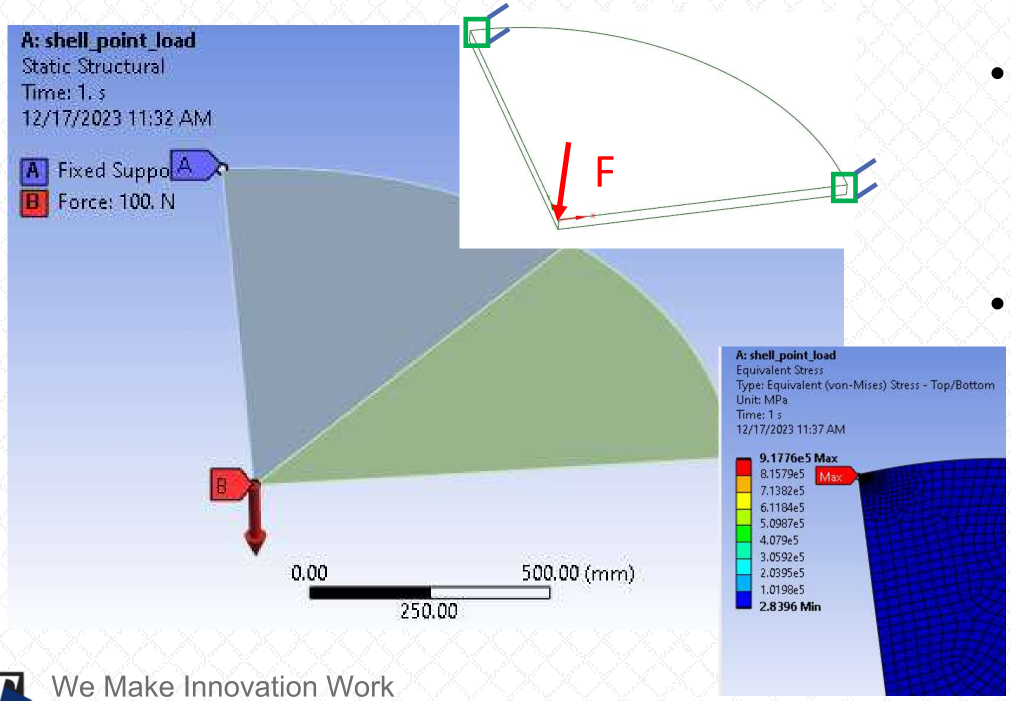
*and all points within the domain have the same convergence behavior as non-convex points on a free boundary

Geometric Singularities: Type I or Type II

Reduced-Order Elements

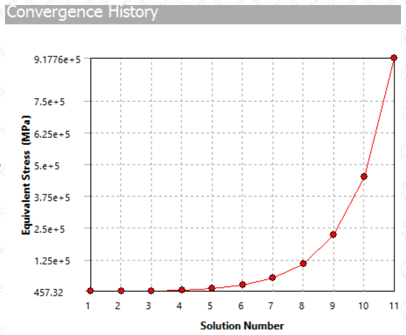
- The recurring theme for plates/shells, is that the principal strain/curvature field requires two linearly independent components (it must have nonzero Gaussian Curvature) in the vicinity of the candidate singularity in order to qualify as a singularity
- This is demonstrated dramatically by simply replacing the fully fixed (clamped) edge of the 1/4 symmetry point-load shell of slide 22 with two clamped vertices as below



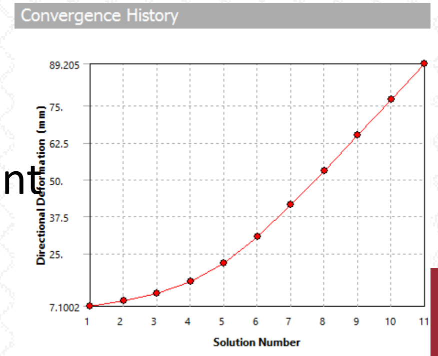


- This happens because the maximum stress is now associates with the clamped vertices
- Since these can support moments, the curvatures near these points (which *must* be multi-directional for this model) goes to infinity

- stress diverges



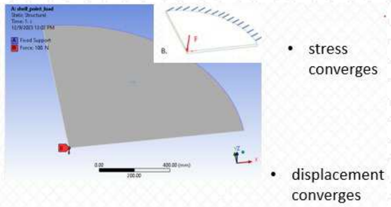
- displacement diverges

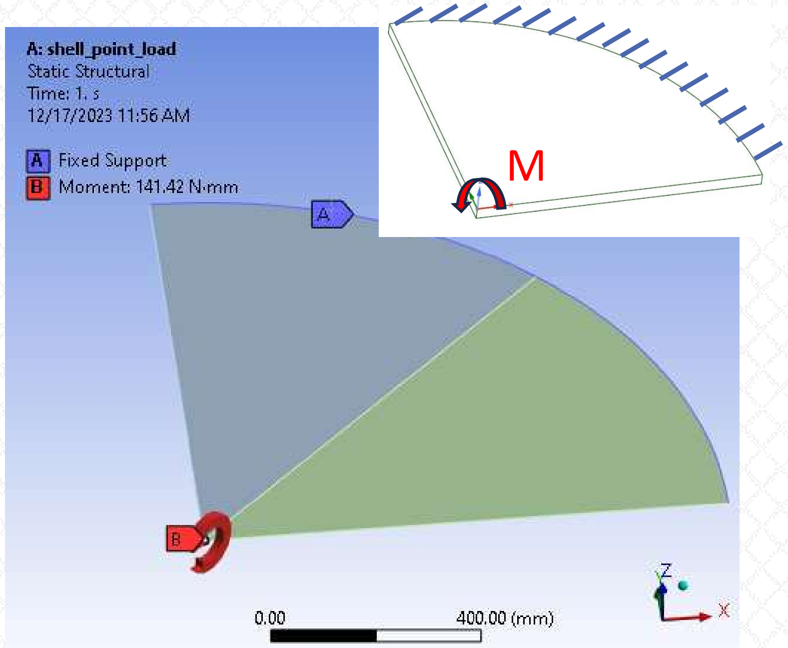


Geometric Singularities: Type I or Type II

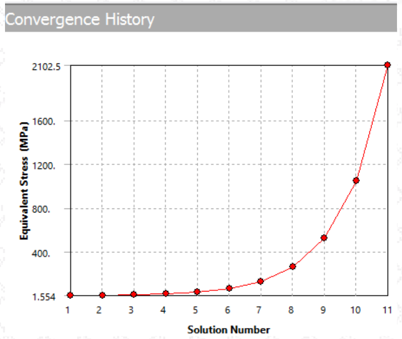
Reduced-Order Elements

- In fact, users should verify for themselves that ANY applied point moments in plates/shells induce either Type I or Type II singularities (depending on boundary condition and plate dimensions) as shown below by returning to the ¼-symmetry circular plate of slide 22 with clamped boundary and replacing the central point force with a moment

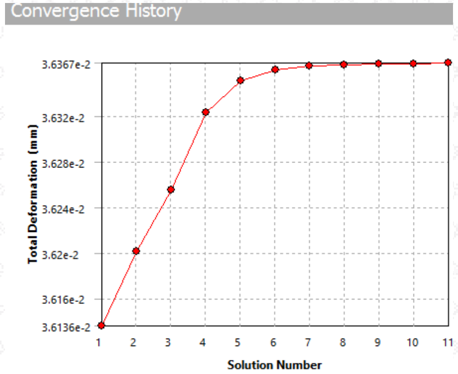




- stress diverges



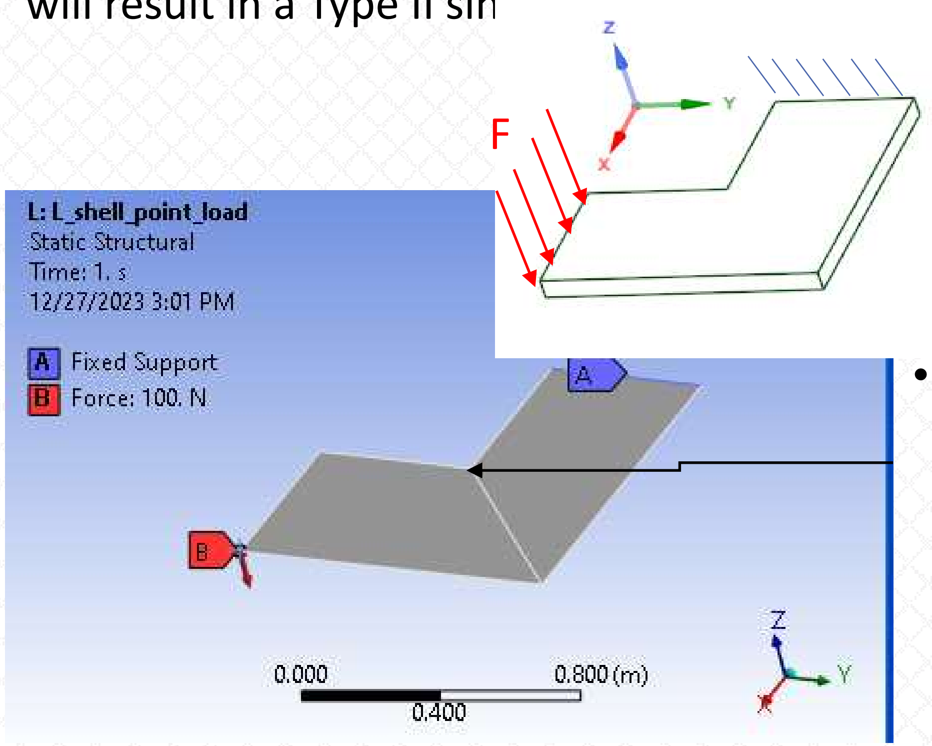
- displacement converges



Geometric Singularities: Type I or Type II

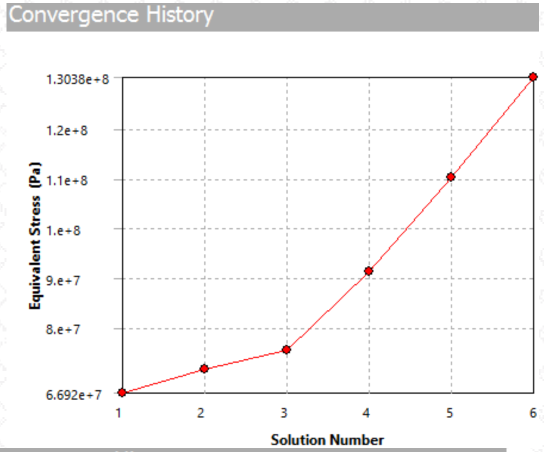
Reduced-Order Elements

- So far, we’ve looked at point forces and moments on plate/shell models. What about other types of singular points?
- Any situation in which a nonzero Gauss Curvature field is interrupted by a material point will result in a Type II sin

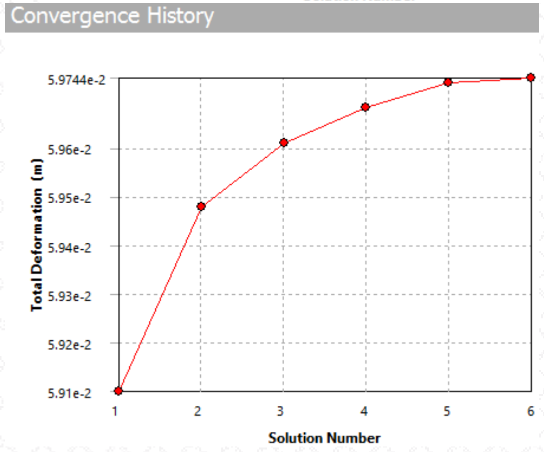


- Type II singularity in re-entrant corner

- stress diverges



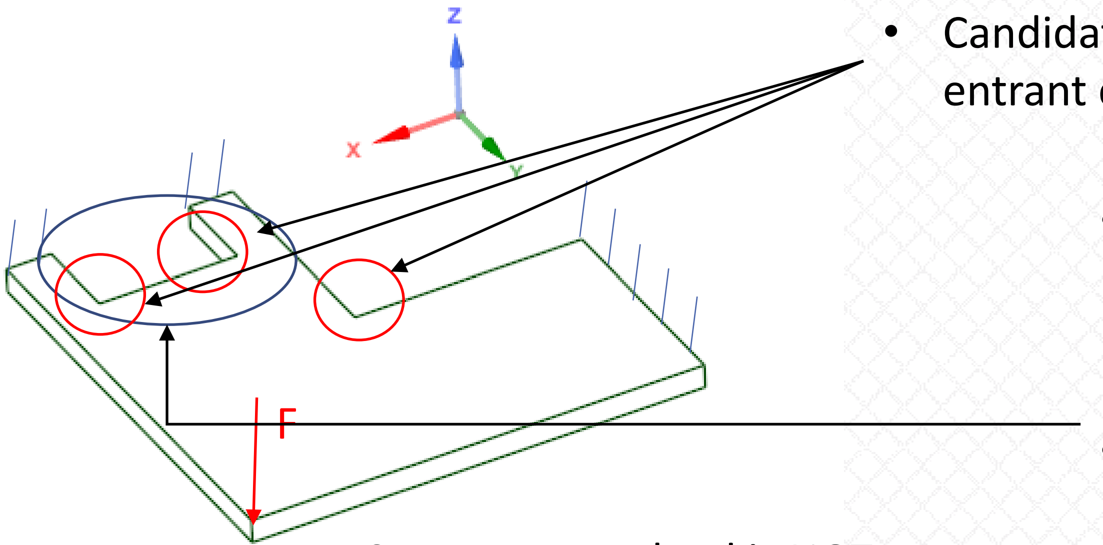
- displacement converges



Geometric Singularities: Type I or Type II

Reduced-Order Elements

- From what we've discussed so far, users should be able to predict the likely location of singular points in an arbitrary plate/shell model configuration.
- For example, in the model below, we should be able to make the following predictions

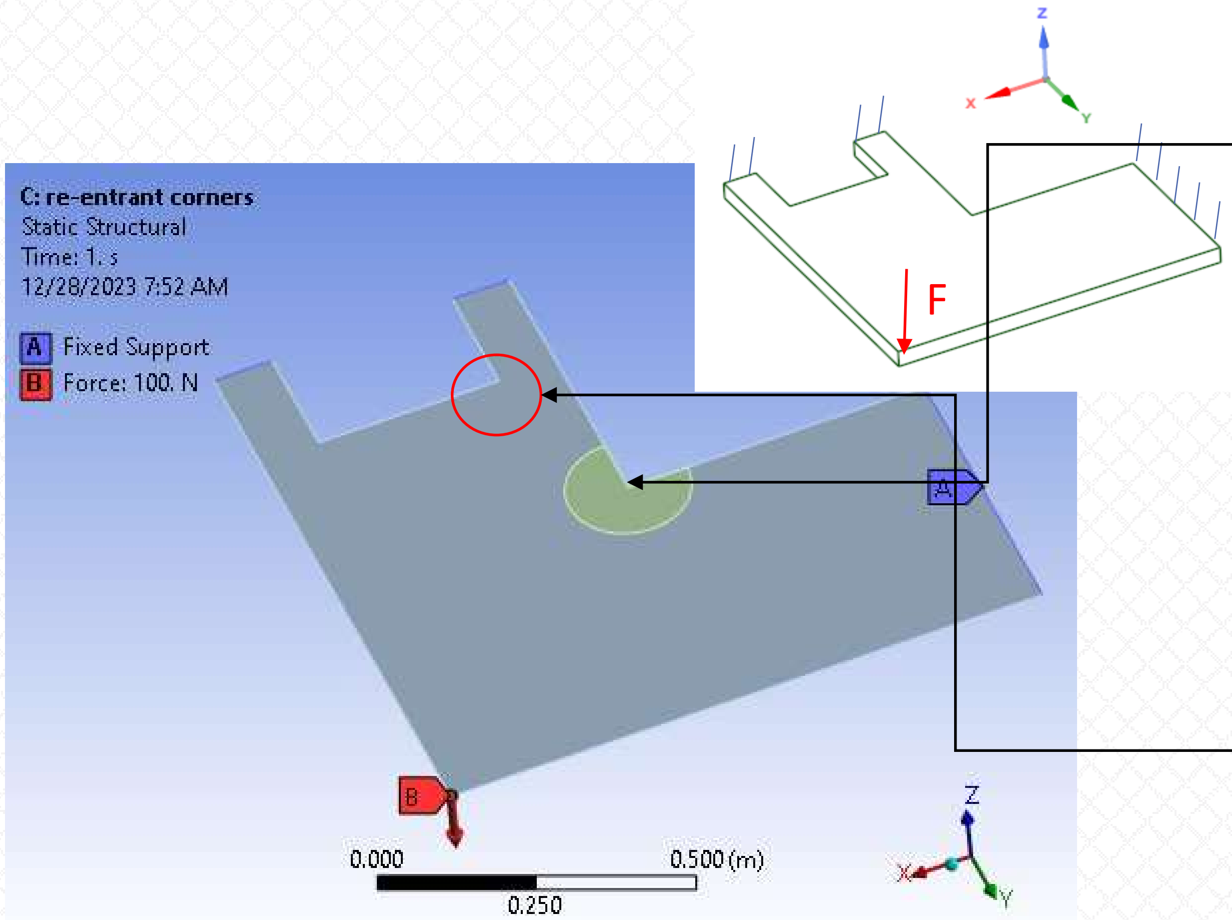


- Candidate singular points (three re-entrant corners)
- Nonzero Gauss curvature is most likely in this region (due to twisting)
- And so one of these two re-entrant corners is likely to be a singular point
- Convex corner load is NOT a singular point

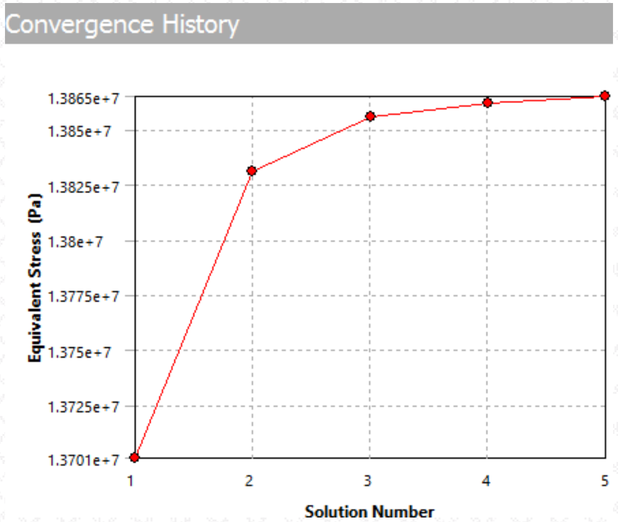
Geometric Singularities: Type I or Type II

Reduced-Order Elements

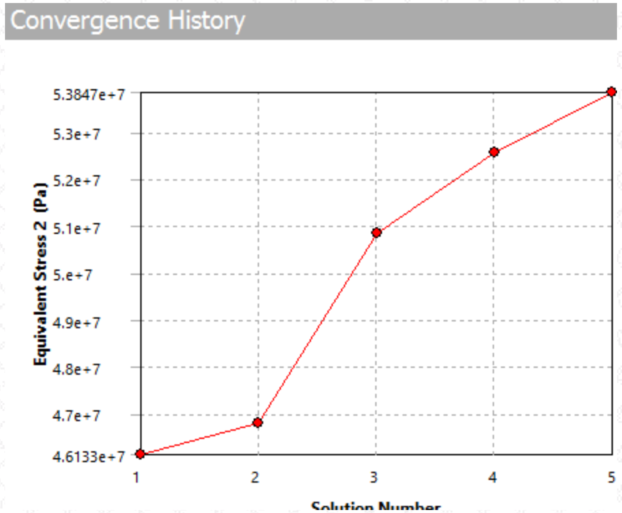
- Running a convergence study on this model confirms our hunch



- stress converges



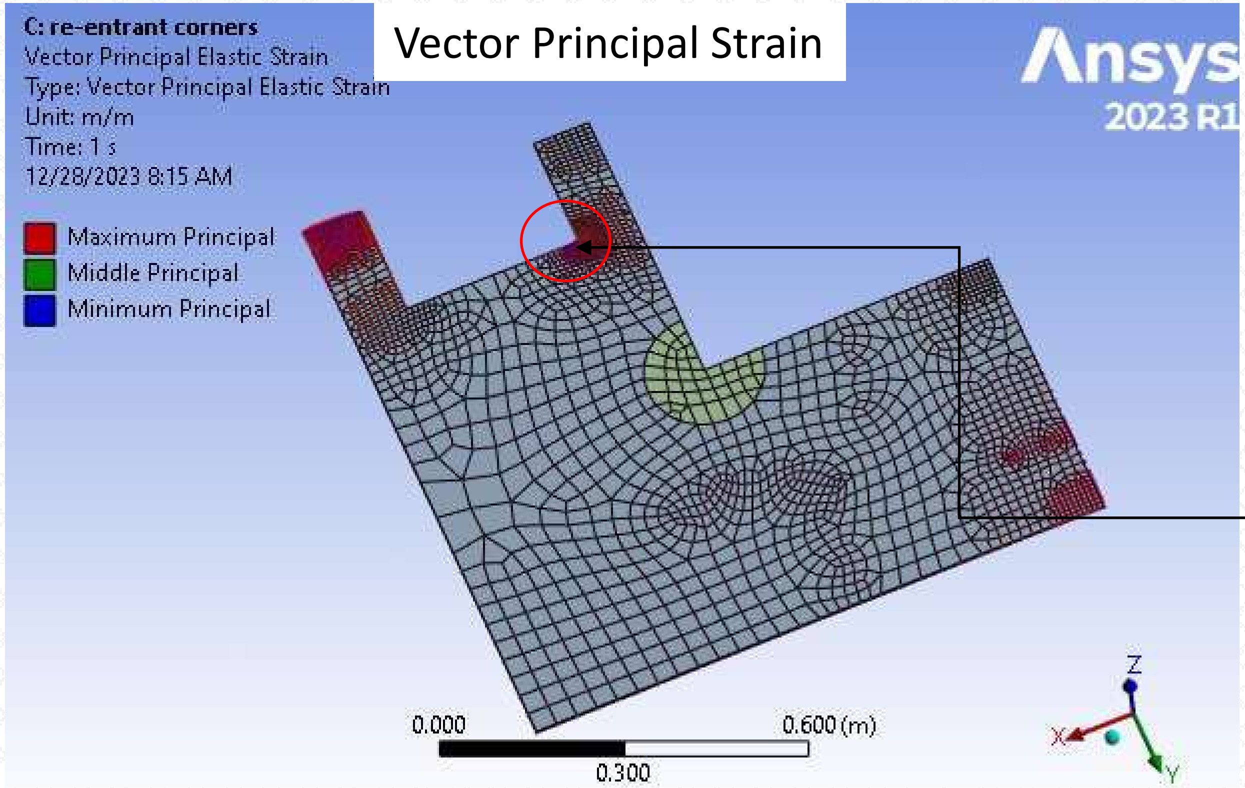
- stress diverges



Geometric Singularities: Type I or Type II

Reduced-Order Elements

- And once again, we can gain further insight from a plot of vector principal stress or strain as below

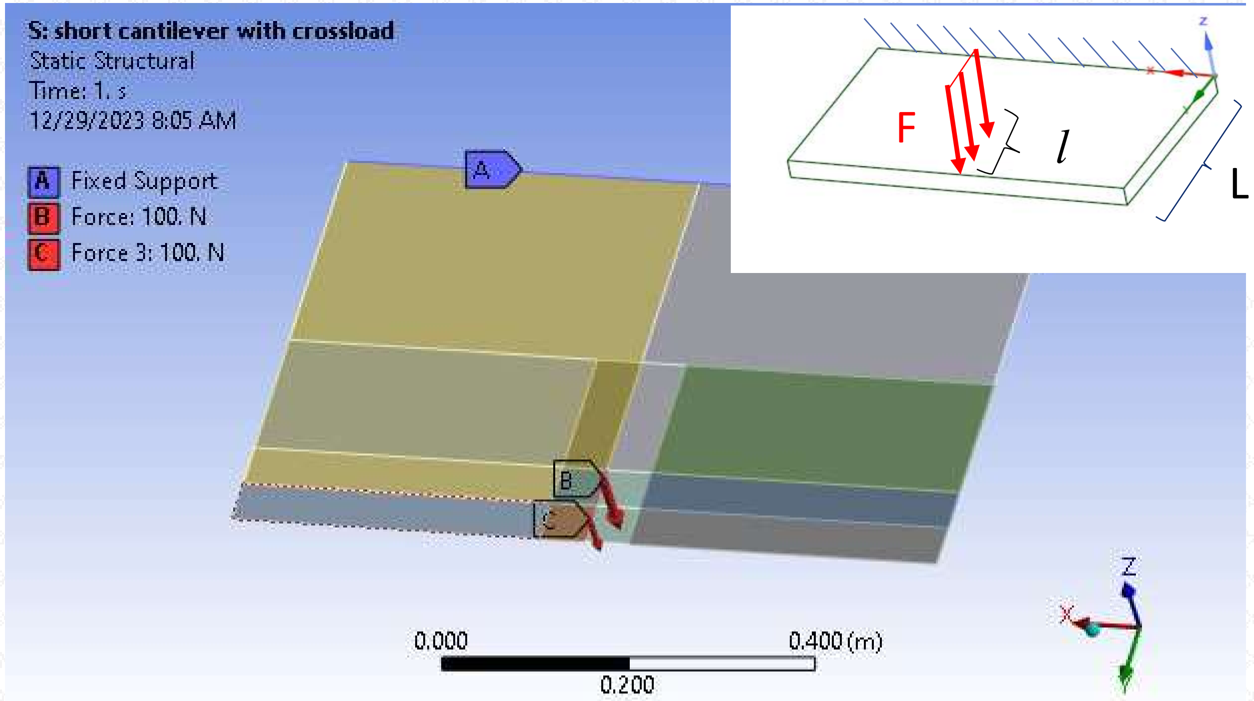


- Locations of high stress gradient stand out in relief in a vector principal strain plot
- Singularity!

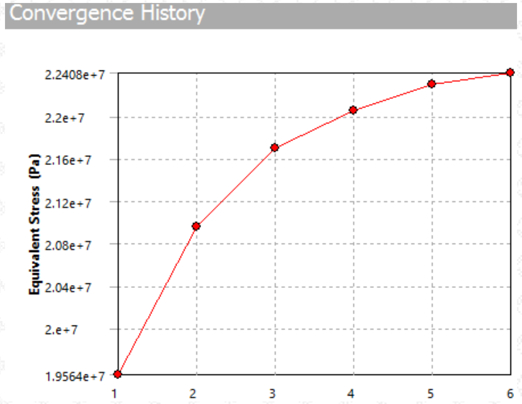
Geometric Singularities: Type I or Type II

Reduced-Order Elements

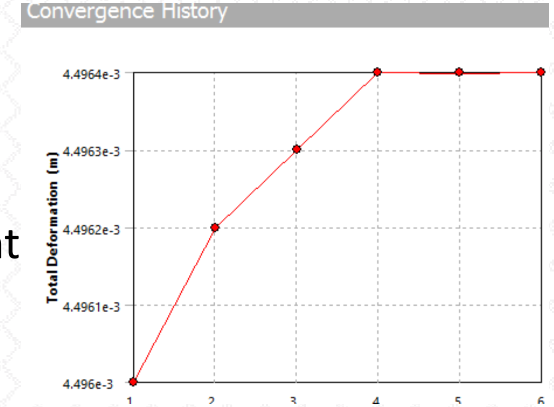
- What about line loads?
- In the model below, a line load (units N/m) is applied over a short line segment of length, l parallel to the short dimension, L.



- stress converges



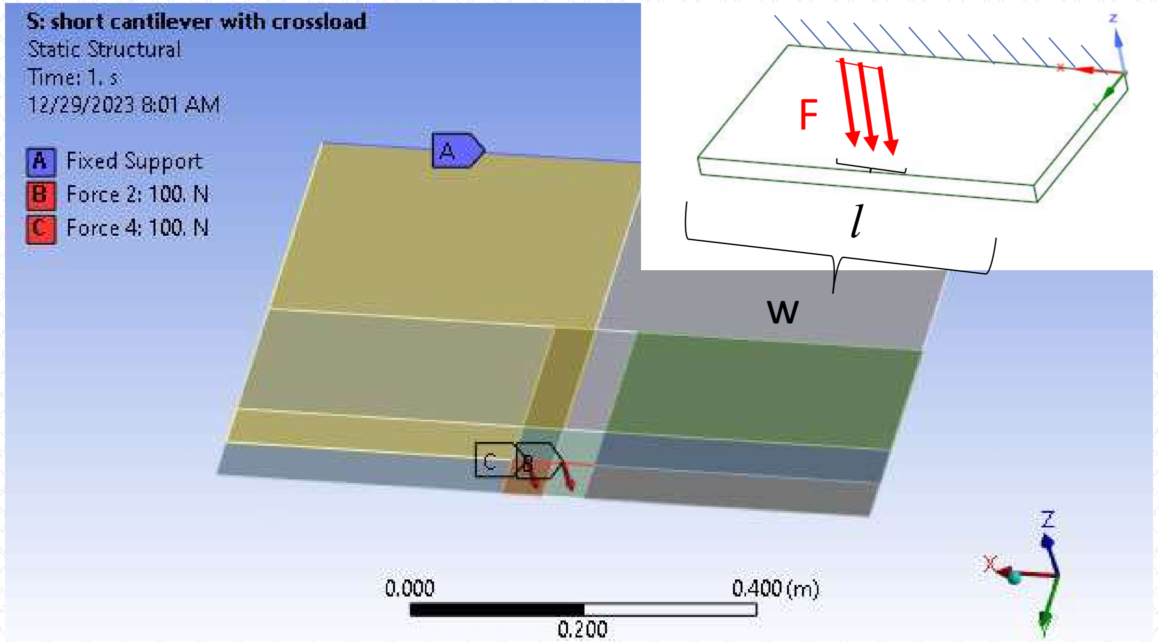
- displacement converges



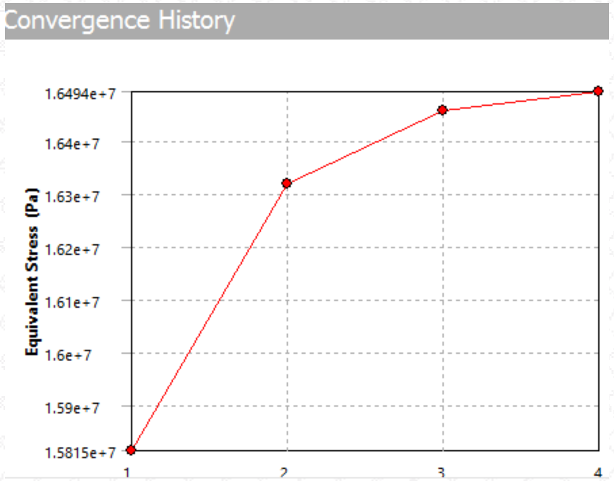
Geometric Singularities: Type I or Type II

Reduced-Order Elements

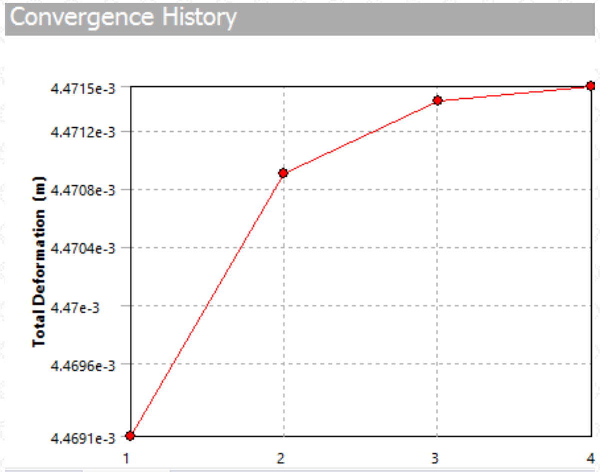
- Line loads of any finite length and configuration converge…



- stress converges



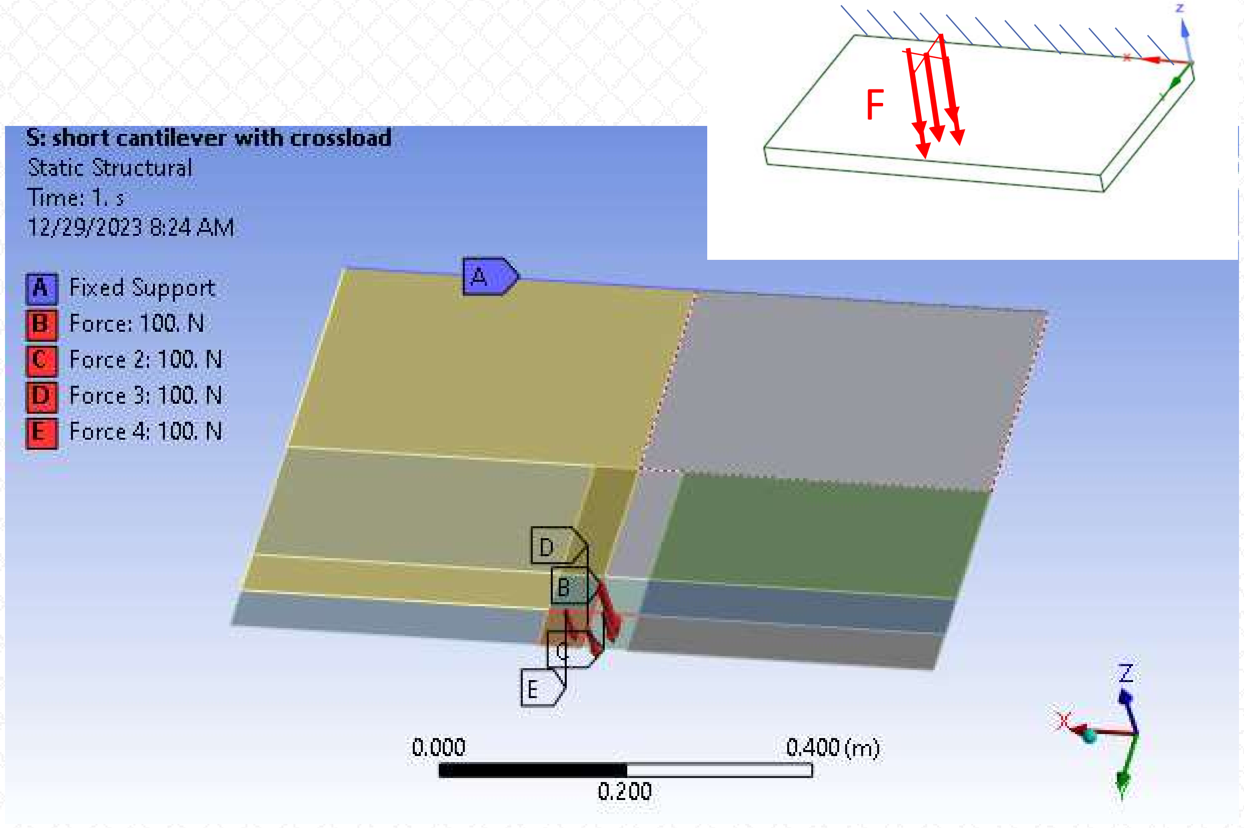
- displacement converges



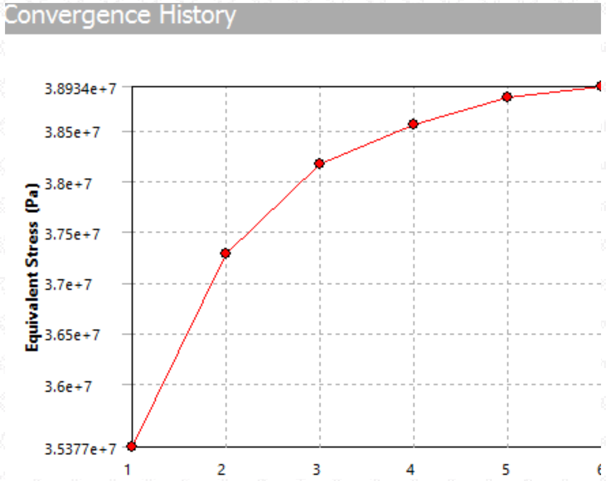
Geometric Singularities: Type I or Type II

Reduced-Order Elements

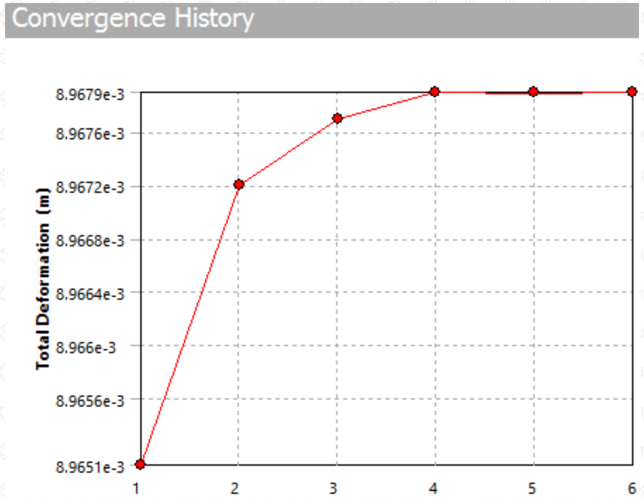
- …All result values continue to converge for any combination of line loads…



- stress converges



- displacement converges



Reduced-Order Elements

- We can now answer the questions raised on slide 8 for plate/shell elements:
  1. Are point loads always singularities?

     A: No. Not on convex edges or corners
  2. If the answer to 1. is 'no', when can we expect them?

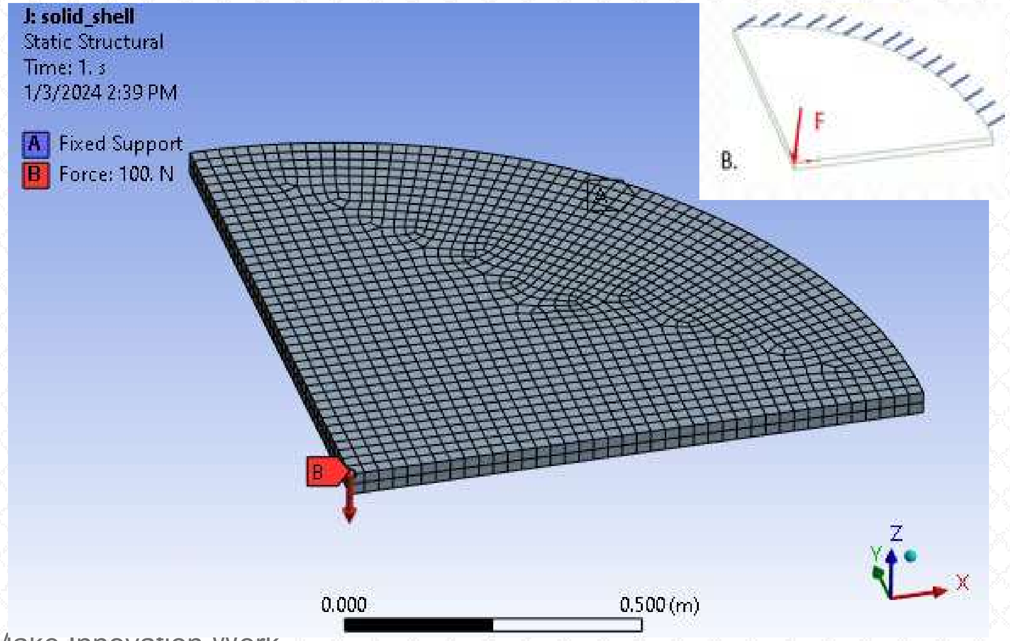
     A: On non-convex (concave or flat) edges and interior points
  1. Can other sorts of boundary conditions or geometry lead to this behavior?

     A: We can expect them in re-re-entrant corners which experience twisting, point loads at locations other than convex edges and corners, and at any locations with applied point moments (or at fixed points which support moment reactions)

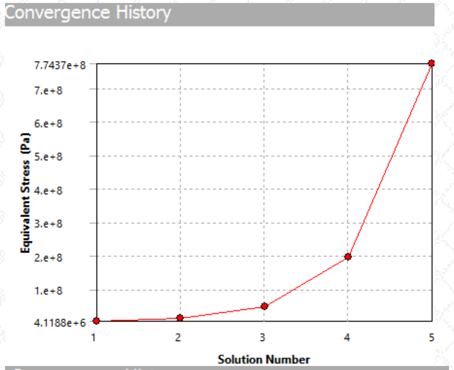
Geometric Singularities:

Full-Order Elements

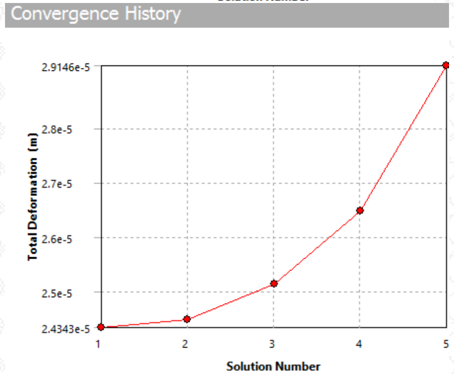
- Plate/Shell elements represent what we've been referring to as 'reduced-order' elements. These are governed by an underlying differential equation whose solution does not require or support the full elastic strain tensor. This is ultimately the reason for some of the behaviors described in the previous slides (we'll discuss this more later)
- The situation changes for elements which DO support the full elastic strain tensor. If we replace any of the models described so far with full-order elements (by extruding the surface in the shell thickness direction), the result is a Type I singularity! We do this below for the model of slide 22



- stress diverges



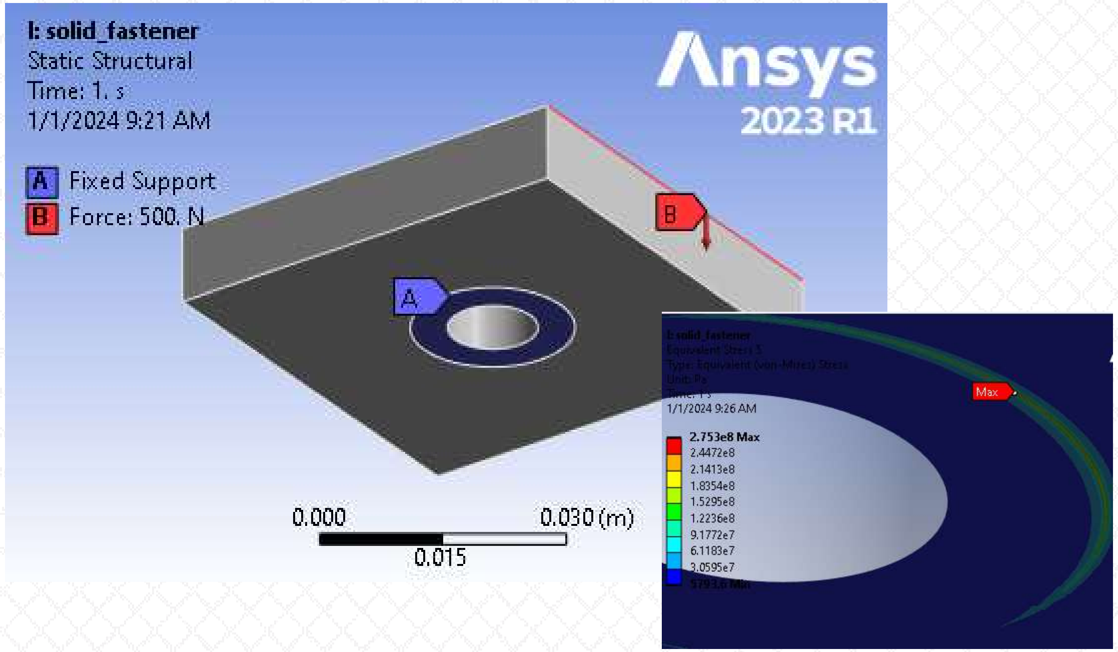
- displacement diverges



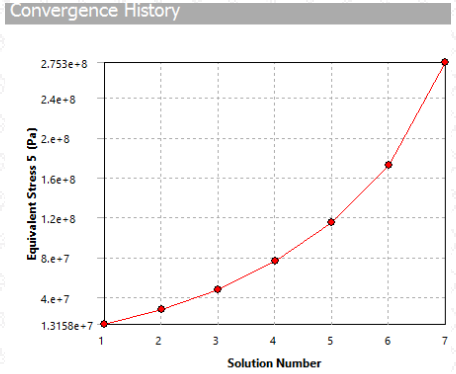
Geometric Singularities:

Full-Order Elements

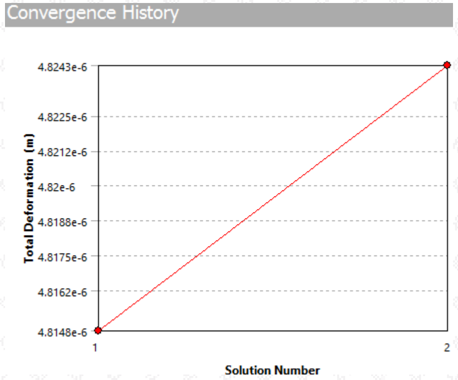
- For full-order elements, geometric singularities are not restricted to geometric points, but also extend to line/curve and surface boundary conditions.
- For these element types, Type I or Type II singularities occur at any and all locations where at least one strain displacement component is discontinuous (for elastic problems). Below is a common example of a Type II singularity arising from the prescribed displacement over an annular surface



- stress diverges



- displacement converges



Geometric Singularities: Type I

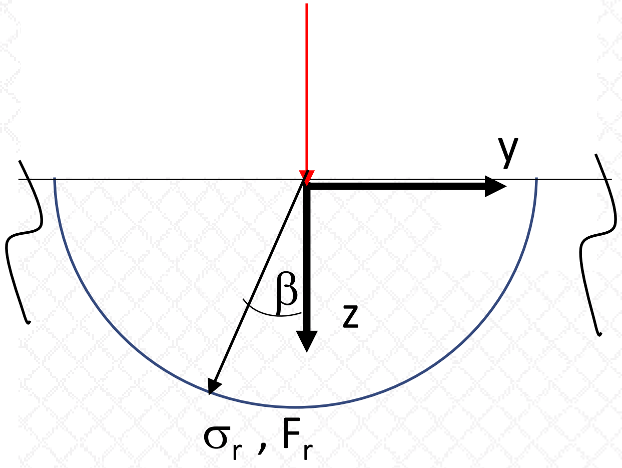
Full-Order Elements:
The Boussinesq Solution for a Point Load over an Elastic Half-Space

- Similar to the Poisson Equation, stress and strain near a point load on the surface of a full elastic continuum are governed by the elastic stress equilibrium equation and may be obtained by the construction shown at the lower right.

$$\nabla \boldsymbol{\sigma} + \boldsymbol{F} = 0 \qquad (14)$$

- We don't need to solve for the full stress tensor to see that, by equilibrium over a surface element at radius r from the point of load application:

$$\sigma_r = A\frac{cos\beta}{r^2} \qquad (15)$$



- In fact, this is sufficient to see that *all* point loads are singular points in a full elastic continuum (regardless of where they're applied)!
- The full stress solution is called Boussinesq's Solution for point loads over elastic half-spaces, and interested readers will find many references (see here, for example)

Geometric Singularities:

Full-Order Elements

- We can now answer the questions raised on slide 8 for full-order elements:
  1. Are point loads always singularities?
     A: Yes (all locations –both convex corners and otherwise)
  2. If the answer to 1. is 'no', when can we expect them?
     A: See above
  3. Can other sorts of boundary conditions or geometry lead to this behavior?
     A: Yes. As shown on slide 43, prescribed displacements at points, curves, or surfaces may result in displacement field discontinuities, which may in turn result in geometric discontinuities

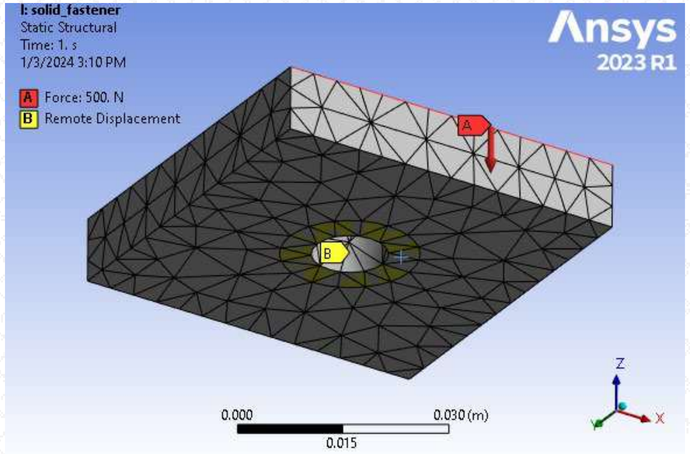
Geometric Singularities: Remedies

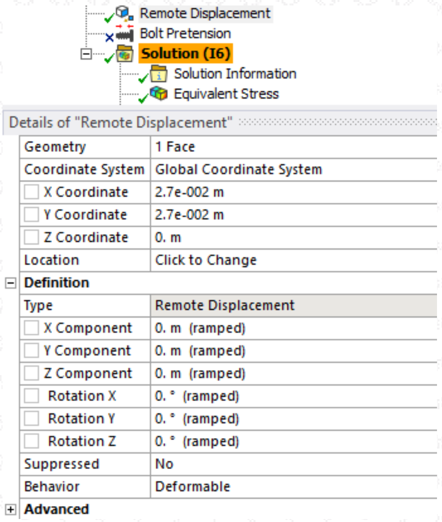
- As we explained on slide 2, we refer readers to the wealth of online discussions about what can be done about geometric singularities when they occur. However, we will add a few observations of our own
- All model problems involve some idealization. Geometric singularities are certainly a result of such idealization. In particular, geometric features with zero surface area or length cannot be found either in nature, or in man-made constructions. Thus, such idealizations serve merely to approximate small features as a modeling convenience. So, when encountered in a model (using the convergence tool, for example), Nick Stevens offers the following remedies (with which we strongly concur):
    - Remove the singularity by replacing the sharp feature with its ‘real-world’ geometric detail. Due to model size limitations, this may require submodeling (see here)
    - Ignore the singularity: Either flag the location(s) as unrealistic and demonstrate convergence (and peak stresses) at nearby location(s), OR cap the contour bar legends at some known value, such as material yield (while demonstrating that only a vanishingly small portion of material exceeds this value)
    - Use stress linearization to demonstrate that bending and membrane stresses converge and are acceptable (even when values near the singularity aren’t)
    - Use an elastic/perfectly plastic material (assuming material yield is the limiting stress value) to demonstrate precisely what portion of material yields

Geometric Singularities: Remedies

this link. And this one

- In addition to the remedies mentioned on the previous slide, we'll add another one
- The elastic Type II singularity of slide 43 is a very common one encountered in structural (elastic) models. Note that it results from the imposition of a displacement boundary condition whose geometric boundary produces a necessarily discontinuous displacement field (the source of this singularity) –regardless of the shape of the boundary
- Such singularities are also commonly associated with surface-to-surface contact
- In the spirit the first remedy of the previous slide, we can alleviate Type II singularities which arise from discontinuous displacement boundary conditions by replacing them with constraints of RBE3 type. In Ansys, this is most easily done by replacing the displacement boundary condition with a remote displacement with "Deformable" behavior as shown below



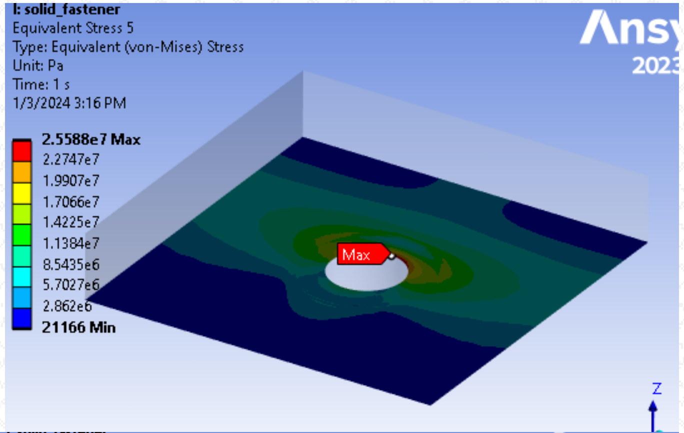


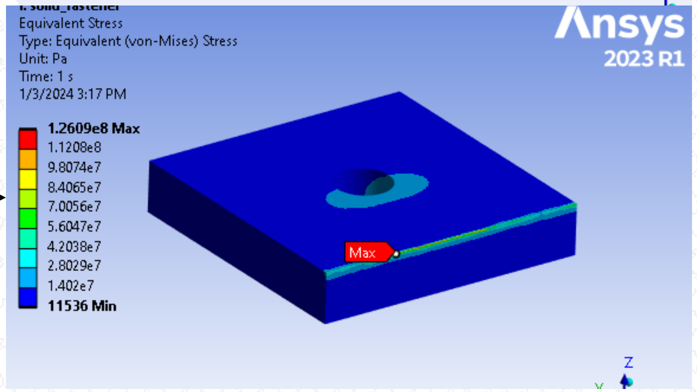
Geometry	1 Face
Coordinate System	Global Coordinate System
X Coordinate	2.7e-002 m
Y Coordinate	2.7e-002 m
Z Coordinate	0. m
Location	Click to Change
Definition	
Type	Remote Displacement
X Component	0. m (ramped)
Y Component	0. m (ramped)
Z Component	0. m (ramped)
Rotation X	0. ° (ramped)
Rotation Y	0. ° (ramped)
Rotation Z	0. ° (ramped)
Suppressed	No
Behavior	Deformable
Advanced	

Geometric Singularities: Remedies

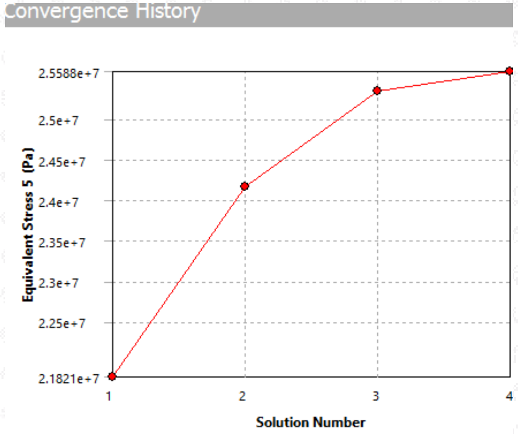
this link. And this one

- When we make this change, we find that the annular (fixed) boundary now converges
- but we STILL have a Type I singularity on the loaded edge (a convergence study on surfaces containing *that* edge would diverge). A remedy for that would involve distributing the load over a finite area (by imprinting a small rectangular strip on the surface geometry, for example)

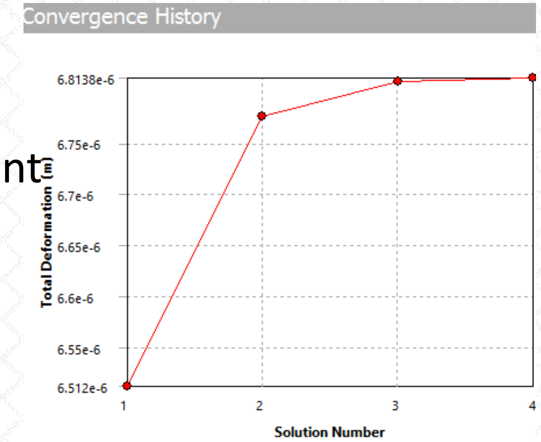




- stress converges



- displacement converges



Geometric Singularities:
Summary

- We define the term ‘geometric singularity’ on slide 8. Such singularities are characterized by a failure of mesh convergence (itself defined on slides 3 – 7) for some solution quantity of interest
- We introduce the two categories of singularity encountered in linear finite element problems: Type I (slide 11) and Type II (slide 19)
  - **Type I** singularities are characterized by a failure of the primary solution variable to converge (and along with it, all its derivatives. See equations 7, 8, and 11 of slides 12, 13, adn 17). It was found that plate/shell elements are largely immune to this type of singuarity, except in the case of applied point moments, or clamped (moment-carrying) points. We also found that the membrane models (plate/shells with no bending capacity) are highly susceptible to this type of singularity (this is because these elements have a full-order formulation in-plane). In general, full-order elements (which we define as those carrying the full stress tensor for their spatial dimensionality) are quite susceptible to these and they’re easier to identify than are Type II singularities
  - **Type II** singularities are characterized by convergence of the primary variable, but divergence of its derivatives (see equation 12 of slide 20). Plate/shell elements are susceptible to this type of singularity for point loads and re-entrant corners, and we summarized the conditions which lead to it (slides 25 thru 28). The general requirement for producing such singularities is a nonzero Gauss curvature in the displaced model shape around any fixed or loaded points.

Geometric Singularities: Summary (Continued)

- The results presented are further summarized on the following table
- Note that these are *potential* singular sites. Especially with full-order elements, whether one of these locations is a geomeric singularity (slide 8) will ultimately depend on other loading and boundary condition details

Element Type	Type I	Type II
Membrane	• point forces (slide 15)	• prescribed displacements (Dirichlet) on points • re-entrant corners
Full-order	• point forces • curve and edge loads (3D) (slide 48)	• prescribed displacements (Dirichlet) on points and curves (3D) • prescribed displacements (Dirichlet) on surfaces which subdivide the bounding surface(s) (slide 43) • re-entrant corners
Plate/shell	• point moment loads with clamped points	• point moment loads with all other bc's (slide 33) • point forces at non-convex and interior locations • re-entrant corners

Geometric Singularities:
Closing Thoughts

- Type I singularities due to point and line loads in full-order element types may be explained by the Poisson-type relationship of the load 'source' term to the underlying elliptic equation as shown on slide 44
- Surface loads (tractions, **t**) in full-order elements must obey the following relation. This places some restrictions on convergent loading scenarios (but not as many as some readers may assume. Recall the Boussinesq solution)

$$\boldsymbol{t} = \int_{\boldsymbol{s}} (\boldsymbol{\sigma} \cdot \boldsymbol{n}) \cdot d\boldsymbol{s}$$

- **s** is the loaded surface
- **n** is a vector-valued function representing the surface normal

- Note that plate/shell elements do NOT obey this relation! In fact, the underlying differential equation for those element types only provide the in-plane components of the stress tensor.
- This provides a high-level explanation for why some point loads are not singular points (the solution lacks stress components in the thickness direction). We hope our discussion on the previous slides has provided more insight.
- Re-entrant (unloaded) corners in full-order elements are potential sites of Type II singularity due to the fact that the displacement derivatives (strain) are not defined at such locations (but the displacements should still converge. Read more about this here).
- However, note that not all such sites will be singular locations as we've defined them (Type I or Type II singularities). This determination will further depend on the geometry and other boundary conditions.
- Want more? Interested readers may find this link, as well as this one useful.

Final Notes

- The main goal of this article is to help analysts get an intuitive understanding of what causes geometric singularities in finite element models and when they occur
- There is much confusion in the Engineering community over what constitutes a stress singularity
- Much of this confusion can be traced to the fact that model problems (not just finite element models) are idealizations of real Engineering assemblies, systems, or components. This line of thought suggests that the problem goes away if modeled more 'realistically' (by eliminating all zero-length and zero-area features, for example). It follows that stress singularities are fictions produced by simplifying assumptions
- While there is truth to this, the analyst's choices are restricted to those that will produces solutions in a reasonable timeframe with the resources she has. The level of model fidelity required to achieve singularity-free realism is often beyond reach
- Even when it's possible to replace all stress singularities with convergent stress fields (their more well-behaved cousins –stress concentrations), one often finds many of the same issues persist. In particular, a small region exceeds a stress allowable. Whether this small region is a point or convergent mesh region, the analyst is faced with the same choices listed on slide 46.
- Beyond modeling practicalities, what we've been calling geometric singularities tell us something very important about a model or design
- Regardless of how we treat them, these are locations which require special attention as they are likely sites of material failure if failure occurs.