Lessons and Tricks Learned in Using ANSYS

v1.3 – Part 1

Eric Sun

Nov 20, 2020

Outline

1. **Debugging**
2. **Worksheet, Mismatched Meshing, and Integration Points**
3. **Shared Topology, Node Merging, and Contacts**
4. **Stress Concentration**
5. **Shear Locking, Hourglassing, and Solid Shell Elements**
6. **Pressure Vessels**

Lessons Learned – Debugging

1. The support engineers are not as motivated as you. Don’t rely on them to solve your problem.

2. Use a simple model with simple boundary conditions for testing or learning.

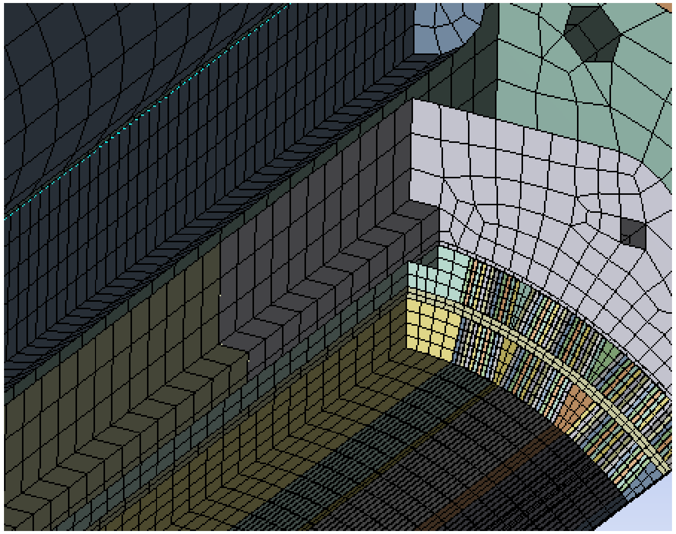
3. Don’t ignore the most obvious, even the unit.

Don’t Rely on Support Engineers

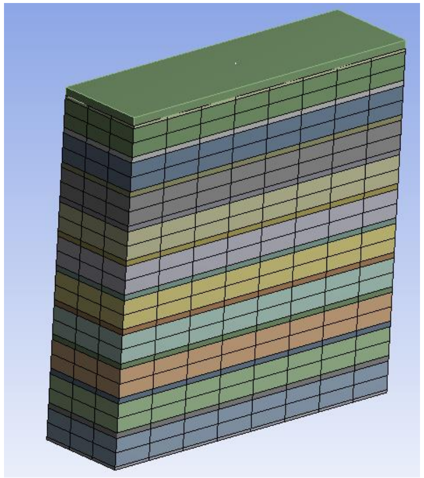
- Support engineers are great to solve the problems if the problems are well known to them, but not to you.
- Support engineers are usually knowledgeable in the operation and special features of the software.
- Support engineers are not great if they never see the problems before.
- Support engineers are slow to respond your questions if it needs their effort to verify your findings even in the simple model.
- Support engineers are not motivated to solve the problem for you.
- Only you are motivated. No one else.

Start from a Simple Model

- Simple model is great because it can be solved in a few seconds.
- If a simple model can't be solved easily, don't hold the illusion that a similar case in the production model will be solved.
- A support engineer once told me every case was different in nonlinear analysis.
  – I disagree. If it doesn't work with a simple model, there is little hope with a complicated model.
- Use a simple model to test the boundary conditions and loadings.



Solve time = 4 day



Solve time = 30 sec

Don’t Ignore the Obvious

- Even the most unlikely could be wrong for ANSYS in a special situation, like units.
  - The production model produced crazy results for no reason.
  - Created a simple model to test it. Used the meter unit because the material was defined with meters.
  - Still got crazy results. And it shouldn’t because I got good results using 2019 R1.
  - The unit was the only thing that was different.
  - A unit bug of gaskets was confirmed in versions before 2020R2.
  - The unit bug had many symptoms.
  - The bug was fixed in 2020R2.

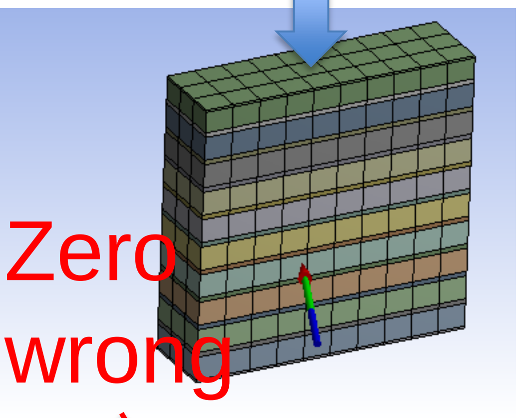
- The root cause of the bug was related to the tolerance threshold set in the gasket algorithm. The original threshold was 1.0E-8 for both mm and meter. If the unit is meter, the threshold should be 1.0E-11, not 1.0E-8.

Unit Bug - Gaskets

Results are identical

2019 R1
Unit: meter

Load = 0 N (0 s)
-> 11638 N (1 s)
-> 24258 N (2 s)



Zero wrong

Tabular Data

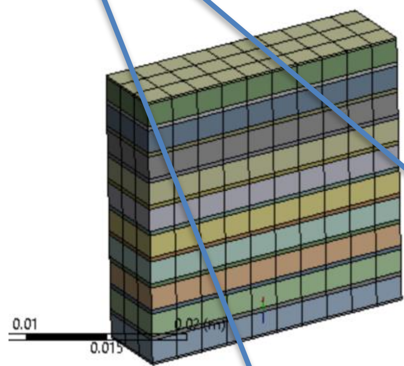
	Time [s]	Force Reaction (X) [N]	Force Reaction (Y) [N]
1	1.e-002	0.56351	2.4286e-017
2	2.e-002	0.3833	1.648e-017
3	3.5e-002	0.43296	2.6021e-018
4	5.75e-002	0.51407	8.5001e-017
5	9.125e-002	0.63063	-2.7756e-017
6	0.14188	0.78914	-2.949e-017
7	0.21781	1.0315	4.1633e-017
8	0.33172	1.2452	3.8164e-017
9	0.50258	1.1859	-5.2042e-017
10	0.75258	-0.24229	-2.0817e-017
11	1.	-4.0377	4.8572e-017
12	1.05	-4.0377	1.3878e-017
13	1.1	-4.0377	-1.0408e-016
14	1.175	-4.0377	1.249e-016
15	1.2875	-4.0377	-9.7145e-017
16	1.4563	-4.0377	9.0206e-017
17	1.7063	-4.0377	-1.5266e-016
18	1.8531	-4.0377	2.7756e-016
19	2.	-4.0377	-6.245e-017

2019 R1
unit: mm

Tabular Data

	Time [s]	Force Reaction (X) [N]	Force Reaction (Y) [N]
1	1.e-002	0.58756	37.53
2	2.e-002	0.41389	73.87
3	3.5e-002	0.48051	126.21
4	5.75e-002	0.58167	183.86
5	9.125e-002	0.72793	265.38
6	0.14188	0.93154	381.84
7	0.21781	1.2918	682.03
8	0.33172	2.0541	2047.3
9	0.50258	3.1614	4451.9
10	0.75258	3.8424	7661.9
11	1.	3.4103	11638
12	1.05	3.8187	12269
13	1.1	4.2271	12900
14	1.175	4.8396	13846
15	1.2875	5.7585	15266
16	1.4563	7.1368	17396
17	1.7063	9.1787	20551
18	1.8531	10.378	22404
19	2.	11.578	24258

2020R2
unit: meter

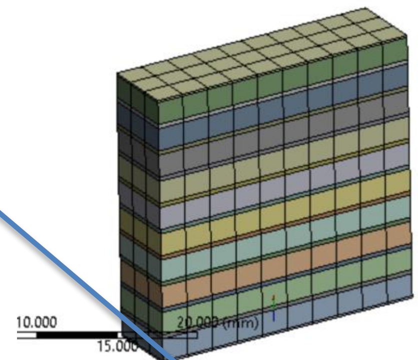


Tabular Data

	Time [s]	Force Reaction (X) [N]	Force Reaction (Y) [N
1	1.e-002	0.58756	37.53
2	2.e-002	0.41389	73.87
3	3.5e-002	0.48051	126.21
4	5.75e-002	0.58167	183.86
5	9.125e-002	0.72793	265.38
6	0.14188	0.93154	381.84
7	0.21781	1.2918	682.03
8	0.33172	2.0541	2047.3
9	0.50258	3.1614	4451.9
10	0.75258	3.8424	7661.9
11	1.	3.4103	11638
12	1.05	3.8187	12269
13	1.1	4.2271	12900
14	1.175	4.8396	13846
15	1.2875	5.7585	15266
16	1.4563	7.1368	17396
17	1.7063	9.1787	20551
18	1.8531	10.378	22404
19	2.	11.578	24258

5 Messages No Selection Metric (m, kg, N, s,

2020R2
unit: mm



Tabular Data

	Time [s]	Force Reaction (X) [N]	Force Reaction (
1	1.e-002	0.58756	37.53
2	2.e-002	0.41389	73.87
3	3.5e-002	0.48051	126.21
4	5.75e-002	0.58167	183.86
5	9.125e-002	0.72793	265.38
6	0.14188	0.93154	381.84
7	0.21781	1.2918	682.03
8	0.33172	2.0541	2047.3
9	0.50258	3.1614	4451.9
10	0.75258	3.8424	7661.9
11	1.	3.4103	11638
12	1.05	3.8187	12269
13	1.1	4.2271	12900
14	1.175	4.8396	13846
15	1.2875	5.7585	15266
16	1.4563	7.1368	17396
17	1.7063	9.1787	20551
18	1.8531	10.378	22404
19	2.	11.578	24258

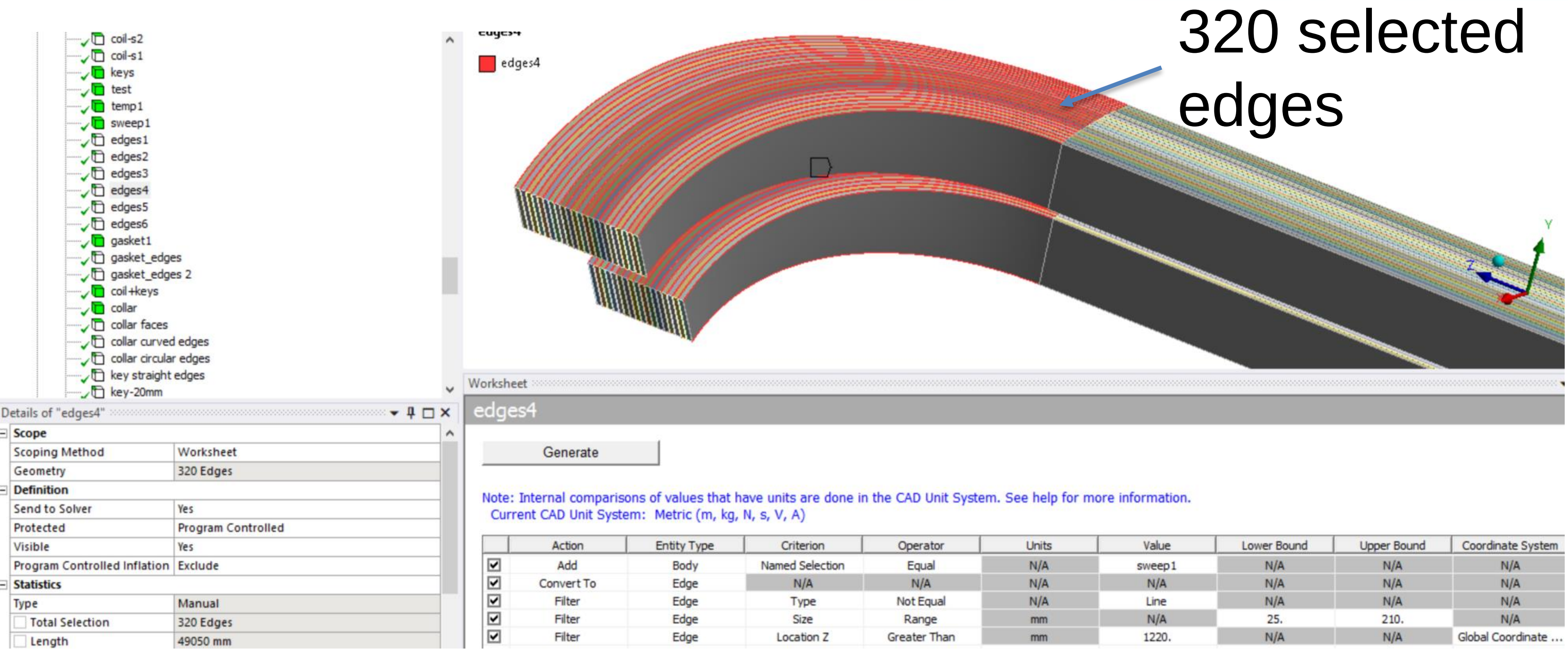
3 Messages No Selection Metric (mm, kg, N, s,

Reaction force at the bottom

Tricks Learned – Use Worksheet to Select Edges for Meshing

4. Worksheet is a beautiful tool to select hundreds of edges with a bunch of rules. Applying mesh size control to these selected edges could produce a consistent mesh pattern among many bodies.

Example – Use Worksheet to Select Edges



Rules to select edges

	Action	Entity Type	Criterion	Operator	Units	Value	Lower Bound	Upper Bound	Coordinate System
☑	Add	Body	Named Selection	Equal	N/A	sweep1	N/A	N/A	N/A
☑	Convert To	Edge	N/A	N/A	N/A	N/A	N/A	N/A	N/A
☑	Filter	Edge	Type	Not Equal	N/A	Line	N/A	N/A	N/A
☑	Filter	Edge	Size	Range	mm	N/A	25.	210.	N/A
☑	Filter	Edge	Location Z	Greater Than	mm	1220.	N/A	N/A	Global Coordinate ...

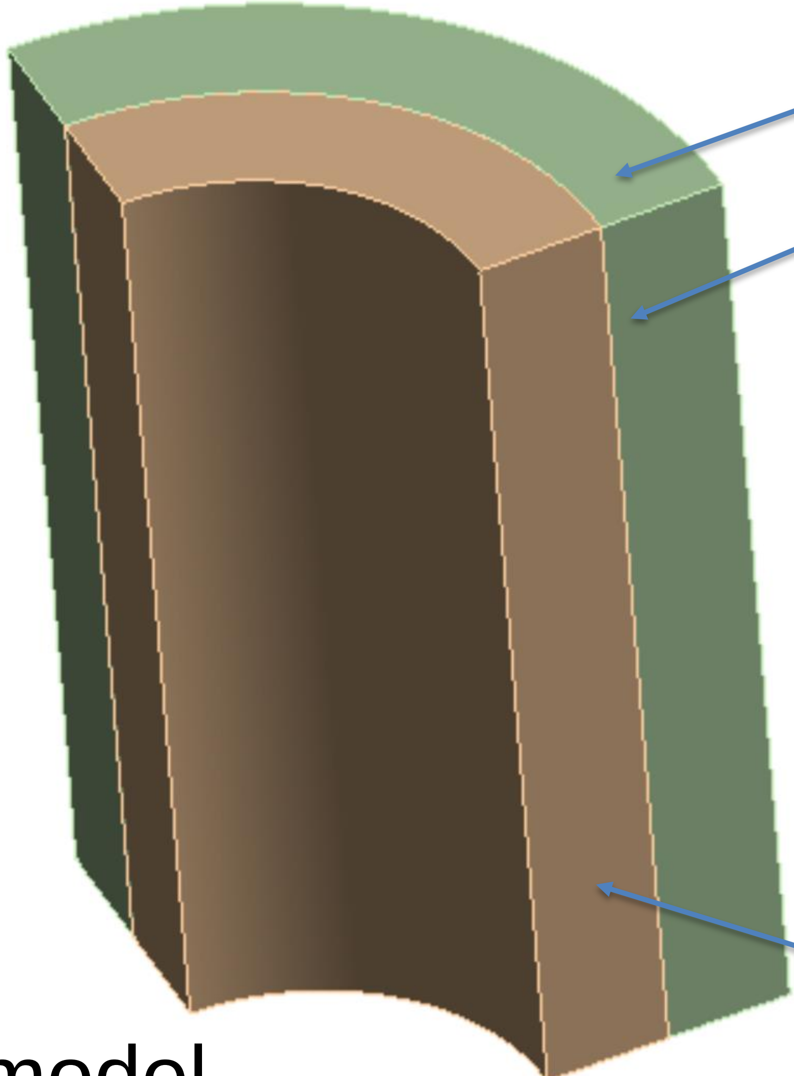
Lessons Learned – Mismatched Meshing

5. Matching meshes on the curved contact interfaces are of paramount importance for good stress or strain patterns if linear elements are used.

6. If quadratic elements are used, mismatched meshes of interfaces produce reasonable results.

7. If quadratic elements are used, matched meshes of interfaces produce the best results.

Problem Definition



Aluminum

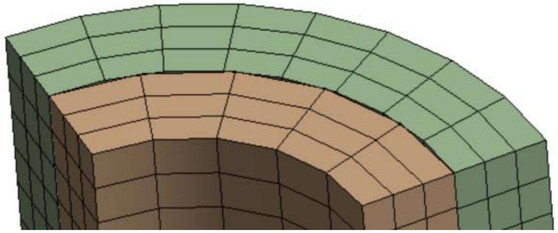
Frictional interface

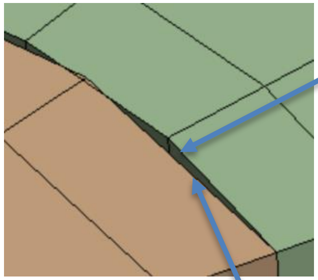
Load: cool-down from 295 K to 4.2K

Stainless steel

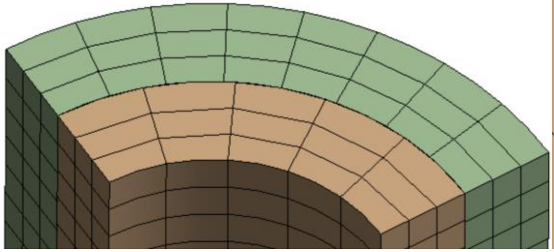
1/8th model

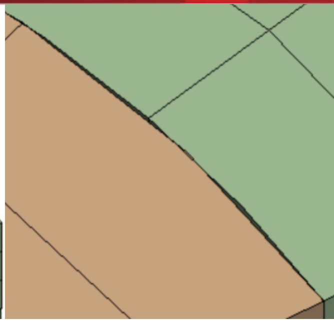
Mismatched Meshes – Wrong Target





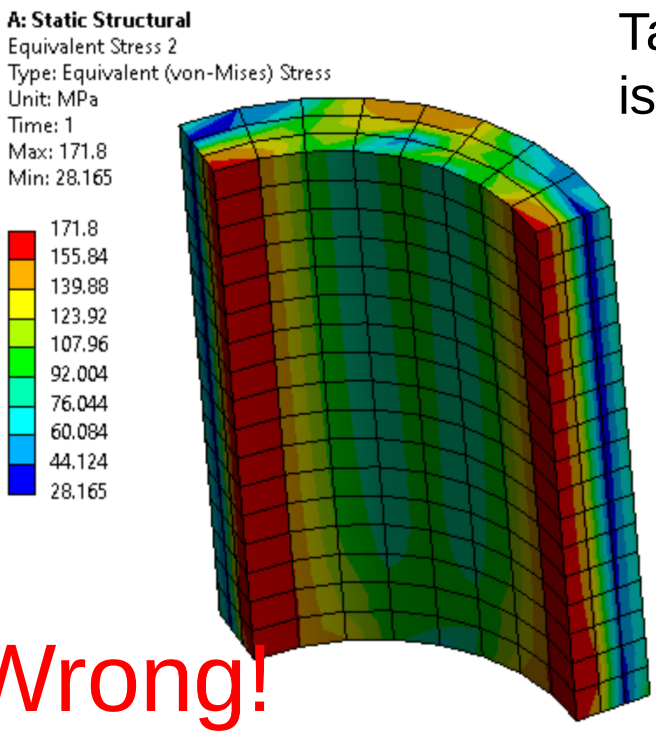
Contact





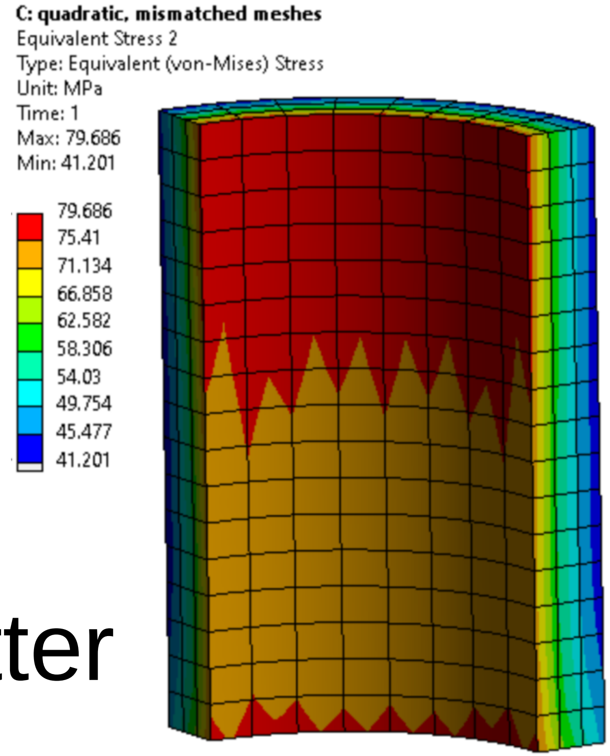
Linear, mismatched

Quadratic, mismatched



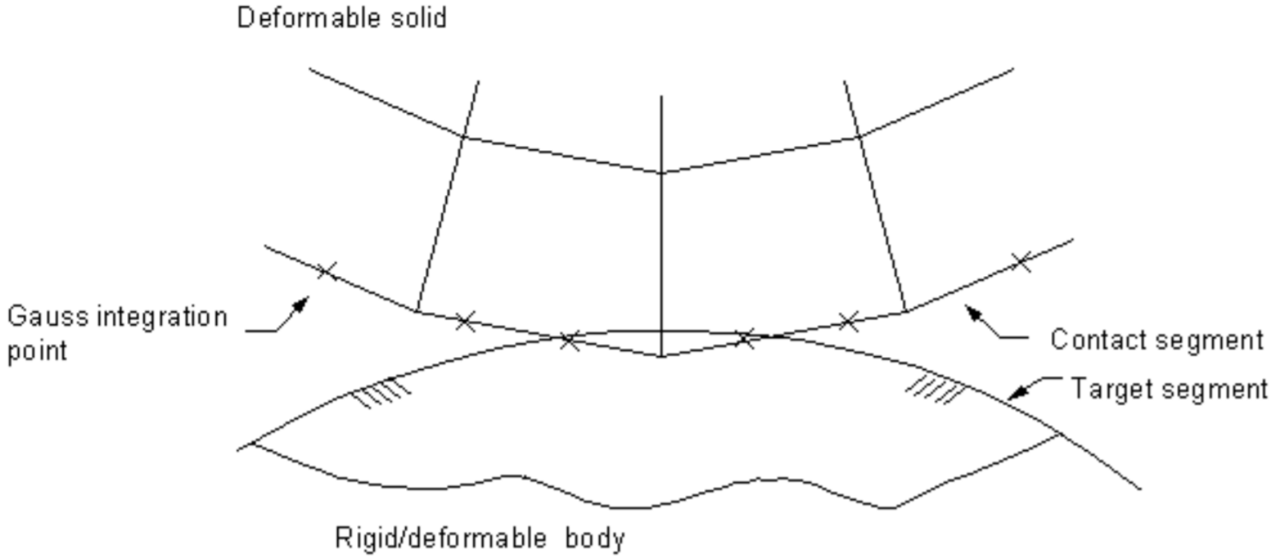
Target surface is wrong.

Wrong!



Better

How to Achieve Better Contact Results per ASNYS Help



1. Contact elements can’t penetrate the target surface. However, target elements can penetrate through the contact surface.
2. The target surface is always the rigid surface; the contact surface, deformable surface.
3. For flexible-to-flexible contact, the choice of which is contact or target can cause a different amount of penetration and therefore affect the solution accuracy.

How to Achieve Better Contact Results per ANSYS Help

1. The flat/concave surface should be the target surface.

2. The coarse mesh should be the target surface and the fine mesh should be the contact surface and

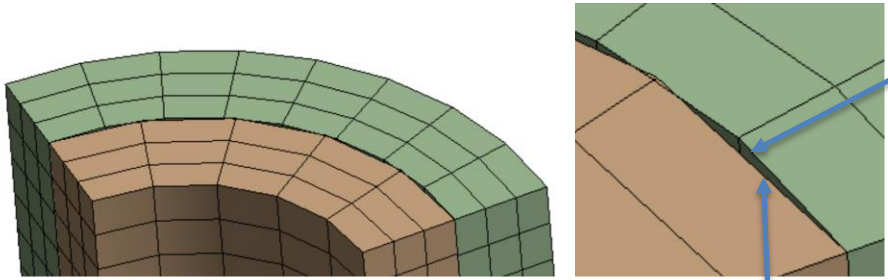
3. The softer surface should be the contact surface and the stiffer surface should be the target surface.

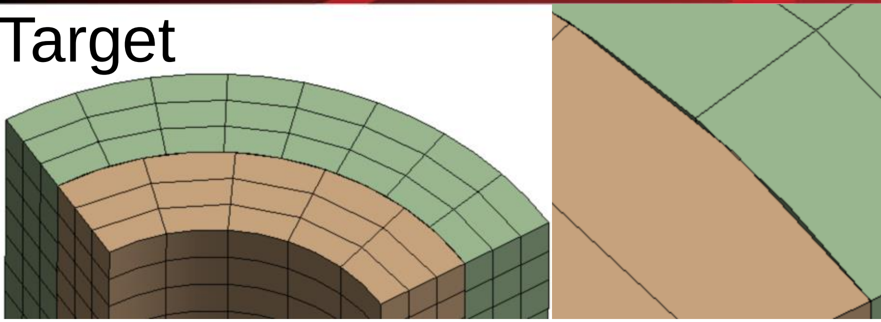
4. The surface with the underlying higher-order elements should be the contact surface and the other surface should be the target.

5. The larger surface should be the target surface.

Mismatched Meshes – Correct Target

Target

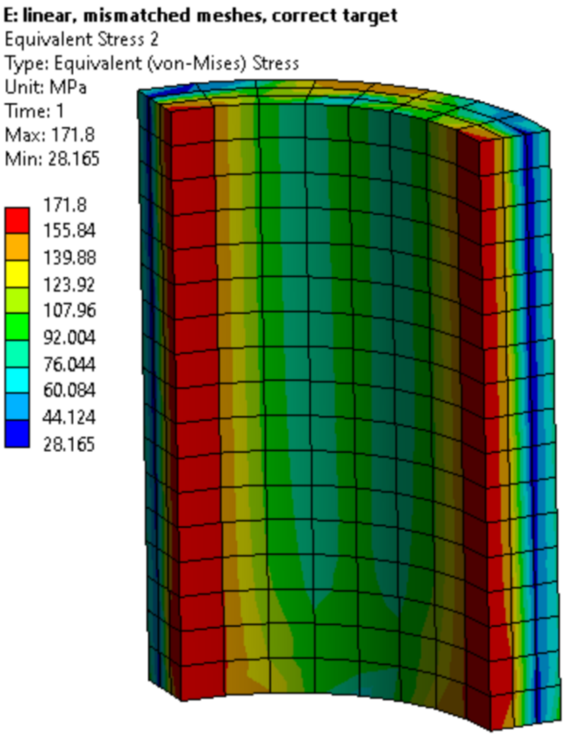




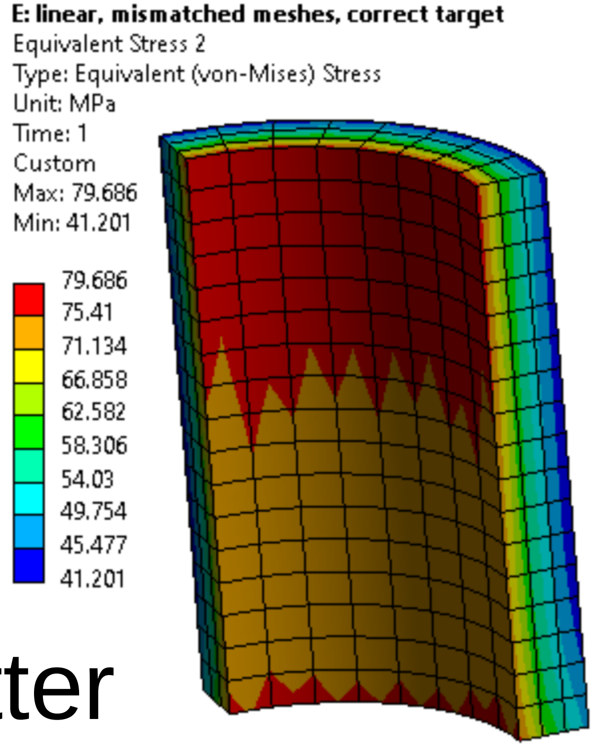
Linear, mismatched

Quadratic, mismatched

Contact

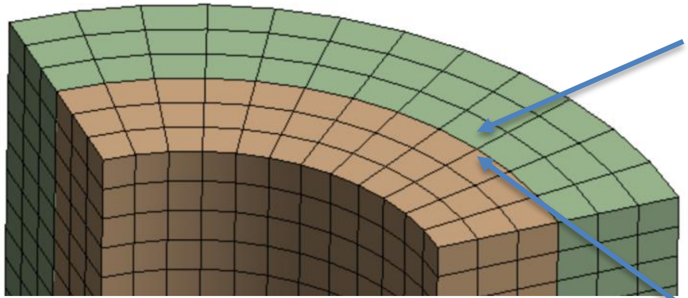


Still wrong with correct target surface!

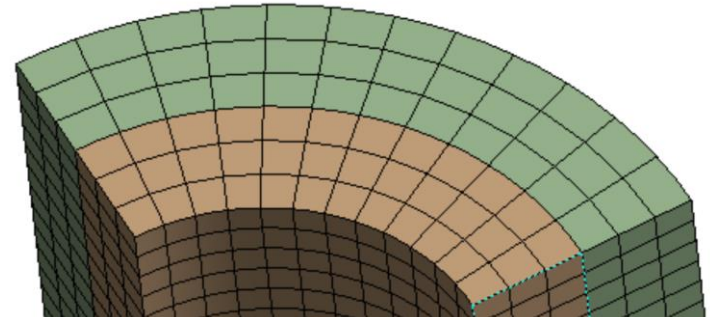


Better

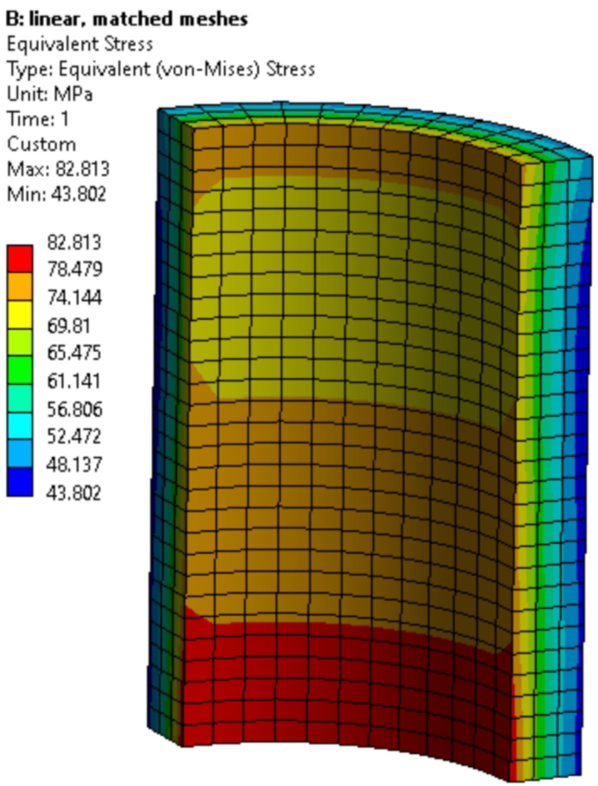
Matched Meshes

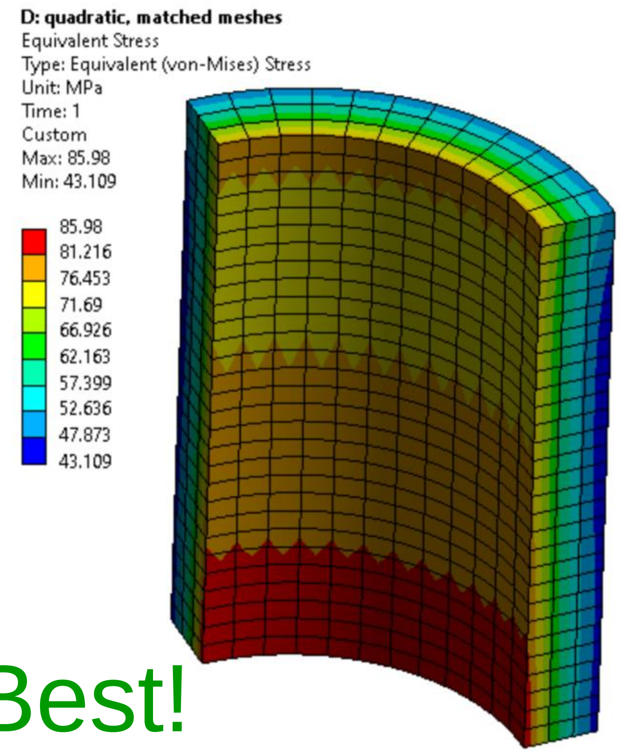


Linear, matched



Quadratic, matched





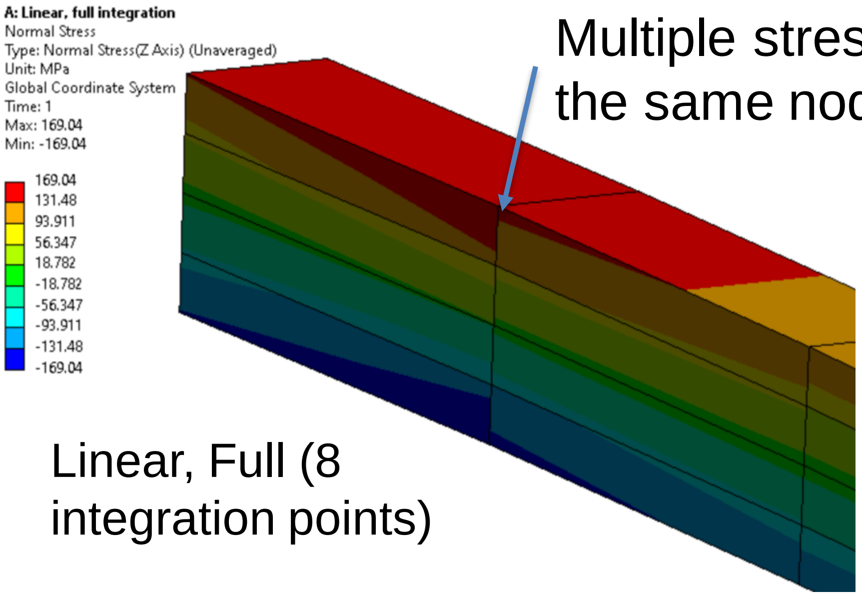
Best!

Lessons and Tricks Learned–Integration Points

8. The strain and stress are computed at the integration point, not at the node. One integration point corresponds to only one strain and one stress. More integration points mean more displacements inside the elements.

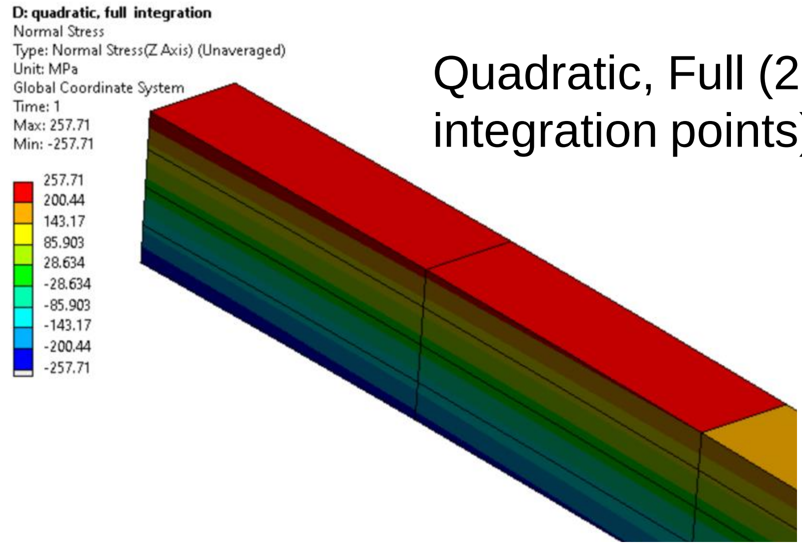
9. There are multiple computed strains at the same node from adjacent elements and thus multiple computed stresses. They are usually averaged. A converged value can be reached with finer mesh sizes at locations without stress concentration.

Tricks Learned – Integration Points

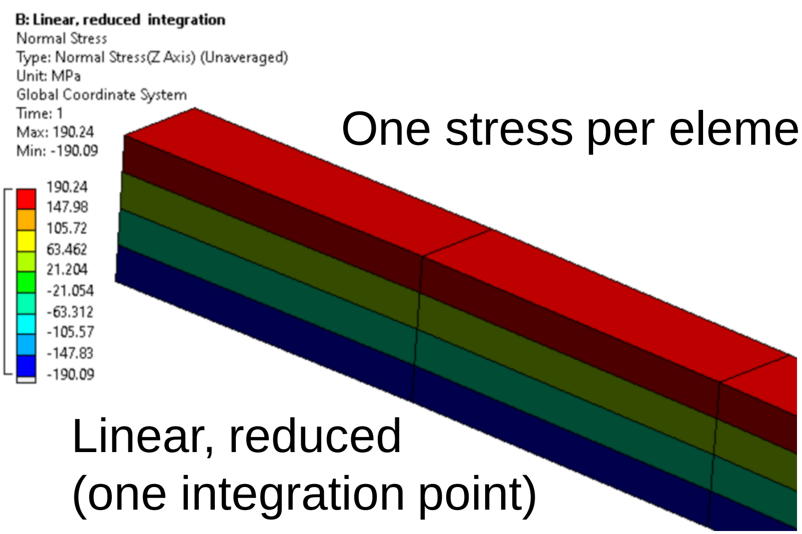


Multiple stresses at the same node

Linear, Full (8 integration points)

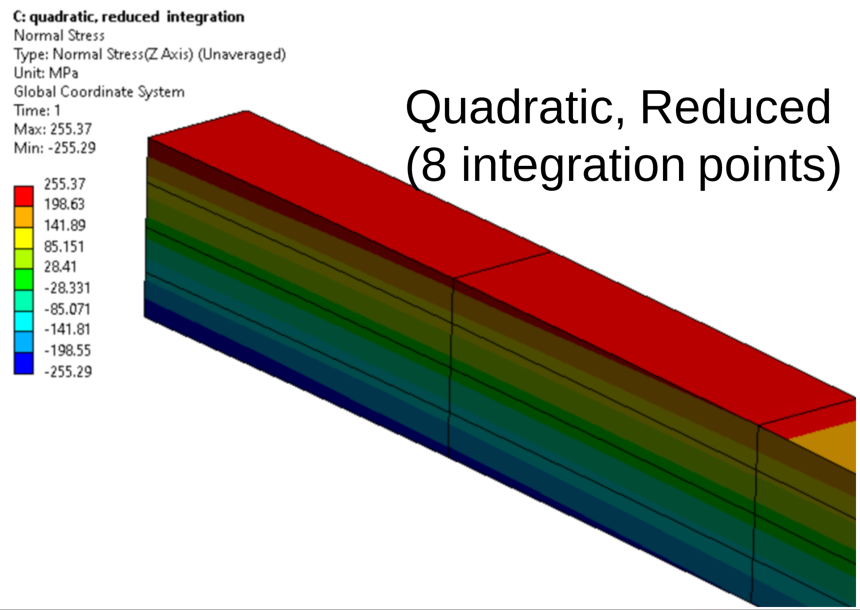


Quadratic, Full (27 integration points)



One stress per element

Linear, reduced (one integration point)



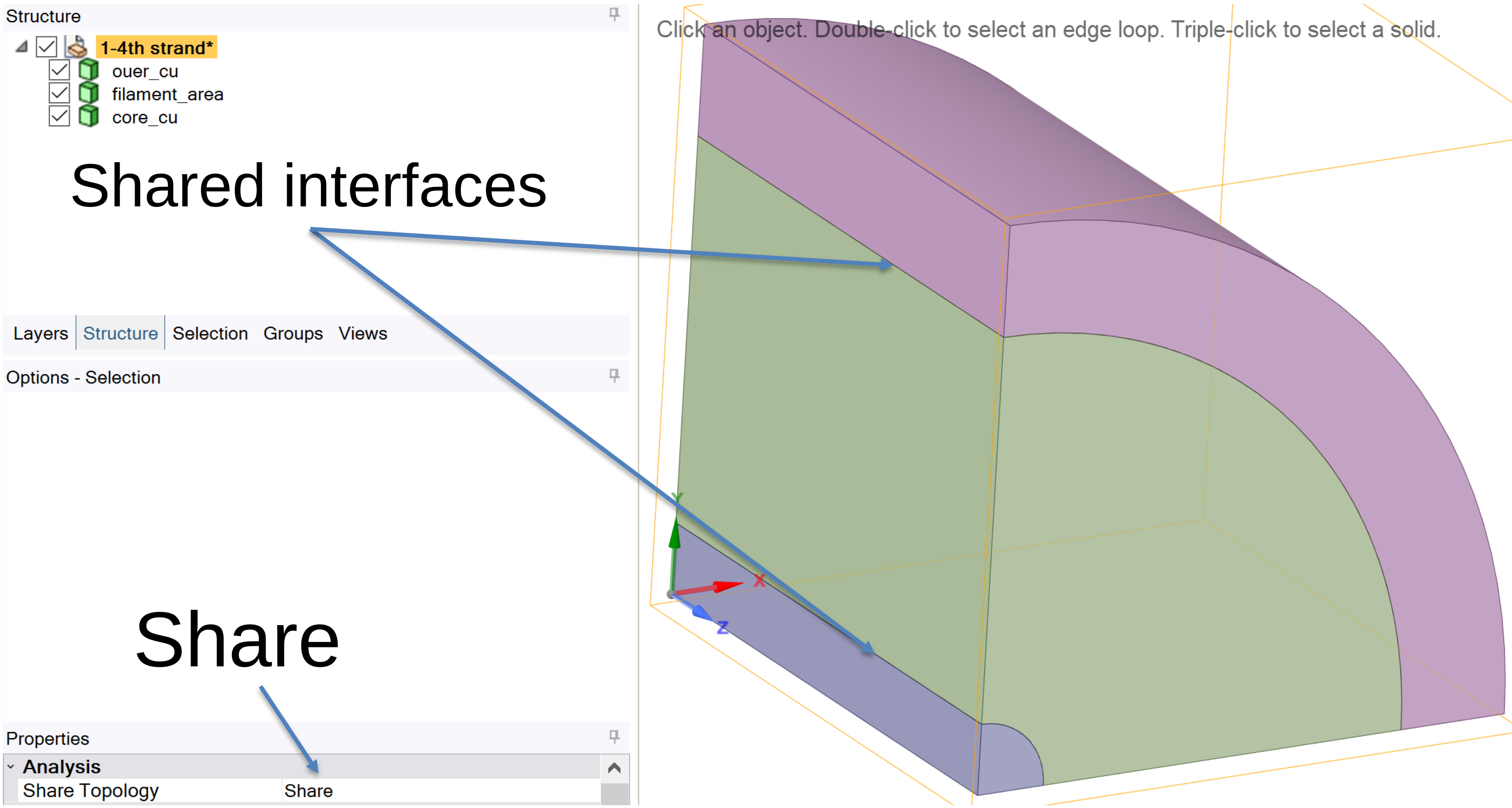
Quadratic, Reduced (8 integration points)

Tricks Learned – Shared Interfaces and Node Merging

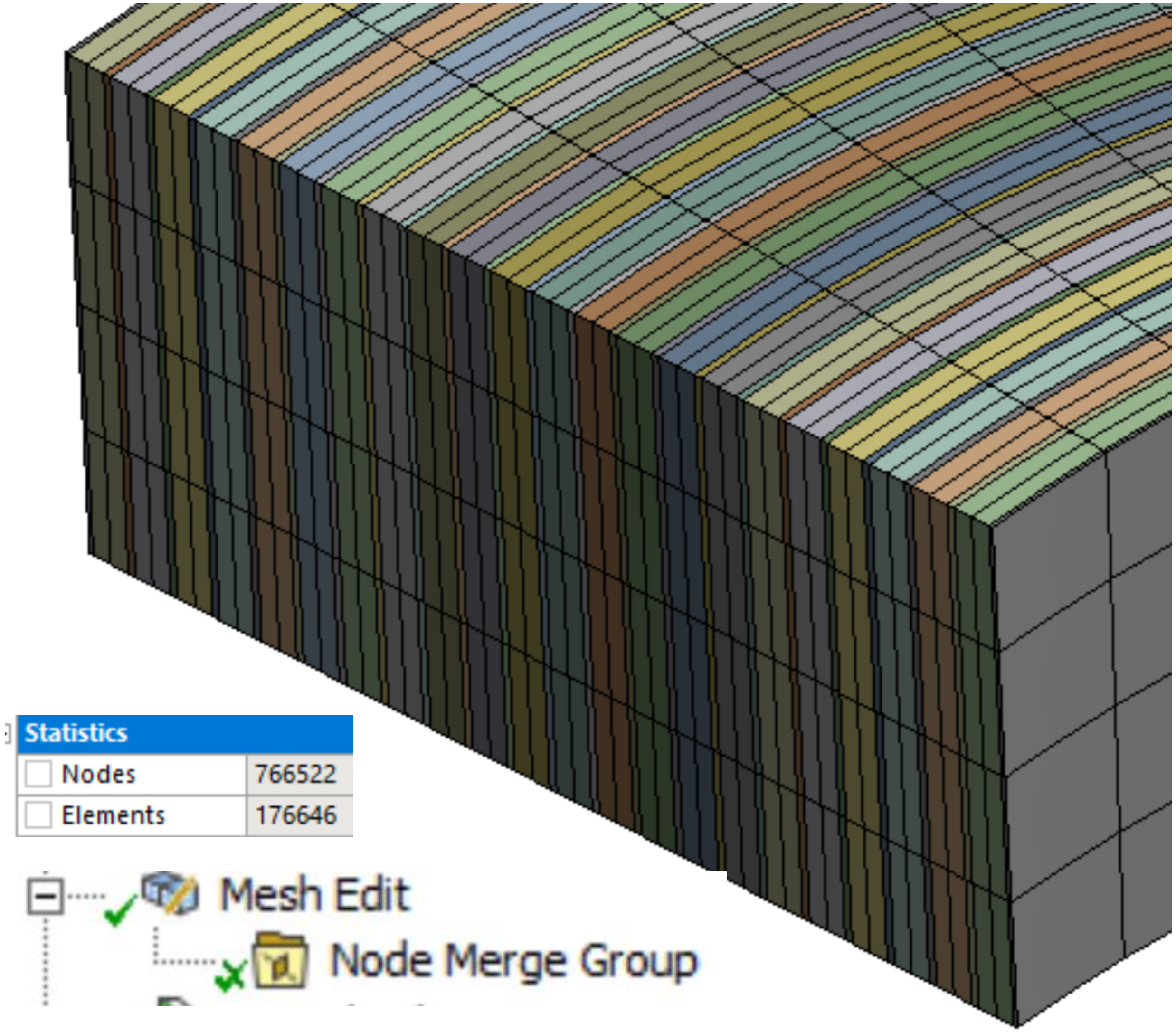
10. It is always the goal to minimize the number of bonded contacts if possible. Contacts would create non-continuous stresses. Use Share Topology [Share] in SpaceClaim to reduce the number of contacts.

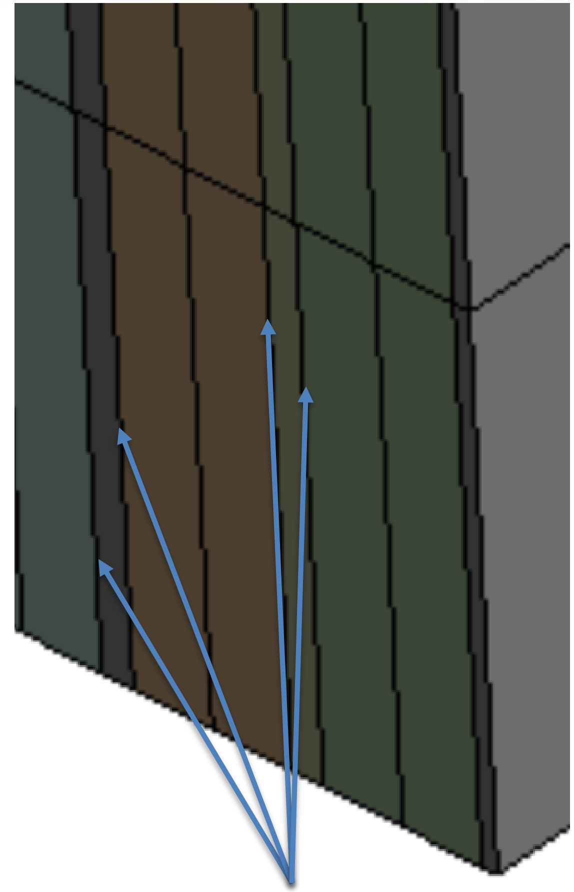
11. Sometimes, ANSYS will choke if a lot of Share conditions are imposed on the complicated geometry. A better way is to mesh each body individually but use the same mesh size and pattern at their shared surfaces. Use Node Merging Tool in Mechanical to merge the nodes at the same locations.

Examples – Shared Interfaces



Example - Node Merging





Nodes on interfaces are merged.

Automatic node merge operation merged 297190 node(s).

Nodes are down 28%.

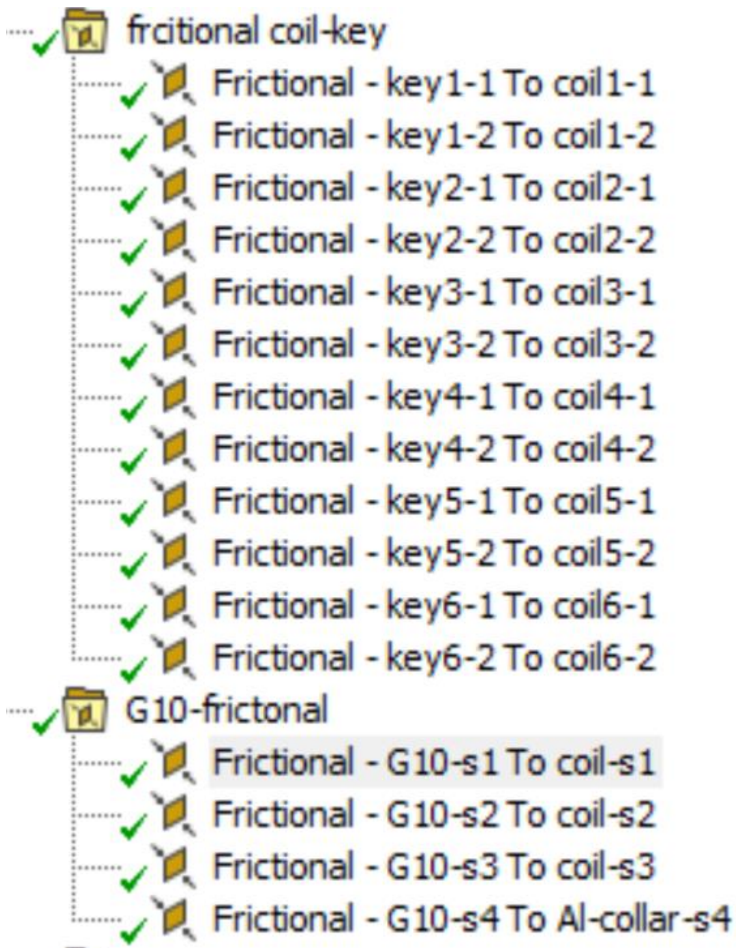
Tricks Learned – Define Contacts Manually if Possible

12. If possible, define contact manually using properly pre-defined named faces.

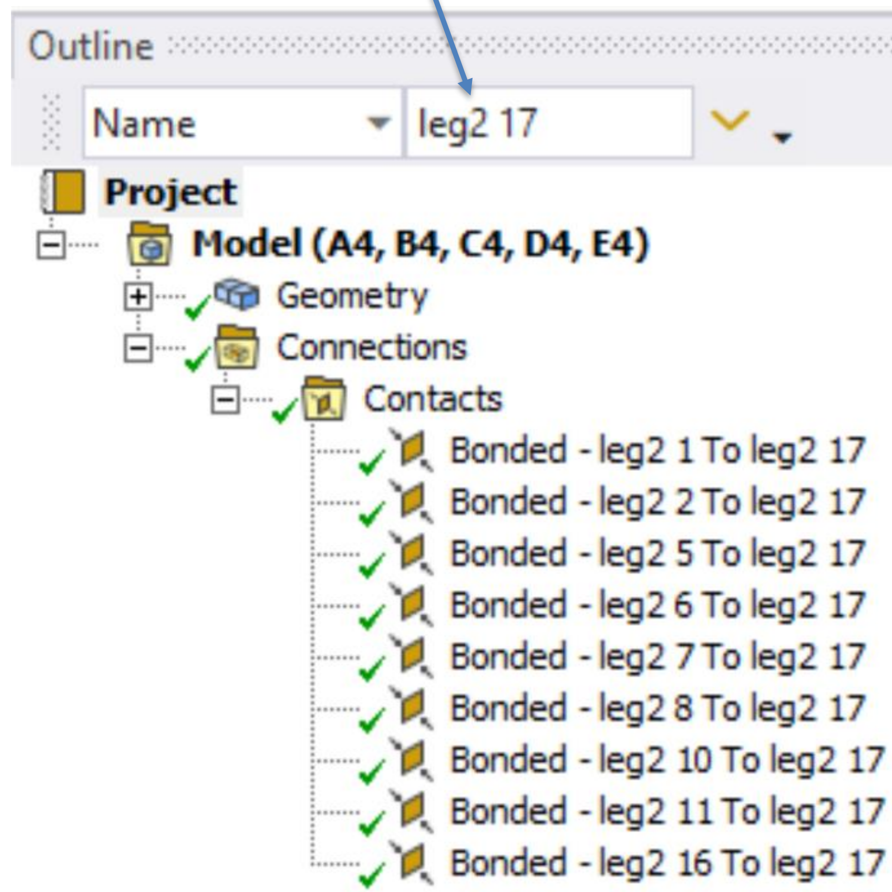
13. If not possible, properly name the interested components, define the contact scope and search for these components to make sure automatic contact definitions are adequate.

Examples - Define Contacts

Surfaces are named properly in advance.



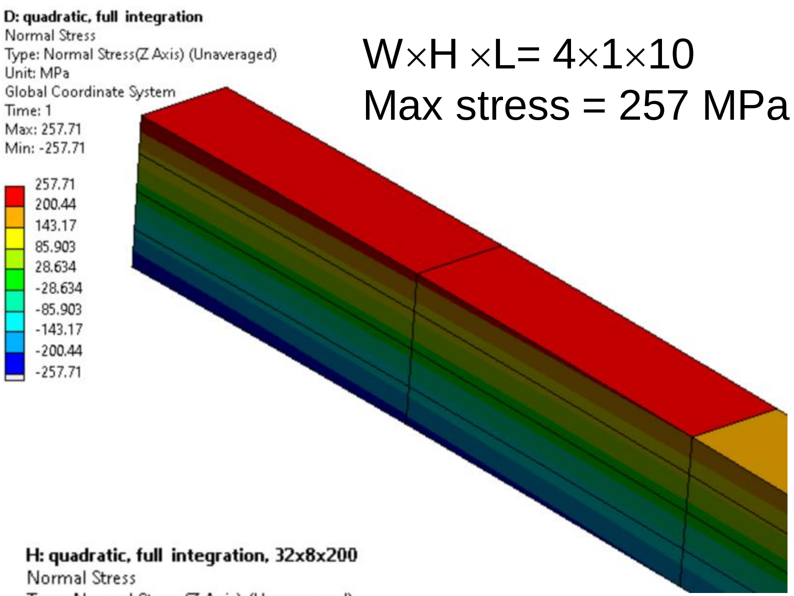
Search for “leg2 17”



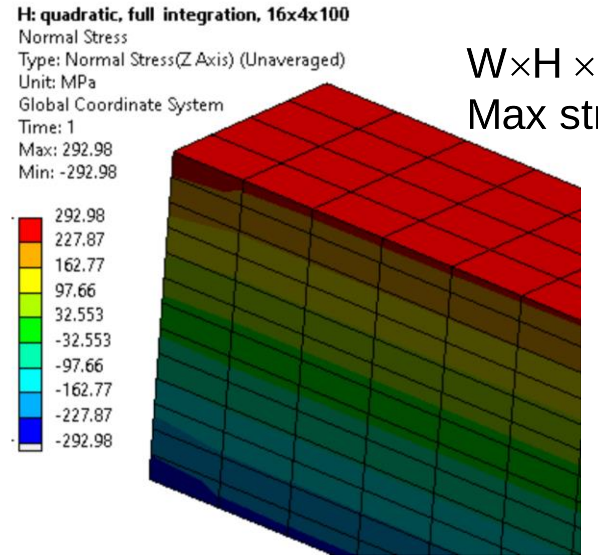
Lessons Learned – Stress Concentration

14. For linear elastic analysis, there is no converged stress at the areas with stress concentration. The maximum stress is infinite just as the theory predicts. Thus, if there is stress concentration, the maximum stress in a linear elastic analysis is not a useful value to validate the structure.

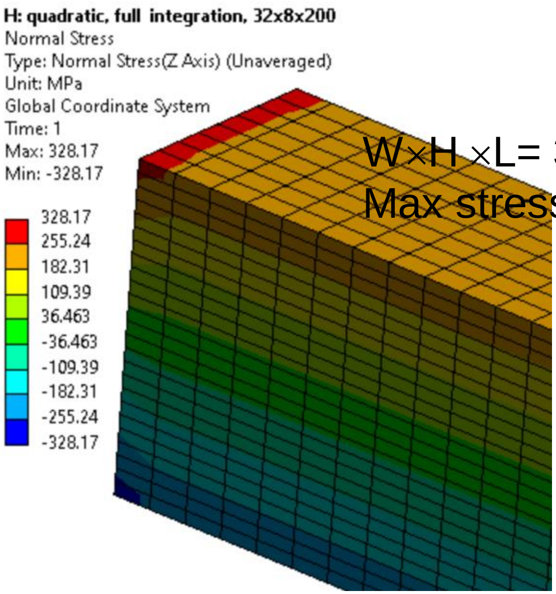
Stress Concentration- An Example



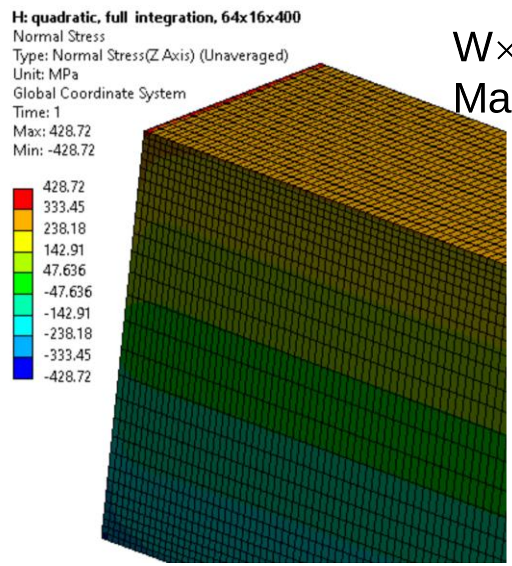
W×H ×L= 4×1×10
Max stress = 257 MPa



W×H ×L= 16×4×100
Max stress = 293 MPa



W×H ×L= 32×8×200
Max stress = 328 MPa



W×H ×L= 64×16×400
Max stress = 429 MPa

Stress Concentration – A Real Example

- Pictures couldn't convey my amazement how beefy the shipping frame was.
- I guessed that beam elements were not used.

Lessons Learned – Shear Locking

15. Fully integrated linear elements, such as 8-node solid elements, may suffer shear locking. False results may be produced with fully integrated 8-node solid elements.

Shear Locking Mechanism Explained

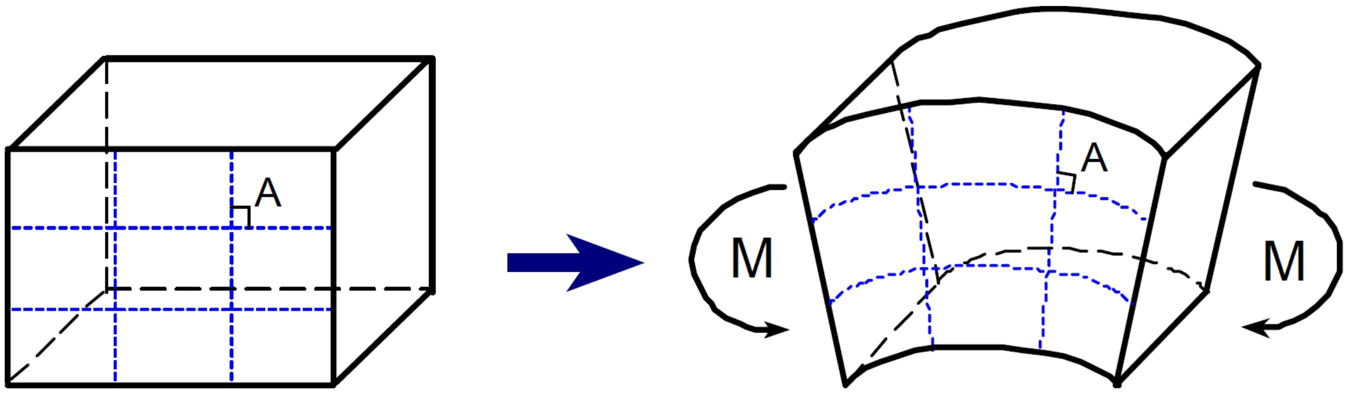


Fig. 2 Shape Change of the Material Block under the Moment in the Ideal Situation

Deformed edges are straight.

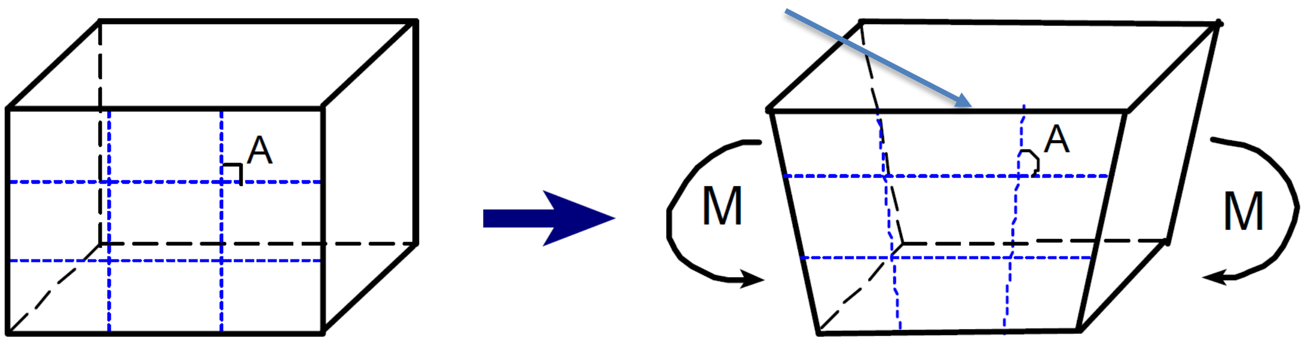


Fig. 3 Shape Change of the Fully Integrated First Order Element under the Moment

Fully integrated linear elements couldn't simulate the ideal situation. They are too rigid.

Simple Beam Example

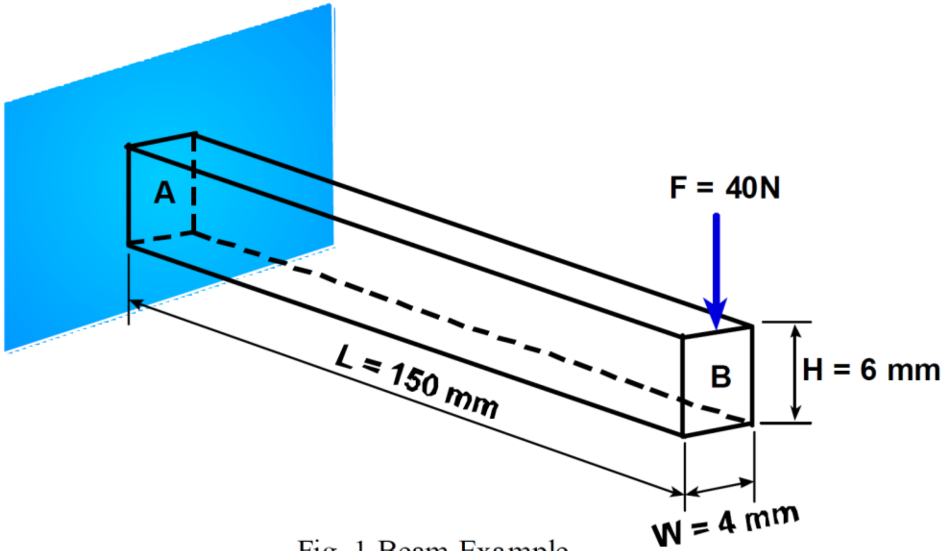
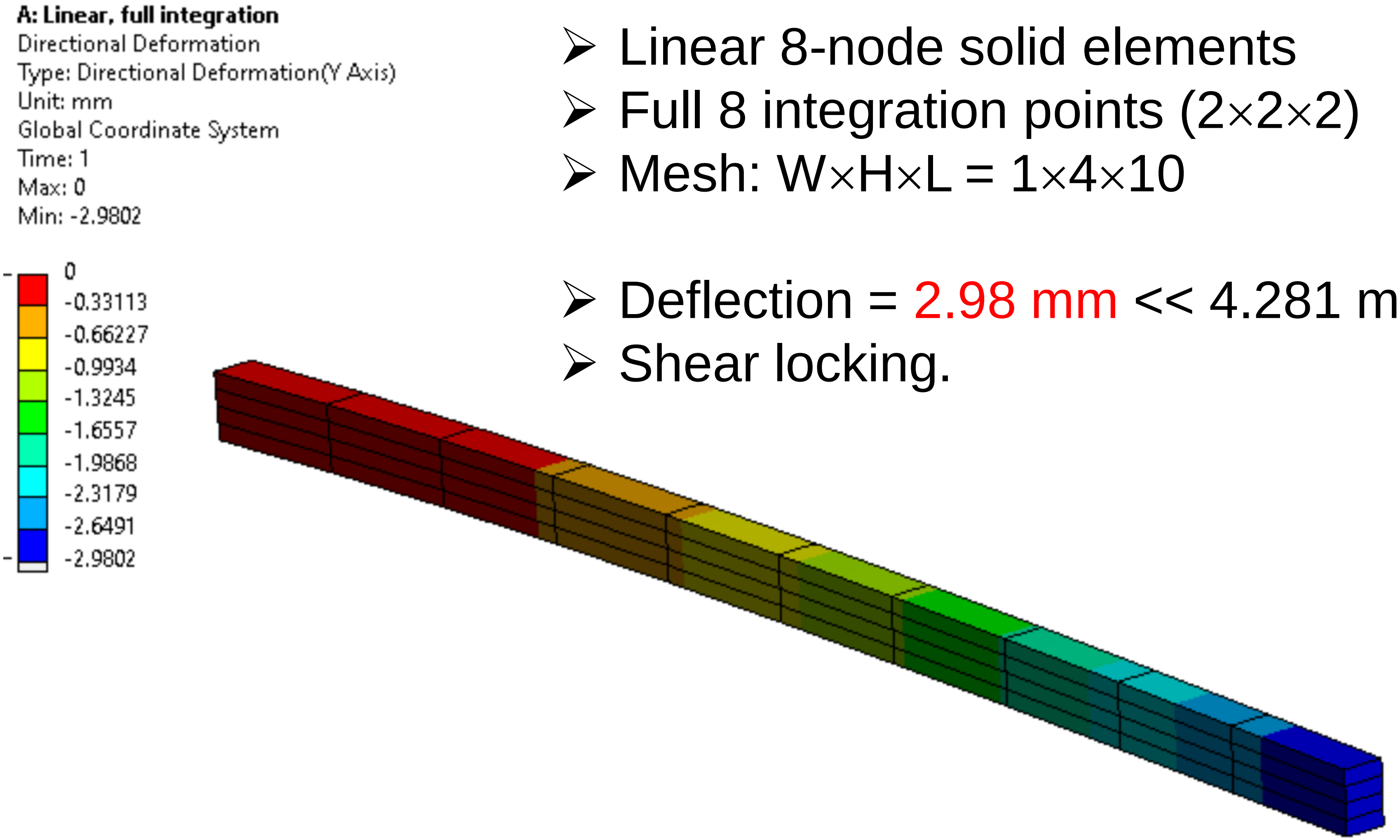


Fig. 1 Beam Example

$$E_{11} = 146.9GPa = 146.9\times10^{3}\ N/mm^{2}$$
$$E_{22} = E_{33} = 10.89GPa = 10.89\times10^{3}\ N/mm^{2}$$
$$G_{12} = G_{13} = 10.89GPa = 10.89\times10^{3}\ N/mm^{2}$$
$$G_{23} = 6.4GPa = 6.4\times10^{3}\ N/mm^{2}$$
$$\upsilon_{12} = \upsilon_{23} = 0.38$$
$$\upsilon_{23} = 0.776$$
$$\rho = 1.5\times10^{3}\ kg/m^{3} = 1.5\times10^{-9}\ ton/mm^{3}$$

Deflection = 4.281 mm
First mode = 284 Hz
Bending stress = 250 MPa

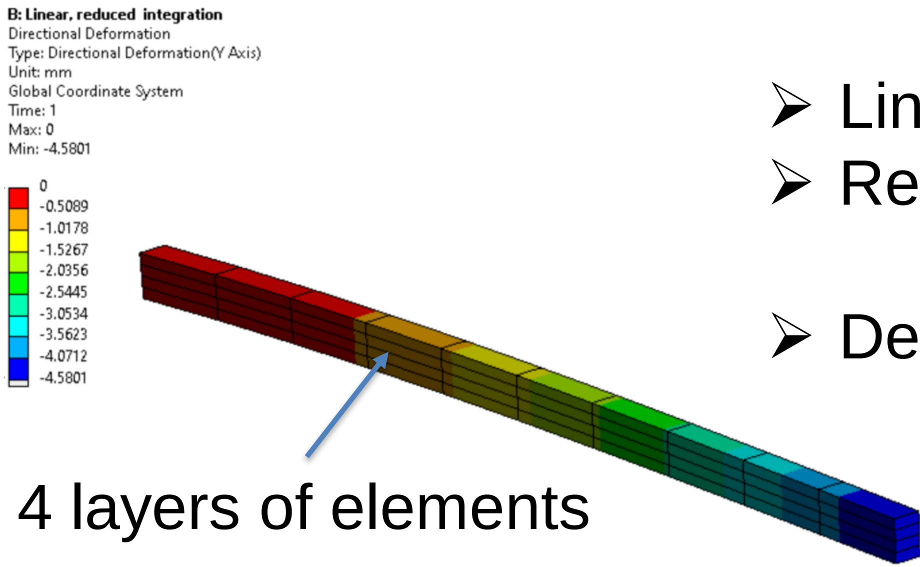
Shear Locking



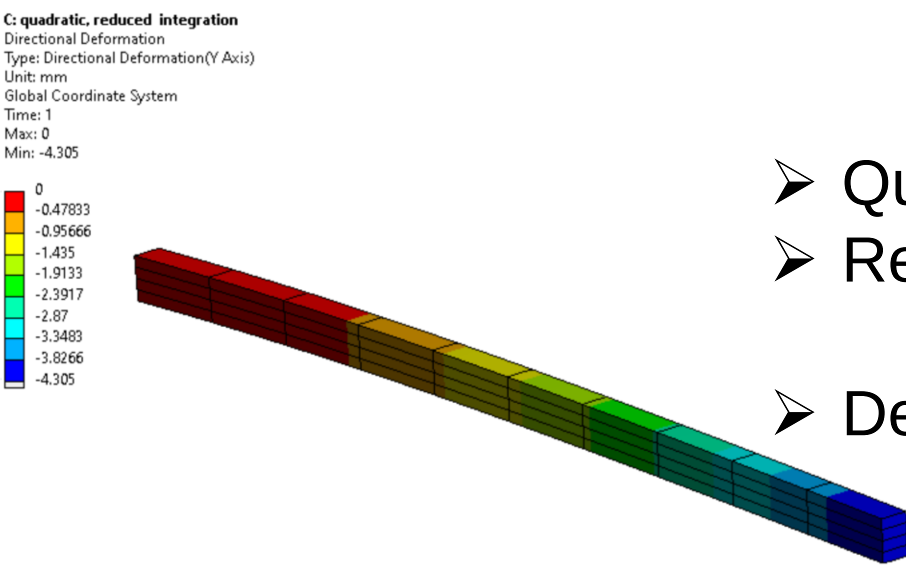
- Linear 8-node solid elements
- Full 8 integration points (2×2×2)
- Mesh: W×H×L = 1×4×10

- Deflection = 2.98 mm << 4.281 mm
- Shear locking.

Shear Locking - Remedies

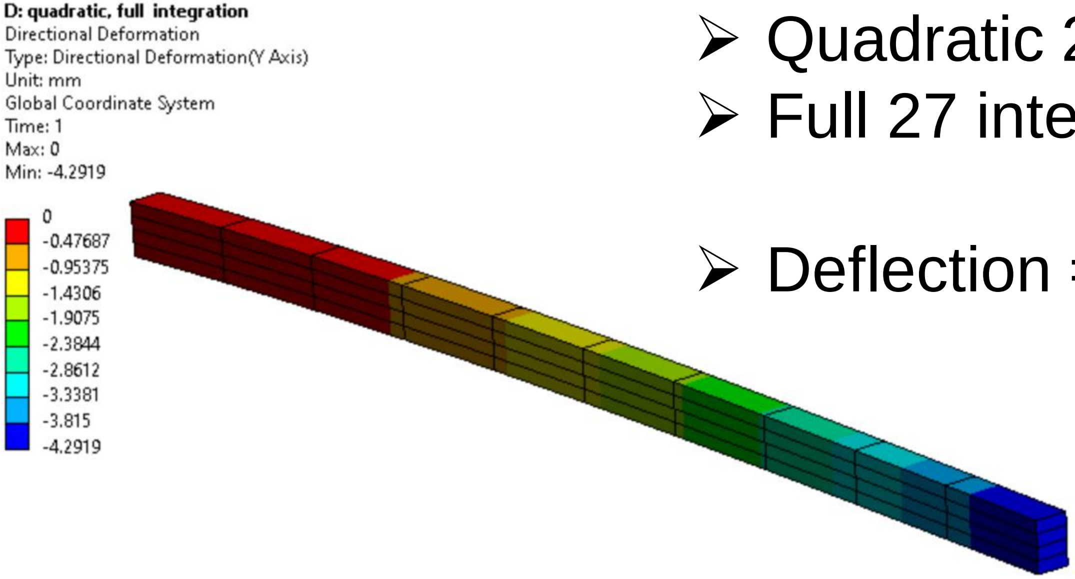


- ➢ Linear 8-node solid elements.
- ➢ Reduced integration = 1 point.
- ➢ Deflection = 4.580 mm vs 4.281 mm



- ➢ Quadratic 20-node solid elements.
- ➢ Reduced integration = 8 points.
- ➢ Deflection = 4.305 mm vs 4.281 mm

Shear Locking - Remedies



- Quadratic 20-node solid elements.
- Full 27 integration points.

- Deflection = 4.292 mm vs 4.281 mm

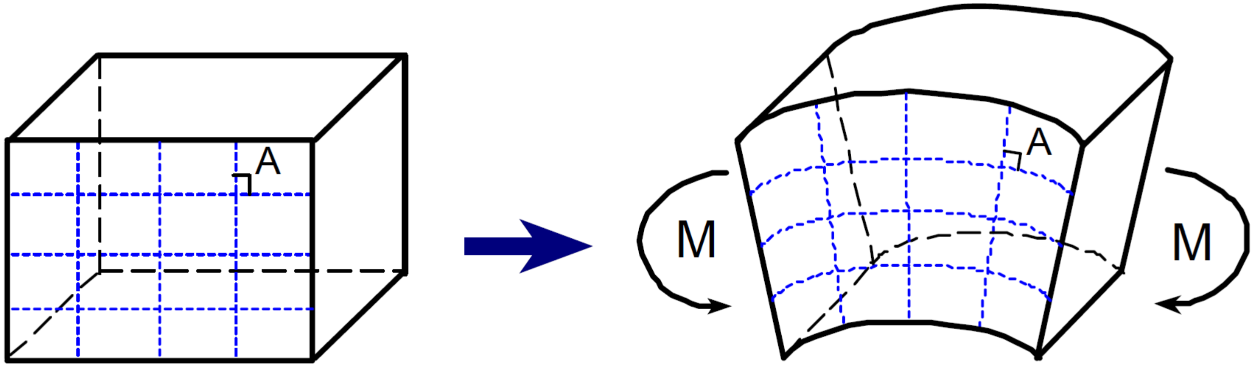


Fig. 4 Shape Change of the Fully Integrated Second Order Element under the Moment

Lessons Learned – Hourglassing

16. The reduced integration of linear 8-node solid elements leads to numerical difficulty called hourglasing because the reduced 8-node solid elements tend to be excessively flexible.

- Linear 8-node solid elements.
- Reduced integration = 1 point.

- Mesh: W×H×L = 1×4×10
- Deflection = 4.580 mm vs 4.281 mm

Hourglassing Explained

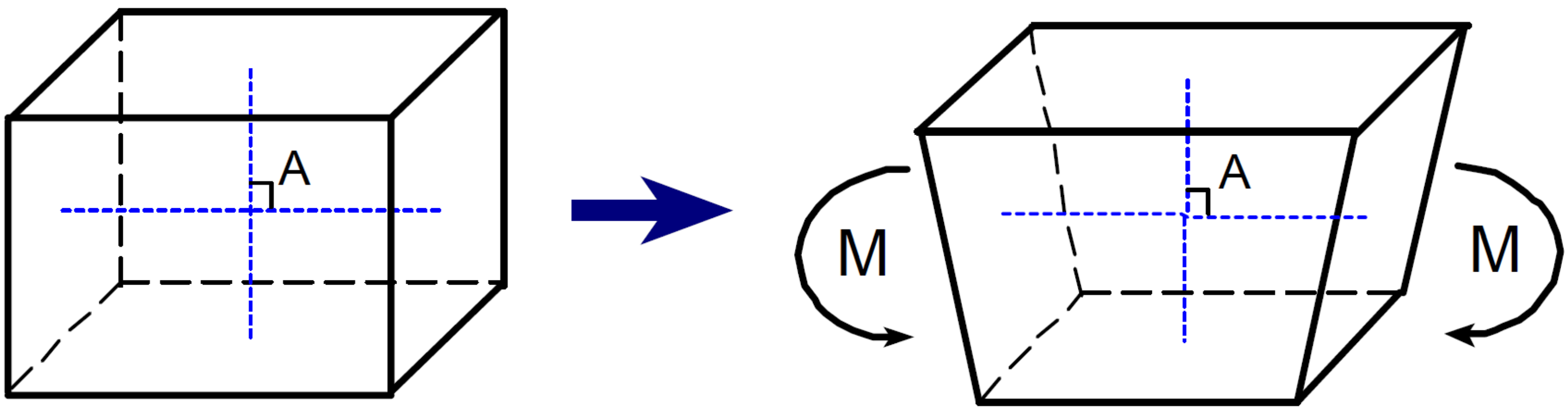
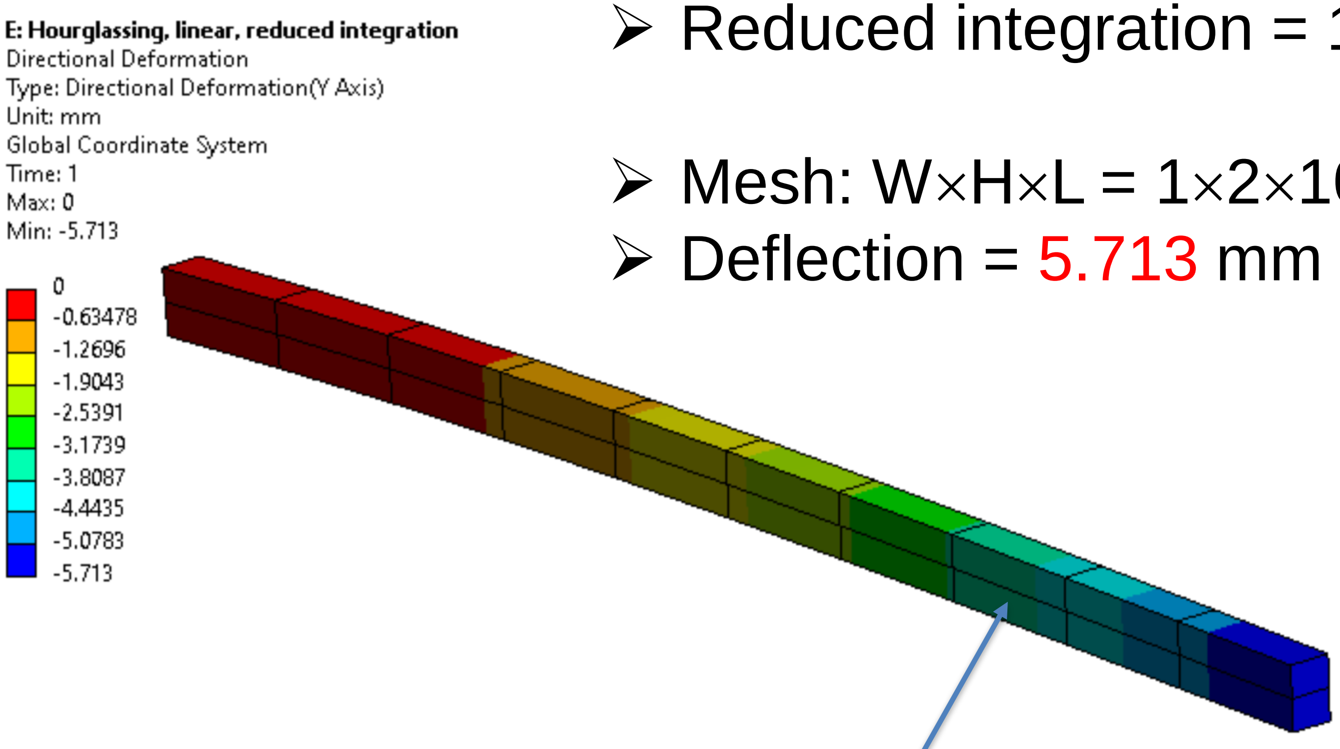


Fig. 5 Shape Change of the Reduced Integration Element under the Moment

Reduced integration of 8-node solid element has one integration point only.

Hourglassing - Example

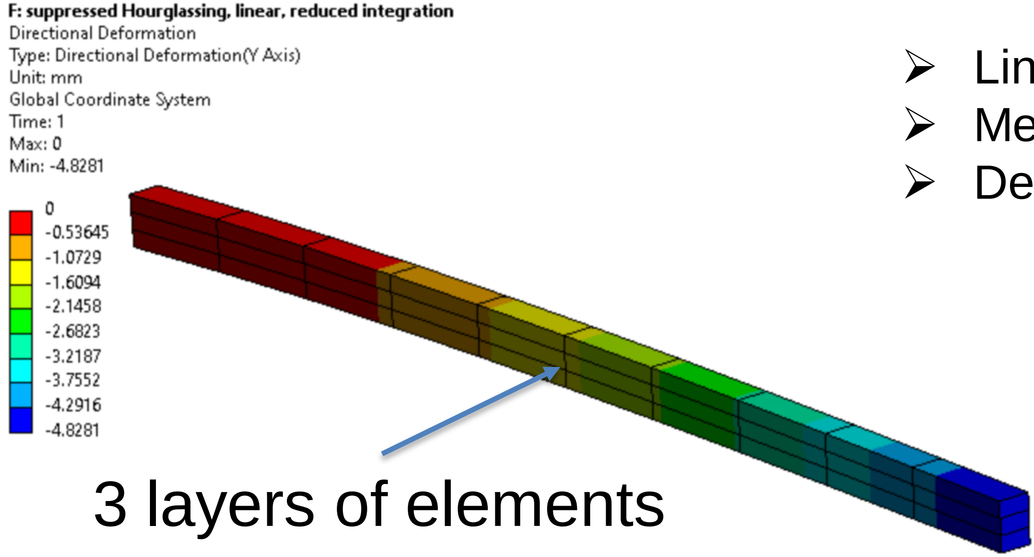


- ➢ Linear 8-node solid elements.
- ➢ Reduced integration = 1 point.

- ➢ Mesh: W×H×L = 1×2×10
- ➢ Deflection = 5.713 mm vs 4.281 mm

2 layers of elements

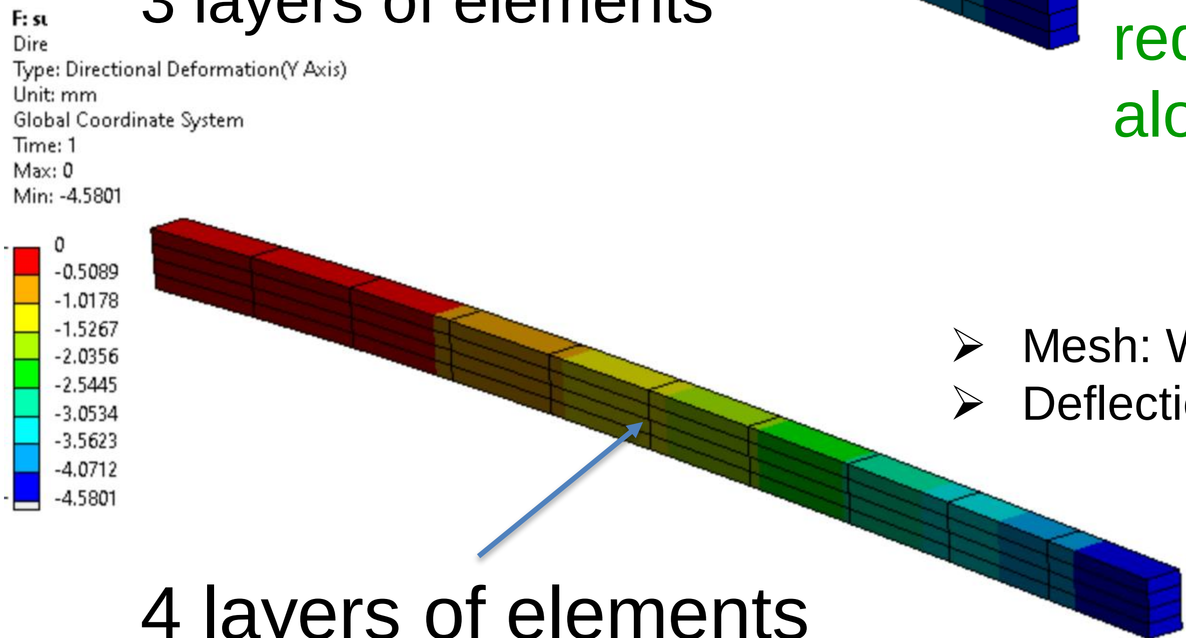
Hourglassing - Remedies



3 layers of elements

- Linear, reduced 8-node solid elements.
- Mesh: W×H×L = 1×3×10, 88 nodes.
- Deflection = 4.828 mm vs 4.281 mm

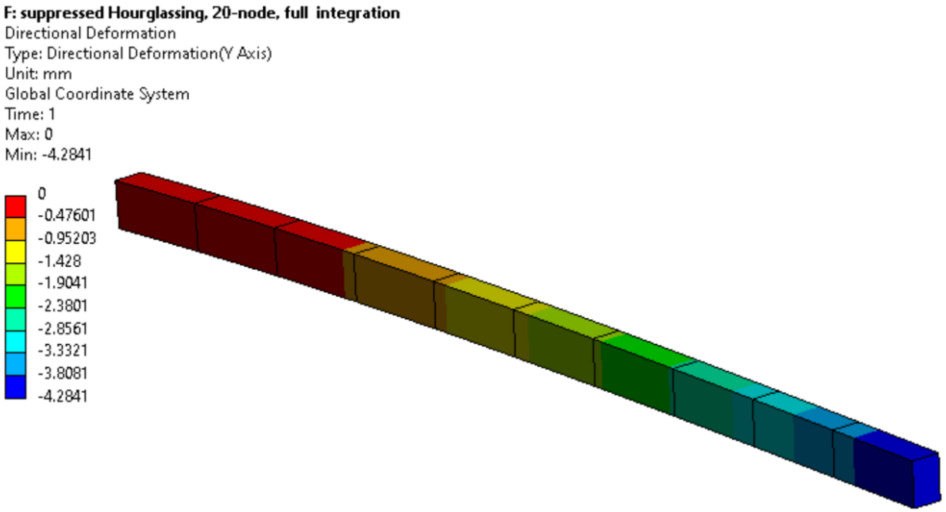
Solution:
Use 3 or more layers of reduced 8-node elements along the thickness.



- Mesh: W×H×L = 1×4×10, 110 nodes.
- Deflection = 4.580 mm vs 4.281 mm

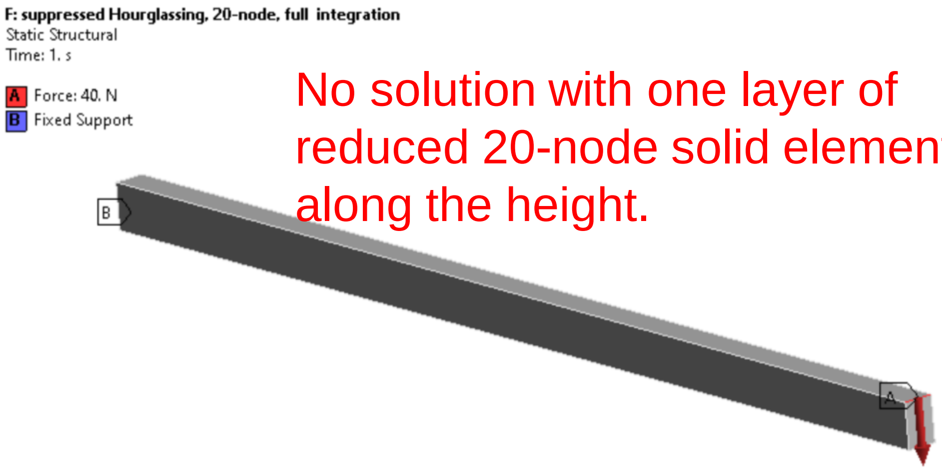
4 layers of elements

Hourglassing - Remedies



- Quadratic 20-node solid elements.
- Full integration.
- 128 nodes.

- Mesh: W×H×L = 1×1×10
- Deflection = 4.284 mm vs 4.281 mm



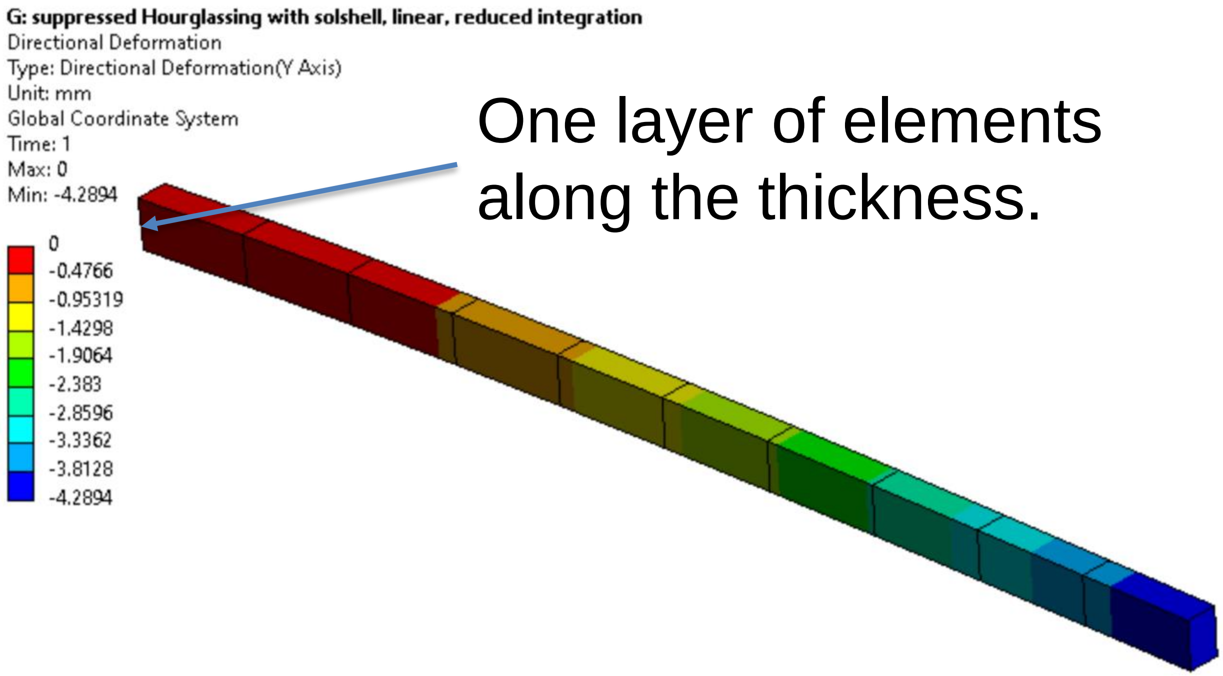
No solution with one layer of reduced 20-node solid elements along the height.

Solution:
Use one layer of fully integrated 20-node elements or 2 layers of reduced 20-node elements.

Hourglassing - Remedies

Details of "Sweep Method" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Sweep
Algorithm	Program Controlled
Element Order	Linear
Src/Trg Selection	Manual Thin
Source	1 Face
Free Face Mesh Type	All Quad
Sweep Num Divs	Default
Element Option	Solid Shell



- Linear 8-node solid-shell elements.
- 44 node
- Mesh: W×H×L = 1×1×10
- Deflection = 4.289 mm vs 4.281 mm

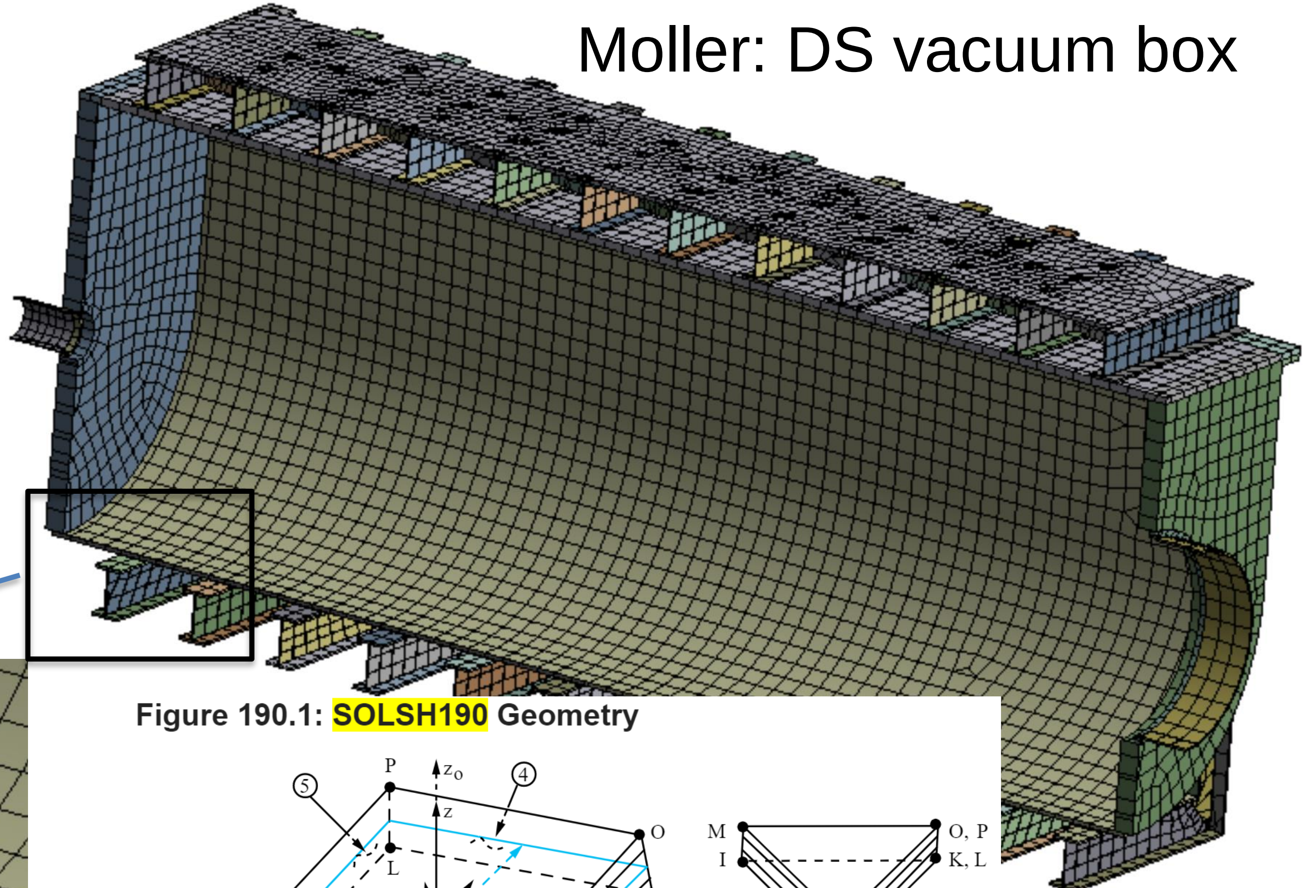
Solution:
Solid shell elements

When no shell section definition is provided, the element is treated as single-layered and uses two integration points through the thickness.

Example using Sold-Shell Elements

Moller: DS vacuum box

SOLSH190 has 8 nodes and supports multilayer structures.



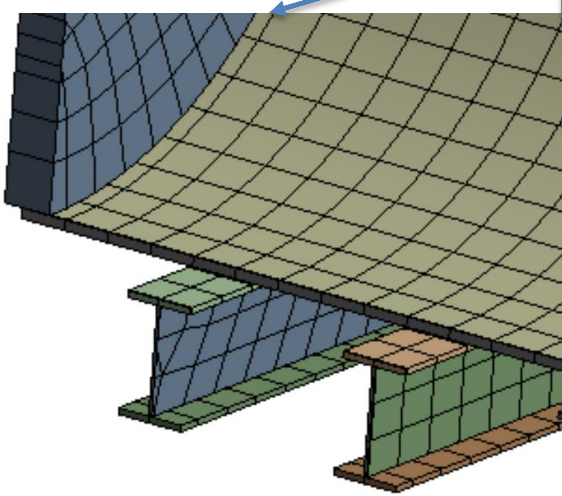
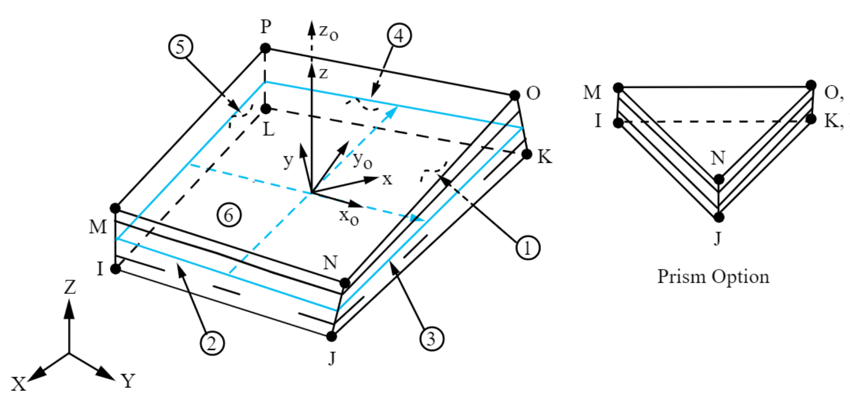


Figure 190.1: SOLSH190 Geometry



Tricks Learned - ASME Pressure Vessel Code

17. [ASME 2019 BPVC, Division 2, Part 5 Design by Analysis, Section 5.2] Elastic-Plastic Stress Analysis Method of Division 2 can be used to qualify Division 1 vessels by elevating the design factors from 2.4 (Division 2) to 3.5 (Division 1).

18. Stress-strain curves of any materials used in elastic-plastic analysis shall be computed according to ASME, Section VIII, Division 2, Annex 3-D STRENGTH PARAMETERS (2019, page 148). Don't use measured stress-strain curves.

ASME 2019, BPVC Section VIII Division 2, page 610

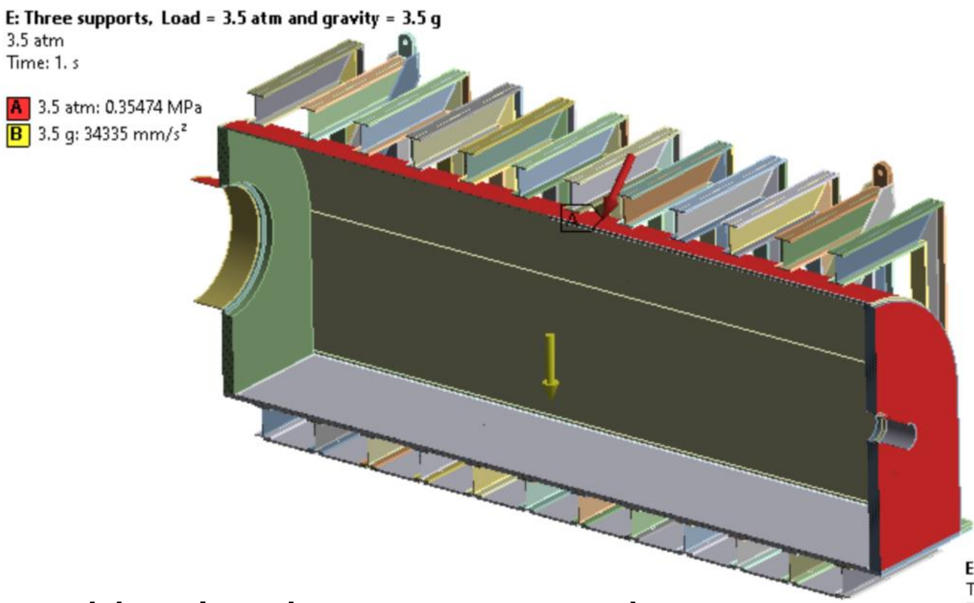
Table 5.5
Load Case Combinations and Load Factors for an Elastic–Plastic Analysis

Criteria	Required Factored Load Combinations
Design Conditions	
Global	(1) $\beta(P + P_s + D)$ (2) $0.88\beta(P + P_S + D + T) + 1.13\beta L + 0.36\beta S_S$ (3) $0.88\beta(P + P_S + D) + 1.13\beta S_s + 0.71\beta L$ or $0.36\beta W$ (4) $0.88\beta(P + P_S + D) + 0.71\beta W + 0.71\beta L + 0.36\beta S_S$ (5) $0.88\beta(P + P_S + D) + 0.71\beta E + 0.71\beta L + 0.14\beta S_S$
Local	$1.7(P + P_s + D)$
Serviceability	Per User's Design Specification, if applicable; see 5.2.4.3(b)

ASME 2019 BPVC Section VIII Division 1, page 584

(-c) When applying the elastic–plastic stress analysis method in Division 2, 5.2.4, in conjunction with Division 2, Table 5.5, β shall equal 3.5.

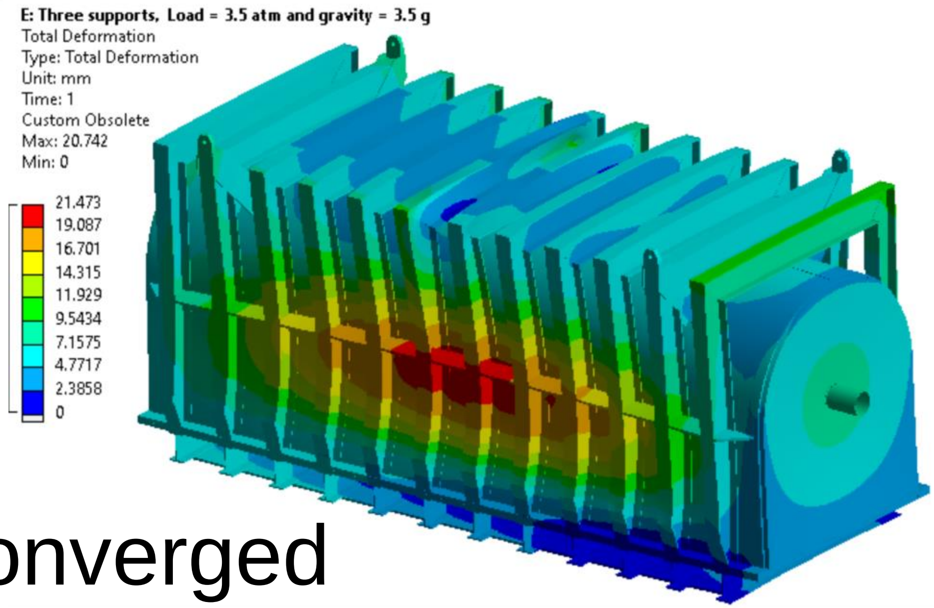
Division 1 Vessel: Design for Analysis



Design factor (β) = 3.5, not 2.4

Analysis load = 3.5 atm and 3.5 g

Working load = 1.0 atm and 1.0 g



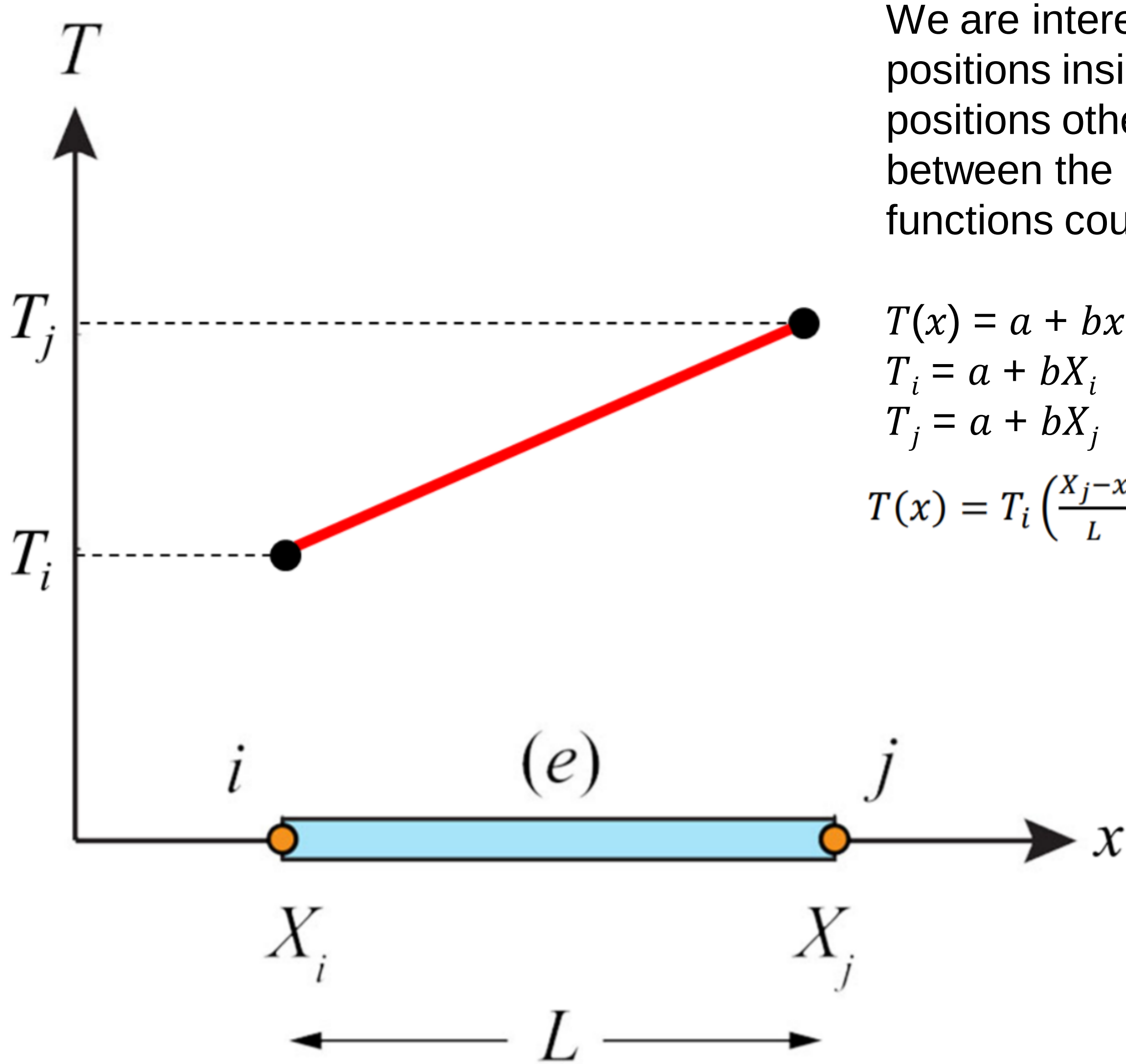
Converged

Summary – Lessons and Tricks Learned

1. **Debugging**
   - You are the only one who is motivated. Don’t rely on support engineers.
   - Always use a simple model; don’t ignore the obvious, even the unit.
2. **Worksheet, Mismatched Meshing, and Integration Points**
   - Worksheet is a useful tool to select a large number of entities.
   - Mismatched meshing patterns of contact interfaces could be problomatic.
   - Strain and stress are first computed at integration points, not at nodes.
3. **Shared Topology, Node Merging, and Contacts**
   - Share Topology and Node Merging should be use to minimize the number of bonded contact.
   - Don’t let ANSYS to automatically define the contacts for you if possible. Name the contact interfaces.
4. **Stress Concentration**
   - Maximum stress value doesn’t convey much information in linear elastic analysis if there is stress concentration. Beam elements should be used for beam-like structures.
5. **Shear Locking, Hourglassing, and Solid Shell Elements**
   - Be ware of the potential errors of analysis results due to shear locking and hourglassing
   - Solid shell elements are great to simulate thin shell vessels.
6. **Pressure Vessel**
   - ASME Section VIII Division 1 vessels can be analyzed with Division 2’s Rules of Design by Analysis.

Backup

Shape Functions



We are interested in the value of the solution at positions inside the element. To calculate values at positions other than the nodes, we interpolate between the nodes using shape functions. Shape functions could be linear or quadratic.

$$T(x) = a + bx$$
$$T_i = a + bX_i$$
$$T_j = a + bX_j$$

$$T(x) = T_i\left(\frac{X_j - x}{L}\right) + T_j\left(\frac{x - X_i}{L}\right)$$

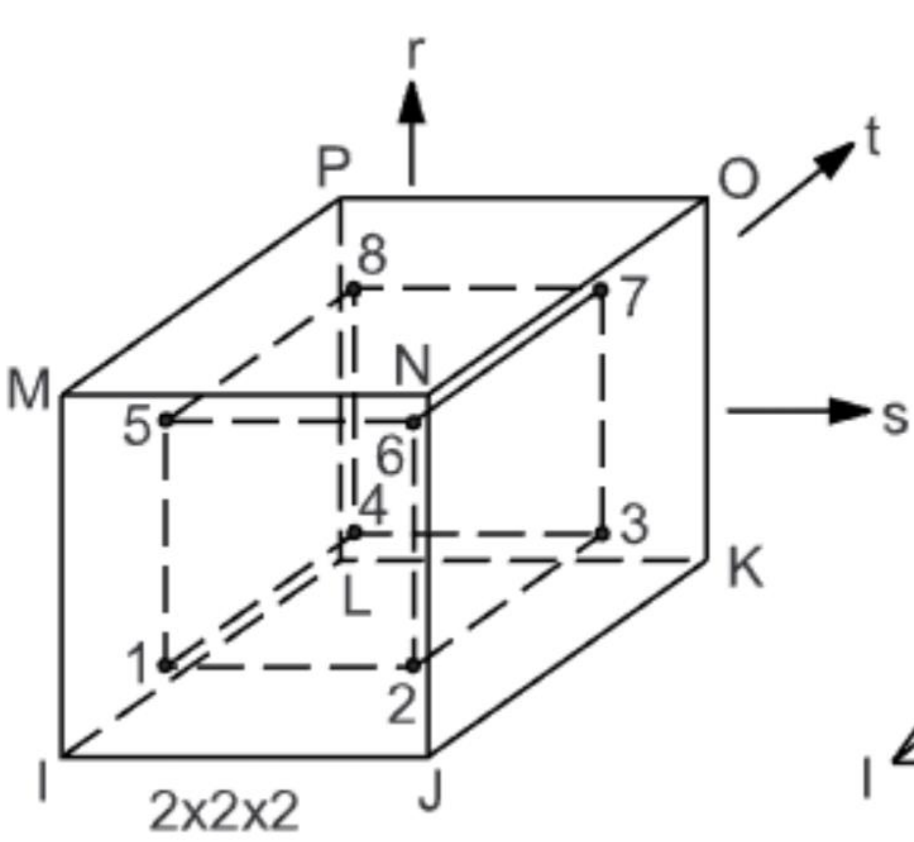
$$T(x) = S_i T_i + S_j T_j$$

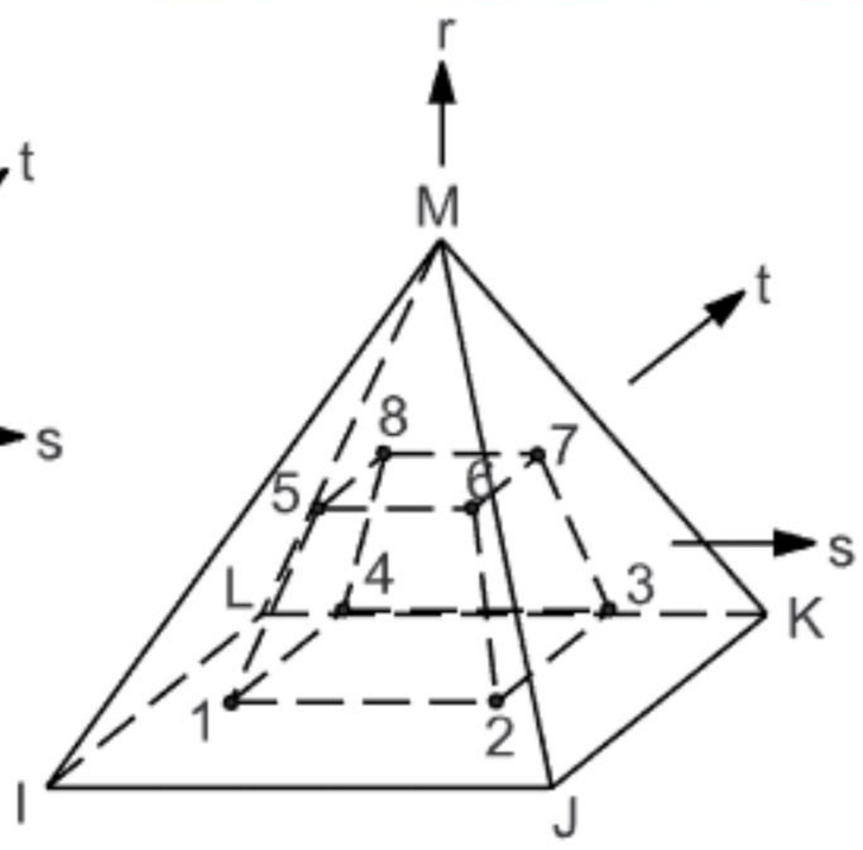
Shape functions

$$S_i = \left(\frac{X_j - x}{L}\right)$$

$$S_j = \left(\frac{x - X_i}{L}\right)$$

Integration Points





While forces and displacements are computed at nodal locations, stresses and strains are computed at integration points.

FEA Solution Process

1. Compute matrices and vectors for all elements. This implies to set up an element-level equation system using matrices and vectors.

2. Assemble the matrices of each element and form a global matrix and global vector and therefore a global equation system.

3. Solve the global equation system for displacements at nodes.

4. Determine strains, inside an element, at its integration points, using the displacement differentiation matrix.

5. Calculate stresses with the Hook's law for each element.