1.3-GHz Superconducting Radio Frequency Cavity Stress and Buckling Analysis

Ashish KUMAR

CASA, Accelerator Laboratory,

High Energy Accelerator Research Organization (KEK)

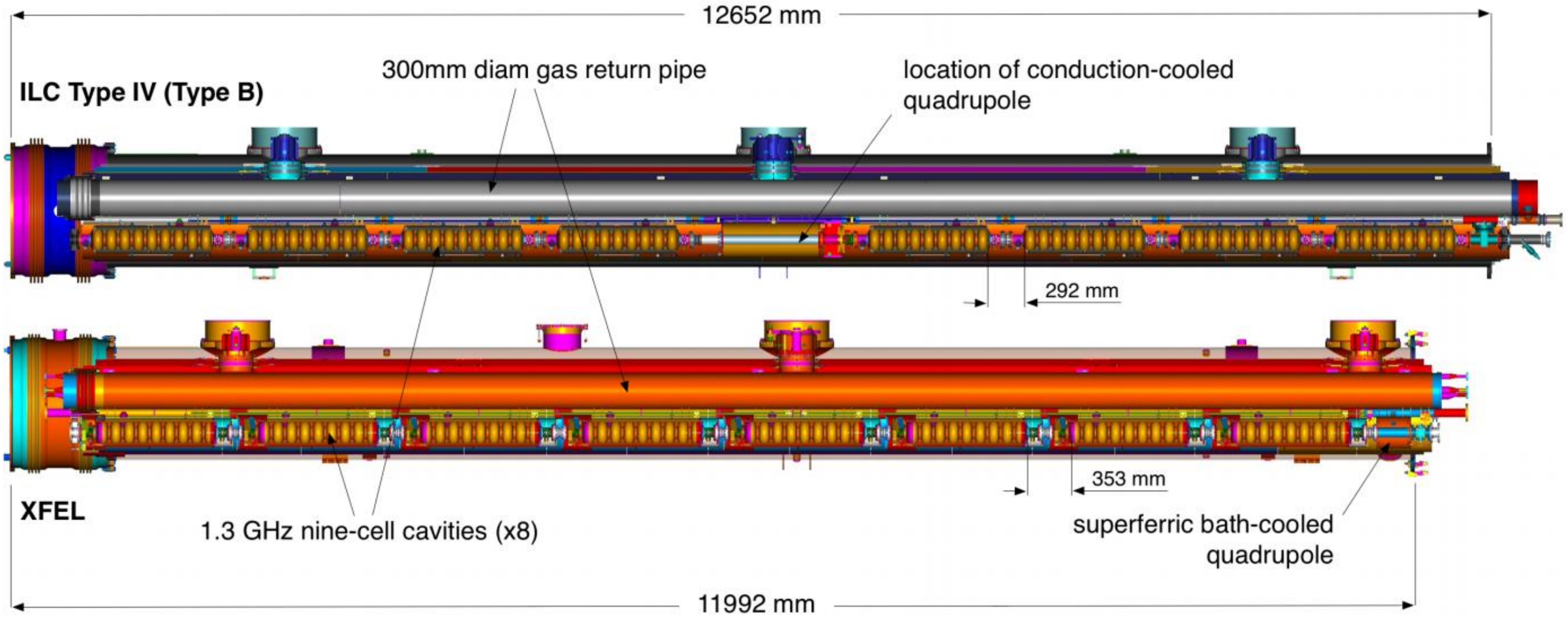
2021/09/09

Contents

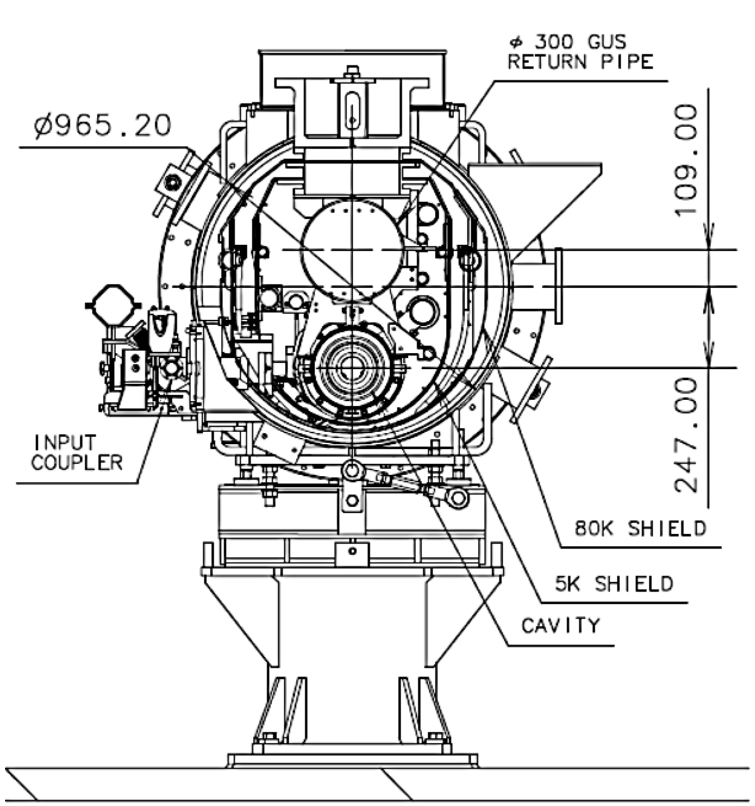
1. Cryomodule Assembly
2. Pressure Vessel Compliance for Cryomodules
3. Structural Analysis using FEA
4. 1.3 GHz 3-Cell SRF Cavity Structural Analysis
5. Eigen value Buckling Analysis
6. 1.3 GHz 3-Cell SRF Cavity Buckling Analysis
7. 1.3 GHz 3-Cell SRF Cavity Non-linear Buckling Analysis

Cryomodule Assembly

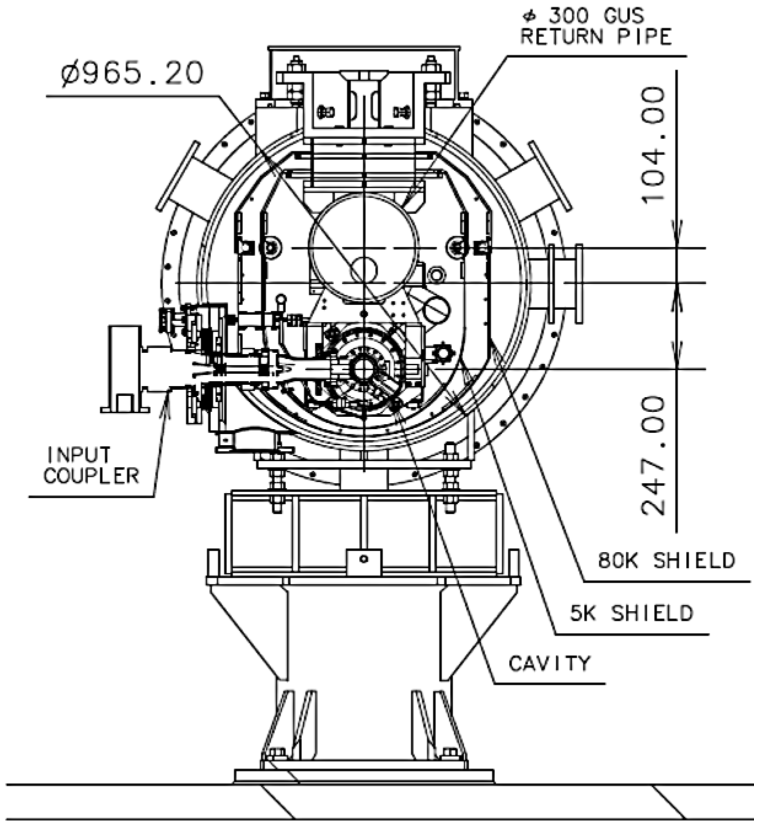
ILC Cryomodule



Cryomodule Cross-sectional View



(a) Module-C for FNAL cavity



(b) Module-A for KEK cavity

- 2 K two-phase superfluid helium pipeline
- 2 K gas helium return line.
- 5 K shield line and 80 K or 40 K shield line.
- Cold mass at 2 K is enclosed with two thermal shields
- No issues with pressure vessel compliance for these lines.

Pressure Vessel Compliance

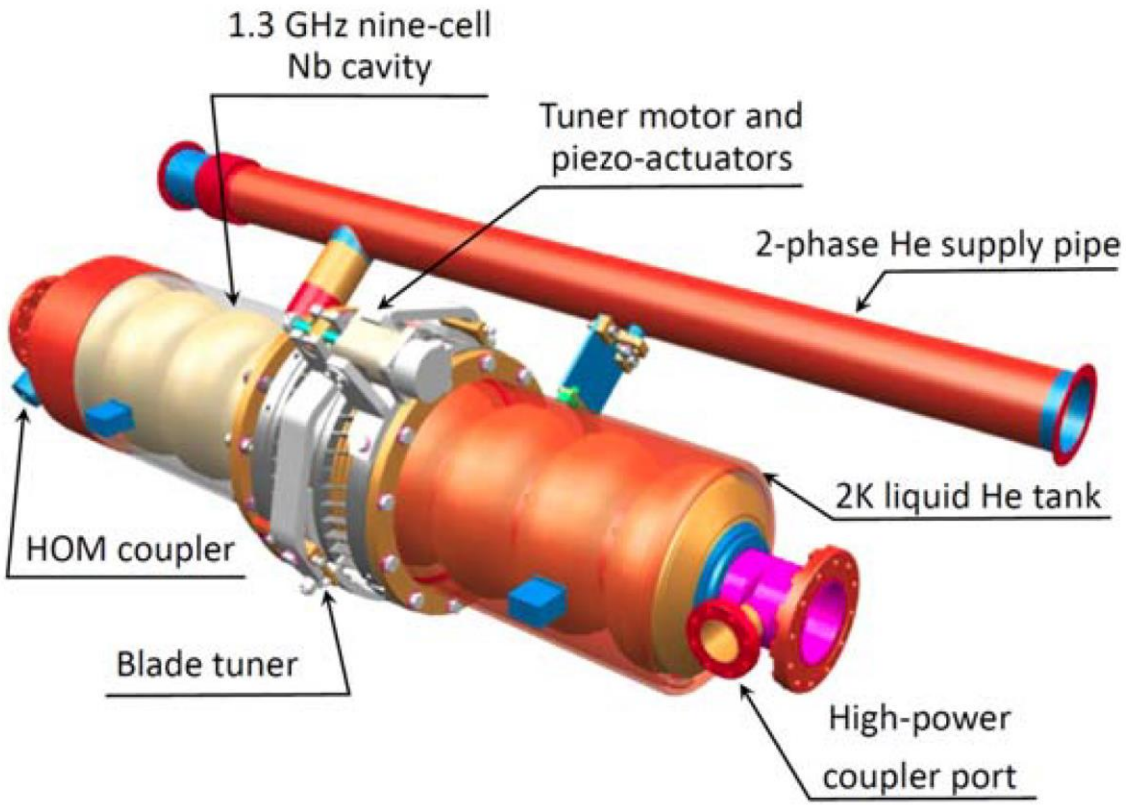
1.3 GHz 9-Cell SRF Cavity Assembly

Niobium SRF Cavity with its Titanium jacket is considered as a pressure vessel

Design must be cleared by high-pressure gas safety authorities

Determine maximum allowable stress and buckling pressure using ANSYS.

Nb and NbTi not listed as a material for high pressure vessel design



Cited from: The International Linear Collider: A Global Project, arXiv:1903.01629 [hep-ex]

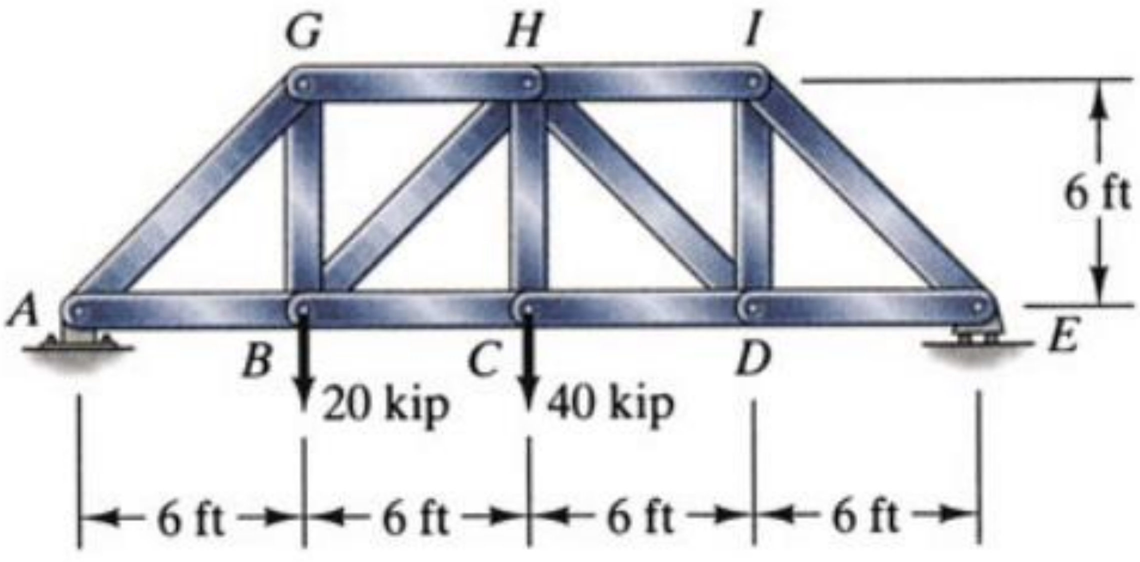
Steps Involved in Preparing Documentation

- Description of the SRF cavity assembly.
- Mechanical properties of the materials at room and in liquid helium temperatures.
- Mechanical properties of welded joints like Nb-Nb, Nb-Ti, Ti-Ti welds etc.
- Stress and buckling analysis of the cavity assembly using CAE software at maximum allowable working pressure (0.2 MPa) and tuner displacement.
- Cavity fabrication information.
- Pressure test and examination reports.
- Documentation to summarize above items to be submitted to the high-pressure gas safety authority.

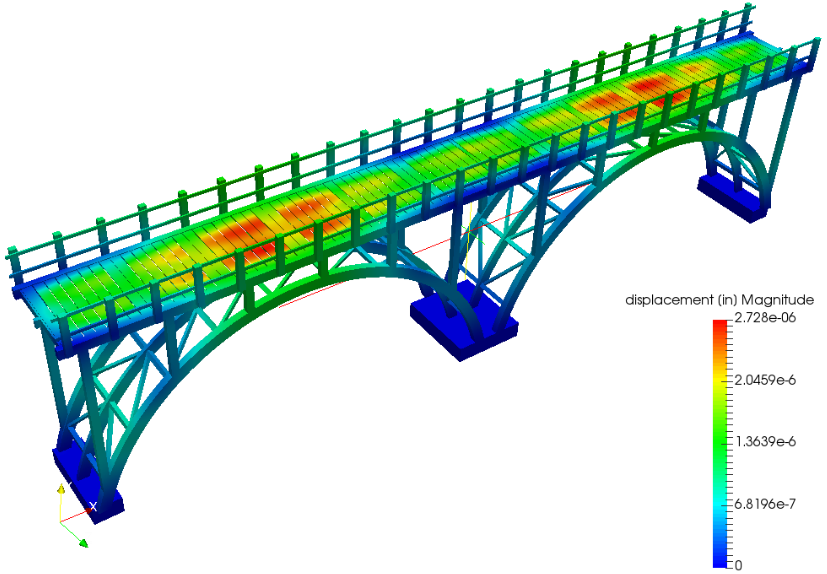
Structural Analysis

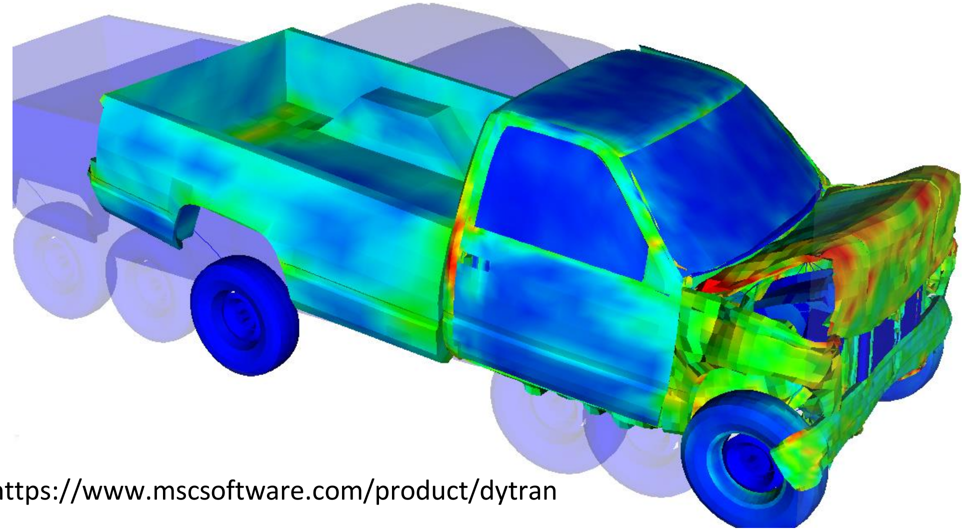
Structural Analysis

- Structural analysis determines the integrity of structure to withstand loads.
- Structural analysis incorporates the fields of mechanics and dynamics as well as the many failure theories.
- From a theoretical perspective, the primary goal of structural analysis is the computation of deformations, internal forces, and stresses.
- In practice, structural analysis reveals the structural performance of the engineering design and ensures the soundness of structural integrity in design without dependence on direct testing.



A. Bedford and K.M Leichti, Mechanics of materials, second edition, Springer.





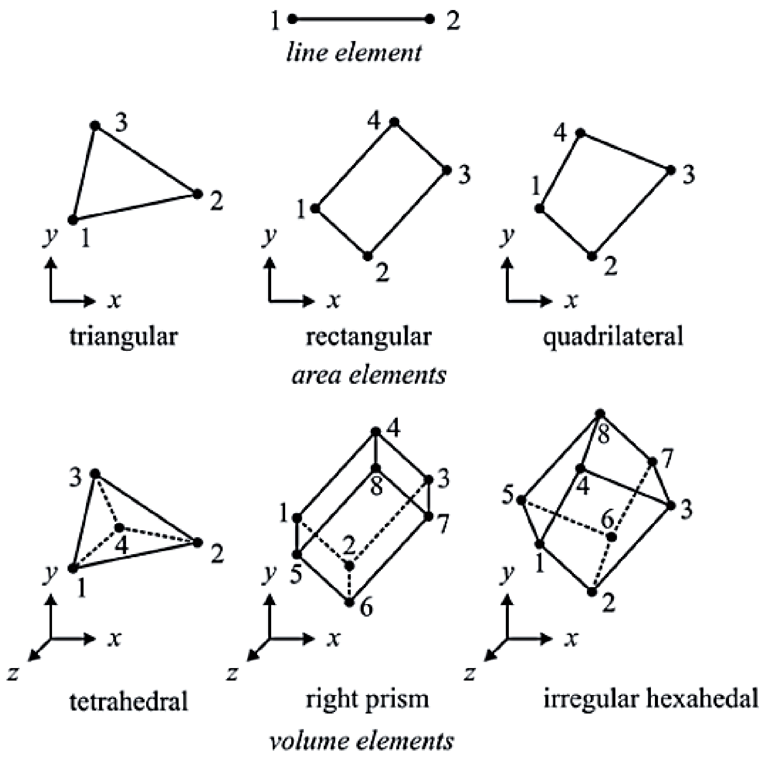
https://www.mscsoftware.com/product/dytran

Finite Element Analysis

- FEA is a powerful computational technique for approximate solutions to a variety of "real-world" problems.
- It relies on decomposition of domain (solid, liquid or gas) into a finite number of sub-domains.
- Reduces the problem into finite number of unknowns by dividing the domain into elements and by expressing the unknown field variable in terms of the assumed approximating functions within each element.

Finite Element Analysis

- Discretization of the domain into finite number of sub-domains.
- Selection of interpolating functions.
- Development of the elemental matrix of the sub-domain.
- Assembly of the element matrices for each subdomain to obtain global matrix.
- Imposition of boundary conditions.
- Solution of equations.
- Additional data-analysis if necessary.



Cited from: E Madenci and I Guven., The finite element method and applications in engineering using ANSYS®, second edition, Springer 2015.

Direct Approach: Linear Spring System

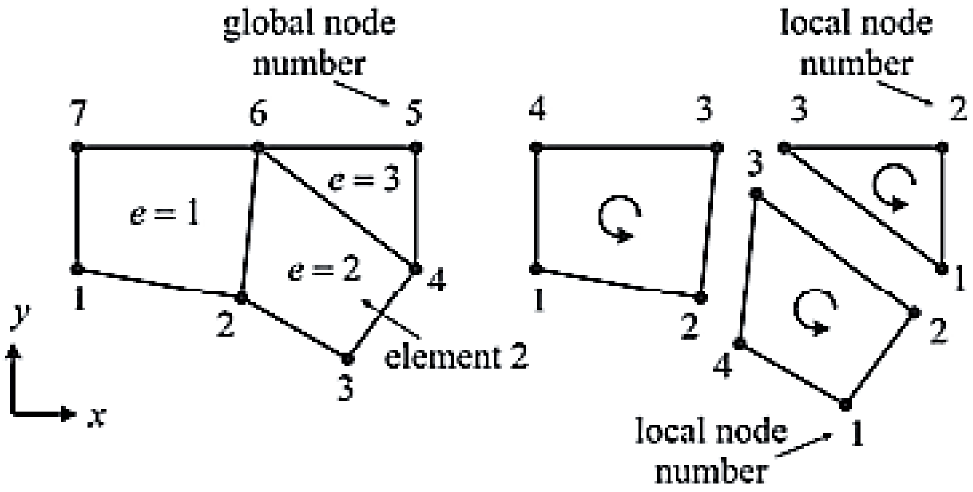
- Suitable for simpler problems but fundamental step of a typical FEA.
- Global system of equations can be cast into:

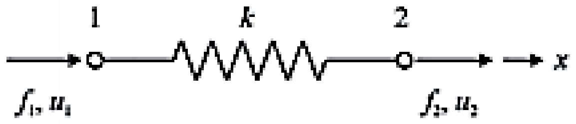
$$\boxed{\mathrm{Ku} = \mathrm{F}}$$

$$u = u_1 - u_2 \quad \rightarrow \quad f_1 = \mathrm{ku} = \mathrm{k}(u_1 - u_2) \quad \rightarrow \quad f_2 = -f_1$$

$$\boxed{\begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix} \; or \; k^{(e)} u^{(e)} = f^{(e)}}$$

Here, $u^{(e)}$ is the vector of nodal unknowns representing displacement, $k^{(e)}$ is the element (stiffness) matrix and $f^{(e)}$ is the element (force) vector. The stiffness matrix can be represented in its indicial form as $k_{ij}^{(e)}$.

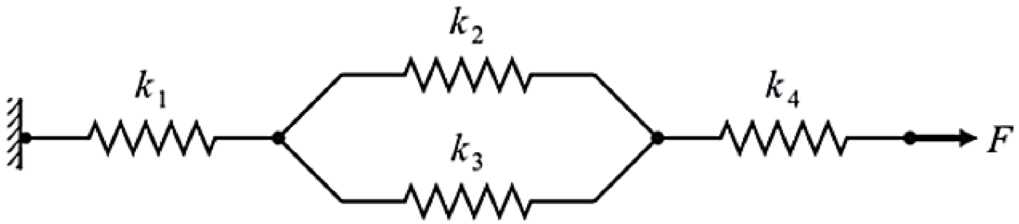


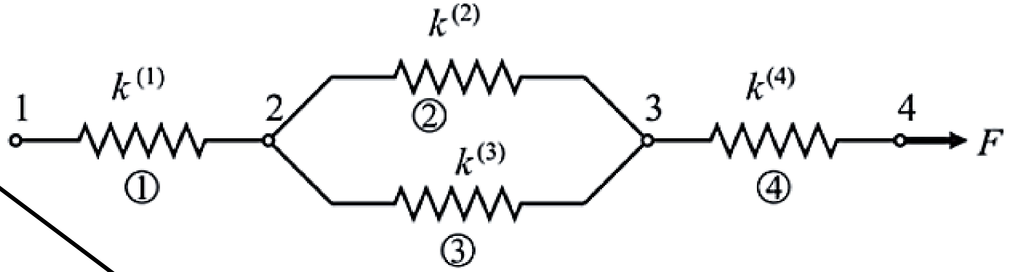


Cited from: E Madenci and I Guven., The finite element method and applications in engineering using ANSYS®, second edition, Springer 2015.

Assembly of Global System Matrix

- Engineering problem model with finite elements requires the assembly of element characteristic (stiffness) matrices and element right-hand-side vectors.





$$K = \sum_{e=1}^{E} k^{(e)}$$

$$K\boxed{u} = F$$

$$F = \sum_{e=1}^{E} f^{(e)}$$

$$K = \begin{bmatrix} k_{11}^{(1)} & k_{12}^{(1)} & 0 & 0 \\ k_{21}^{(e)} & \begin{pmatrix} k_{22}^{(1)} + k_{11}^{(2)} \\ + k_{11}^{(3)} \end{pmatrix} & \left(k_{12}^{(2)} + k_{12}^{(3)}\right) & 0 \\ 0 & \left(k_{21}^{(2)} + k_{21}^{(3)}\right) & \begin{pmatrix} k_{22}^{(2)} + k_{22}^{(3)} \\ + k_{11}^{(4)} \end{pmatrix} & k_{12}^{(e)} \\ 0 & 0 & k_{21}^{(e)} & k_{22}^{(4)} \end{bmatrix}$$

$$u = \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = \begin{bmatrix} u_1^{(1)} \\ u_2^{(1)} = u_1^{(2)} = u_1^{(3)} \\ u_2^{(2)} = u_2^{(3)} = u_1^{(4)} \\ u_2^{(4)} \end{bmatrix}$$

$$F = \begin{bmatrix} f_1 \\ f_2 \\ f_3 \\ f_4 \end{bmatrix} = \begin{bmatrix} f_1^{(1)} \\ f_2^{(1)} + f_1^{(2)} + f_1^{(3)} \\ f_2^{(2)} + f_2^{(3)} + f_1^{(4)} \\ f_2^{(4)} \end{bmatrix}$$

Cited from: E Madenci and I Guven., The finite element method and applications in engineering using ANSYS®, second edition, Springer 2015.

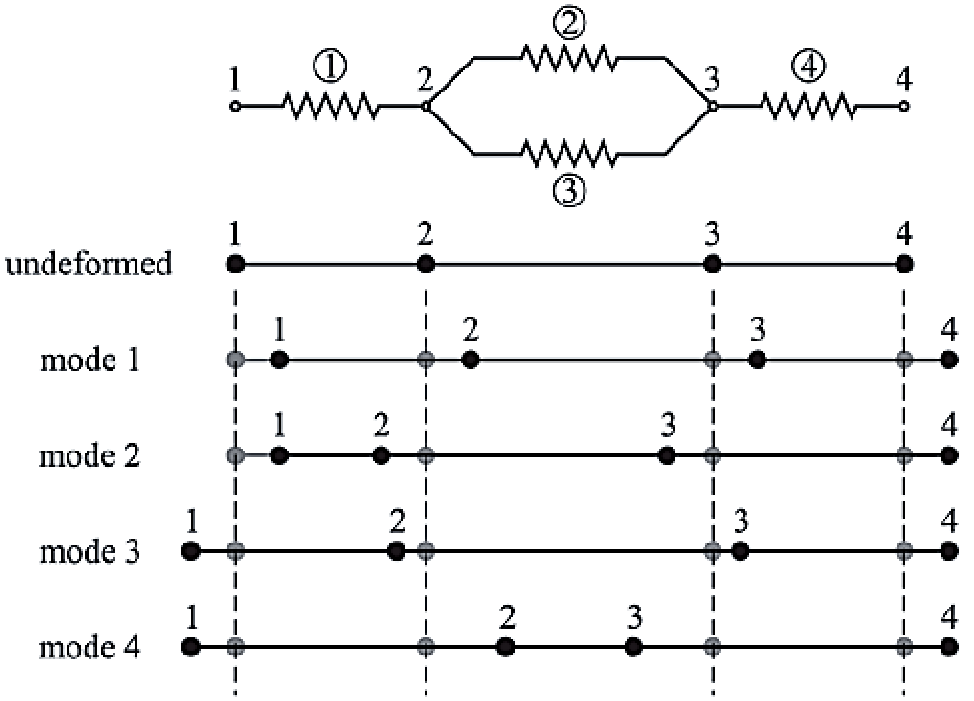
Solution of Global Matrix

- For the specific values $k_{11}^{(e)} = k_{22}^{(e)} = k^{(e)}$ and $k_{21}^{(e)} = k_{12}^{(e)} = -k^{(e)}$, the global system matrix becomes

$$K = k^{(e)} \begin{bmatrix} 1 & -1 & 0 & 0 \\ -1 & 3 & -2 & 0 \\ 0 & -2 & 3 & -1 \\ 0 & 0 & -1 & 1 \end{bmatrix}$$

- The eigenvalues are $\lambda_1 = 0, \lambda_2 = 2, \lambda_3 = 3 - \sqrt{5}$, and $\lambda_3 = 3 - \sqrt{5}$. The corresponding eigenvectors are

$$\mathrm{u}^{(1)} = \begin{bmatrix} 1 \\ 1 \\ 1 \\ 1 \end{bmatrix}, \mathrm{u}^{(2)} = \begin{bmatrix} 1 \\ -1 \\ -1 \\ 1 \end{bmatrix}, \mathrm{u}^{(3)} = \begin{bmatrix} 1 \\ 2-\sqrt{5} \\ -2+\sqrt{5} \\ 1 \end{bmatrix}, \mathrm{u}^{(4)} = \begin{bmatrix} 1 \\ 2+\sqrt{5} \\ -2-\sqrt{5} \\ 1 \end{bmatrix}$$



Cited from: E Madenci and I Guven., The finite element method and applications in engineering using ANSYS®, second edition, Springer 2015.

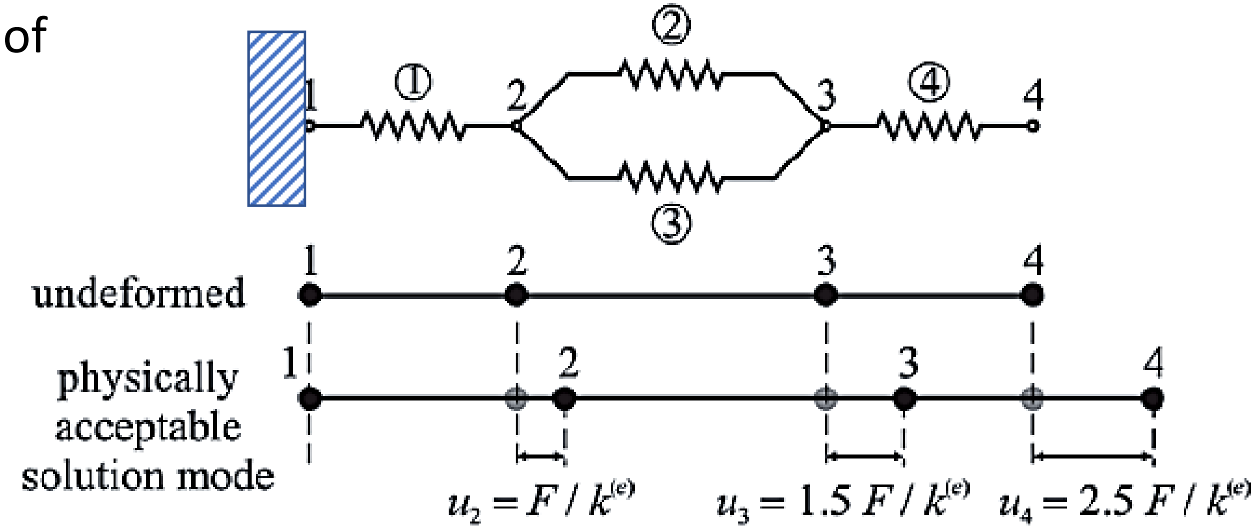
Applying Boundary Condition

- Possible to obtain unique solution with boundary conditions.

- If $u_1 = 0$, the nodal force f_1 still remains an unknown and the corresponding nodal forces have values of $f_2 = f_3 = 0$, and $f_4 = F$.

$$k^{(e)} \begin{bmatrix} 1 & -1 & 0 & 0 \\ -1 & 3 & -2 & 0 \\ 0 & -2 & 3 & -1 \\ 0 & 0 & -1 & 1 \end{bmatrix} \begin{bmatrix} 0 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = \begin{bmatrix} f_1 \\ 0 \\ 0 \\ F \end{bmatrix}$$

$$u_2 = \frac{F}{k^{(e)}}, u_3 = \frac{3}{2}\frac{F}{k^{(e)}}, u_4 = \frac{5}{2}\frac{F}{k^{(e)}}$$



Cited from: E Madenci and I Guven., The finite element method and applications in engineering using ANSYS®, second edition, Springer 2015.

Stress-strain Relationship in ANSYS

- Stress-strain relationship in ANSYS for linear materials is defined as:

$$\boxed{\{\sigma\} = D\{\varepsilon^{el}\}}$$

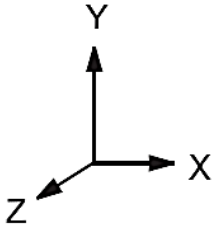
$\{\sigma\}$ = Stress vector = $[\sigma_x\ \sigma_y\ \sigma_z\ \sigma_{xy}\ \sigma_{yz}\ \sigma_{xz}]^T$

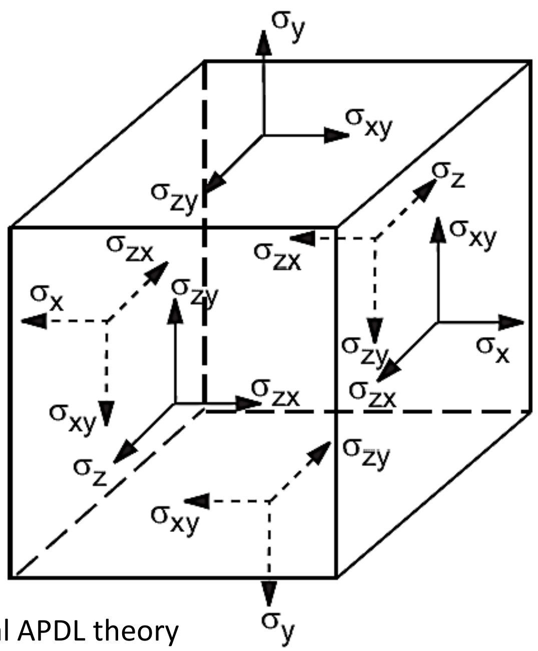
[D] = Elasticity or elastic stiffness matrix or stress-strain matrix.

$\{\varepsilon^{el}\}$ = $\{\varepsilon\}$ − $\{\varepsilon^{th}\}$ = elastic strain vector

$\{\varepsilon\}$ = total strain vector = $[\varepsilon_x\ \varepsilon_y\ \varepsilon_z\ \varepsilon_{xy}\ \varepsilon_{yz}\ \varepsilon_{xz}]^T$

$\{\varepsilon^{th}\}$ = thermal strain vector





Cited from: P. Kohnke, ANSYS® Mechanical APDL theory reference, ANSYS® Inc.

- Also,

$$\{\varepsilon\} = \{\varepsilon^{th}\} + [D]^{-1}\{\sigma\},$$

- For the 3-D case, the thermal strain vector is:

$$\{\varepsilon^{th}\} = \Delta T[\alpha_x^{se}\ \alpha_y^{se}\ \alpha_z^{se}\ 0\ 0\ 0]^T,$$

α_x^{se} = Secant coefficient of thermal expansion in the x direction

$\Delta T = T - T_{ref}$

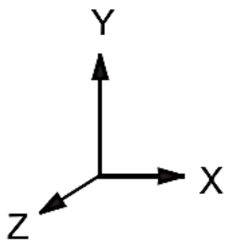
T = current temperature at the point in question

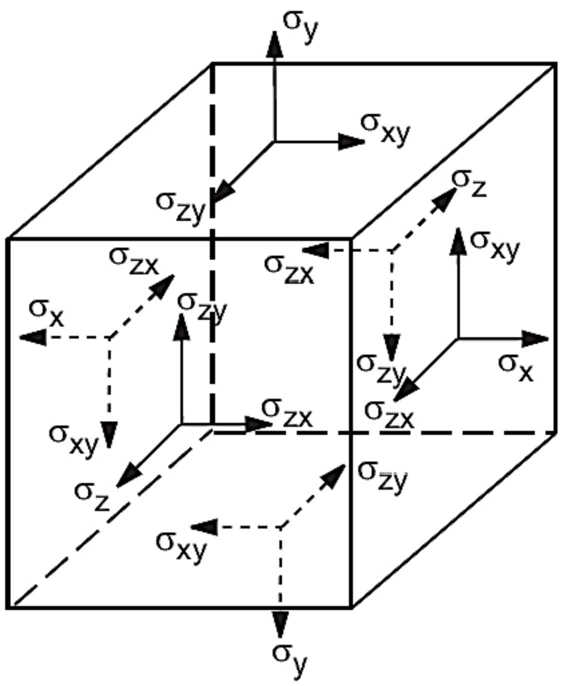
T_{ref} = reference strain free temperature

Stress-strain Relationship in ANSYS

- The flexibility or compliance matrix for the linear material is:

$$
[\mathrm{D}]^{-1} = \begin{bmatrix}
\frac{1}{E_x} & \frac{-\nu_{xy}}{E_x} & \frac{-\nu_{xz}}{E_x} & 0 & 0 & 0 \\
\frac{-\nu_{yx}}{E_y} & \frac{1}{E_y} & \frac{-\nu_{yz}}{E_y} & 0 & 0 & 0 \\
\frac{-\nu_{zx}}{E_z} & \frac{-\nu_{zy}}{E_z} & \frac{1}{E_z} & 0 & 0 & 0 \\
0 & 0 & 0 & \frac{1}{G_{xy}} & 0 & 0 \\
0 & 0 & 0 & 0 & \frac{1}{G_{yz}} & 0 \\
0 & 0 & 0 & 0 & 0 & \frac{1}{G_{xz}}
\end{bmatrix}
$$





Where the typical terms are:

E_x = Young's modulus in the x direction, ν_{xy} = major Poisson's ratio

ν_{yx} = minor Poisson's ratio, G_{xy} = Shear modulus in the xy plane

Cited from: P. Kohnke, ANSYS® Mechanical APDL theory reference, ANSYS® Inc.

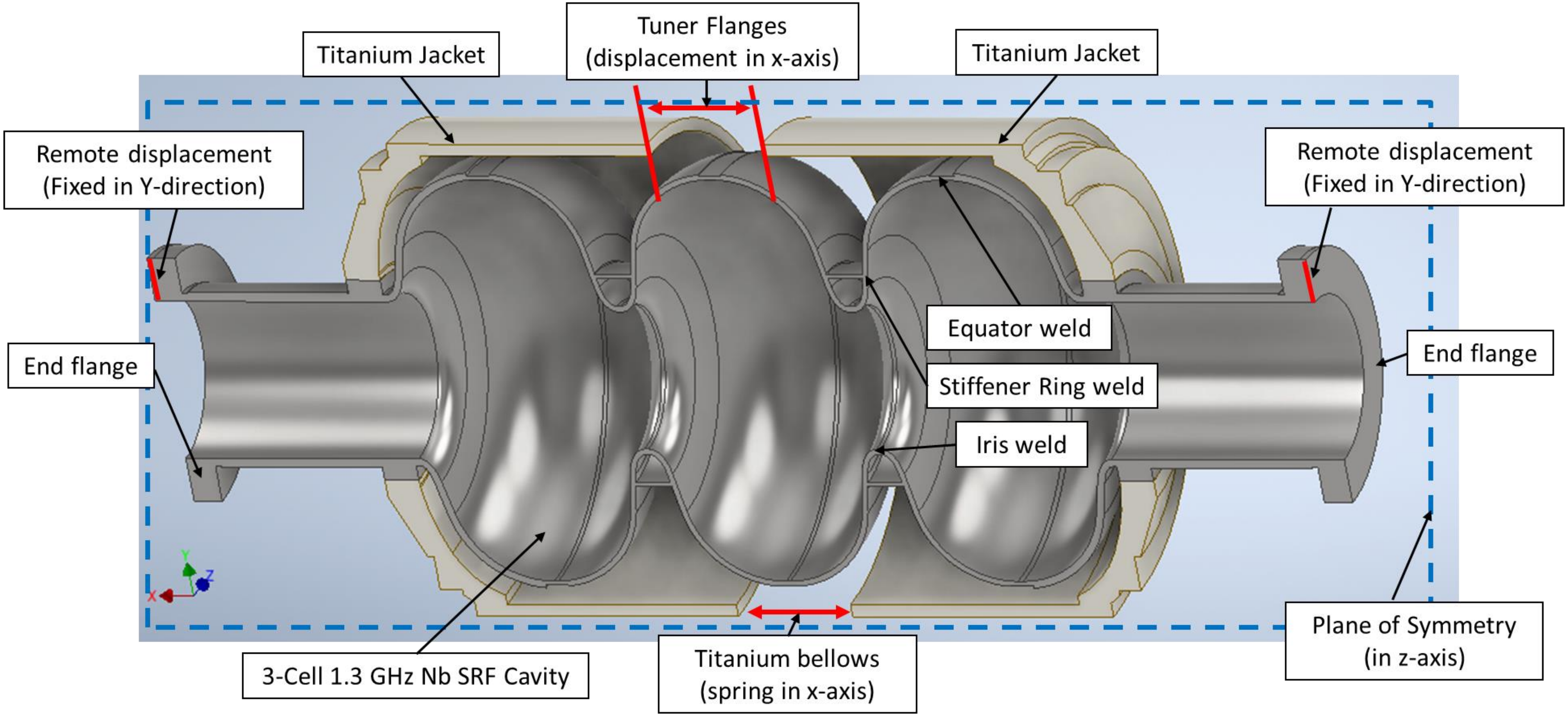
For isotropic materials ($E_x = E_y = E_z$ and $\nu_{xy} = \nu_{yz} = \nu_{xz}$)

1.3-GHz 3-Cell SRF Cavity Stress Analysis

Note

- The results presented here are just an example and the results presented here are not supposed to be considered for 9-Cell SRF Cavity structure.

SRF Cavity Assembly

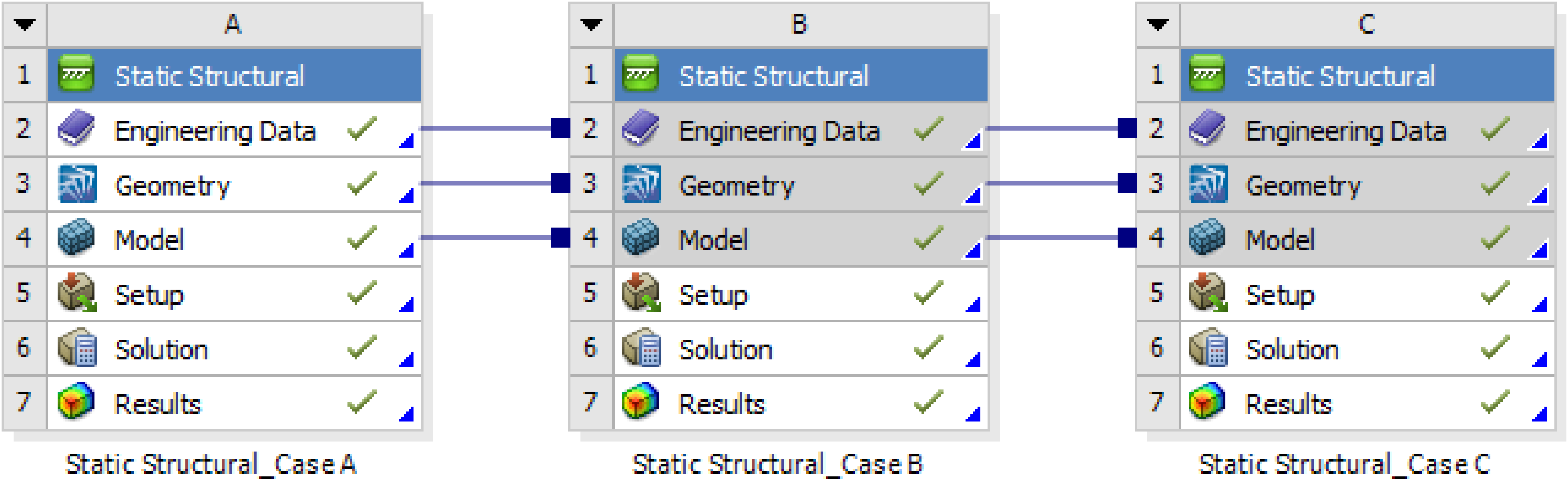


Mechanical Properties

Properties	Unit	Niobium	Titanium
Young's Modulus	GPa	103	107
Poisson's Ratio	-	0.38	0.32
Coefficient of Thermal Expansion	$\times 10^{-6}$ /℃	4.87	5.22

- Elastic properties of the materials will be considered for structure and buckling analysis.
- Coefficient of thermal expansion necessary to include contraction of material during cooldown.

Case Structure in ANSYS



Case Study	Pressure	Tuner Displacement	Temperature
Case A	0.2 MPa	0.65 mm	40 °C
Case B	0.2 MPa	0.65 mm	-271.4 °C
Case C	0.2 MPa	3 mm	-271.4 °C

Engineering Data

	A
1	Static Structural
2	Engineering Data ✓
3	Geometry ✓
4	Model ✓
5	Setup ✓
6	Solution ✓
7	Results ✓

Niobium Properties

Outline of Schematic A2, B2, C2: Engineering Data

	A	B	C	D	E
1	Contents of Engineering Data			Source	Description
2	Material				
3	NIOBIUM			E	"Equation of State and Strength Properties of Selected Materials". Steinberg D.J. LLNL. Feb 1991
4	TITANIUM			E	"Equation of State and Strength Properties of Selected Materials". Steinberg D.J. LLNL. Feb 1991
*	Click here to add a new material				

Properties of Outline Row 3: NIOBIUM

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	8590	kg m^-3		
4	Isotropic Secant Coefficient of Thermal Expansion				
5	Coefficient of Thermal Expansion	4.87E-06	C^-1		
6	Isotropic Elasticity				
7	Derive from	Young's ...			
8	Young's Modulus	1.03E+05	MPa		
9	Poisson's Ratio	0.38			
10	Bulk Modulus	1.4306E+11	Pa		
11	Shear Modulus	3.7319E+10	Pa		

Titanium Properties

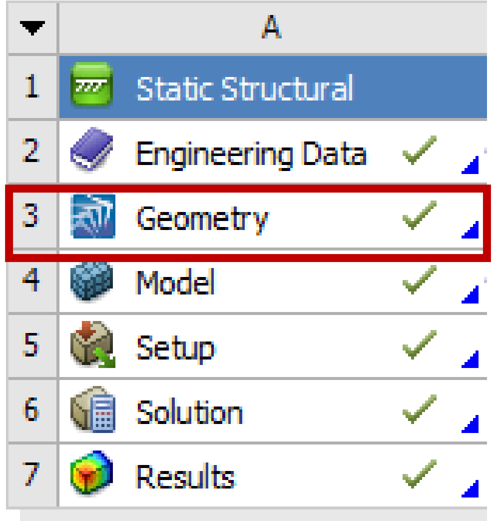
Outline of Schematic A2, B2, C2: Engineering Data

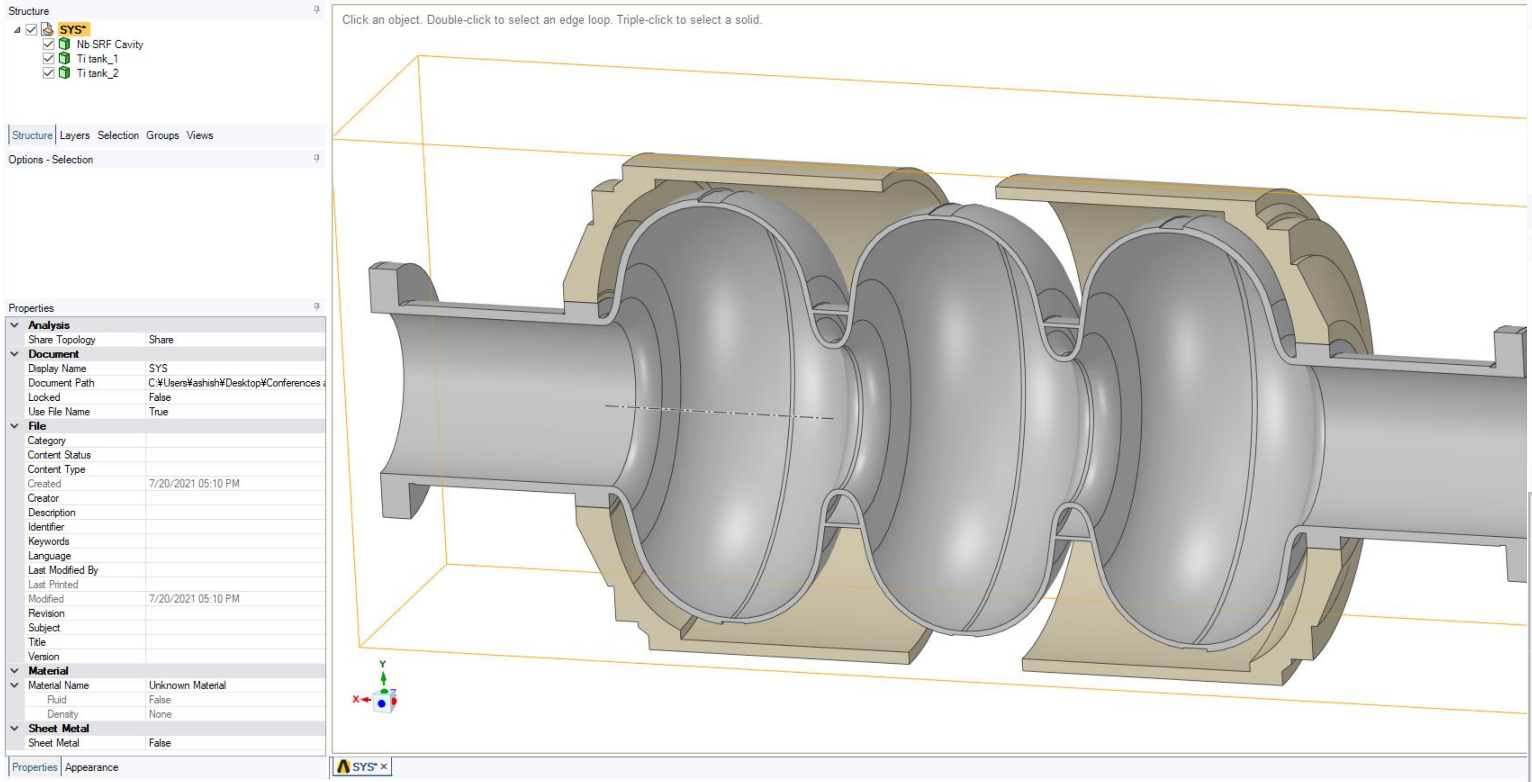
	A	B	C	D	E
1	Contents of Engineering Data			Source	Description
2	Material				
3	NIOBIUM			E	"Equation of State and Strength Properties of Selected Materials". Steinberg D.J. LLNL. Feb 1991
4	TITANIUM			E	"Equation of State and Strength Properties of Selected Materials". Steinberg D.J. LLNL. Feb 1991
*	Click here to add a new material				

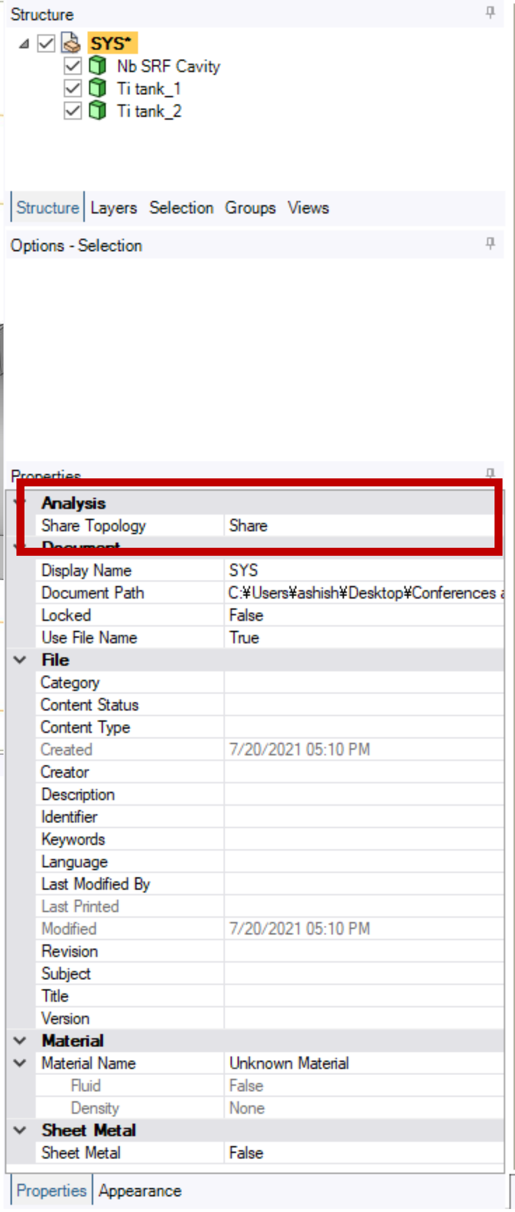
Properties of Outline Row 4: TITANIUM

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	4510	kg m^-3		
4	Isotropic Secant Coefficient of Thermal Expansion				
5	Coefficient of Thermal Expansion	5.22E-06	C^-1		
6	Isotropic Elasticity				
7	Derive from	Young's ...			
8	Young's Modulus	1.07E+05	MPa		
9	Poisson's Ratio	0.32			
10	Bulk Modulus	9.9074E+10	Pa		
11	Shear Modulus	4.053E+10	Pa		

Geometry

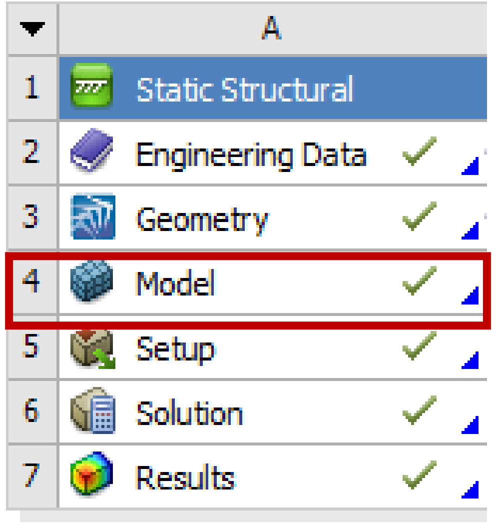


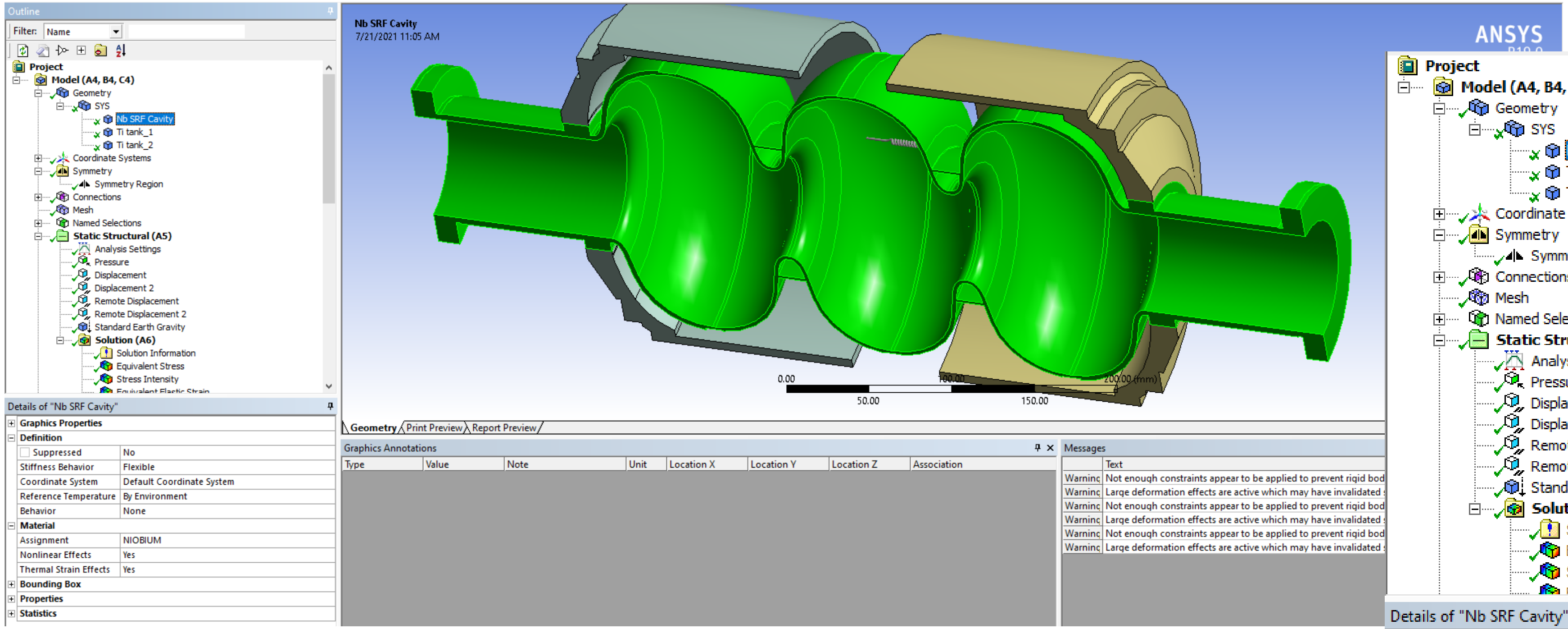


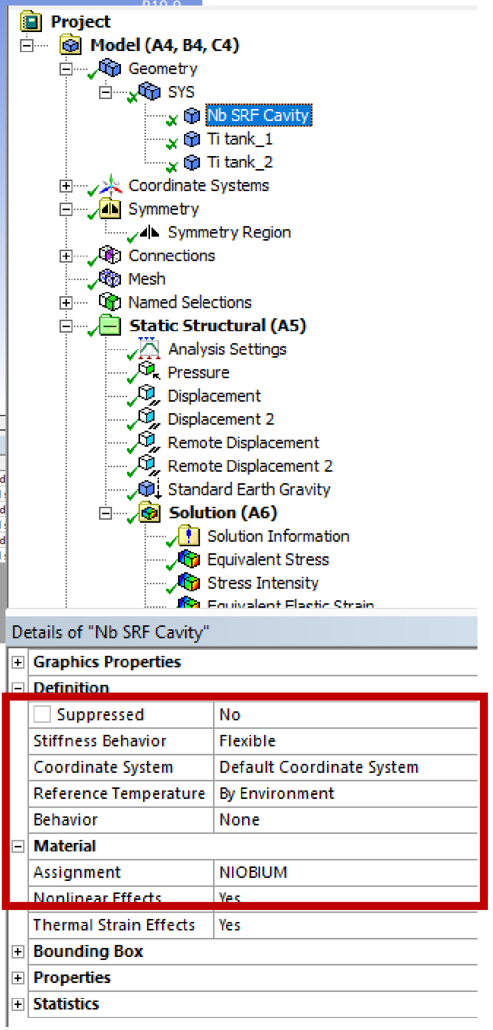


- CAD Model is uploaded to Geometry
- Topology is set to Share

Model – Material Assignment

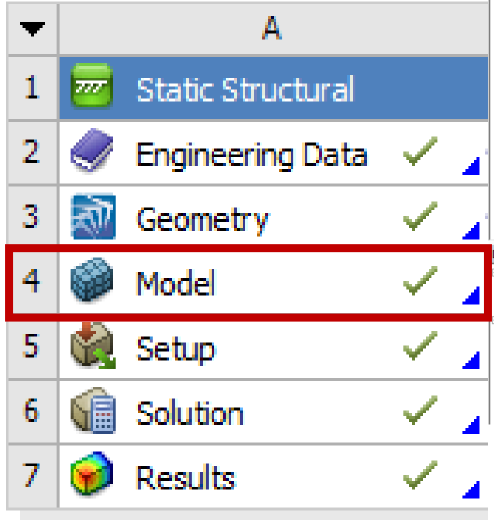


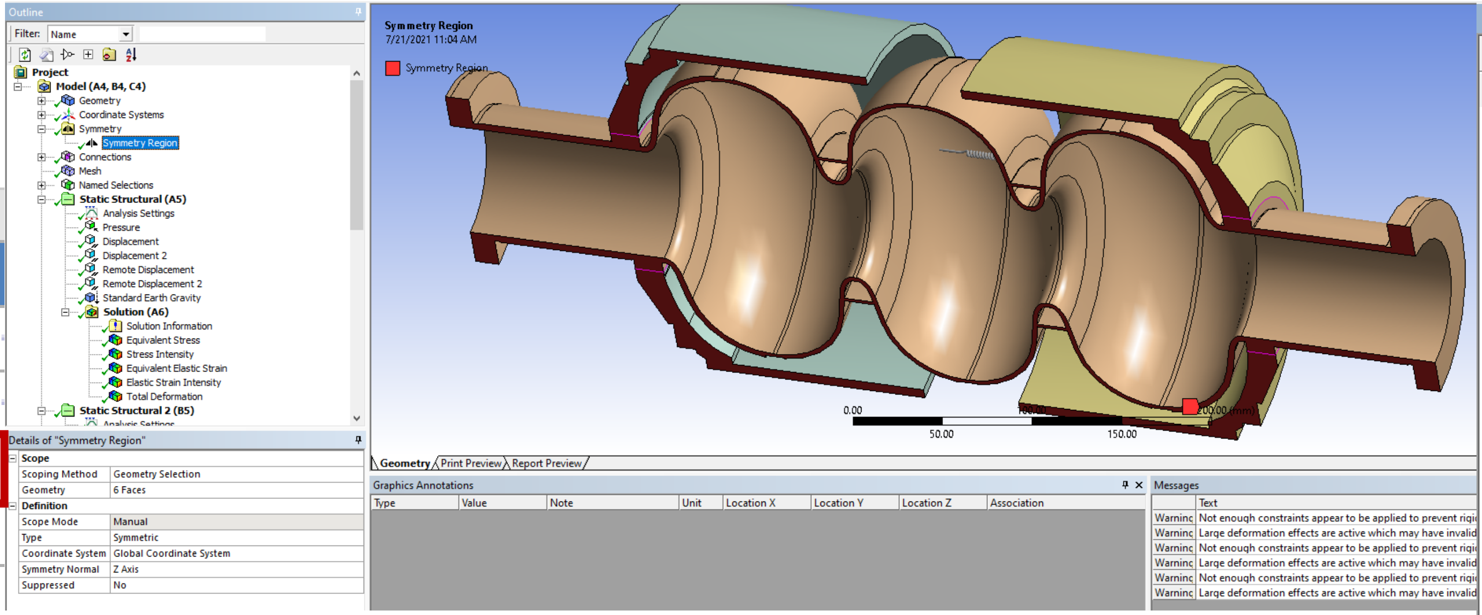


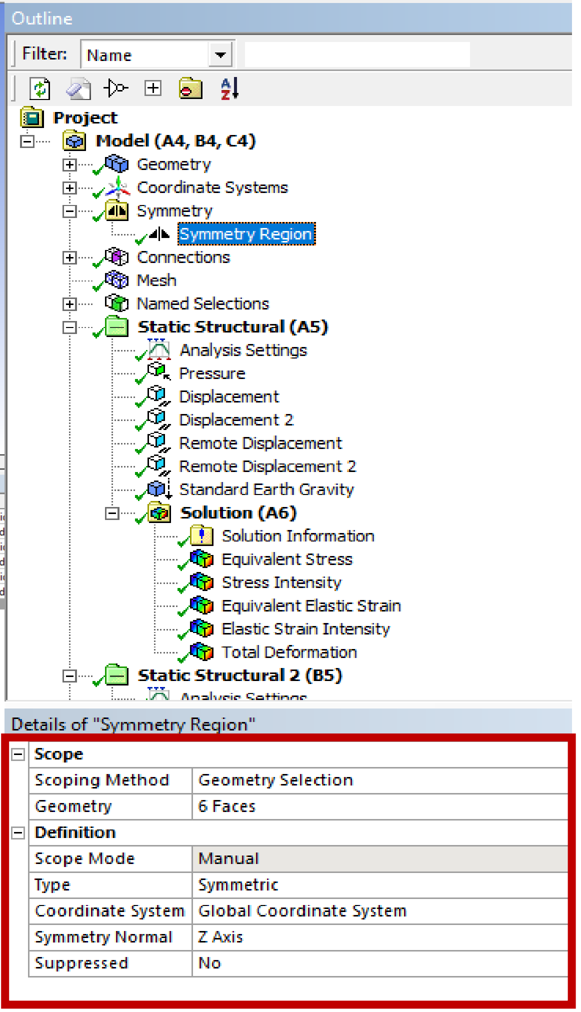


- Material is assigned to each geometry.
- Stiffness behavior is set to Flexible.

Model – Symmetry

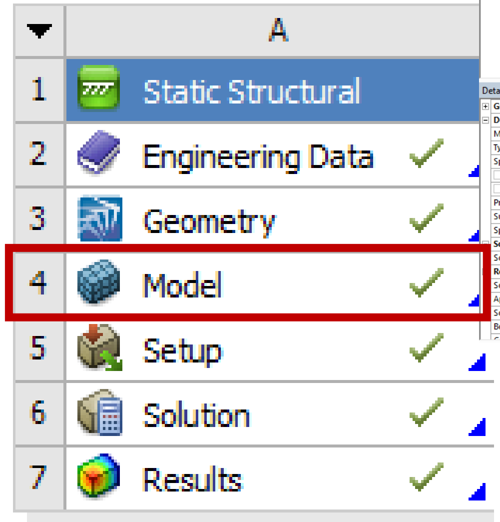


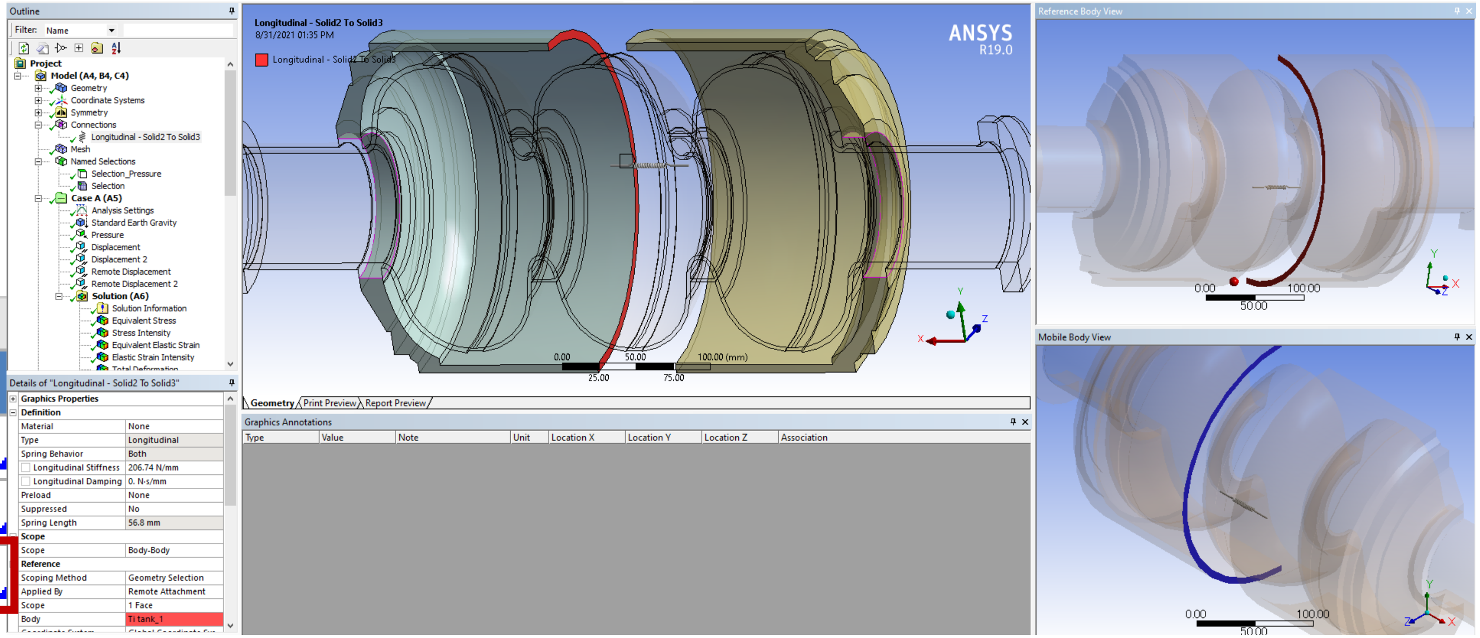


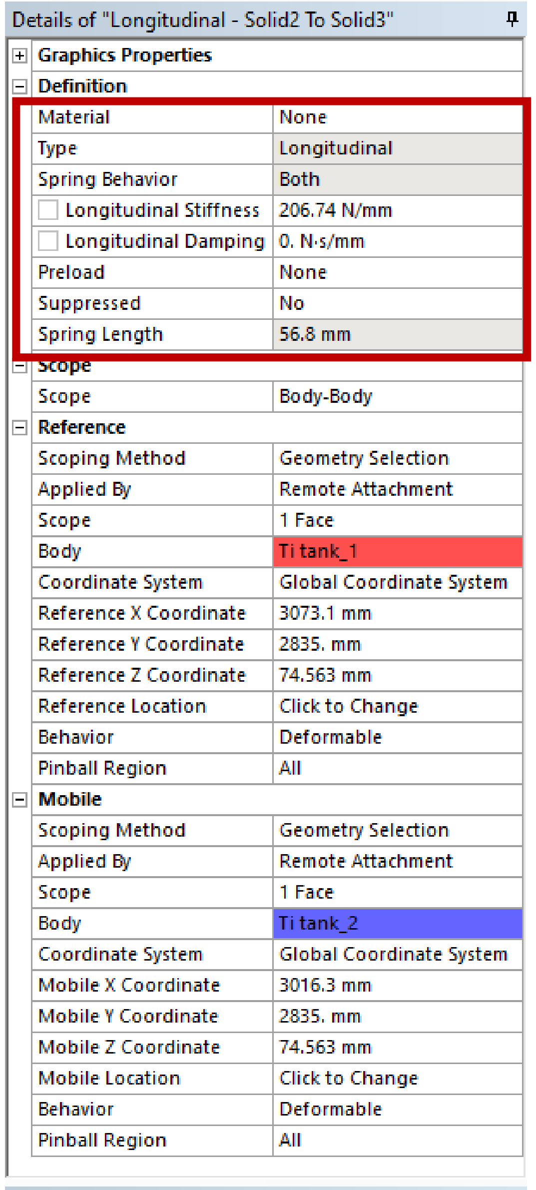


- <u>Model symmetry</u> along z-axis defined.
- Faces in Red specifies the symmetry region.

Model – Virtual Spring





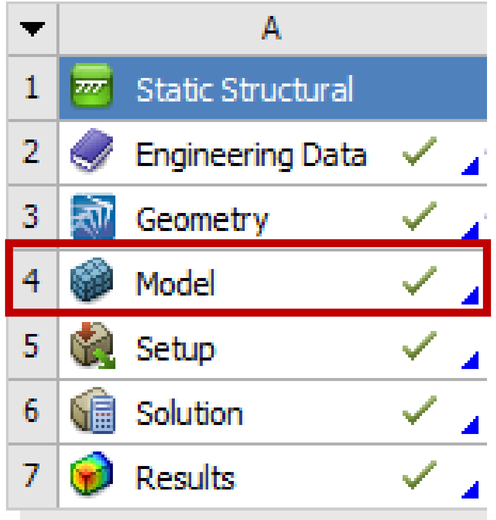


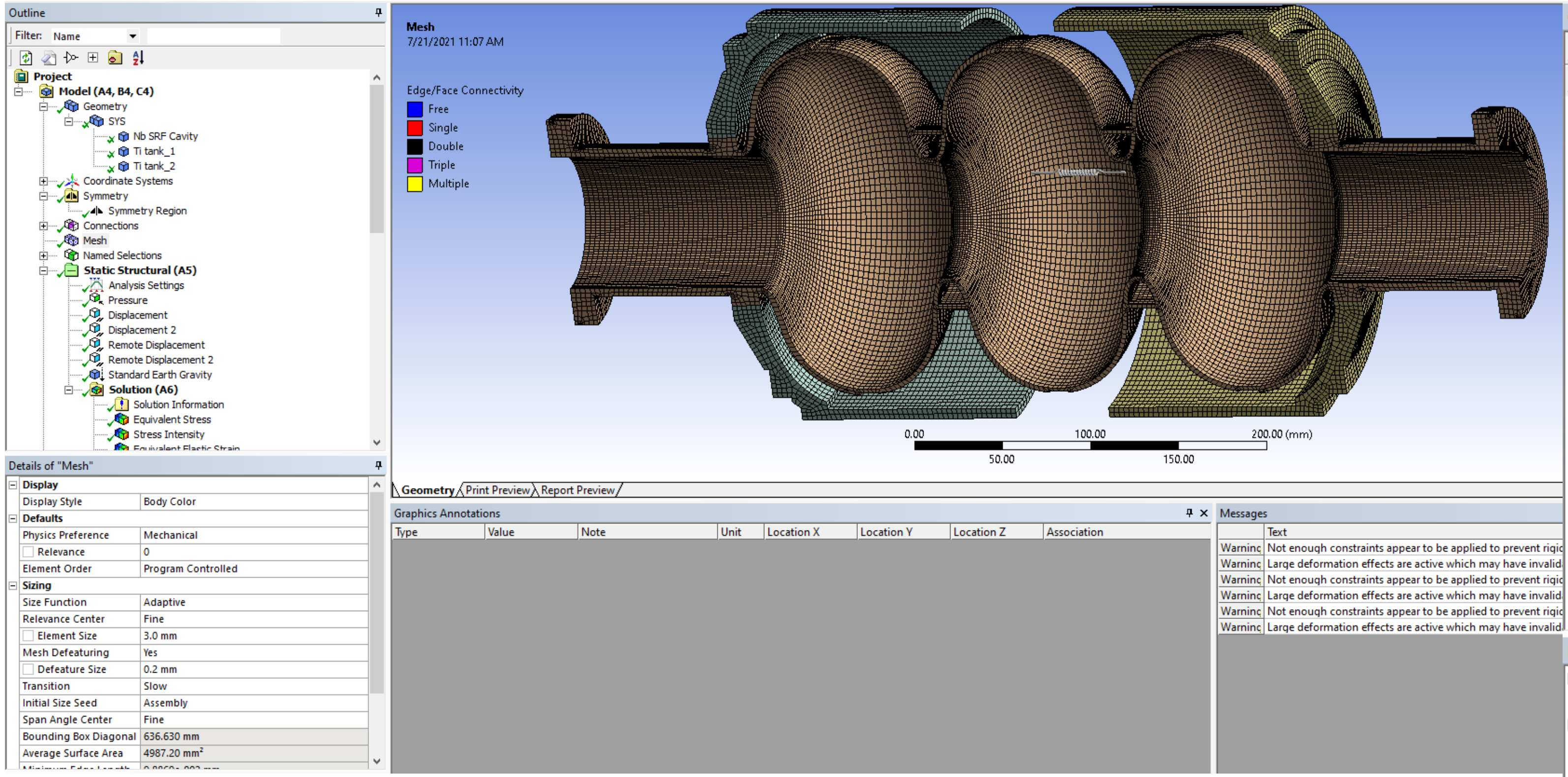
Details of "Longitudinal - Solid2 To Solid3"

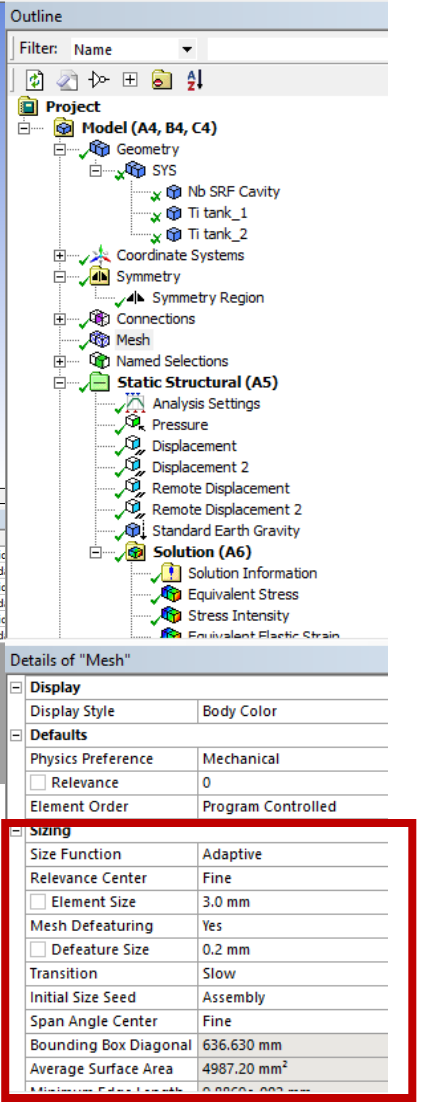
Graphics Properties	
Definition	
Material	None
Type	Longitudinal
Spring Behavior	Both
Longitudinal Stiffness	206.74 N/mm
Longitudinal Damping	0. N·s/mm
Preload	None
Suppressed	No
Spring Length	56.8 mm
Scope	
Scope	Body-Body
Reference	
Scoping Method	Geometry Selection
Applied By	Remote Attachment
Scope	1 Face
Body	Ti tank_1
Coordinate System	Global Coordinate System
Reference X Coordinate	3073.1 mm
Reference Y Coordinate	2835. mm
Reference Z Coordinate	74.563 mm
Reference Location	Click to Change
Behavior	Deformable
Pinball Region	All
Mobile	
Scoping Method	Geometry Selection
Applied By	Remote Attachment
Scope	1 Face
Body	Ti tank_2
Coordinate System	Global Coordinate System
Mobile X Coordinate	3016.3 mm
Mobile Y Coordinate	2835. mm
Mobile Z Coordinate	74.563 mm
Mobile Location	Click to Change
Behavior	Deformable
Pinball Region	All

- A virtual spring replaces Ti bellow between Ti jacket with stiffness of 206.7 N/mm.

Model – Meshing

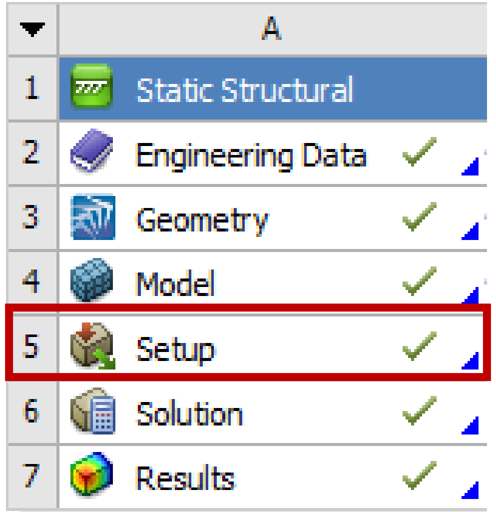


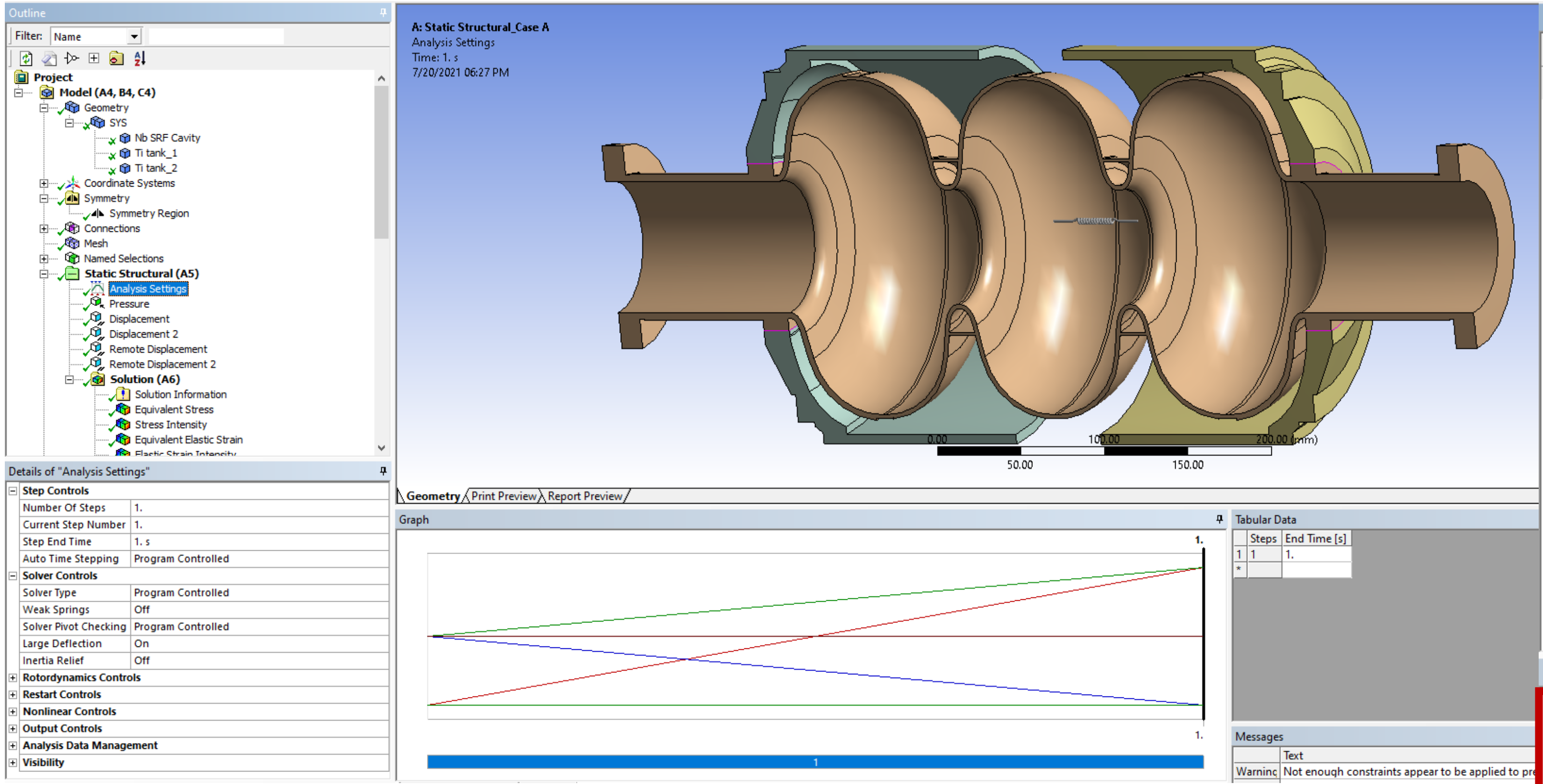


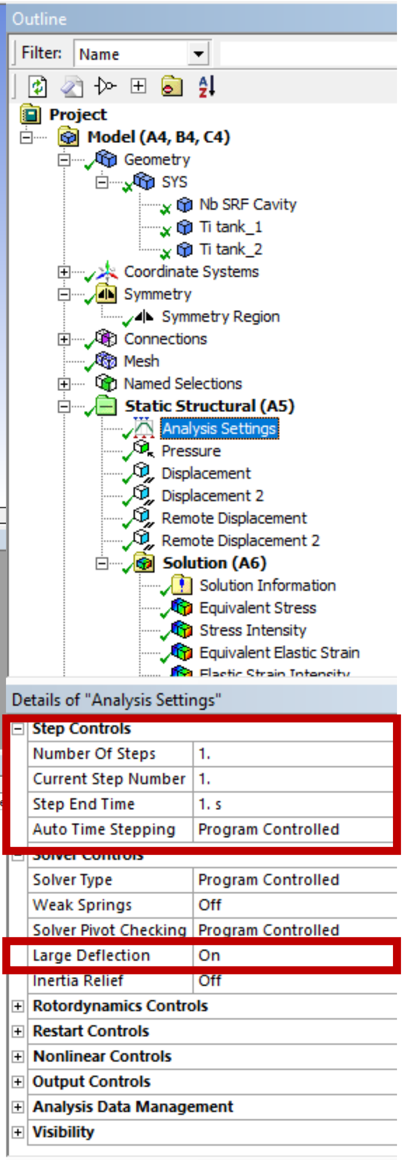


- Model is discretized in this step.
- Meshing generated using <u>Adaptive</u> size control.

Setup – Analysis Settings

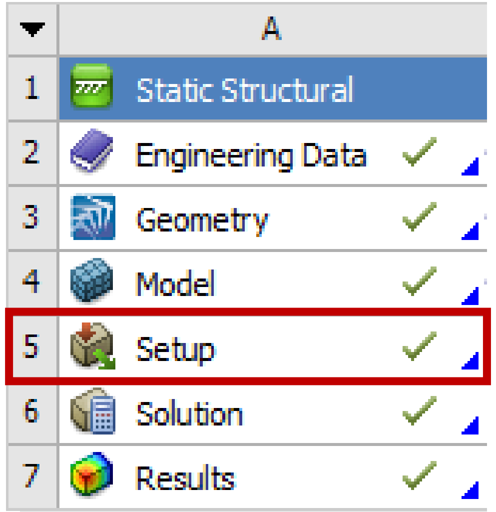


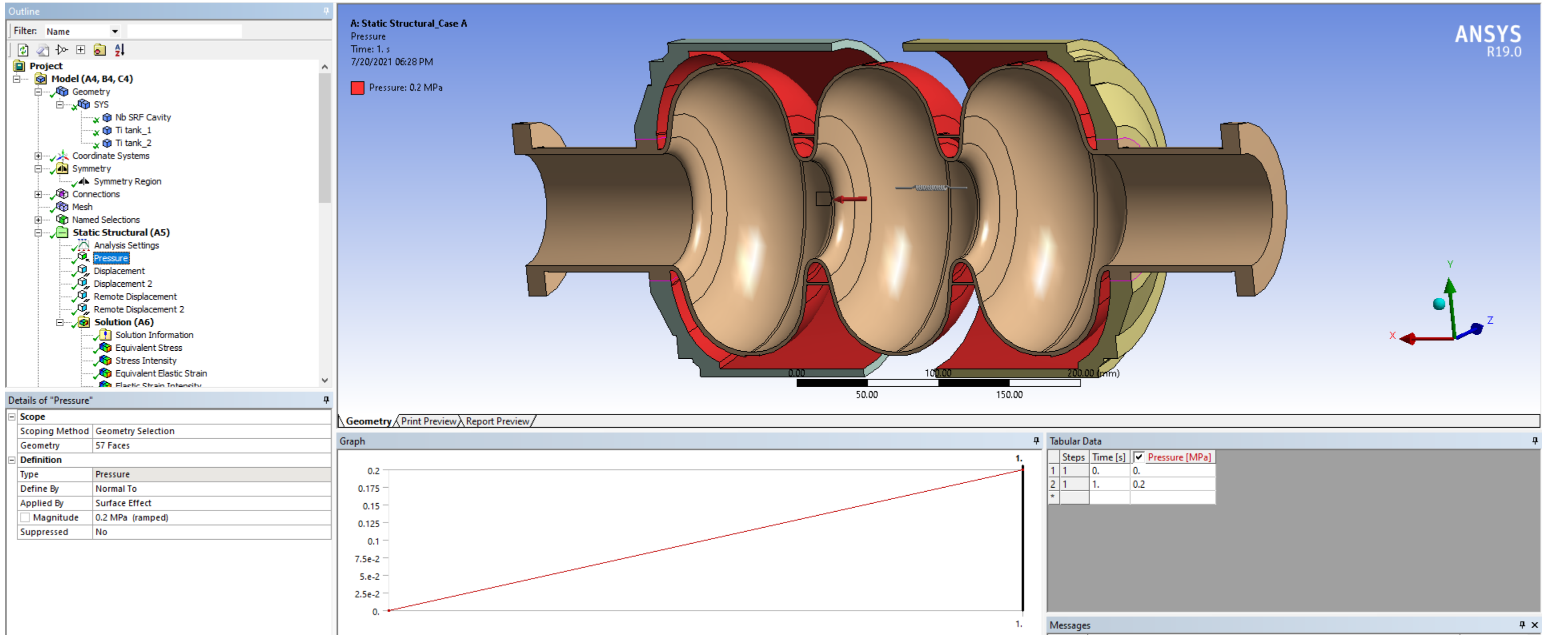


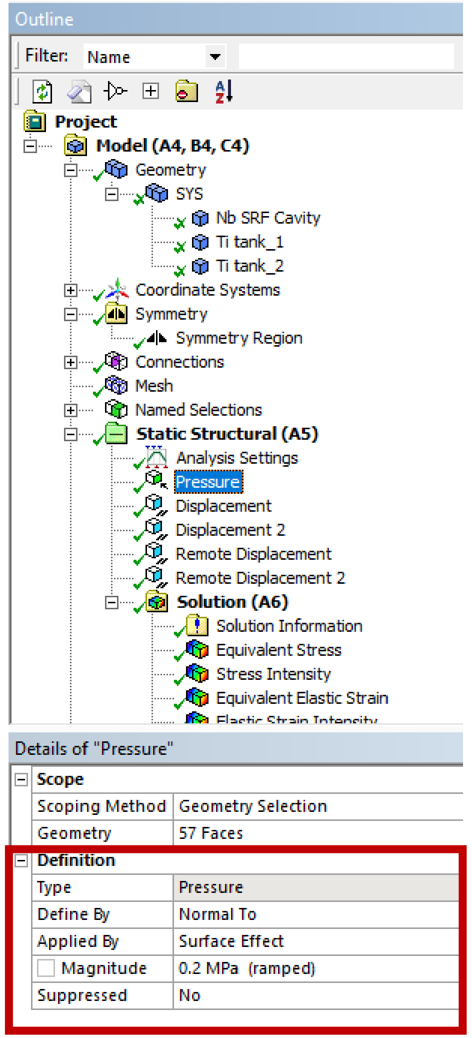


- Set number of steps, sub-steps etc.
- Set Large deflection to ON, when non-linear analysis is expected.

Setup – Pressure Boundary Condition

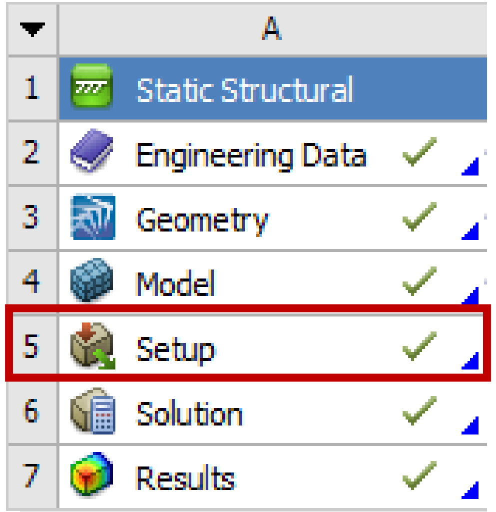


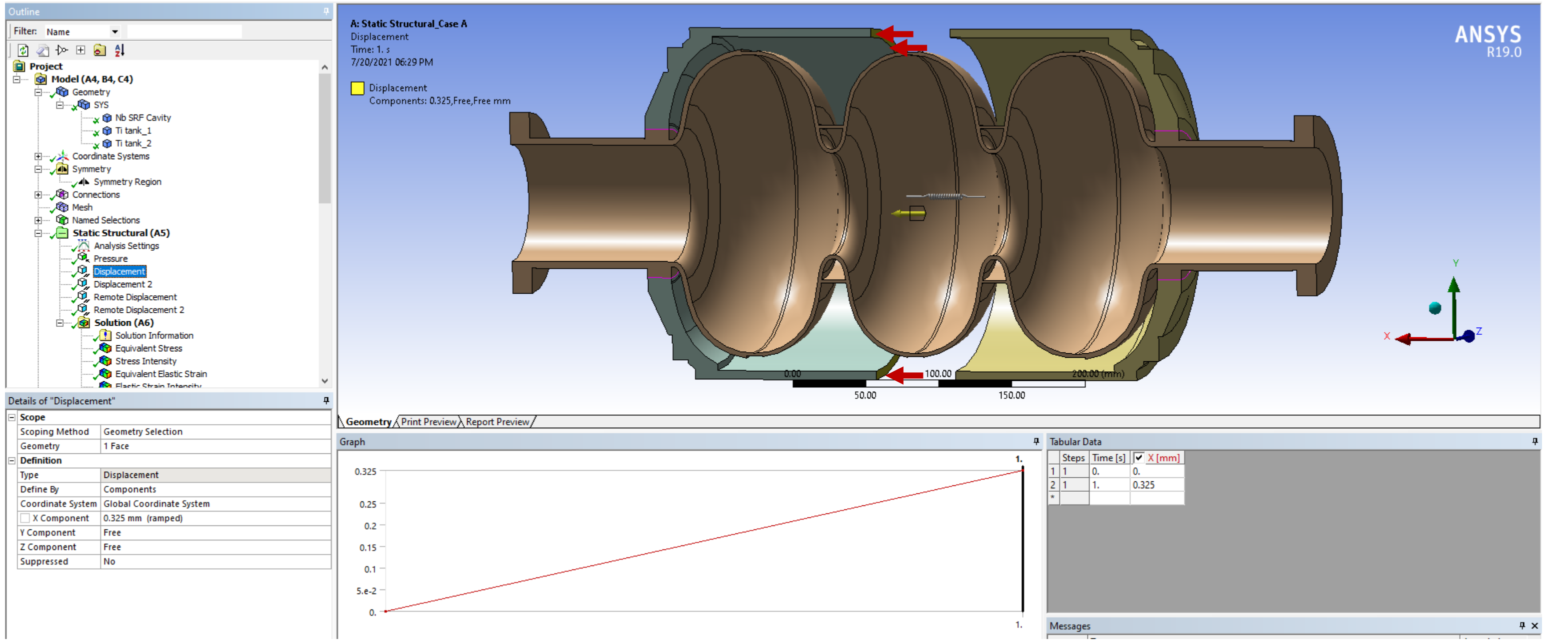


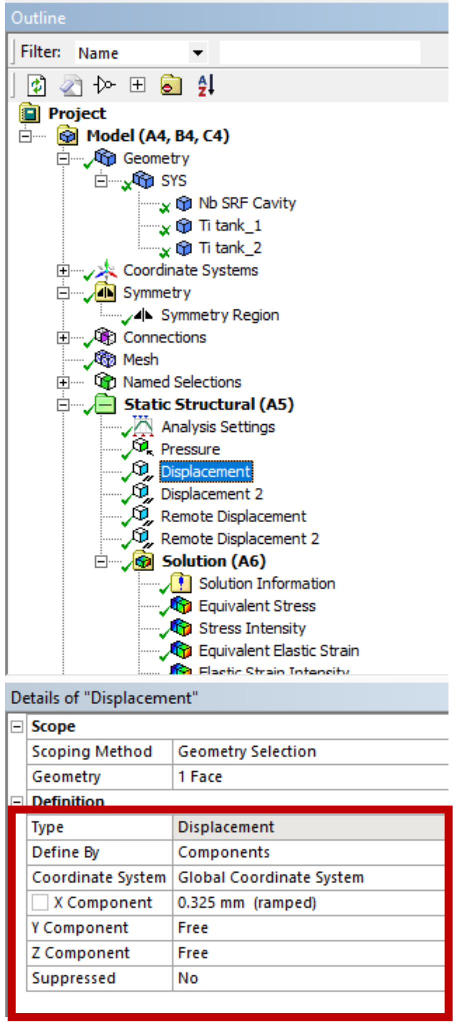


- Pressure boundary condition (BC) to simulate pressurized helium between SRF cavity and Ti jacket at 0.2 MPa.
- Surfaces in RED are provided with pressure boundary condition.

Setup – Displacement Boundary Condition

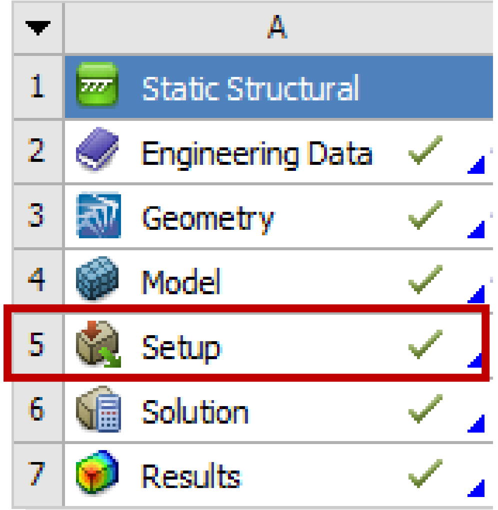


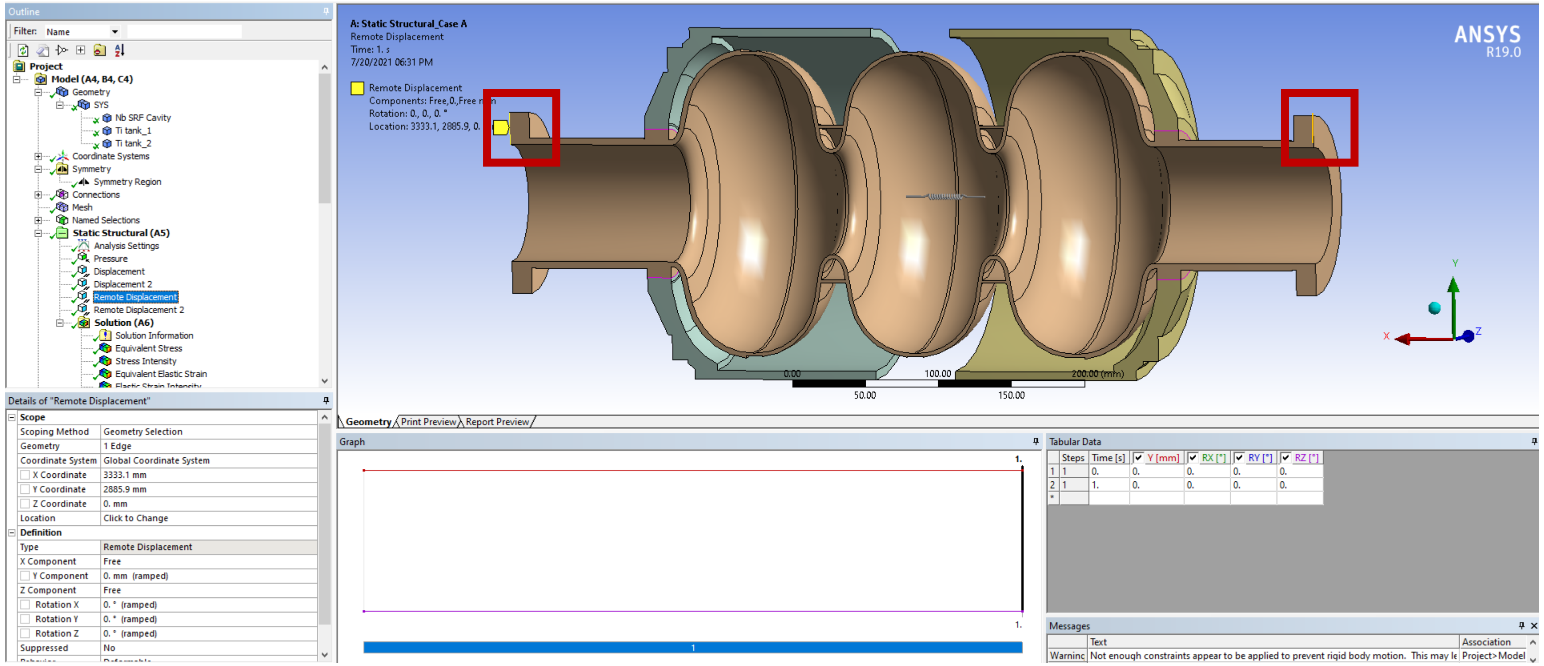


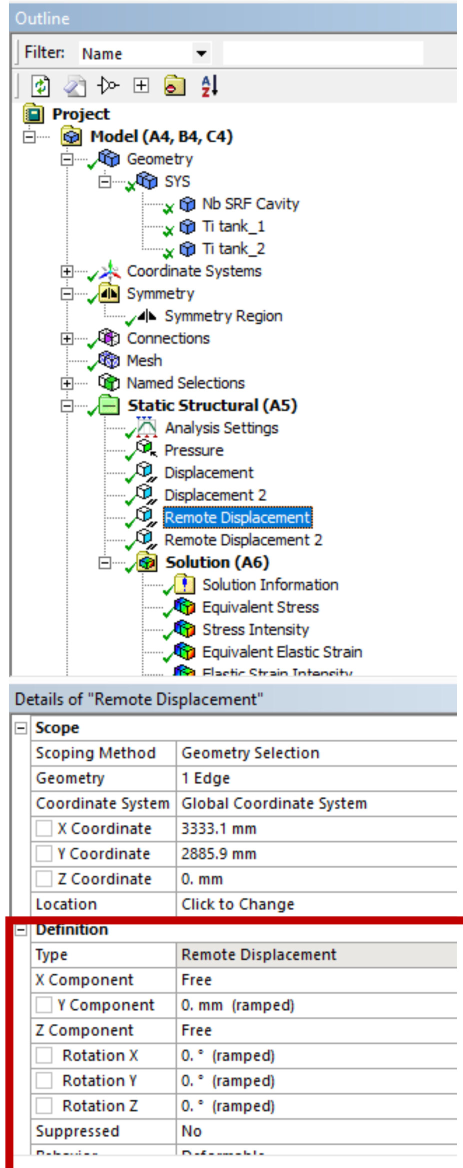


- Displacement BC to simulate tuner movement.
- Displacement in X-direction and movement in other axis are not constrained.

Setup – Remote Displacement Boundary Condition

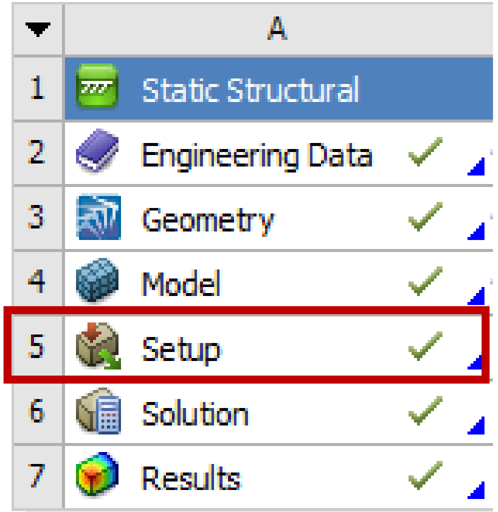


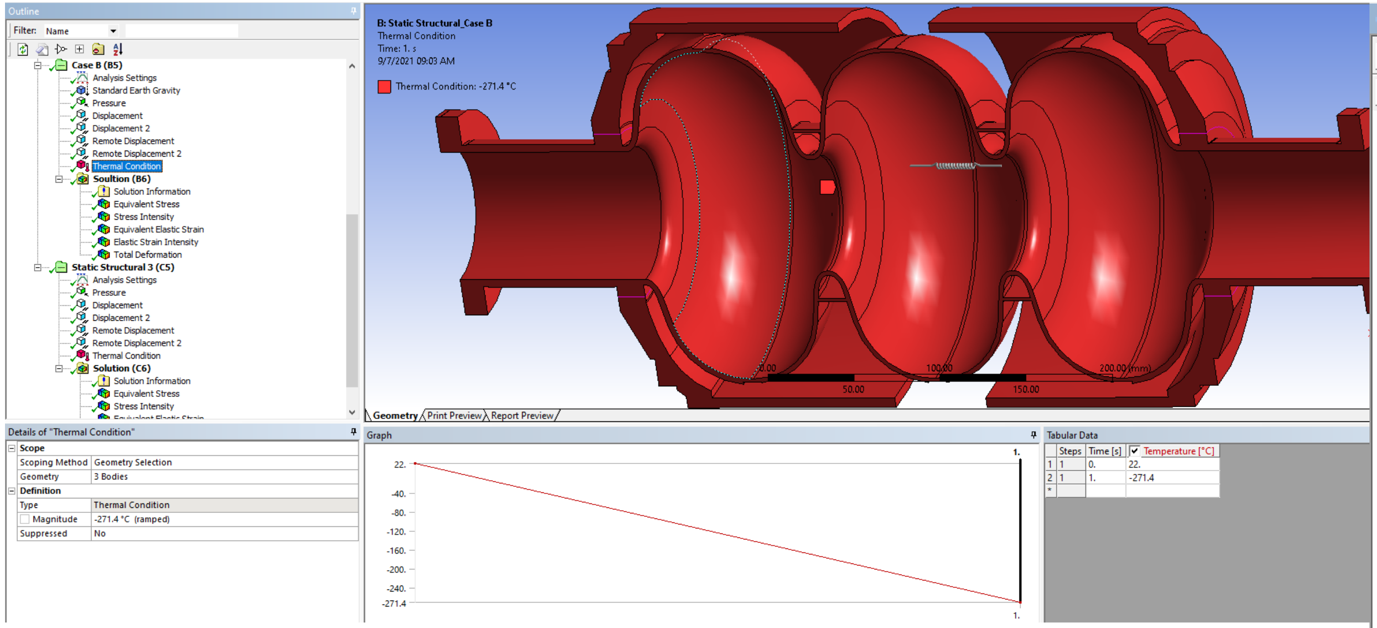


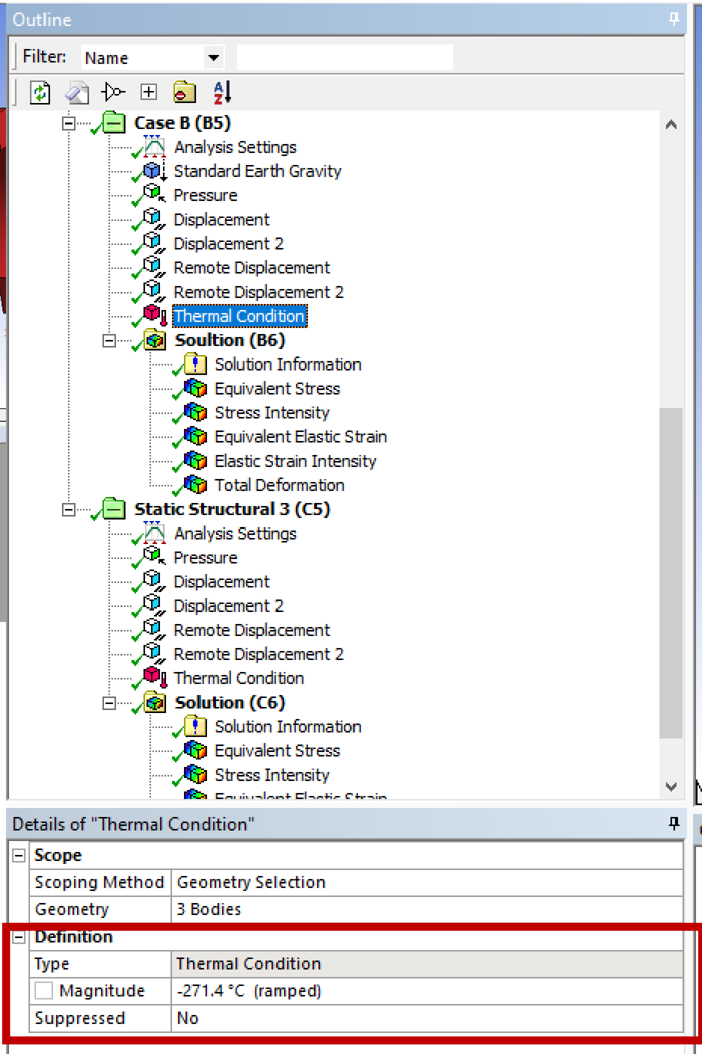


- Remote Displacement BC is similar to displacement but with more control on DOF.
- Line contact on end flanges movement is fixed in Y-axis and no rotation is allowed along any direction.

Setup – Temperature Boundary Condition



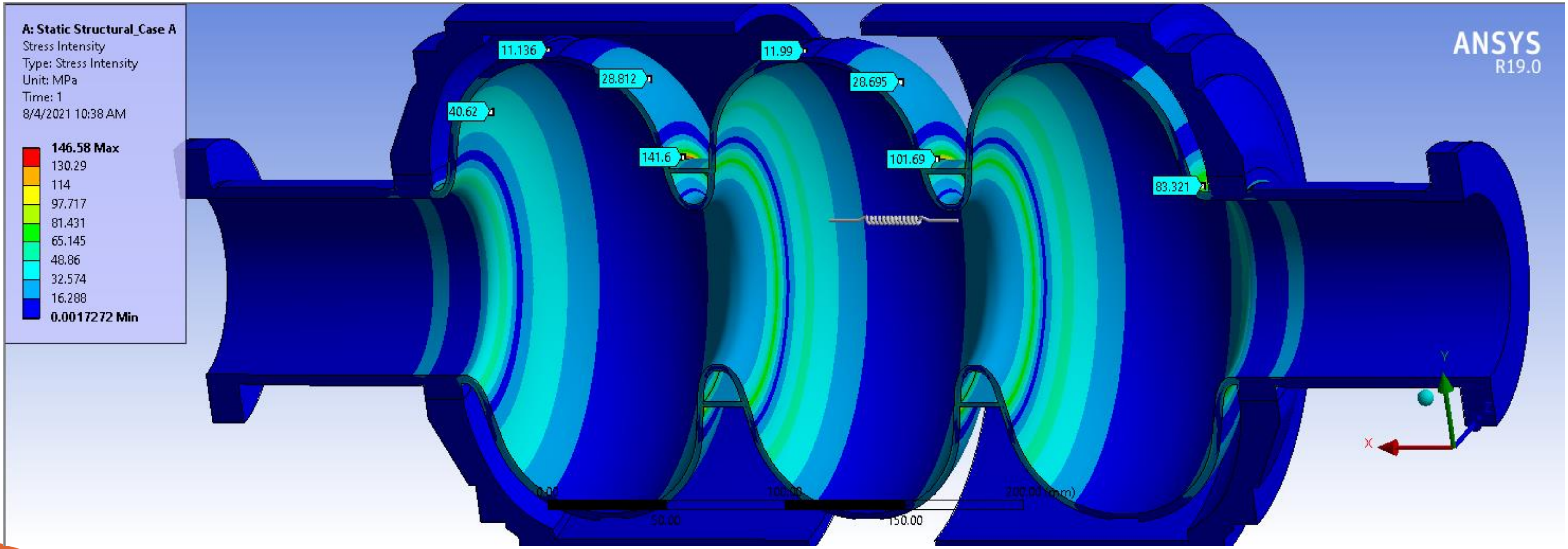




- Applied for Case B and C to simulate cooldown from room temperature to -271.4 °C.
- All bodies are selected.

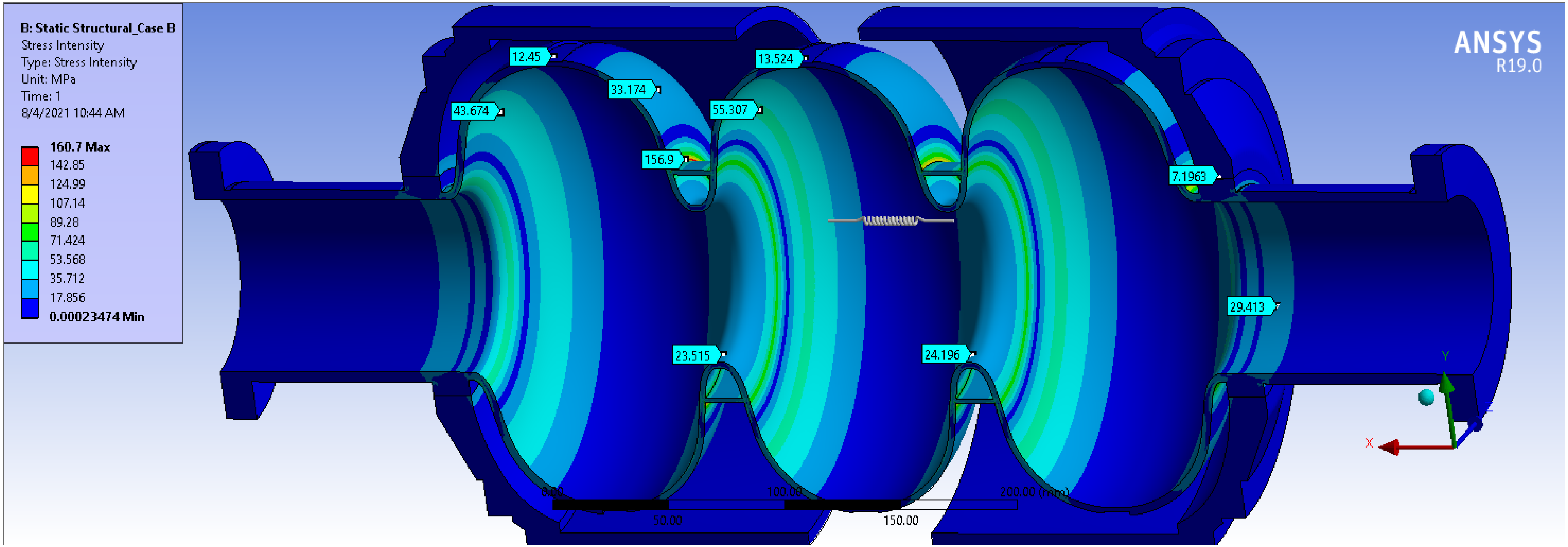
Solution - Case A

- Solution -> insert -> Stress intensity -> select bodies or surface to show
- Stress intensity is the largest of absolute value of difference in principle stresses ($\sigma_1 - \sigma_2$, $\sigma_2 - \sigma_3$, or $\sigma_3 - \sigma_1$).



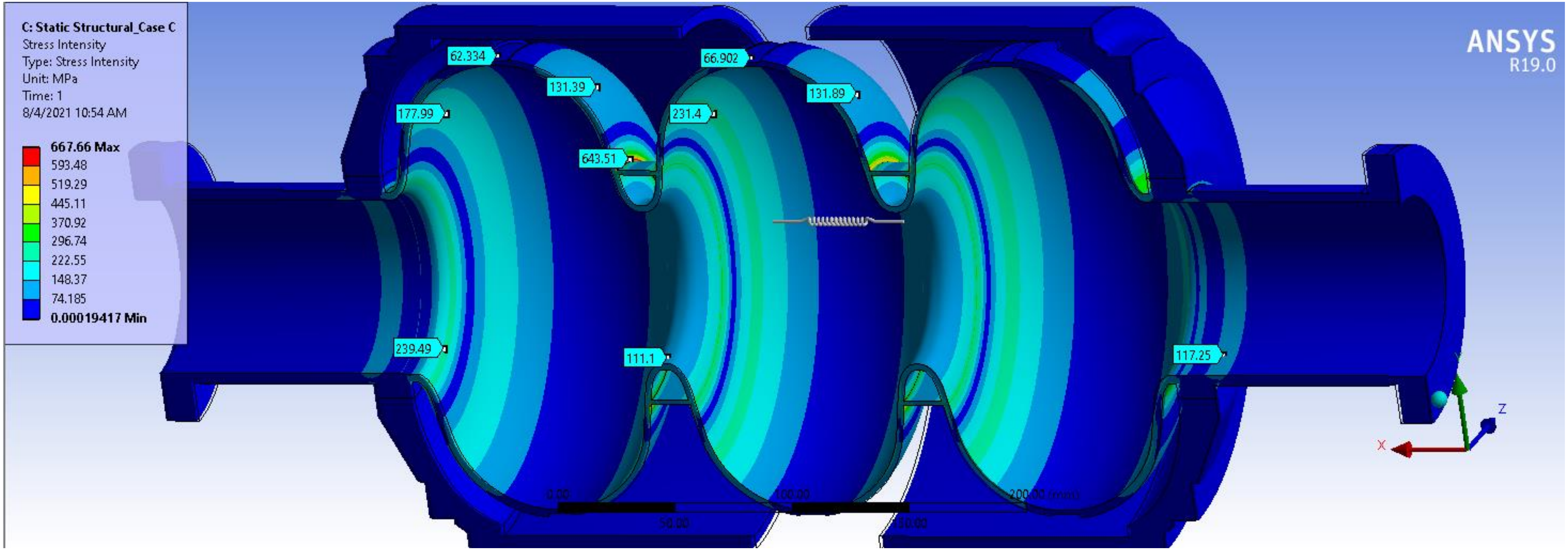
Solution - Case B

- Solution -> insert -> Stress intensity -> select bodies or surface to show
- Stress intensity is the largest of absolute value of difference in principle stresses ($\sigma_1 - \sigma_2$, $\sigma_2 - \sigma_3$, or $\sigma_3 - \sigma_1$).



Solution - Case C

- Solution -> insert -> Stress intensity -> select bodies or surface to show
- Stress intensity is the largest of absolute value of difference in principle stresses ($\sigma_1 - \sigma_2$, $\sigma_2 - \sigma_3$, or $\sigma_3 - \sigma_1$).



Solution: Maximum Stress Intensity

Case	Nb Half Cells [MPa]	Ti Tank [MPa]	Nb-Ti weld [MPa]
A	70	5	12
B	76	5	23
C	300	18	65

Case	Stiffener Ring Weld [MPa]	Iris Weld [MPa]	Equator Region weld [MPa]
A	147	22	12
B	161	24	14
C	668	111	67

- For the high-pressure gas safety regulations, the mechanical properties of the materials should be > 1.5 times the maximum stress intensity for 0.2% Y.S, and > 4 times the maximum stress intensity for T.S, on individual components.
- Data is different for 9-Cell SRF cavity structure, the stress values are lower.

Eigenvalue Buckling Analysis

What is Buckling?

- It is a sudden change of shape of a structural component under compressive or its own load.
- It can occur even though the stresses that develop in the structure are well below those needed to cause failure in the material.
- Further loading may cause significant and somewhat unpredictable deformations.
- Excess loading might or might not lead to complete loss of the structure's load-carrying capacity.

Buckled Structures

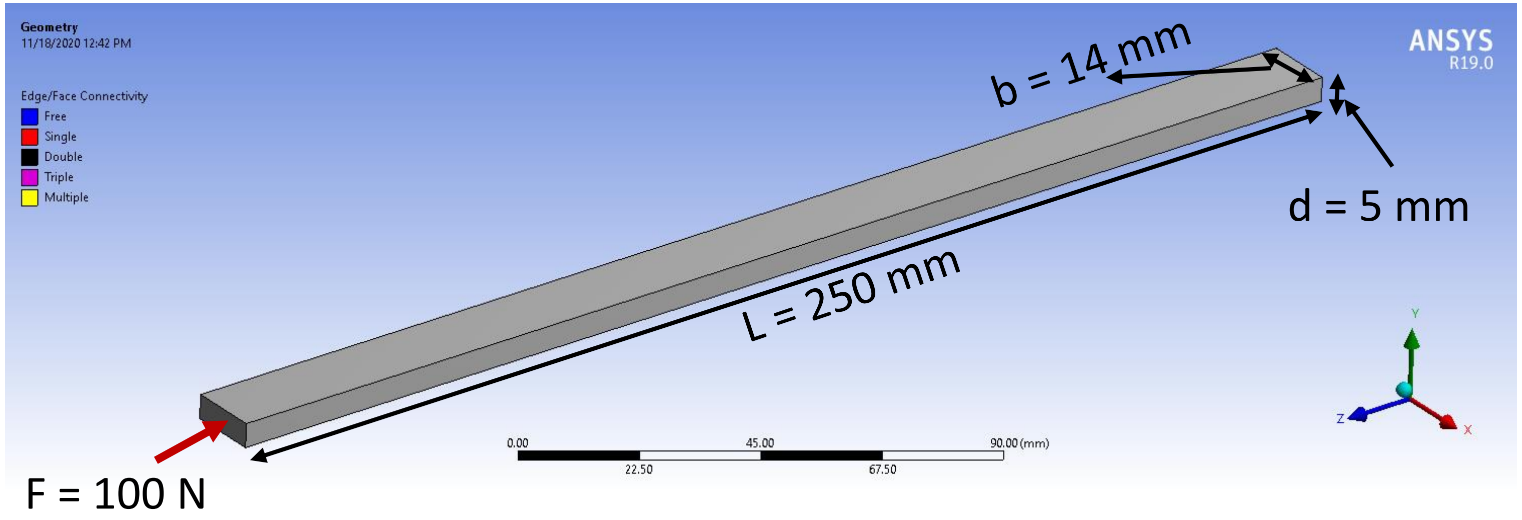
https://www.structuresinsider.com/post/the-difference-between-buckling-compression-shear

https://www.volpe.dot.gov/infrastructure-systems-and-technology/structures-and-dynamics/track-buckling-research

Buckling of a Solid Column

Center for Applied Superconducting Accelerator 応用超伝導加速器センター

Anniversary Since 1971 50th Keep Accelerating

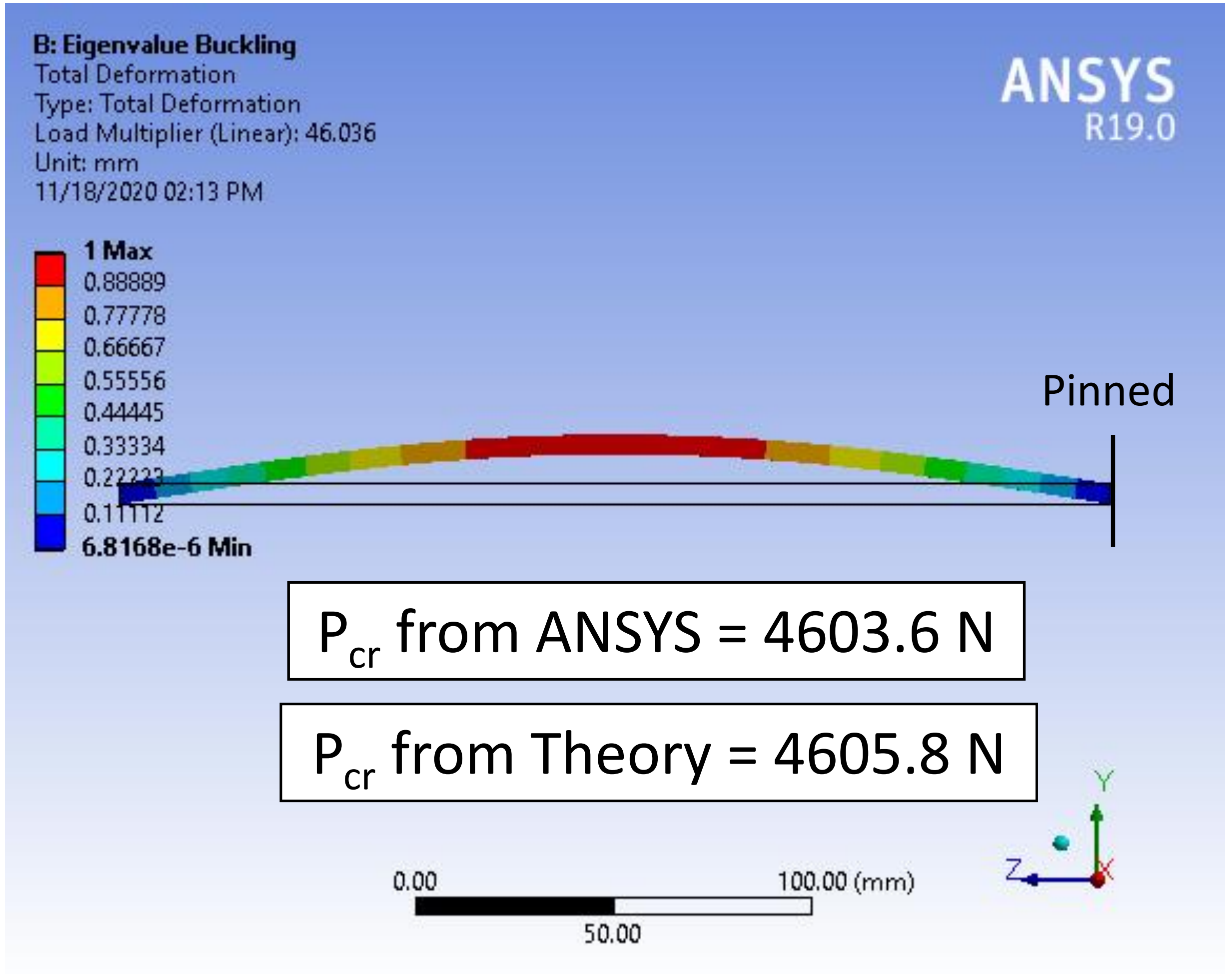


Buckled shape of column shown by dashed line	✓	✓		✓	✓	
Theoretical K value	0.5	0.7	1.0	1.0	2.0	2.0
Recommended design value K	0.65	0.80	1.2	1.0	2.10	2.0
End condition key	Rotation fixed and translation fixed Rotation free and translation fixed Rotation fixed and translation free Rotation free and translation free					

$$\text{Euler's Critical Load}, P_{cr} = \frac{\pi^2 EI}{(KL)^2}$$

- Where ‘E’ is Young's modulus, ‘I’ is moment of inertia, ‘K’ is column effective length factor.
- Stainless Steel E = 200 GPa, $I_{xx} = b*d^3/12$

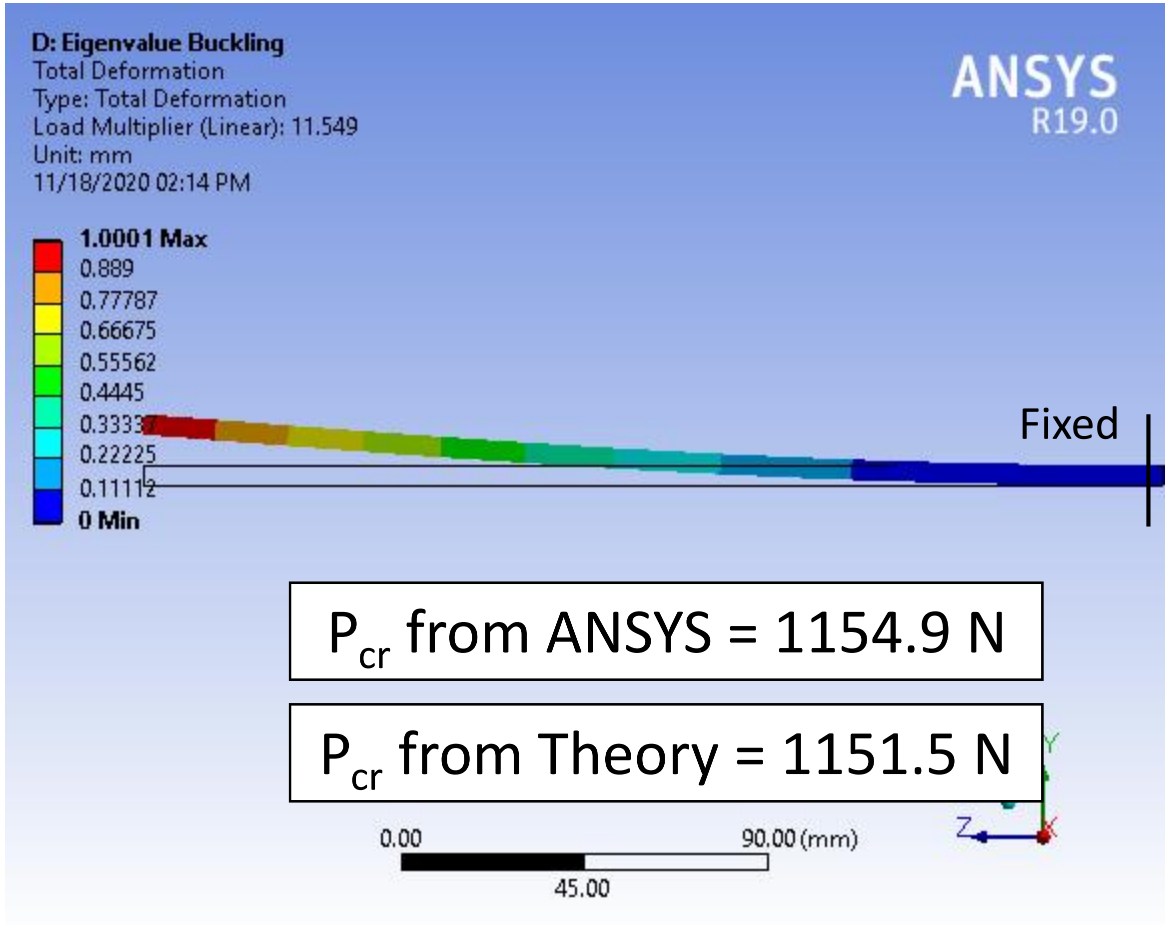
Results – First Mode of buckling



P_{cr} from ANSYS = 4603.6 N

P_{cr} from Theory = 4605.8 N

Pinned-Pinned Column

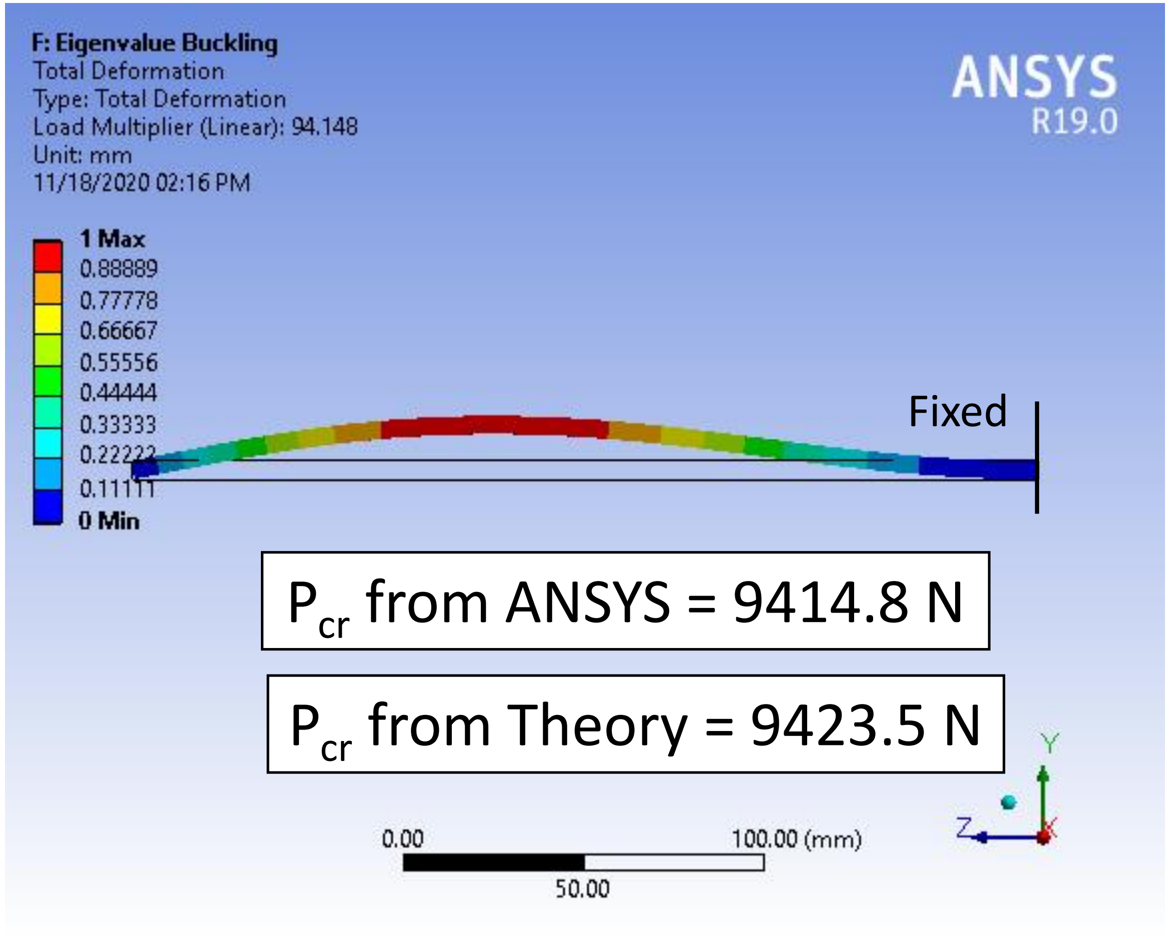


P_{cr} from ANSYS = 1154.9 N

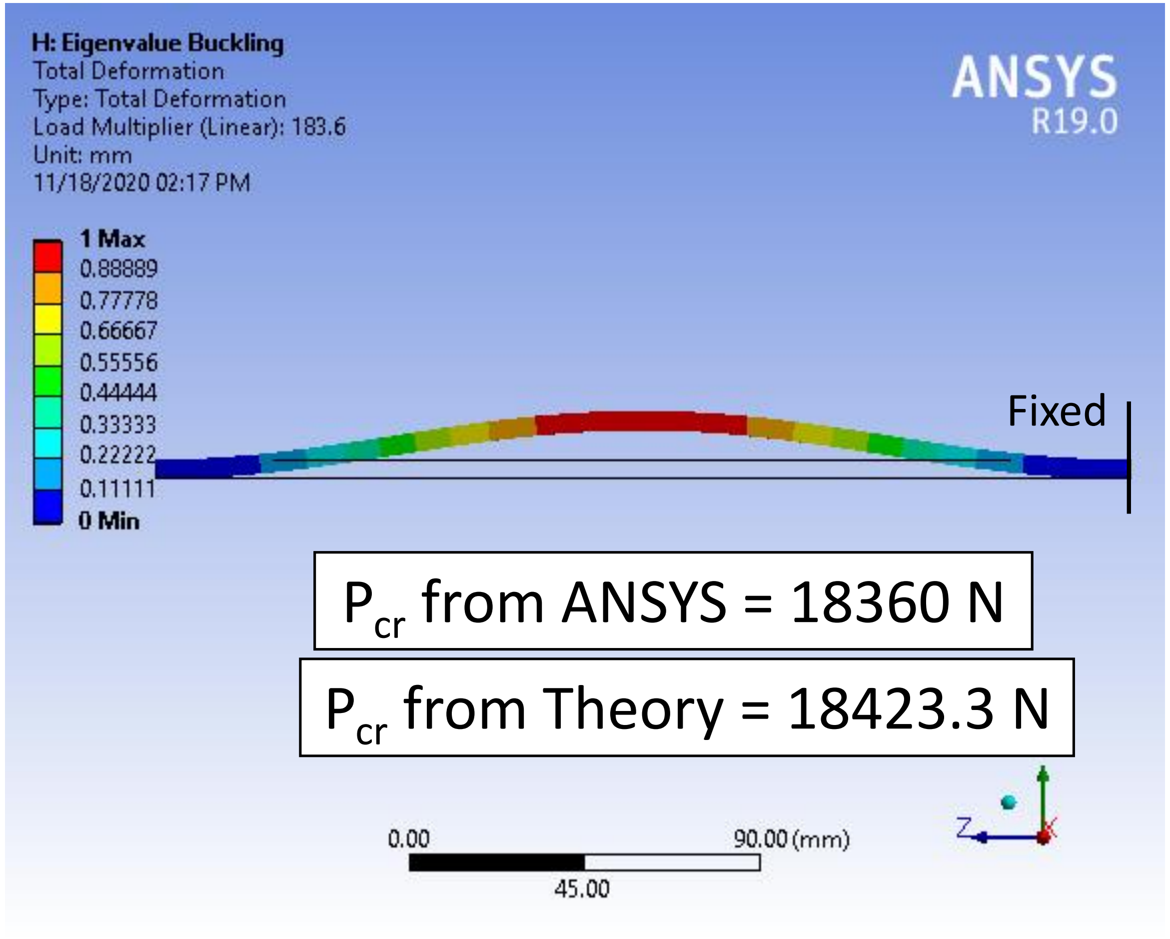
P_{cr} from Theory = 1151.5 N

Fixed-Free Column

Results – First Mode of buckling

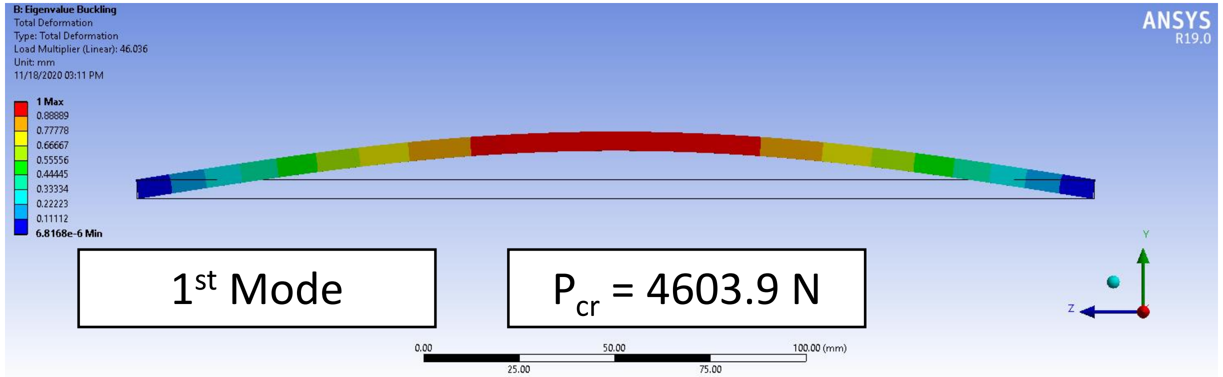


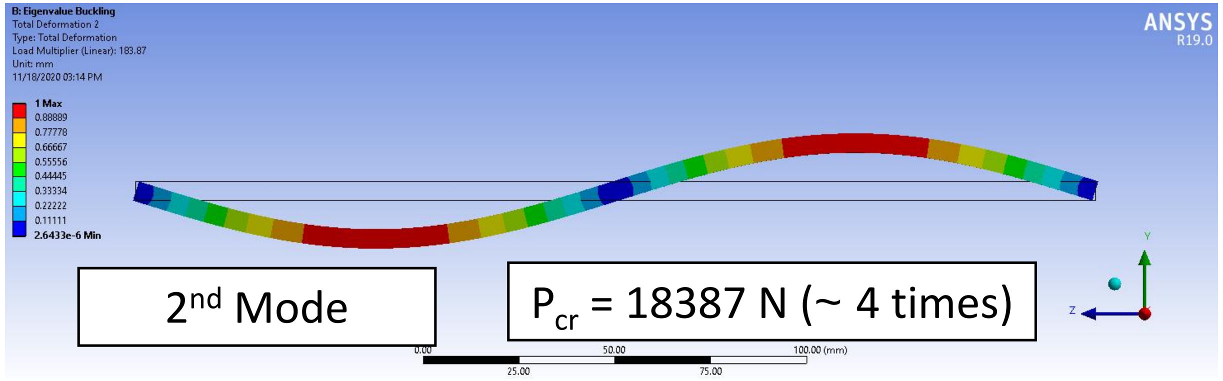
Fixed-Pinned Column

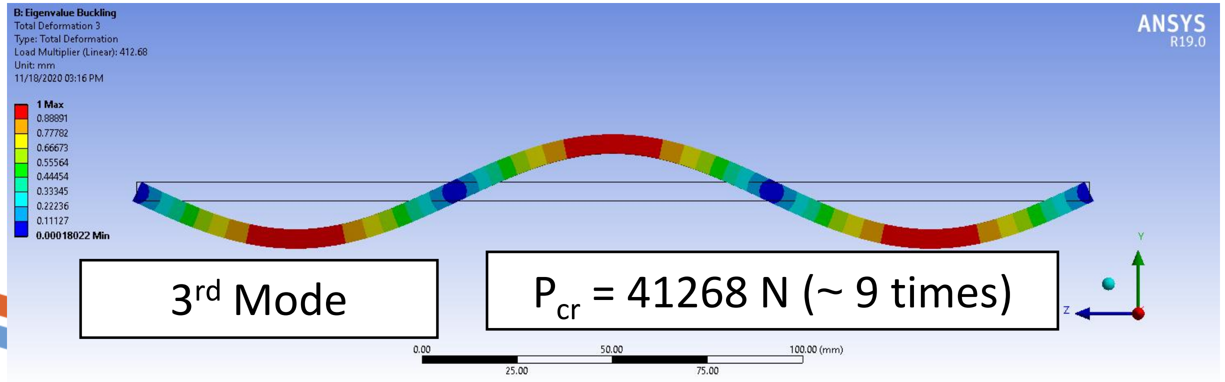


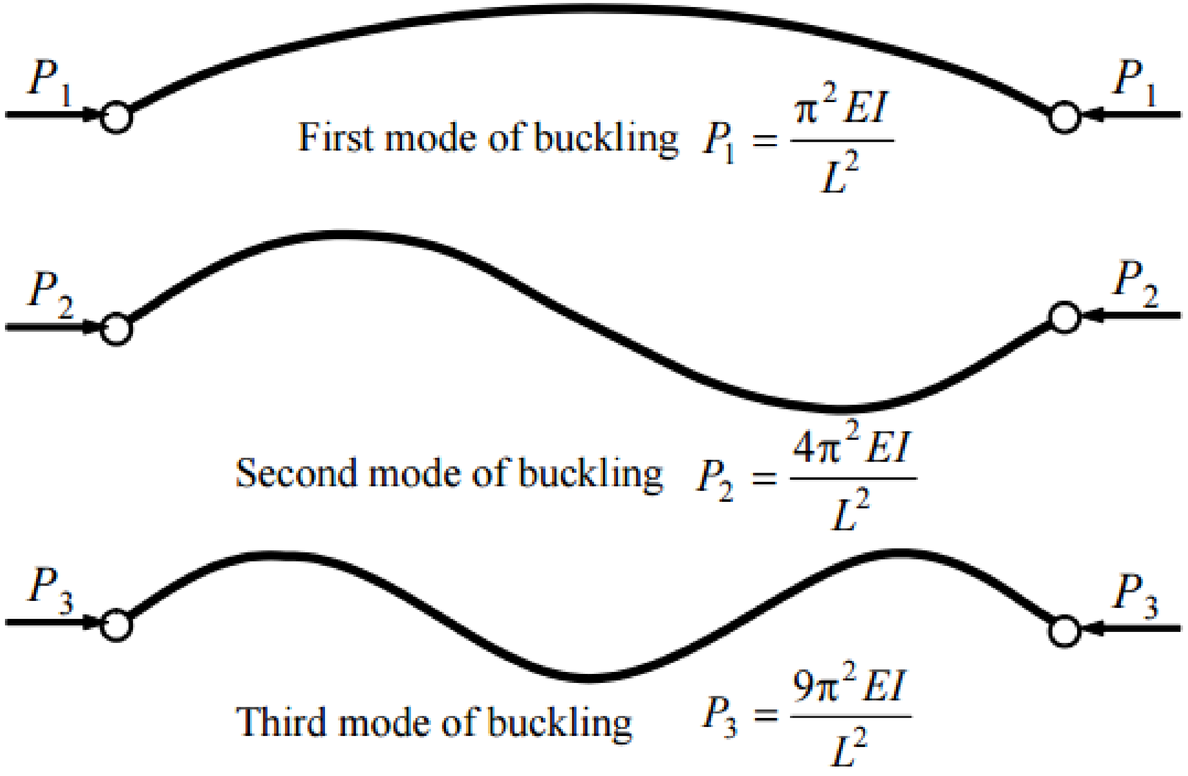
Fixed-fixed Column

Modes of Pinned-Pinned Column Buckling







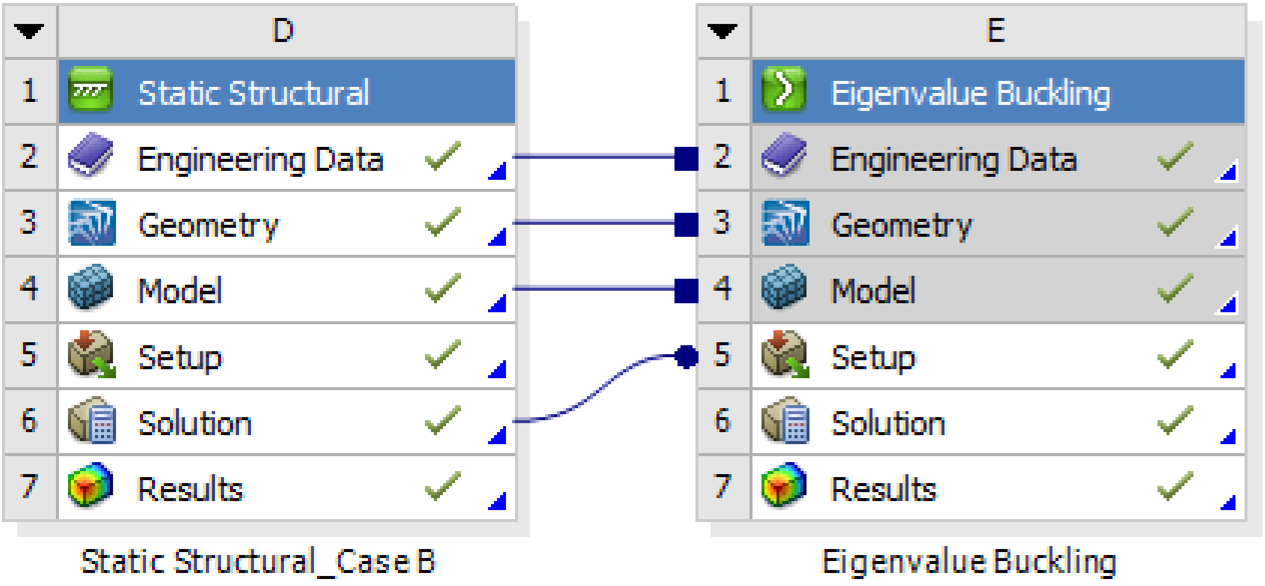


Eigen value Buckling Analysis for 1.3 GHz SRF Cavity

Eigenvalue Buckling of SRF Cavity

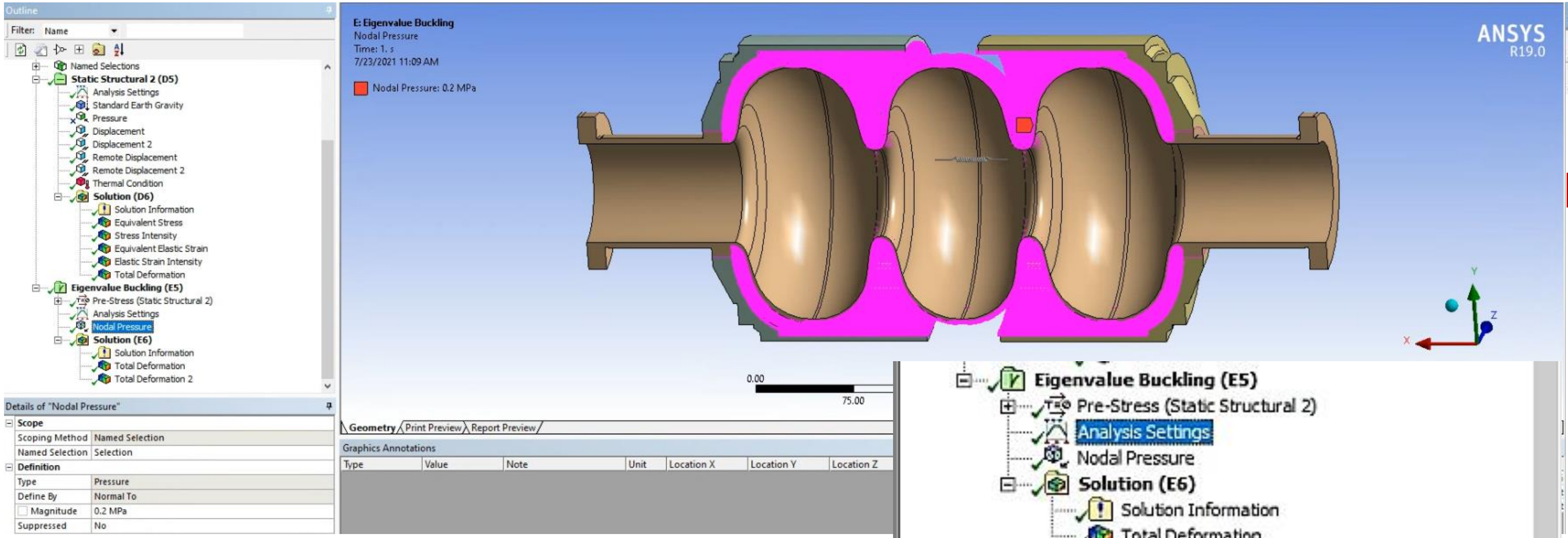
- Goal is to determine the pressure at which the cavity structure will buckle.
- Here only two Eigen modes will be determined.
- For this analysis, Case B is taken into consideration for buckling analysis.
- Cavity structure is easier to buckle in Case B rather than Case C due to lower tuner displacement.

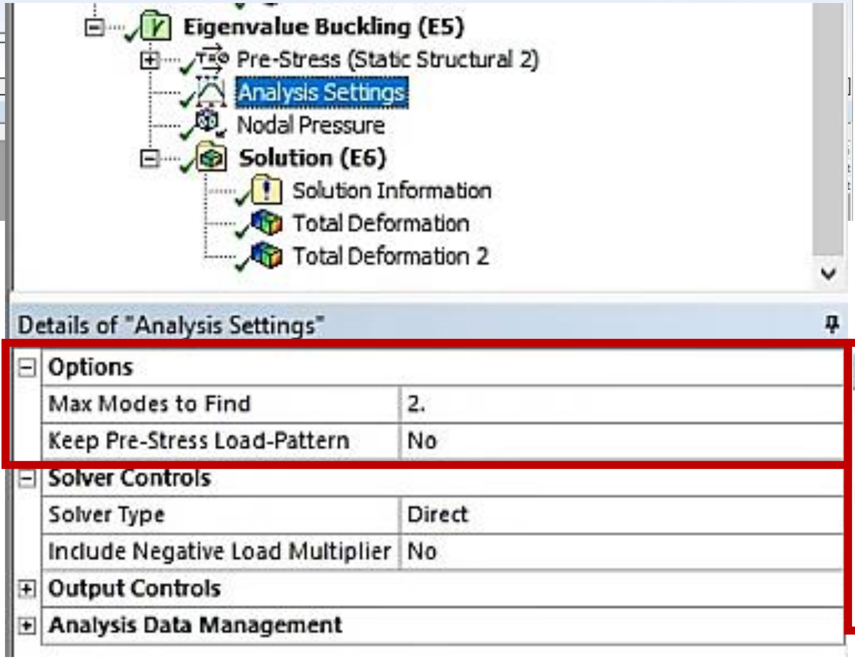
Case Structure

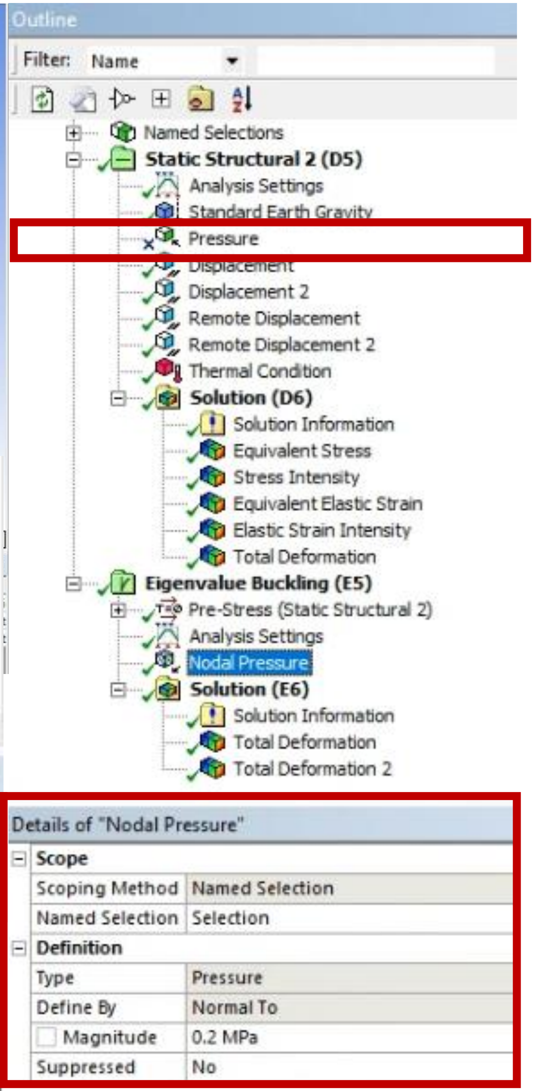


- Same Geometry is transferred by linking.
- Model with the meshing is also linked.
- Setup is transferred, although pressure boundary condition is suppressed.

Eigenvalue Buckling – Analysis Setting

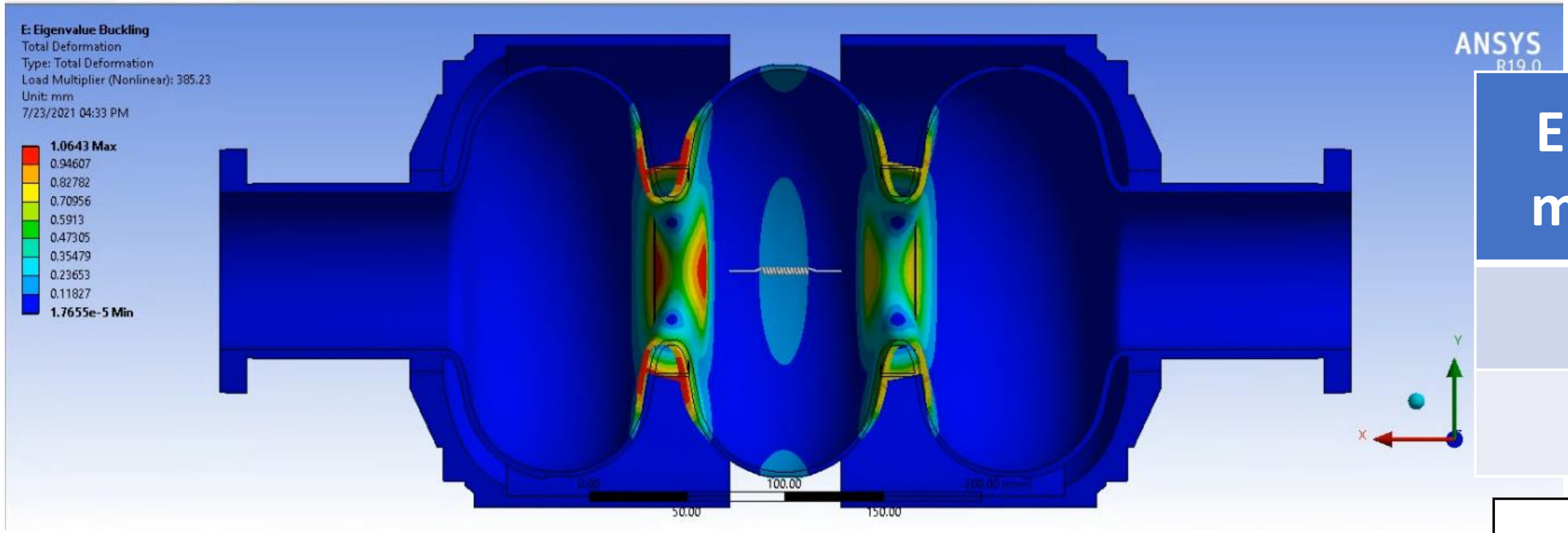


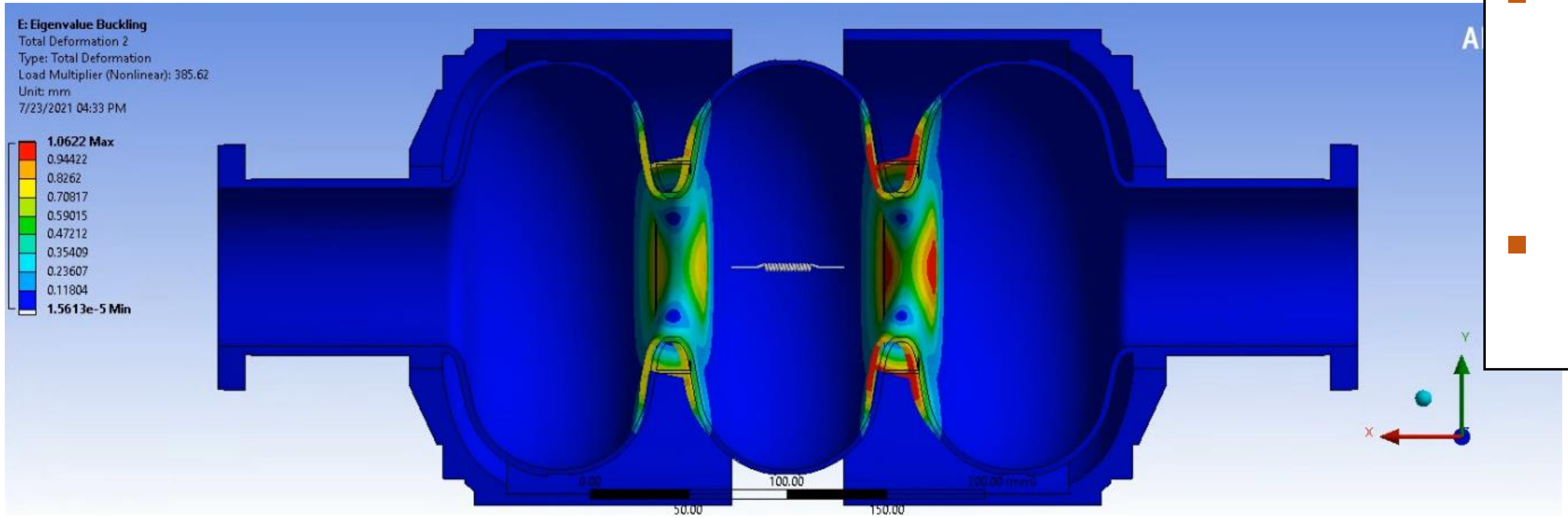




- Nodal pressure condition is provided.
- 0.2 MPa pressure acts on each nodes.

Solution





Eigen mode	Load Multiplier	Buckling Pressure
1st	385.2	77.04
2nd	386.6	77.32

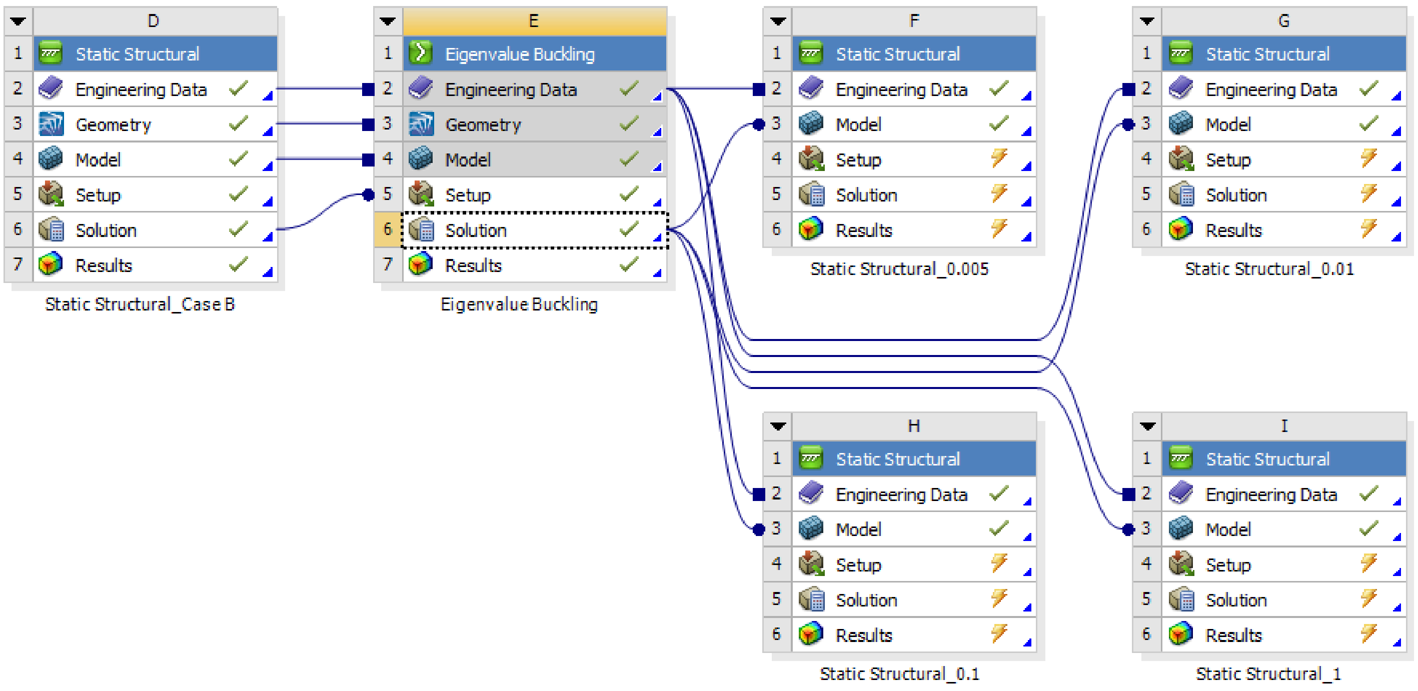
- Eigenvalue buckling pressure is higher than the requirement (> 4*0.2 MPa).
- For 9-Cell SRF cavity it has been calculated to be 33 MPa.

Non-linear Structure Analysis

Non-linear Buckling Analysis

- Continuation of the Eigen value buckling analysis.
- Buckled structure can be transferred for a static structure analysis.
- Possible to control the level of buckling in the structure that would be transferred with *scale factor*, which can be varied from 0 to > 1.
- Original structure scale factor is 0 and 1 being the buckled structure that would be transferred to the static structure analysis.
- Effect of structural deformities on buckling pressure can be determined.
- 1st Eigen mode was transferred to four static structure analysis studies with the scale factors being varied from 0.005, 0.01, 0.1 and 1.

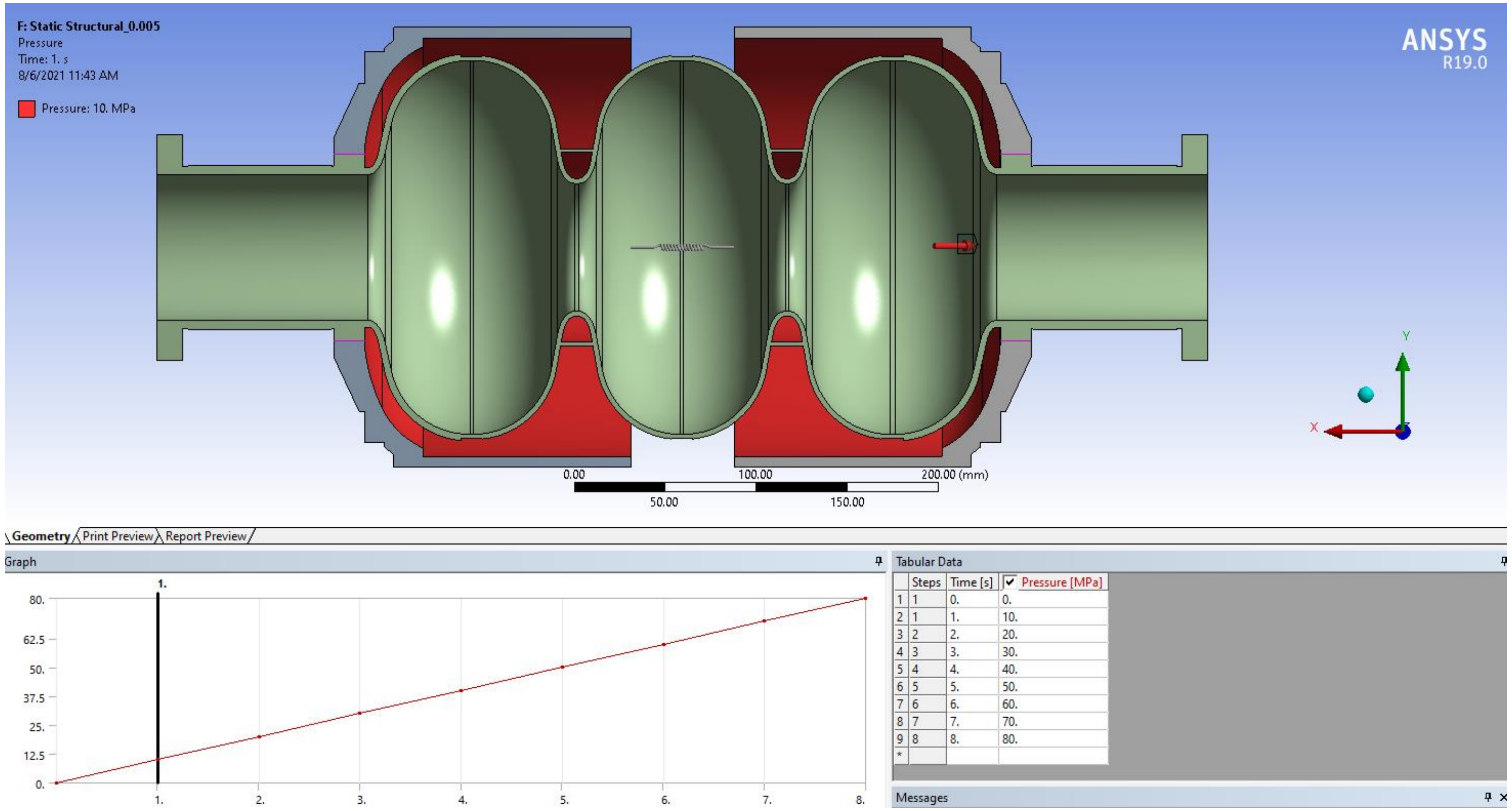
Non-linear Structure Analysis



14	Update Option	Use application default
15	Solve Process Setting	My Computer
16	Queue	
17	Update Settings for Static Structural_0.005 (Component ID: Model 5)	
18	Process Nodal Components	☑
19	Nodal Component Key	
20	Process Element Components	☑
21	Element Component Key	
22	Scale Factor	0.005
23	Mode	1
24	Update Settings for Static Structural_0.01 (Component ID: Model 6)	
25	Process Nodal Components	☑
26	Nodal Component Key	
27	Process Element Components	☑
28	Element Component Key	
29	Scale Factor	0.01
30	Mode	1
31	Update Settings for Static Structural_0.1 (Component ID: Model 7)	
32	Process Nodal Components	☑
33	Nodal Component Key	
34	Process Element Components	☑
35	Element Component Key	
36	Scale Factor	0.1
37	Mode	1
38	Update Settings for Static Structural_1 (Component ID: Model 8)	
39	Process Nodal Components	☑
40	Nodal Component Key	
41	Process Element Components	☑
42	Element Component Key	
43	Scale Factor	1
44	Mode	1

- 1st Eigen mode was transferred to four static structure analysis studies with the scale factors varied from 0.005, 0.01, 0.1 and 1.

Setup – Boundary Condition



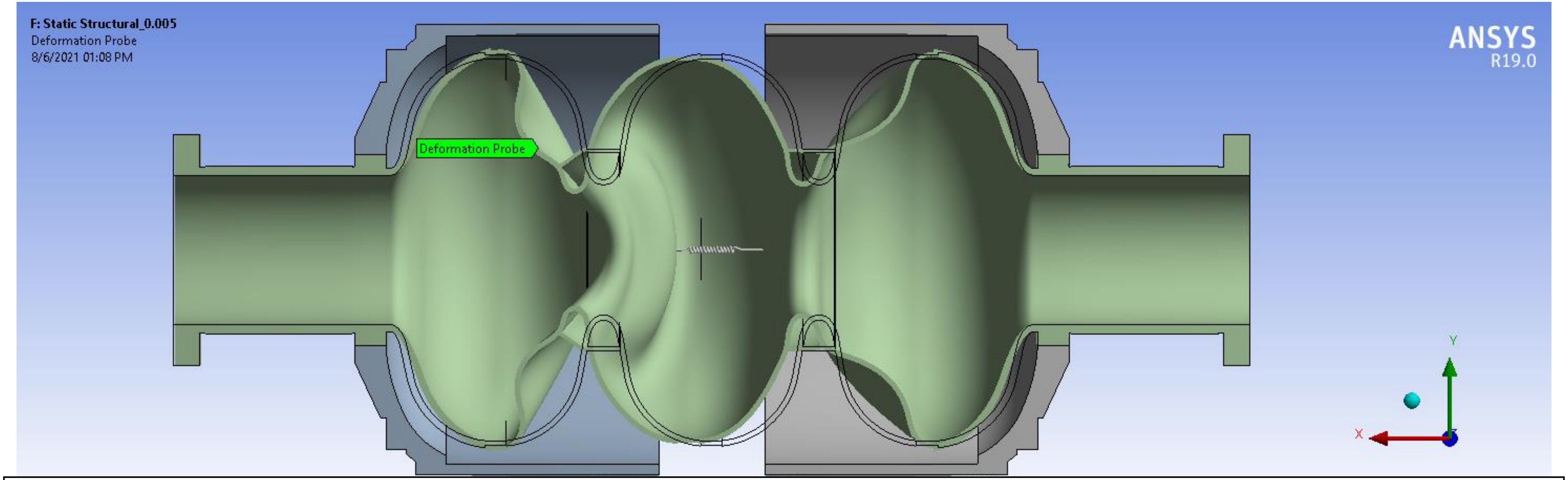
Tabular Data

	Steps	Time [s]	Pressure [MPa]
1	1	0.	0.
2	1	1.	10.
3	2	2.	20.
4	3	3.	30.
5	4	4.	40.
6	5	5.	50.
7	6	6.	60.
8	7	7.	70.
9	8	8.	80.
*			

Messages

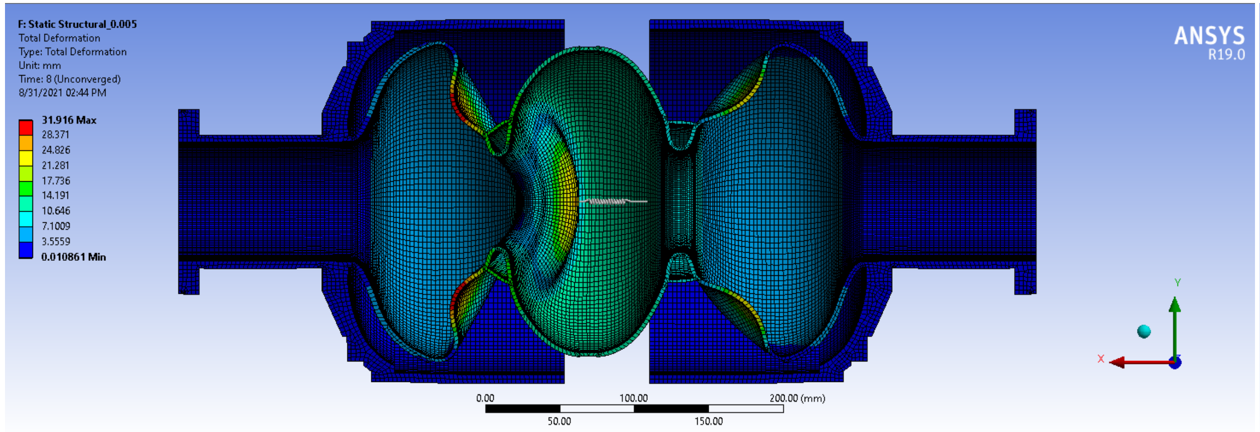
- Boundary conditions remains the same as in Case B, except pressure.
- Pressure (faces in Red) is increased incrementally in steps.
- Pressure increased from 0 – 80 MPa in 8 steps.

Solution – Buckling Pressure

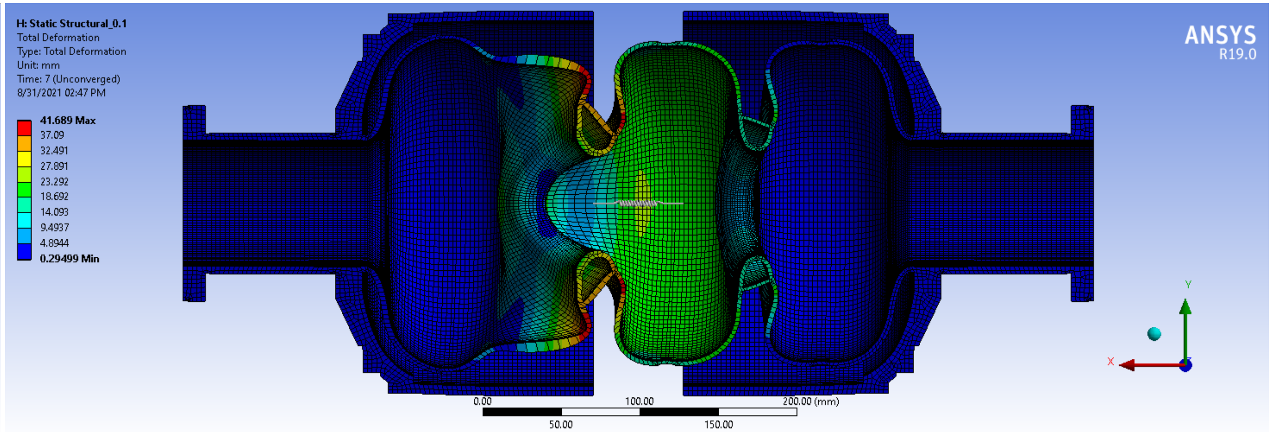


- Buckling pressure was determined by placing a total deformation probe on a point on the cavity structure where the maximum buckling occurs.
- The total deformation obtained from the deformation probe was plotted against the pressure condition for various scaling factors.

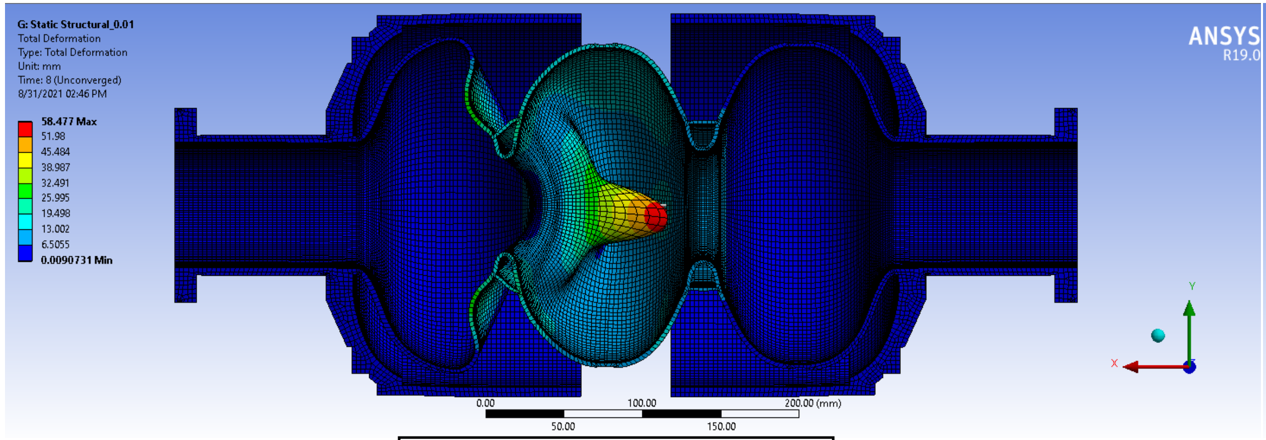
Buckled Structures for Scale factors



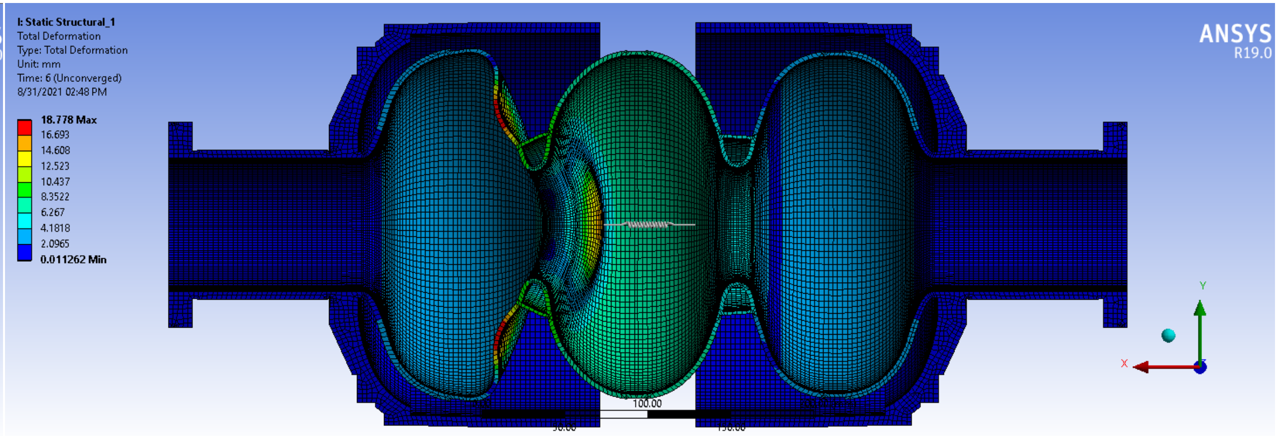
Scale Factor – 0.005



Scale Factor – 0.1



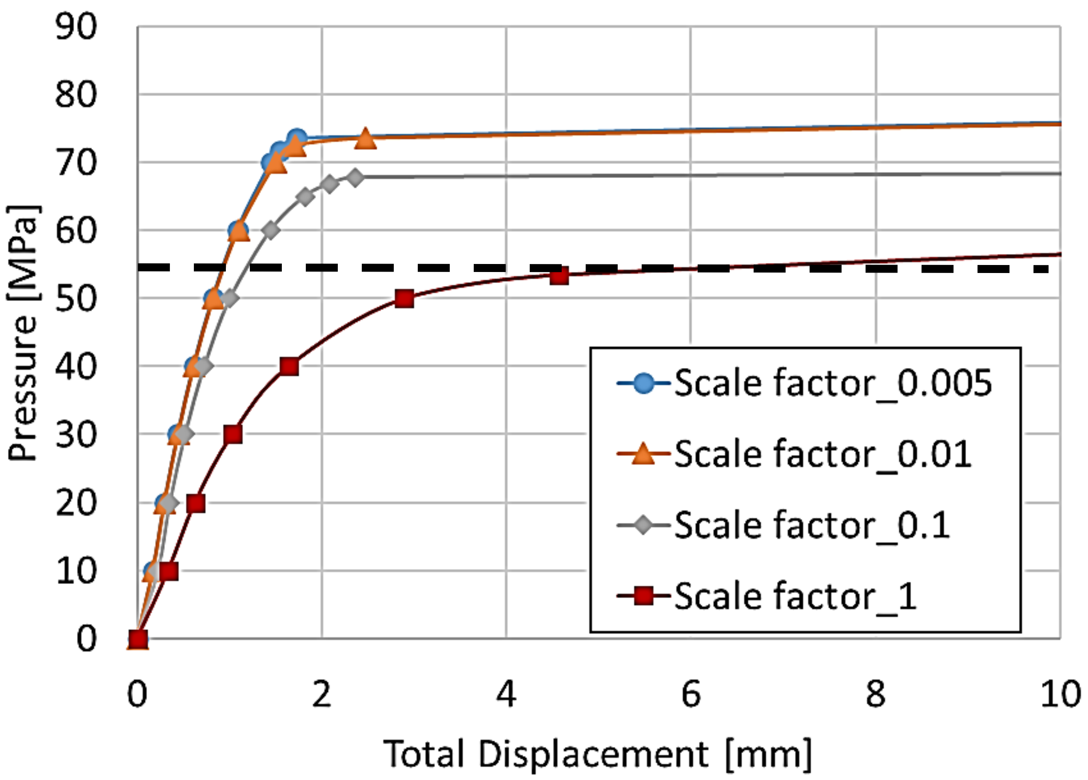
Scale Factor – 0.01



Scale Factor – 1

Results and Discussions

- Buckling pressure was 74 MPa for scale factor of 0.005, similar to Eigenvalue buckling analysis.
- Buckling pressure reduces to 54 MPa for the scale factor of 1.
- For elastic material properties, the 3-Cell cavity will not buckle upto 54 MPa pressure.
- Buckling pressure usually should be > 4 times the MAWP, and in this case the cavity structure has sufficient strength to qualify for that criterion.



Thank you for your attention!