ED5015: Computational Methods in Design

EndSem Project

Name : Tankala Satya Sai | Date : 15-05-2024
Roll No : ED21B065 | Prof. Palaniappan Ramu

Turbine Rotor Disks: An Overview

Function:

A turbine rotor disk is a critical component in a gas turbine engine. It's a flat, circular disc with a central bore that mounts onto a shaft. Turbine blades are attached around the rim of the disk.

Challenges:

- **High Temperature:** The hot gas from combustion flows through the turbine, exposing the disk to extreme temperatures.
- **High-Speed Rotation:** The disk spins at high speeds, generating significant centrifugal forces.

Combined Effect:

This combination of high temperature and high-speed rotation creates significant stresses and deformations within the disk.

Material Selection:

Turbine rotor disks are typically made from high-strength, heat-resistant nickel alloys to withstand these harsh conditions. There's exciting research going on regarding material selections to improve performance.

Design Considerations:

The design of the disk needs to optimize several factors:

- **Strength:** To handle the centrifugal forces without failure.

- **Stiffness:** To minimize deformations that could affect blade performance.
- **Weight:** To reduce overall engine weight and improve efficiency.

Advanced Features:

Modern turbine rotor disks may incorporate features like:

- **Cooling channels:** Internal passages that allow cooling air to circulate, reducing thermal stresses.
- **Blade slots:** Specifically designed grooves for securing the turbine blades.
- **Complex web structure:** A network of support beams within the disk to improve strength and stiffness.

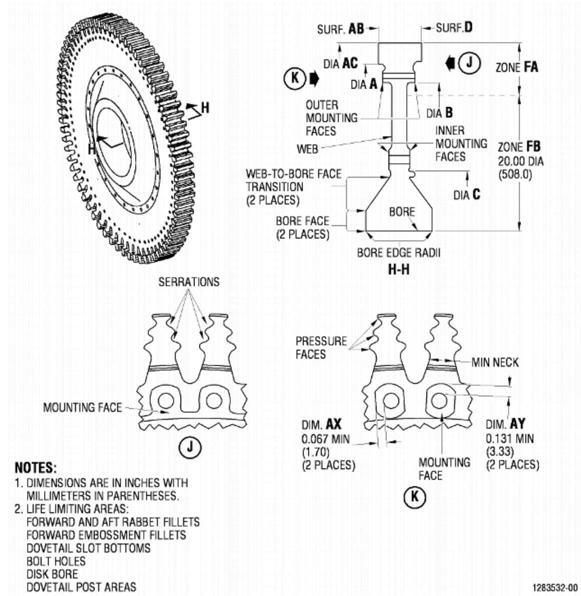
Understanding Stresses and Deformations:

The provided information in the question describes a scenario where the goal is to analyze the stresses and deflections experienced by the turbine rotor disk under specific operating conditions. This analysis is crucial for ensuring the safe and efficient operation of the gas turbine engine.

Based on my research, I learned that this is not precisely a turbine rotor disk; it's just a simple version. Turbine rotor disks have more complex features than this disc. Here are some of the parts of a turbine rotor disk:

- **Disk:** The central round body of the rotor.
- **Bore:** The hole in the disk's center that fits onto the shaft.
- **Rim:** The outer edge of the disk.
- **Blade slots:** The grooves are machined into the disk's rim, which holds the turbine blades.
- **Web:** The disk section connecting the rim to the bore.

Precise turbine rotor disk:



The difference is that the given model doesn't have blades, blade slots, etc.

Problem description

1. A turbine rotor disk shown in Figure 1 is part of a gas turbine, and the blades are attached to its periphery. The gas turbine experiences high temperature and high angular velocity, resulting in large stresses and deformation. The sketch of the cross-section of the turbine disk is given in Figure 2. The aim is to find the various stresses and deflections that act on the turbine rotor disk under the operating conditions provided in Table 1.

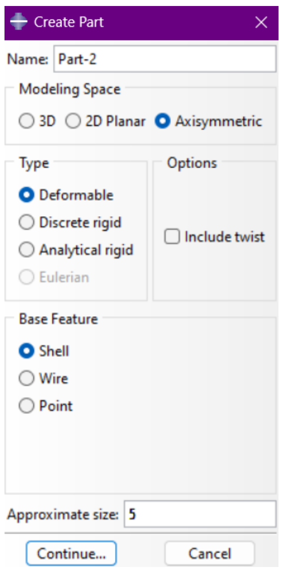
Table 1: Description of parameters for Turbine Disk example

S.No.	Parameters	Unit	Values
1	Rotational velocity, W	rad/s	375
2	Force, F	N	600
3	Young's Modulus, E	Pa	2e11
4	Poisson's Ratio, ν	-	0.3
5	Thermal conductivity, κ	W/mK	60.5
6	Coefficient of thermal expansion, α	1/°C	1.2e-5
7	Temperature at top, T_{top}	°C	600
8	Temperature at bottom, T_{bore}	°C	30
9	Heat transfer coefficient at top, h_{top}	W/m^{2}K	6
10	Heat transfer coefficient at the bottom, h_{bore}	W/m^{2}K	120

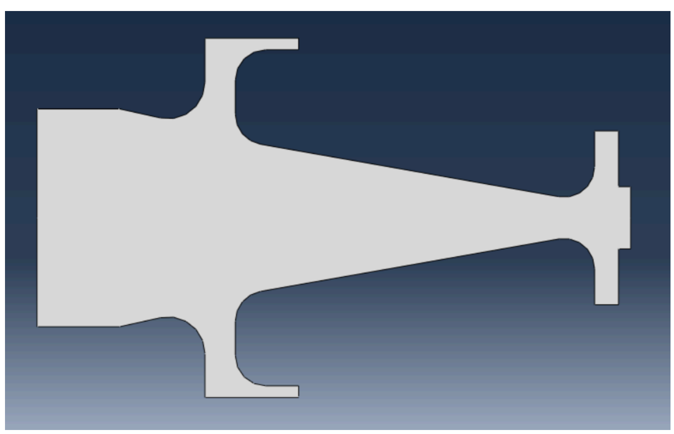
Set up a steady-state thermal-structural simulation model using the finite element procedures in ABAQUS® and report the following in detail:

ABAQUS Analysis Steps

1. Start Abaqus and choose to create a new model database
2. In the model tree double click on the "Parts" node (or right-click on "parts" and select Create)
3. Name the part in the Create Part dialog box and
    a. Select "Axisymmetric."
    b. Select "Deformable"
    c. Select "Shell"
    d. Set approximate size = 5 (SI units: m)
    e. Click "Continue..."



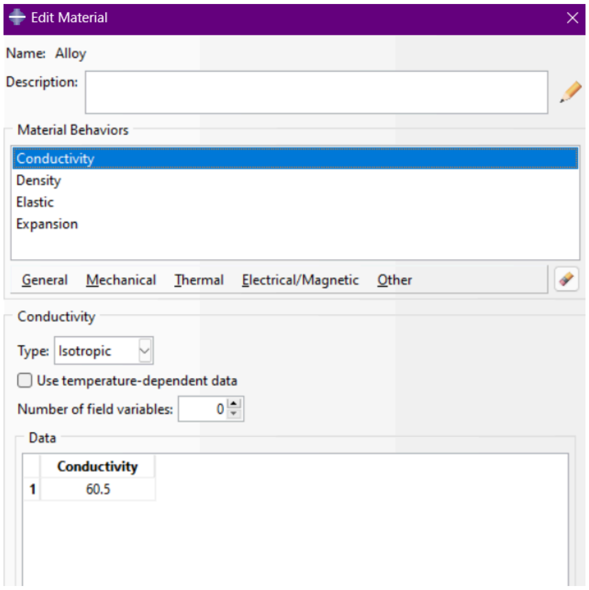
4. Create the geometry shown below:

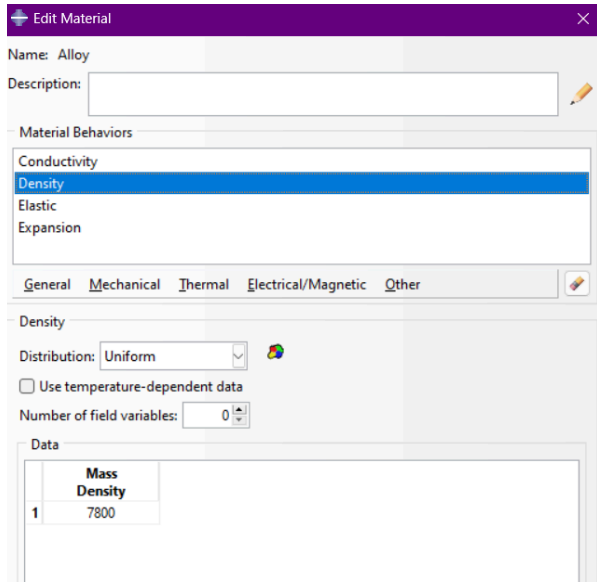


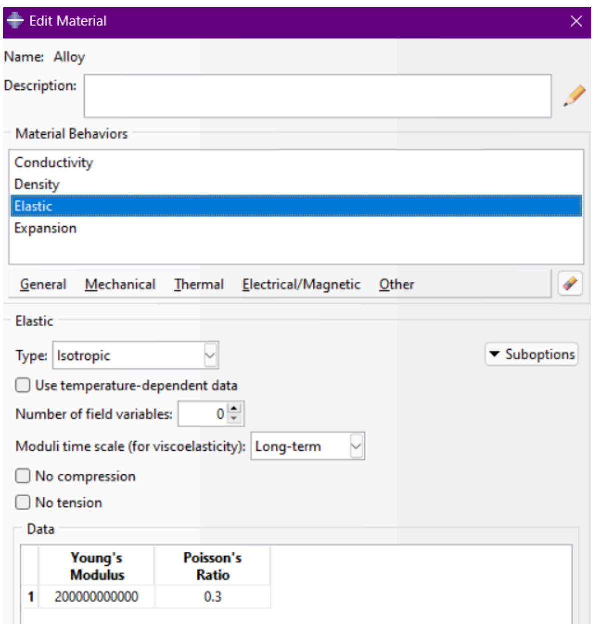
5. Create material property---alloy

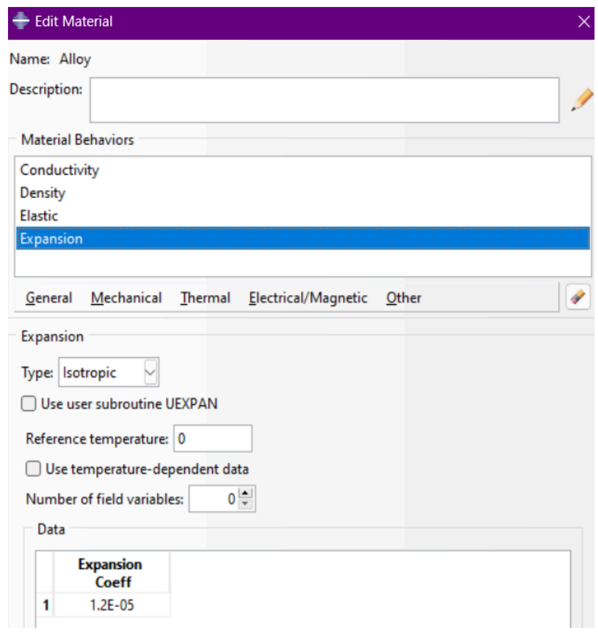
a. Mechanical---Elasticity---Elastic, E=2E11, v=0.3. b. Mechanical---Expansion, coff-1.2E-5 (as 1/C ~ 1/K), Thermal—Conductivity, k = 60.5

b. General—Density, ρ = 7800 [1]





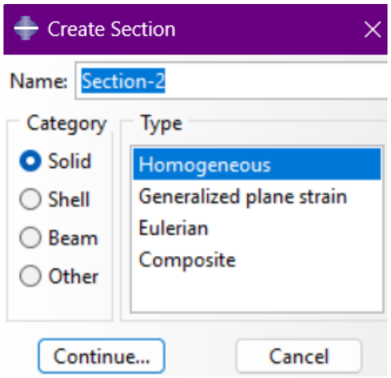


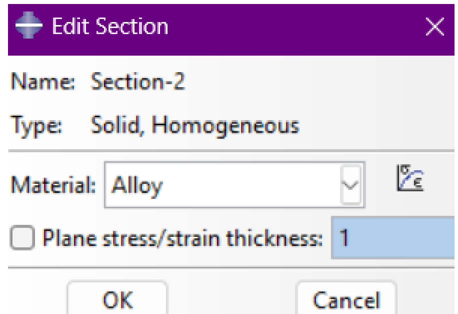


6. Create section

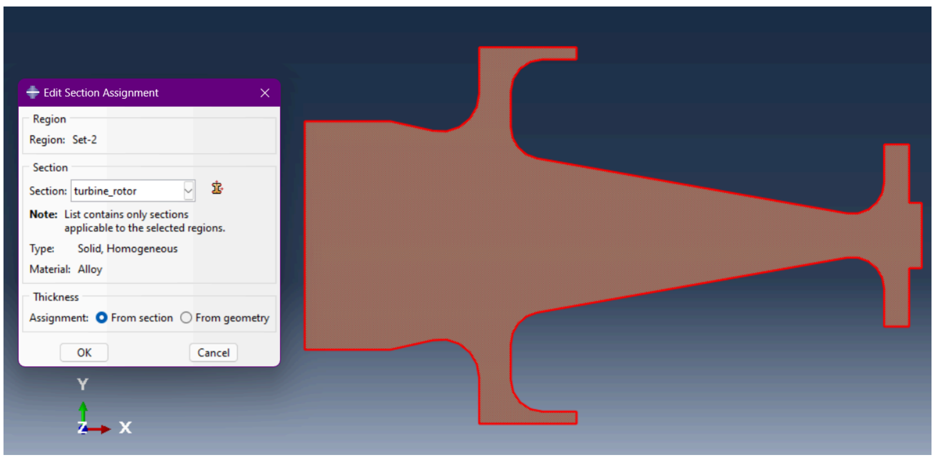
   a. Create a section as below

   b. We need not define the “Plane stress/strain thickness” as it's not given in the question.





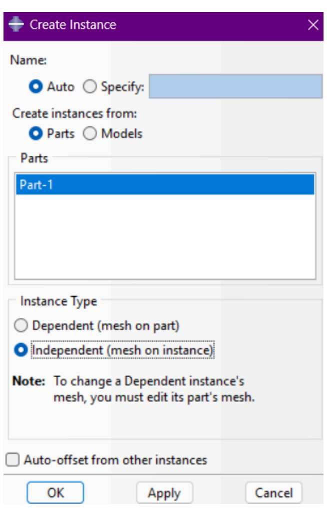
c. assign section



7. Assembly

a. Create instance part

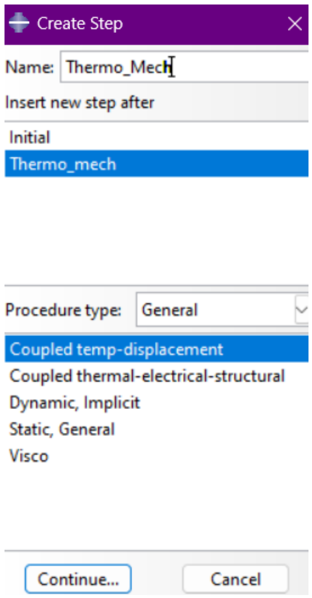
b. Select “Independent (mesh on instance)

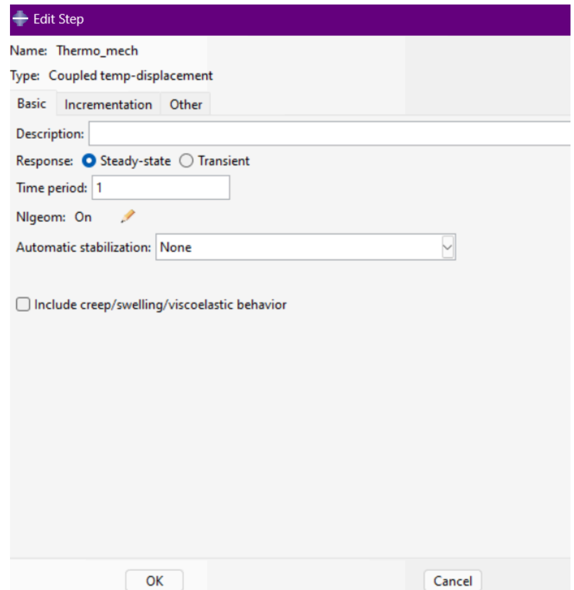


8. Create Step

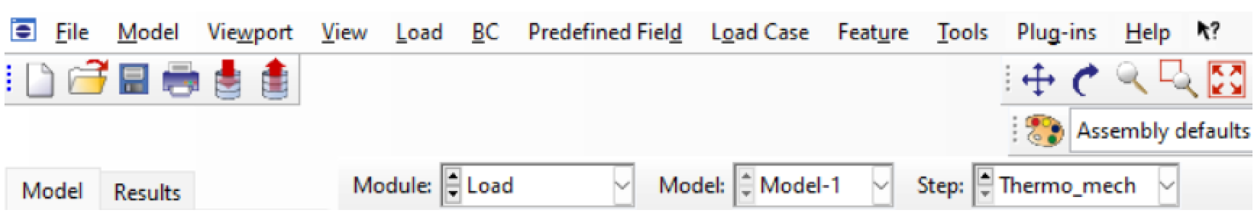
a. Create a “Thermo_Mech” Coupled temp-displacement step

b. Click "Continue..."
c. Select “Steady-state” Response
d. Turn on the Non-linear geometry option. Click “OK...”

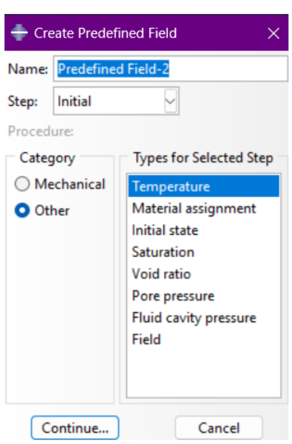


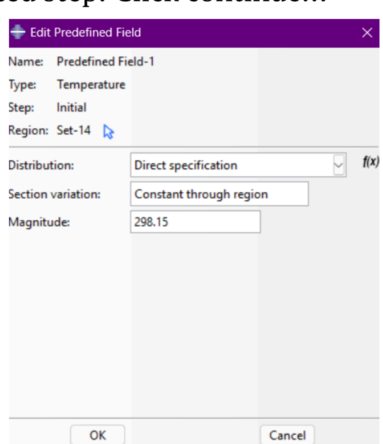


9. Under the Load Module,
Create initial temperature field:
a. Create a predefined field to define the initial temperature of 298.15 kelvin, select "Predefined field" in the main menu(as shown below), and select "Create."



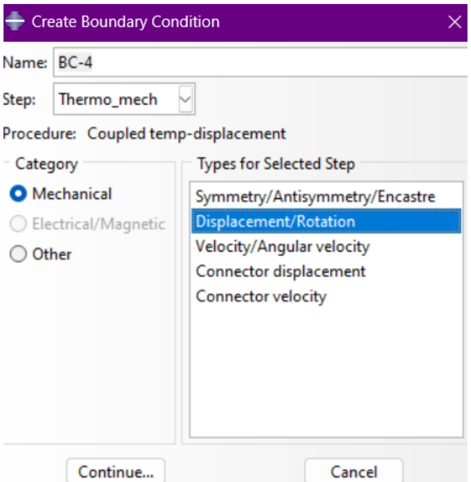
b. Create Predefined field 1, select "initial" in step, "other" in the category, and "temperature" in Types for the selected step. Click continue...

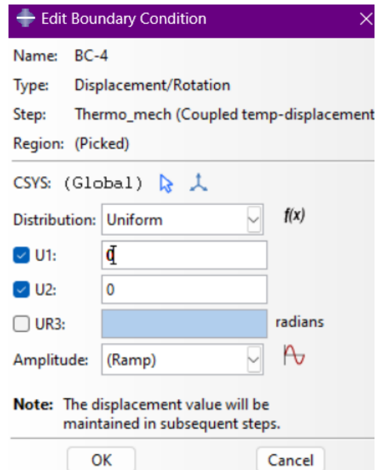




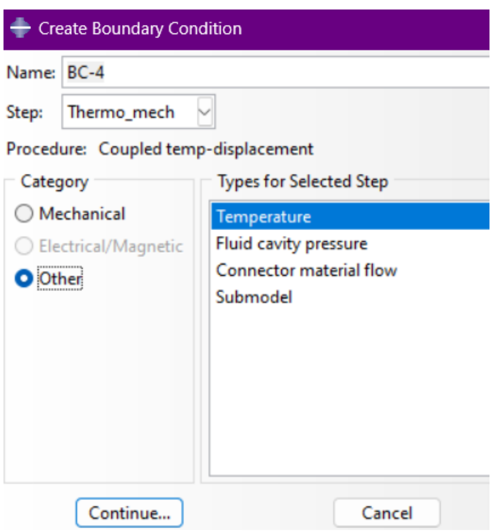
c. Select the whole plate and click "Done."
d. Define the magnitude of the temperature field: 298.15 kelvin. (As shown above)

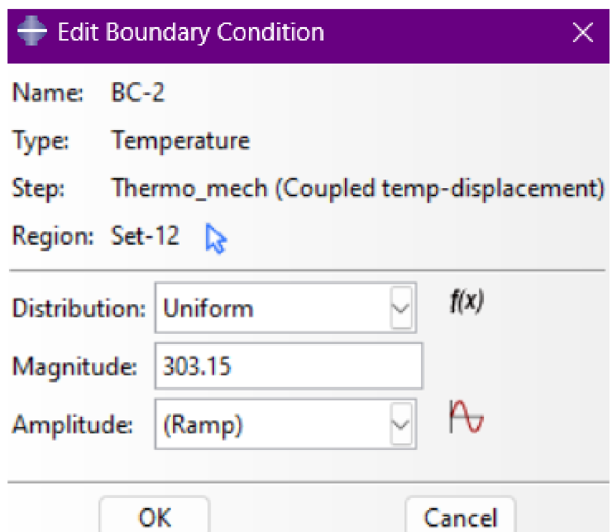
10. Create Boundary conditions
a. To define Z axis-symmetric, we can add a fixed displacement boundary condition as shown below to the bore surface:

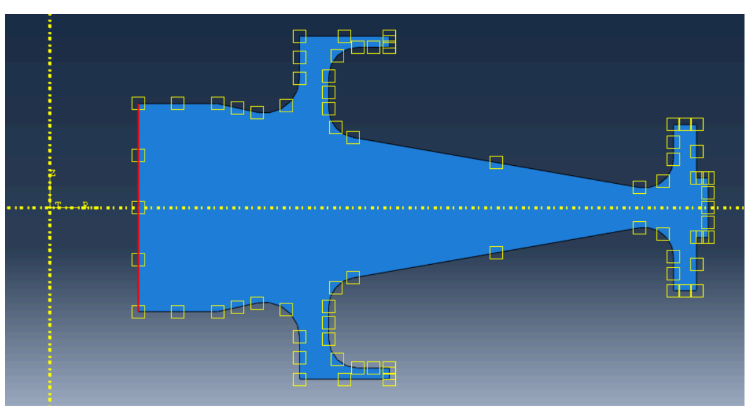




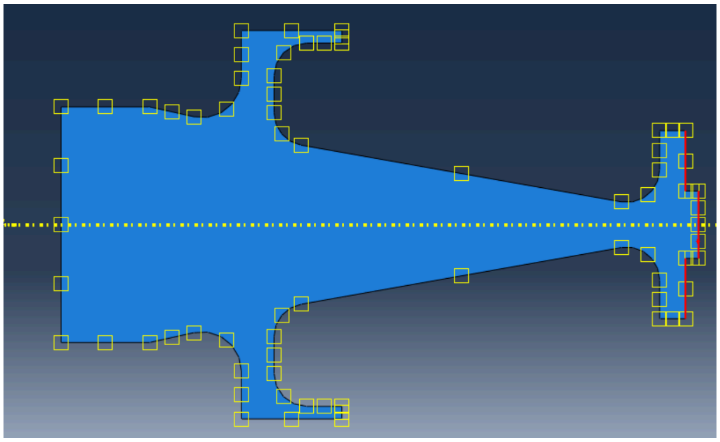
b. Define Temperature at bottom, T_bore:

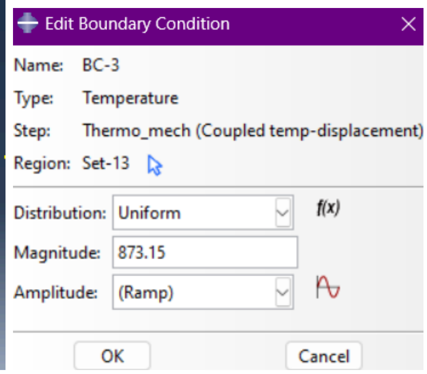






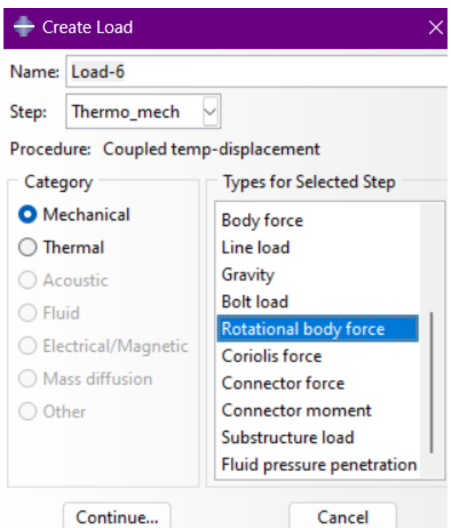
c. Define Temperature at the top, T_top:

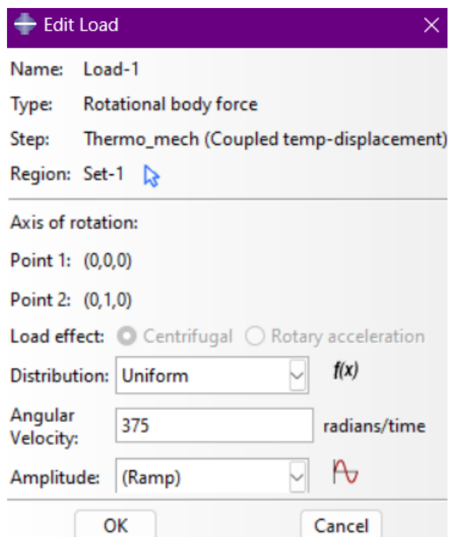


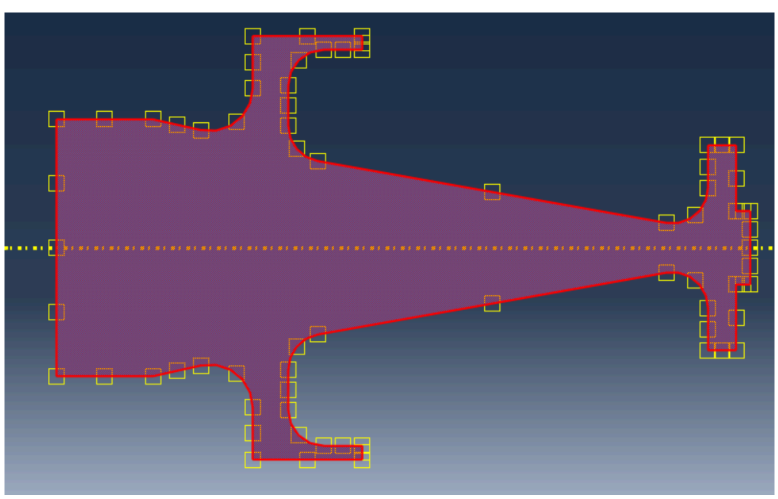


11. Create Loads:

a. Define the W, which is the body force (provided in Table 1) experienced by the rotor due to the rotational velocity.





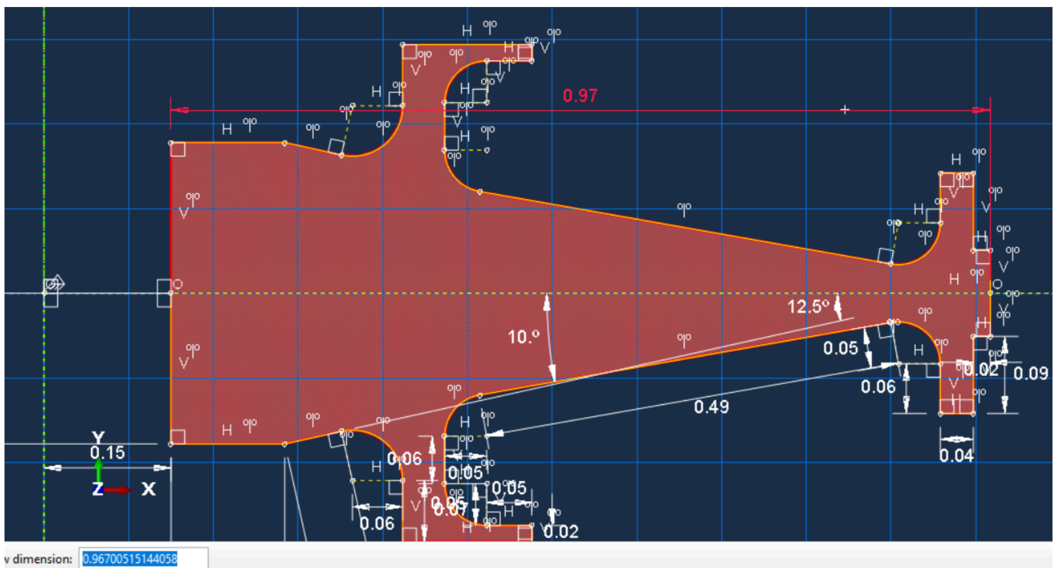


b. Define the F as the force (provided in Table 1) experienced by the rotor in the axial direction due to the weight of the blades.

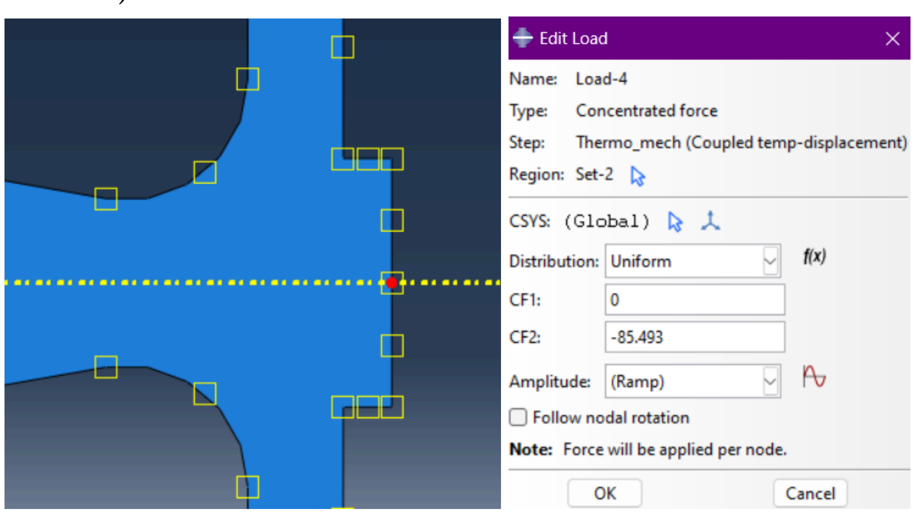
   i. I'm considering a line load that will act along the circumference of the outermost edge center. So we can apply a concentrated force on that point, which will trace out that circle when we revolve the sketch to form a 3d model.

   ii. $F = \frac{F_{total}}{2\pi r} = \frac{600}{2*3.14*r}$, $r = 0.96700515144058 + 0.150 = 1.11700515144058$

   $F = 85.493\ N$



   iii. Apply the load at the point shown below in the negative y direction (axial direction):

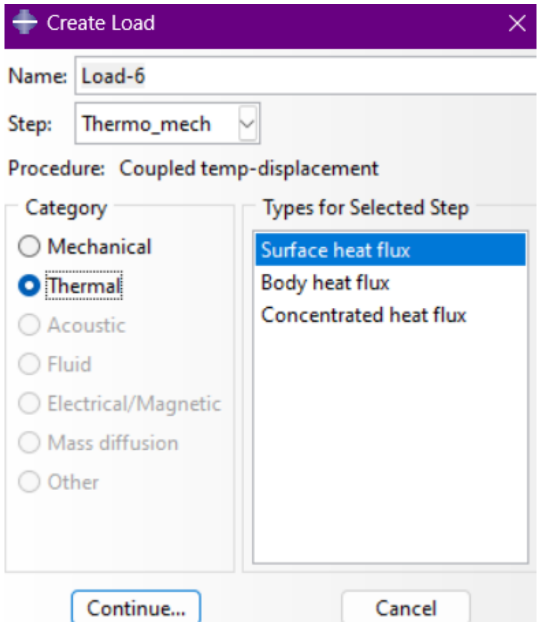


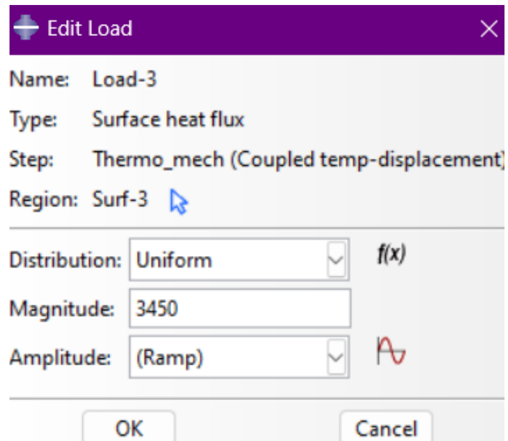
c. Define the Thermal Heat flux to the top surface:

   i. Calculate the thermal heat flux: $q_{top} = h_{top} * (T_{top} - T_{ambient})$

   $q_{top} = 6 * (873.15 - 298.15) = 3,450\ W/m^2$

ii. Apply the “Surface Heat Flux” as shown below and select the top surface (same as the top temperature boundary condition):



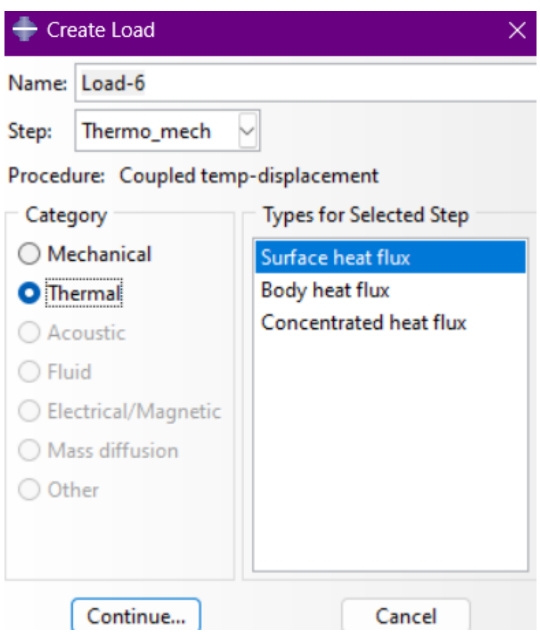


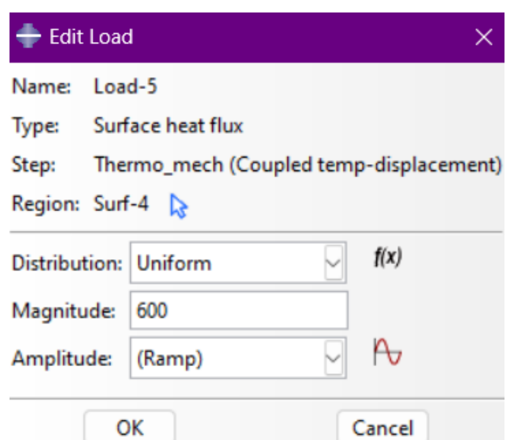
d. Define the Thermal Heat flux to the top surface:

i. Calculate the thermal heat flux: $q_{bore} = h_{bore} * (T_{bore} - T_{ambient})$

$$q_{bore} = 120 * (303.15 - 298.15) = 600\ W/m^2$$

ii. Apply the “Surface Heat Flux” as shown below and select the bore surface (same as the bore temperature boundary condition):

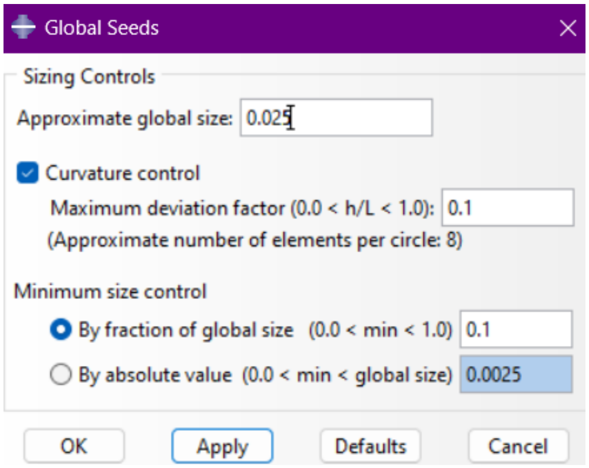




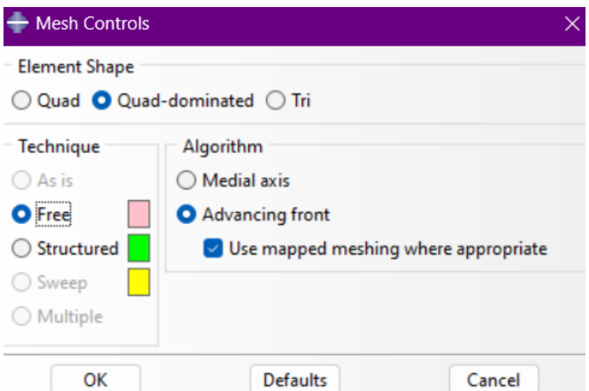
Note: Here, the +ve direction is towards the body and vice versa.

12. Mesh the part:

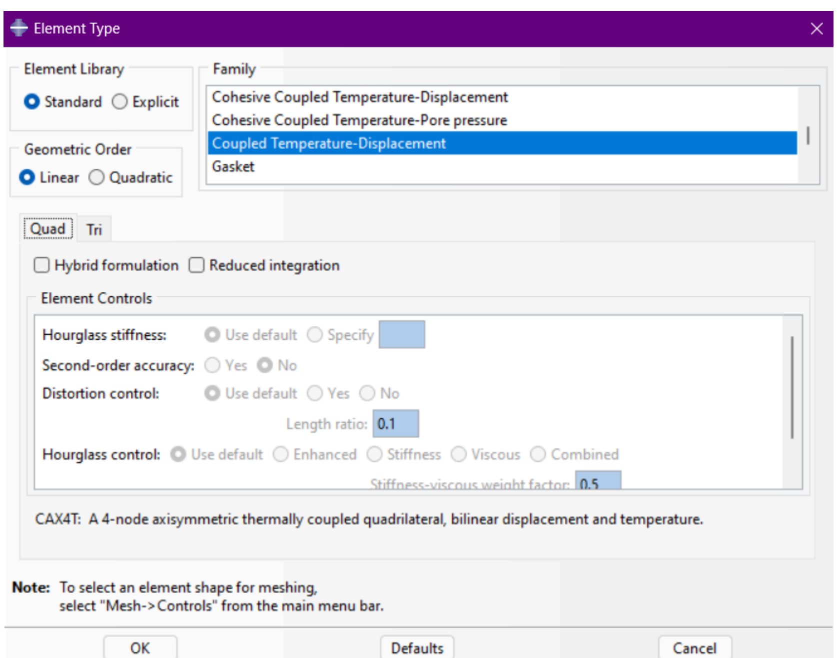
a. Create a seed size: Assign global size as 0.025



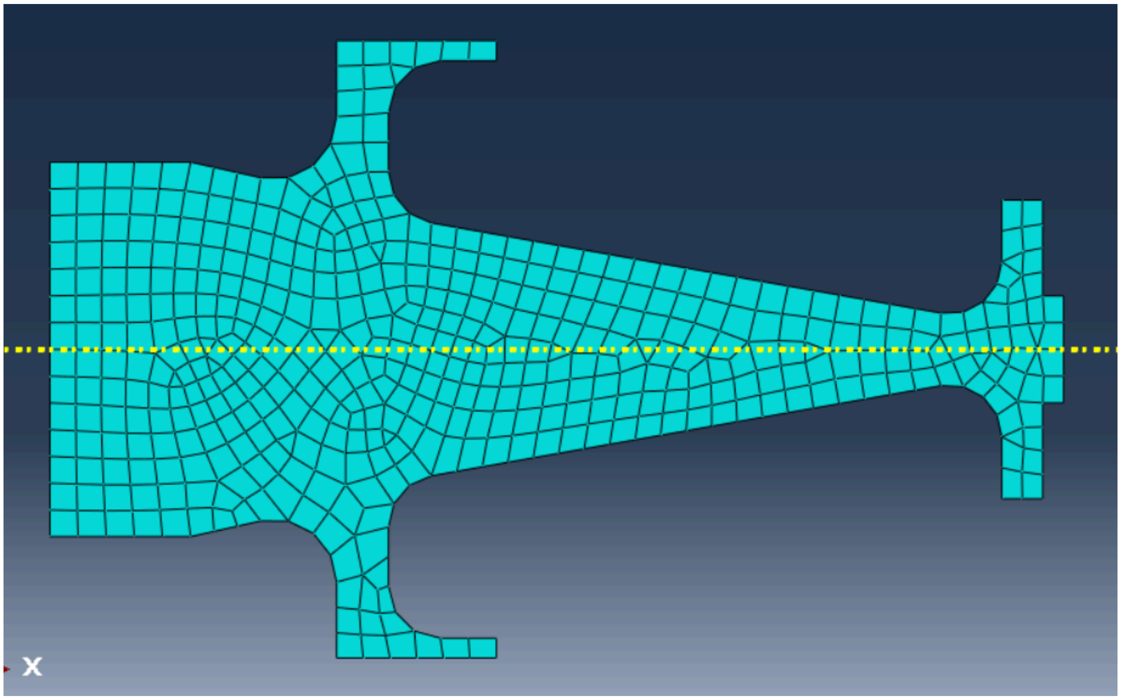
b. Define the mesh controls as below:



c. Assign element type as below:

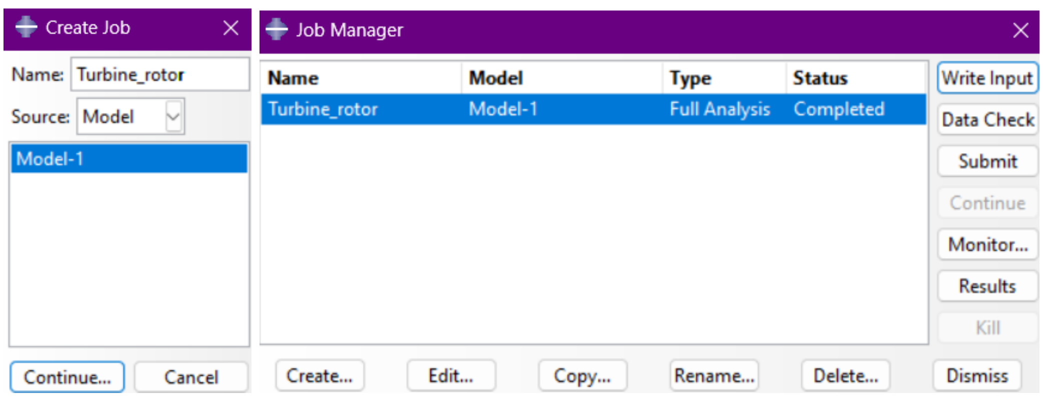


d. Mesh the part:



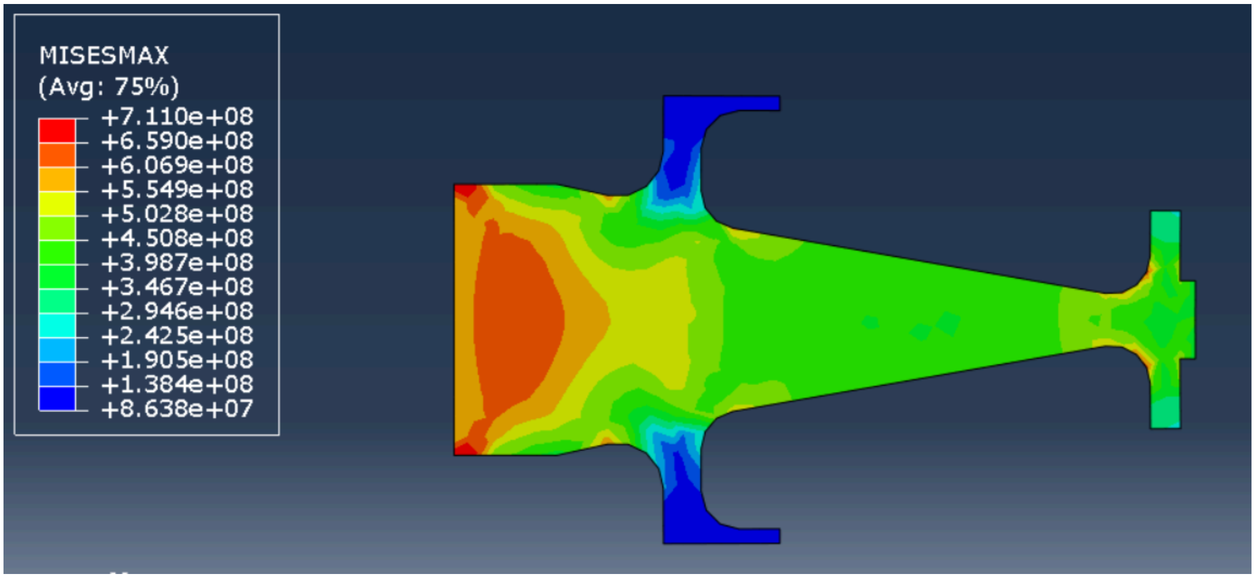
13. Submit the job:

a. Create a job: “Turbine_rotor”, and submit the job:

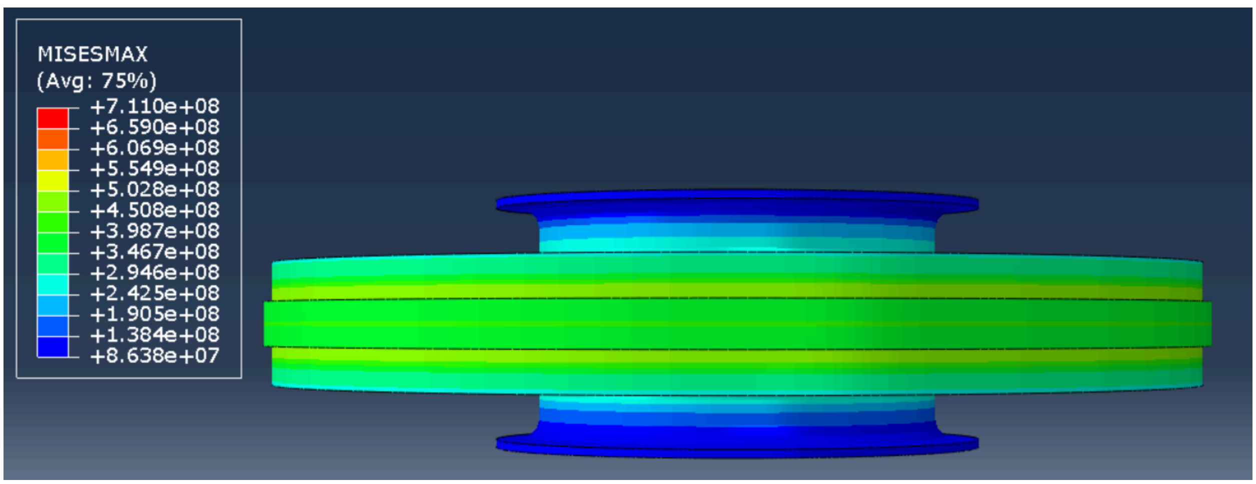


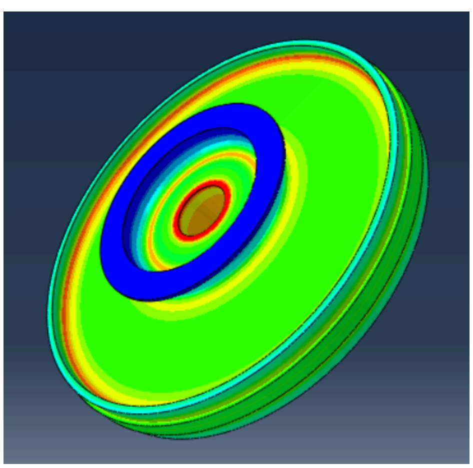
14. Visualise the results:

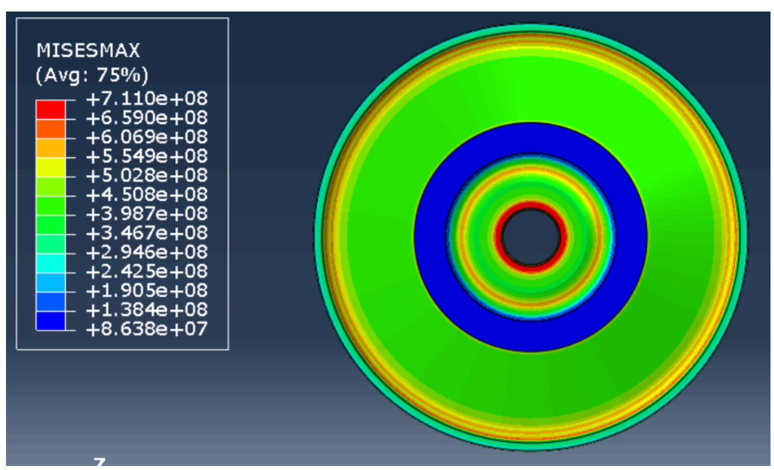
a. Visualize the max misses stress along the cross-section:

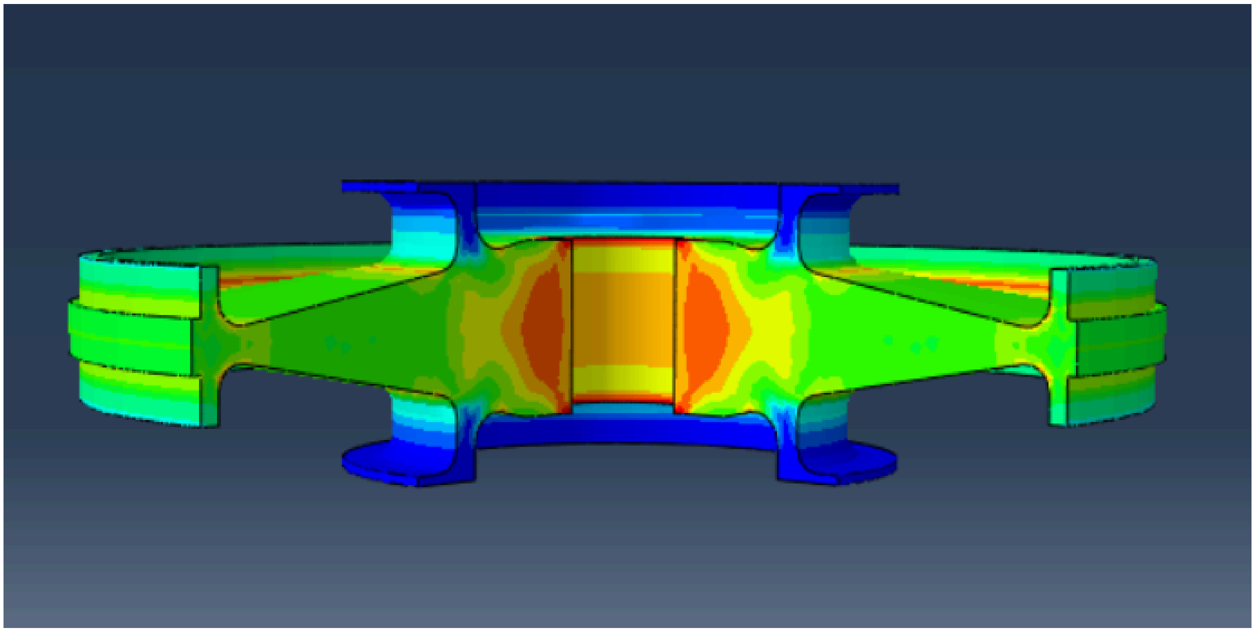


b. Visualize the max misses stress for the whole body:
   i. Go to View—ODB display options—Sweep/extrude—Sweep elements
   ii. You can play with the sweep angle and the number of segments you wish.









Pre-Processing

CAD Model:

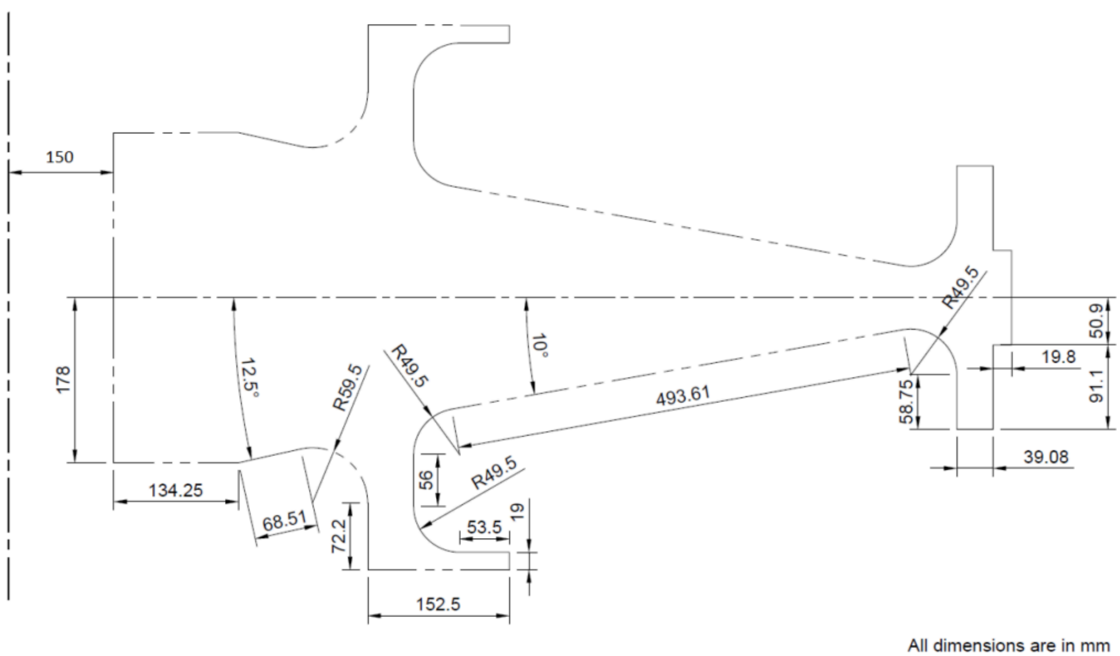
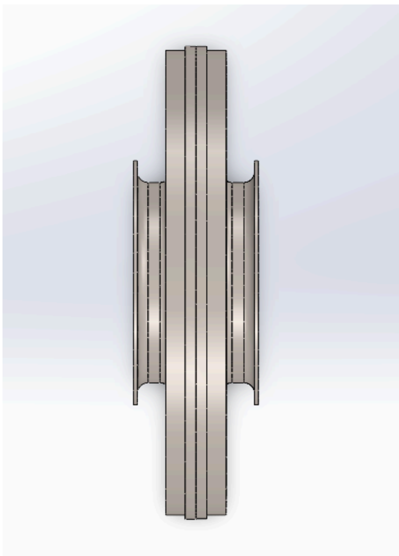
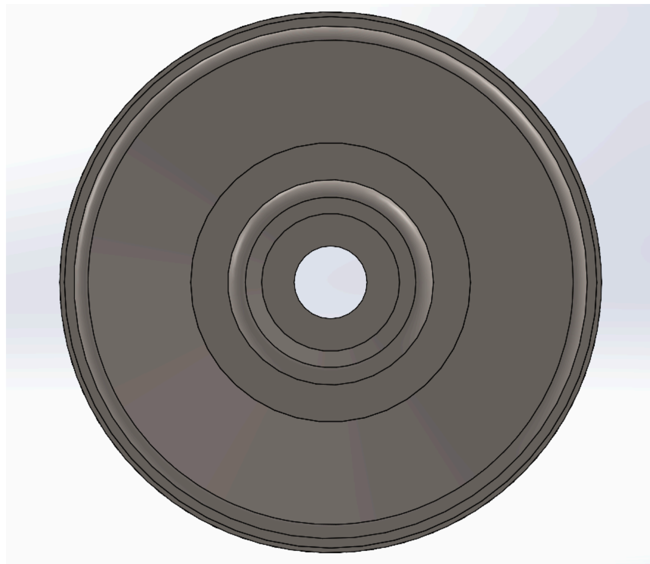
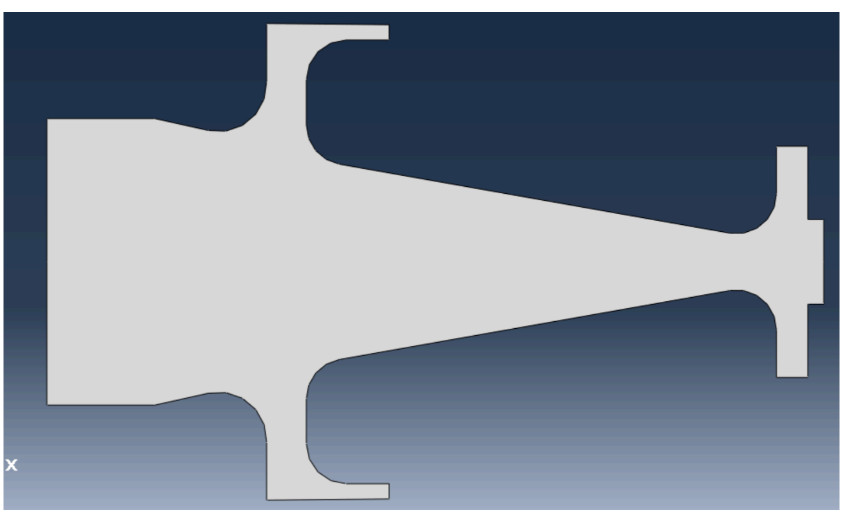


Figure 2:

- Based on the given cross-sectional drawing, I've designed a 3d model in Solidworks 2024 to verify it with the given 3d model side and front views.

- I heard a statement in a lecture last year: "In order to be a good FEA/CEA engineer, you should always exploit the advantages of any kind of symmetry in the model."
- We can observe that there's **axis symmetry**, so to analyze the 3d model, we can approximate this problem as a 2D problem and solve it.
- Why so? Consider a cylindrical coordinate system to represent the space. We can observe that the analysis is independent of the angle θ; we can consider this as an axis-symmetric problem.
- So, I created a 2d part with the cross-section and provided plane stress condition.



Assumptions:

- **Axisymmetric geometry:**
    - Assuming the turbine rotor disk has an axisymmetric geometry, the problem can be simplified to a 2D axisymmetric model.
    - This assumption is justified by the circular cross-section of the disk and the lack of any significant variations in the circumferential direction.
- **Plane stress condition:**
    - Assuming a plane stress condition in the 2D axisymmetric model. This assumption simplifies the stress analysis by reducing the problem to a 2D plane stress formulation.
- **Steady-state thermal-structural analysis:**
    - Assuming a steady-state thermal-structural analysis, neglecting any transient or dynamic effects.
    - This assumption is reasonable given the slow-changing nature of the thermal and structural loads experienced by the turbine rotor disk during regular operation.
- **Linear elastic material behavior:**
    - Assuming that the turbine rotor disk material exhibits linear elastic behavior within the expected range of stresses and deformations.
    - This assumption simplifies the constitutive relationship between stresses and strains, allowing the use of linear elasticity theory.
- As we have a body force in the loads, we need to define the density of the material. So, based on the "Youngs modulus sheet of materials" [1], I learned that ASTM A36 Steel has the same properties as given in the question.

  Regarding material properties, I've defined the density of the material as 7800 kg/m^3.
- I assumed the initial temperature to be room temperature: 298.15 K.
- Instead of defining a Z-axis symmetry boundary condition at the central axis of the bore, I've defined the surface of the bore to be a fixed displacement boundary condition because there will be a shaft at the bore, so considering the analysis w.r.t, this condition is valid.
- **Coordinate system :**
    - The z-axis is aligned with the central axis of the turbine rotor disk, representing the axial direction.
    - The r-axis represents the radial direction, extending from the disk's center to the outer periphery.
    - The θ-axis represents the circumferential direction, with the angular position measured around the central axis.

Meshing

Element selection:

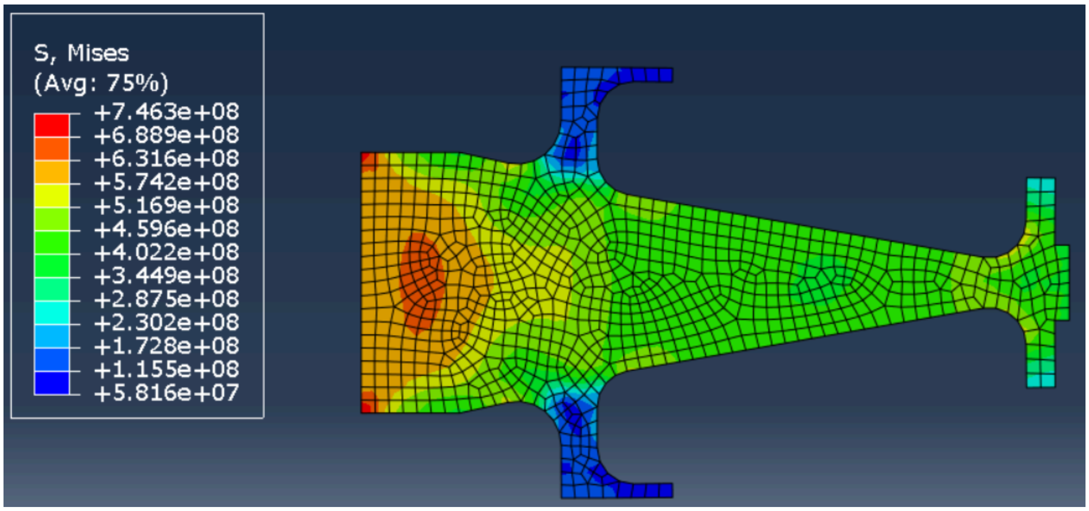
There are multiple element choices:

- Quad Element (Free or Structured):
    - **Linear:** CAX4T: A 4-node axisymmetric thermally coupled quadrilateral, bilinear displacement and temperature.
        - ***Hybrid Formulation:*** CAX4HT: A 4-node axisymmetric thermally coupled quadrilateral, bilinear displacement and temperature, hybrid, constant pressure.
        - ***Reduced Integration:*** CAX4RT: A 4-node thermally coupled axisymmetric quadrilateral, bilinear displacement and temperature, reduced integration, hourglass control.
        - ***Hybrid and Reduced Integration:*** CAX4RHT: A 4-node thermally coupled axisymmetric quadrilateral, bilinear displacement and temperature, reduced integration, hourglass control.
    - **Quadratic:** CAX8T: An 8-node axisymmetric thermally coupled quadrilateral, biquadratic displacement, bilinear temperature.
        - ***Hybrid Formulation:*** CAX8HT: An 8-node axisymmetric thermally coupled quadrilateral, biquadratic displacement, bilinear temperature, hybrid, linear pressure.
        - ***Reduced Integration:*** CAX8RT: An 8-node axisymmetric thermally coupled quadrilateral, biquadratic displacement, bilinear temperature, reduced integration.
        - ***Hybrid and Reduced Integration:*** CAX8RHT: An 8-node axisymmetric thermally coupled quadrilateral, biquadratic displacement, bilinear temperature, hybrid, linear pressure, reduced integration.
- Tri Element (Free or Structured): CAX3T
    - **Linear:** CAX3T: A 3-node thermally coupled axisymmetric triangle, linear displacement and temperature.
    - **Quadratic:** CAX6MT: A 6-node modified axisymmetric thermally coupled triangle, hybrid, linear pressure, hourglass control.
        - ***Hybrid Formulation:*** CAX6MHT: A 6-node modified axisymmetric thermally coupled triangle, hybrid, linear pressure, hourglass control.

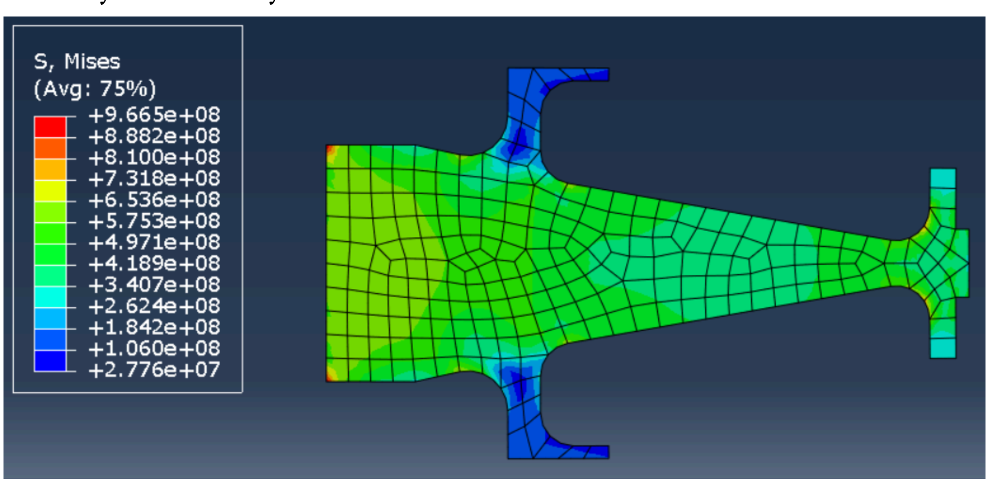
So, considering all the features and choices, I prefer using the following:

1. **CAX4RT:**

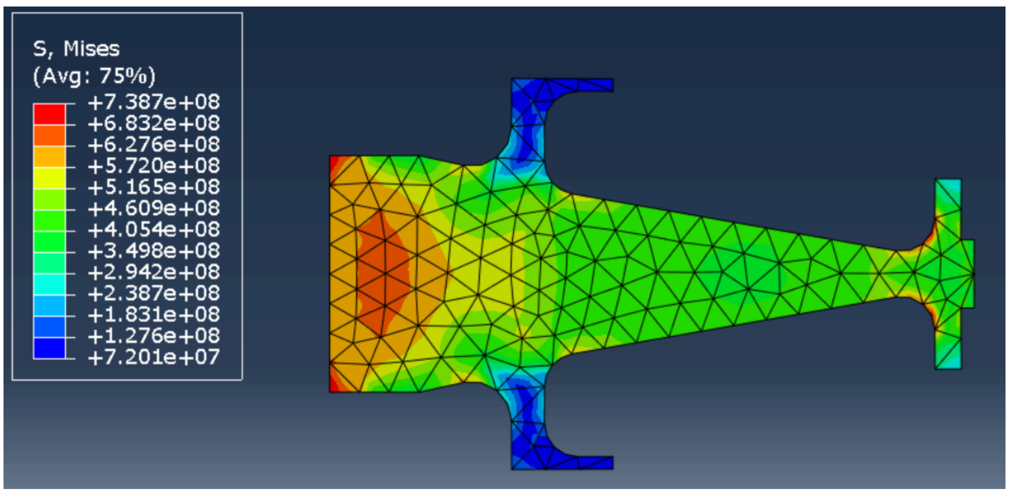
a. This uses reduced integration, which can help reduce the computational cost and improve the accuracy of the solution, especially for problems with bending-dominated behavior.
b. This uses the hourglass control feature, which helps address potential instabilities from reduced integration.
c. The bilinear displacement and temperature formulation provide a good balance between accuracy and computational efficiency.



2. **CAX8RHT:**
   a. This quadratic element type offers higher-order displacement and temperature interpolation, which can improve the accuracy of the solution, especially for problems with smooth fields.
   b. The hybrid formulation, which includes a linear pressure term, can help to improve the performance for problems involving incompressible or nearly incompressible materials.
   c. The reduced integration with hourglass control helps maintain the solution's accuracy and stability.



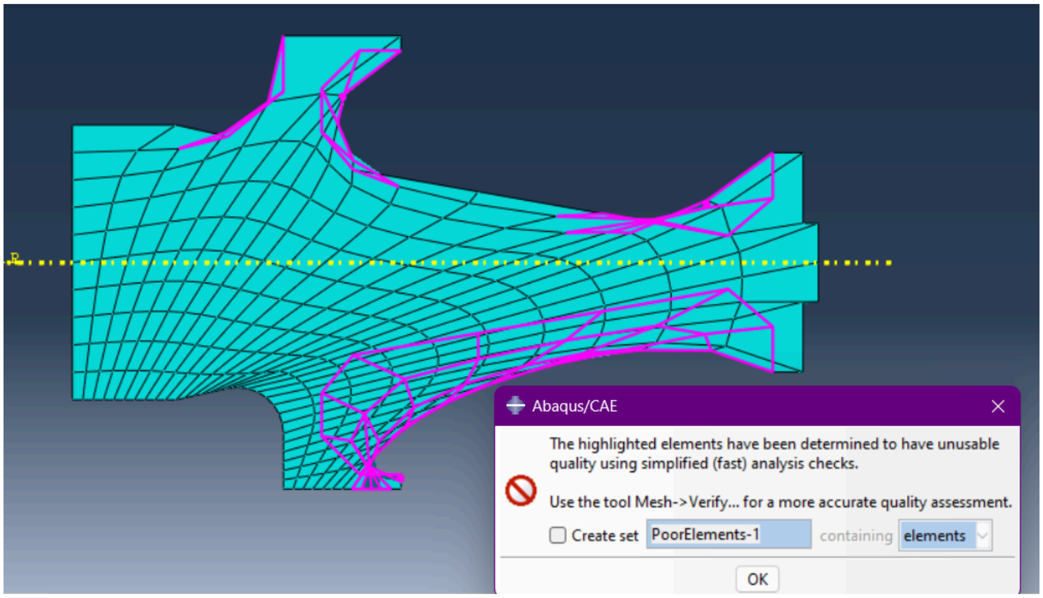
3. CAX6MHT:
    a. It provides a higher-order quadratic displacement and temperature formulation, which can capture more complex field variations than the linear CAX3T element.
    b. The hybrid formulation includes a linear pressure term, which can help improve the performance for problems involving incompressible or nearly incompressible materials.
    c. The hourglass control feature helps address potential instabilities arising from the higher-order formulation.
    d. The combination of the quadratic displacement and temperature, hybrid formulation, and hourglass control makes this element type suitable for a wide range of axisymmetric thermally coupled problems where accurate results are required.



Observations: Comparing the results obtained from three different element types, **CAX8RHT** performed better than the other two because it has higher order elements, which can provide better results with fewer elements than linear. It also has a Hybrid Formulation with Linear Pressure and Reduced Integration with Hourglass Control, which makes it the best choice for this problem-solving.
We can observe from the Quad and tri-element choices that Quad elements converged better and faster comparatively. Hence, it has been proved that quad elements are better than tri elements.

So, by using the **CAX8RHT** element type, we got the maximum mises stress to be **9.665e+08**. So, this is the final maximum mises stress from the analysis.

Using a structured mesh ultimately decreased the quality of the elements:



So, I used free mesh in quad and triangular elements.

Element size:

Two ways to define the element size are Global and edge seeds. I've tried using edge seeds, but as we've higher small and non-linear edges, I don't have much control over the number of elements because we should use integer values while defining seed edges using a number of elements. All the analyses are done while defining Global seed edges.
I've tried out different element types and sizes for each element type and finally preferred **CAX8RHT**.
The maximum element size I could use without any license issues was 0.035.

Mesh convergence study: This study determines the optimal mesh size for our analysis by ensuring that the results converge to a stable solution as the mesh is refined.
H-Refinement:

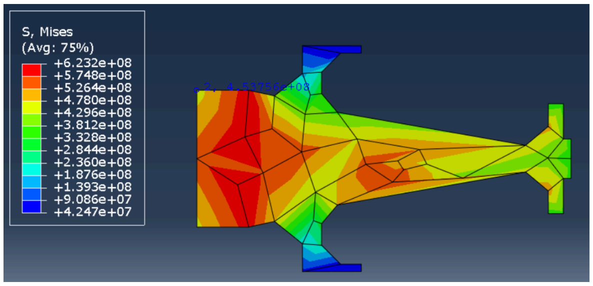
- Start with h1 no of elements and then do the same analysis with h2, then h3.... It goes on until we find convergence.
- h1>h2>h3>.....
- In general, we consider h1, 2h1, 4h1, 8h1,...
- Finally, for finding α, which is the rate of convergence, we need to evaluate the following:

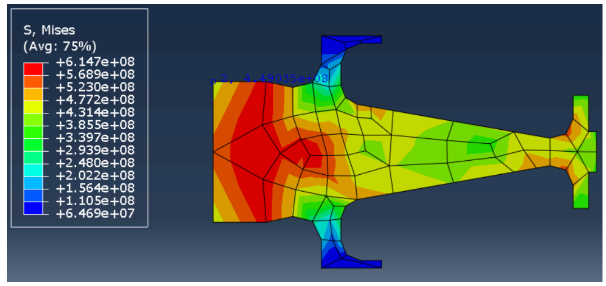
  $\frac{Uh3-Uh2}{Uh3-Uh1} \approx \left(\frac{h2}{h1}\right)^{\alpha}$ Here, we consider a single point and take U at that point.

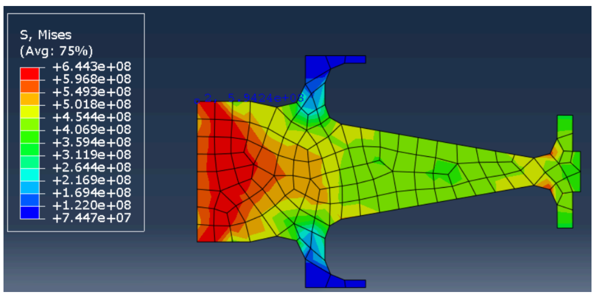
- α Indicates how fast the solution will converge to the exact one.

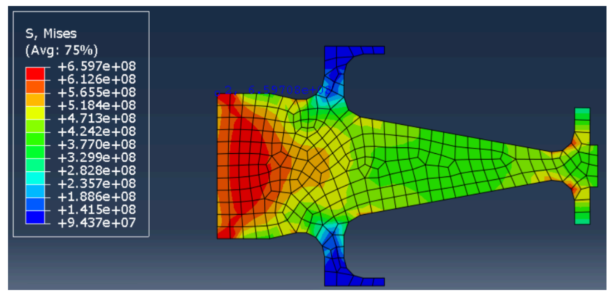
Linear Quad Elements:

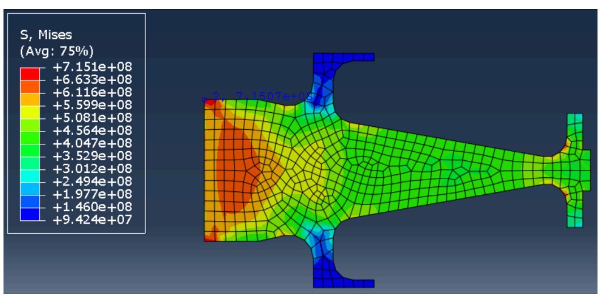
Element used: CAX4T

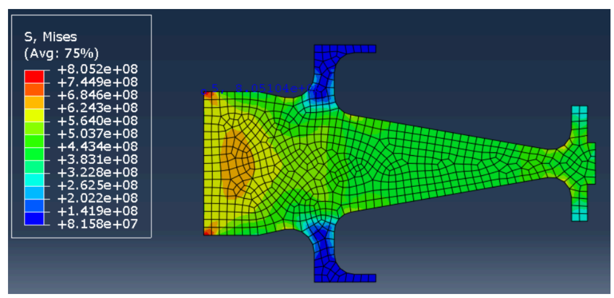












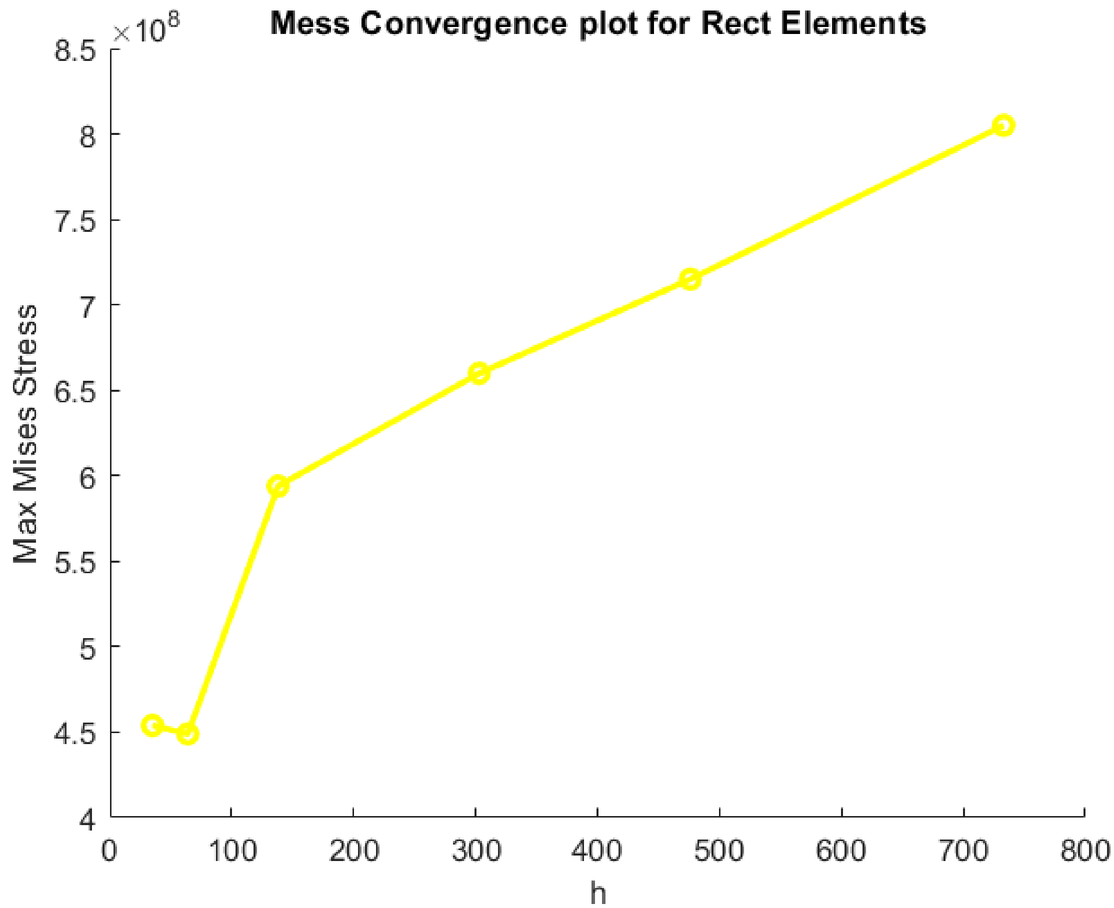
X2 = 6×1

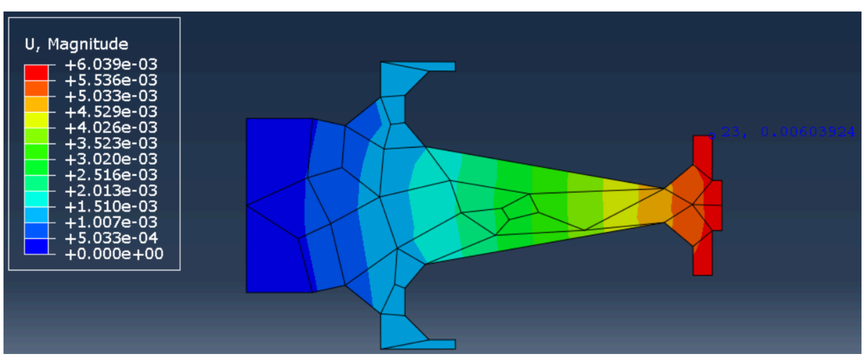
	1
1	35
2	64
3	138
4	303
5	476
6	733

Y2 = 6×1

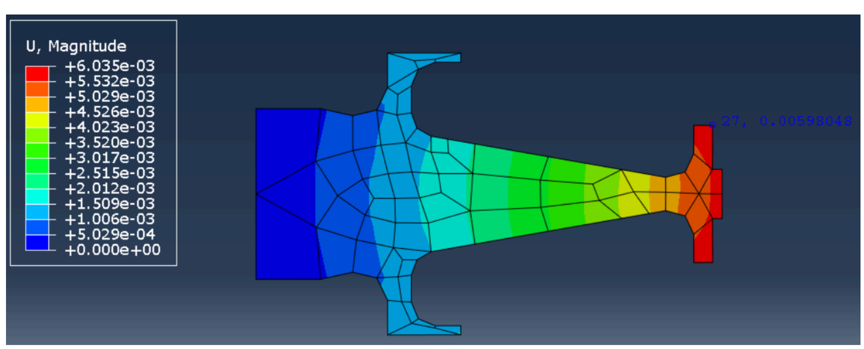
	1
1	454000000
2	449000000
3	594000000
4	660000000
5	715000000
6	805000000

According to the plot, after approx >800 elements, the mesh is converging.

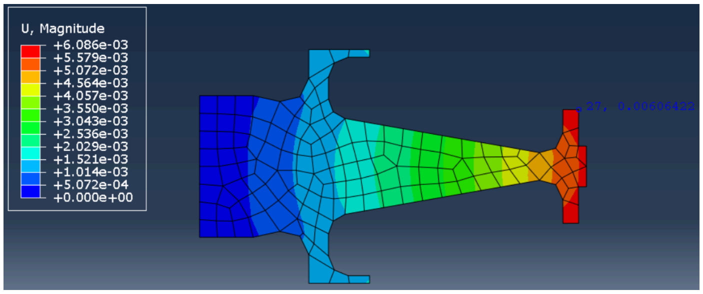




U = 0.006039



U = 0.00598

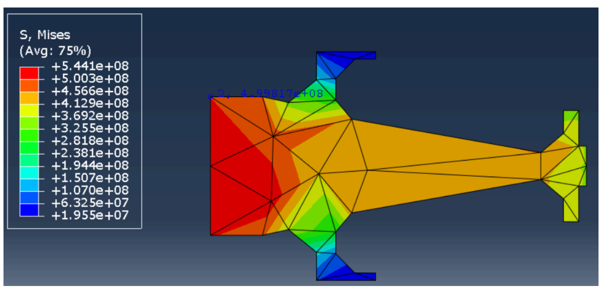


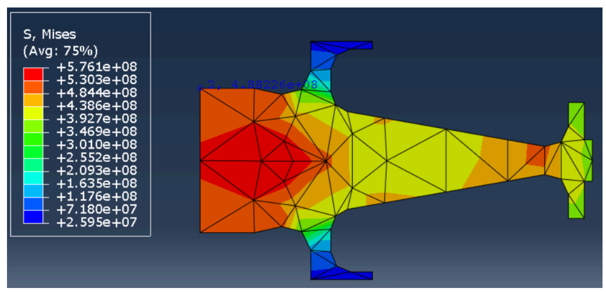
U = 0.00606422

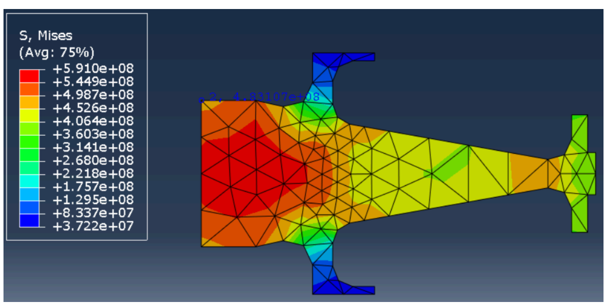
- $\frac{Uh3-Uh2}{Uh3-Uh1} \approx (\frac{h2}{h1})^{\alpha}$ By using this formula and the displacement values from the above, we got the α to be 1.99788

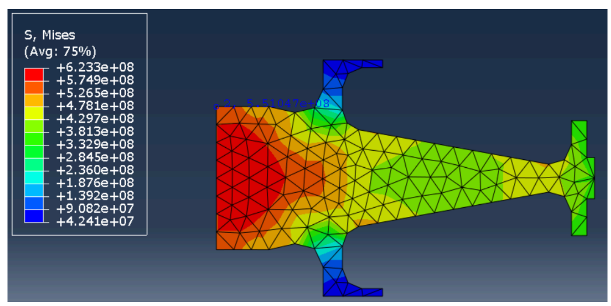
Linear Tri Elements:

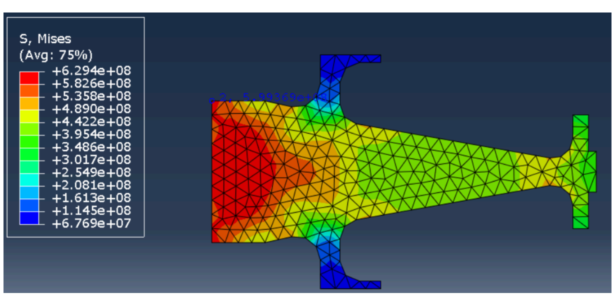
Element used: CAX3T:

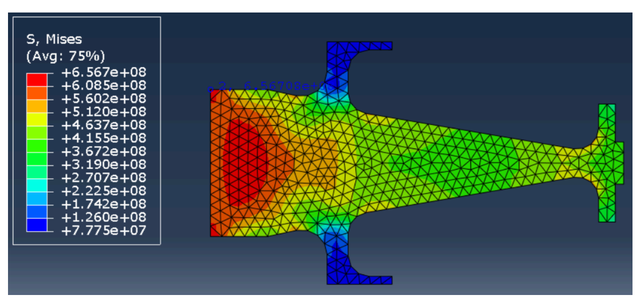


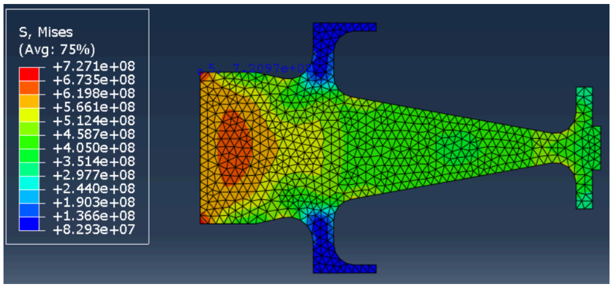


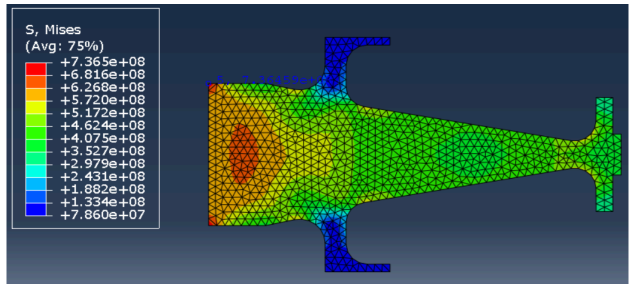












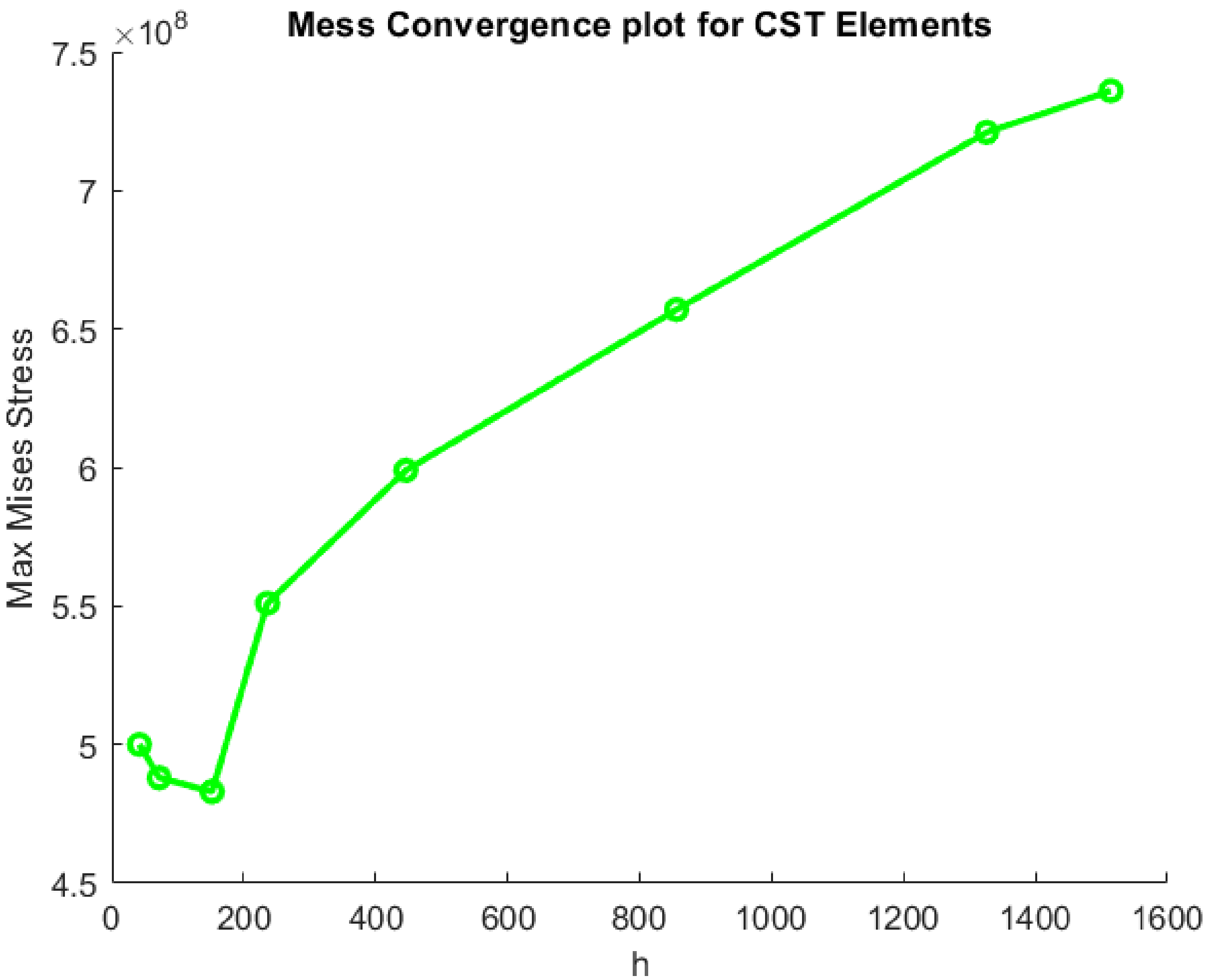
X2 = *8×1*

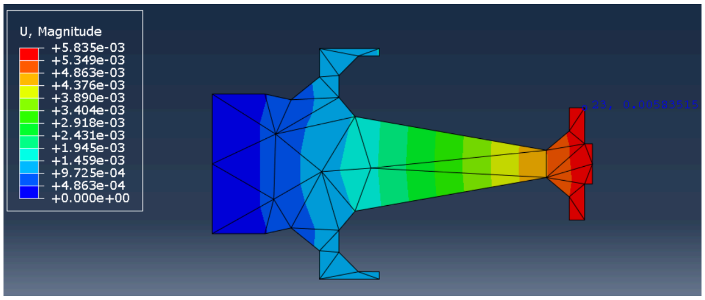
	1
1	42
2	72
3	152
4	236
5	446
6	856
7	1326
8	1514

Y2 = *8×1*

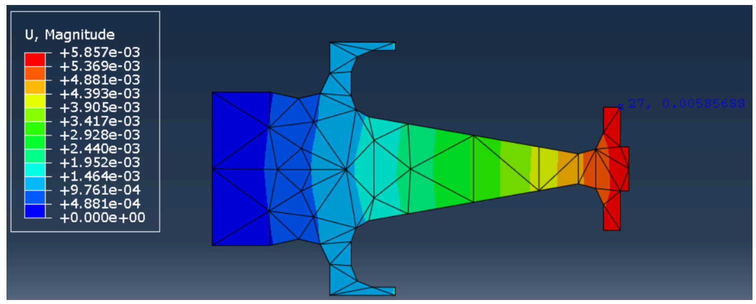
	1
1	500000000
2	488000000
3	483000000
4	551000000
5	599000000
6	657000000
7	721000000
8	736000000

<u>According to the plot, after approx 1.6k elements, the mesh is converging</u>

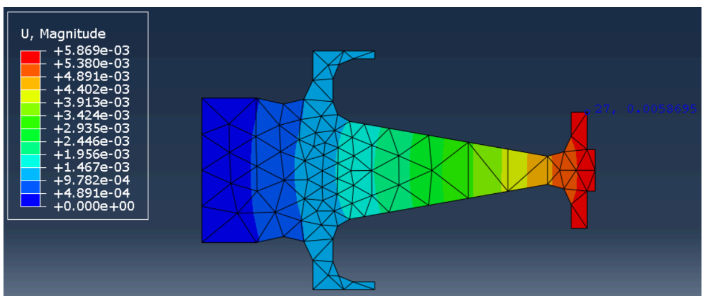




U = 0.00583515



U = 0.00585688

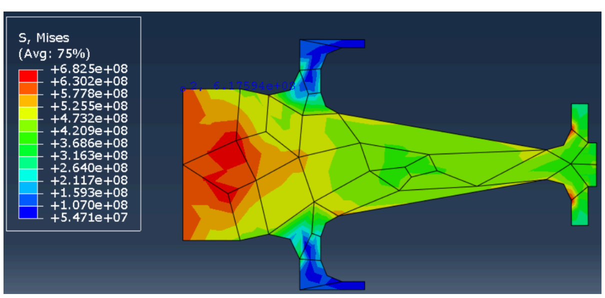


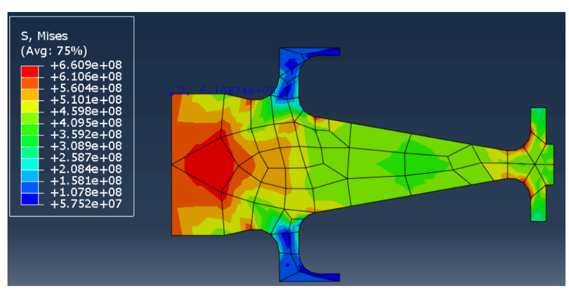
U = 0.00585695

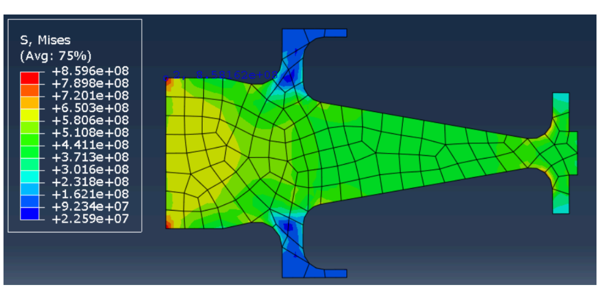
- $\frac{Uh3-Uh2}{Uh3-Uh1} \approx (\frac{h2}{h1})^{\alpha}$ By using this formula and the displacement values from the above, we got the α to be -10.6516

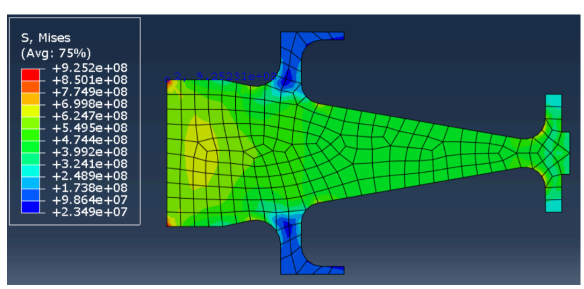
Quadratic Quad Elements:

Element used: CAX8RT:









X2 = *4×1*

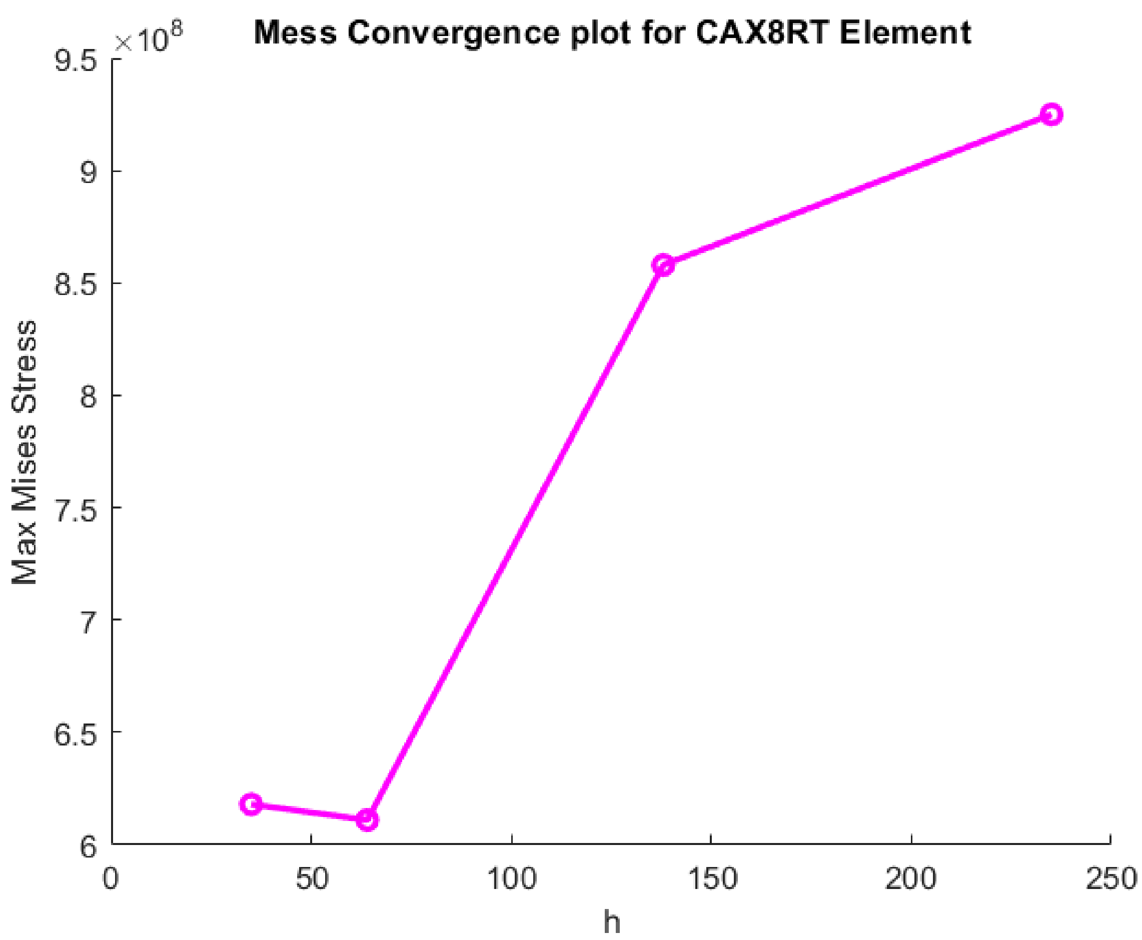
	1
1	35
2	64
3	138
4	235

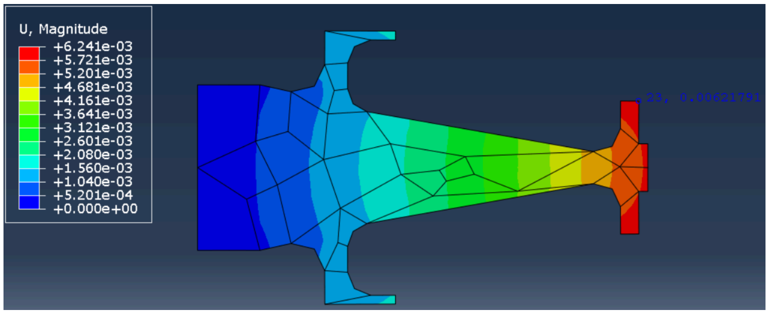
Y2 = *4×1*

	1
1	618000000
2	611000000
3	858000000
4	925000000

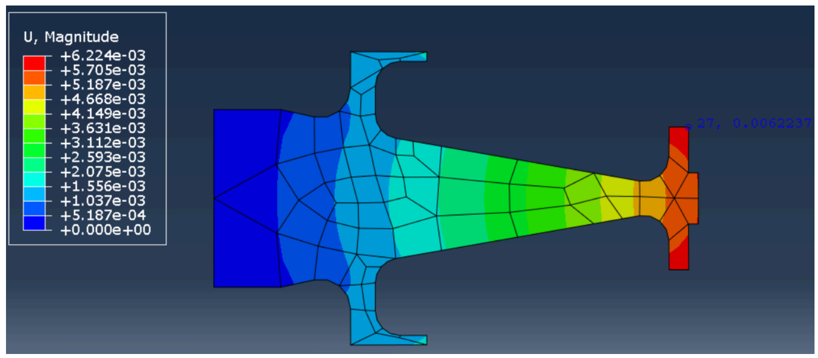
According to the plot, after approx 250 elements, the mesh is converging

Element Sizes	1	0.5	0.05	0.035

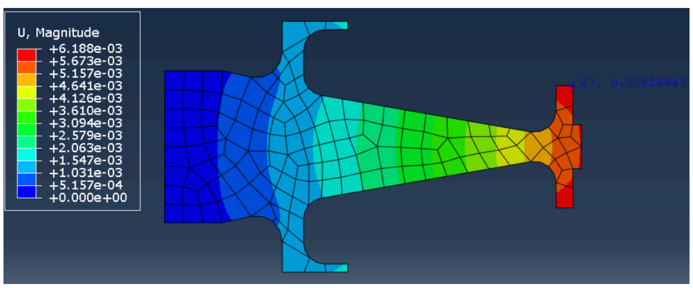




U = 0.00621791



U = 0.00622373



U = 0.00618843

- $\frac{Uh3-Uh2}{Uh3-Uh1} \approx (\frac{h2}{h1})^{\alpha}$ By using this formula and the displacement values from the above, we got the α to be 0.2985260011

Boundary conditions

a. I've already stated a few points in the "Abacus Analysis steps" section. Here, I'll discuss a few more points.

   i. The body force, W, experienced by the turbine rotor disk due to the rotational velocity creates a centrifugal force on the disk, which acts as a body force. The magnitude of this body force can be calculated using the formula:

   $W = mw^2r$, Where:

   - m is the mass of the rotor disk
   - ω is the angular velocity in rad/s
   - r is the radius of the disk

ii. The mass of the rotor disk is calculated by the abacus solver using the density and the dimensions of the disk. That is why density plays a vital role in having body forces.
iii. This body force, acting radially outward, will contribute to the stresses and deformation experienced by the turbine disk under the given operating conditions. The analysis should capture the effects of this rotational body force on the overall behavior of the disk.

Let’s discuss the effect of body force on the rotor.

- It will contribute to the overall stress state in the turbine disk, including generating radial, hoop and von Mises stresses.
- Radial Stress:
    - It will create radial stresses within the turbine disk. These radial stresses will be compressive near the center (bore) of the disk and tensile near the outer periphery. This is because the centrifugal forces acting on the material are relatively small at the center, leading to compressive radial stresses. In contrast, the centrifugal forces are larger at the outer regions, resulting in tensile radial stresses. The magnitude of these radial stresses will be directly proportional to the square of the angular velocity and the distance from the disk's center.
    - The magnitude of the radial stresses will be directly proportional to the square of the angular velocity and the distance from the disk's center.
- Hoop Stress:
    - It will also generate hoop (circumferential) stresses in the turbine disk. This will be tensile, with the highest values occurring near the outer periphery of the disk. The hoop stresses are induced by the centrifugal expansion of the disk material as it rotates.
- Von Mises Stress:
    - Combining the radial and hoop stresses and other stress components (e.g., axial stresses due to the force F) will result in a complex stress state within the turbine disk.
    - The von Mises stress, which is a scalar measure of the distortion energy in the material, will capture the overall intensity of the stress state.
    - The maximum von Mises stress is an important design criterion, as it should be kept within the allowable limits to prevent failure of the turbine disk.

b. I’ve already stated a few points and calculations in the “Abacus Analysis steps” section. Here, I’ll discuss a few more points.

I. This axial force, F, will contribute to the turbine disk's overall stress state and deformation, in addition to the radial stresses and deformations caused by the rotational body force, W.

II. The axial force, F, will generate axial stresses within the turbine disk, which will superimpose the radial and hoop stresses, resulting in a complex three-dimensional stress field. The analysis should capture the combined effect of the rotational body force, W, and the axial force, F, on the turbine disk's overall stress distribution and deformation.

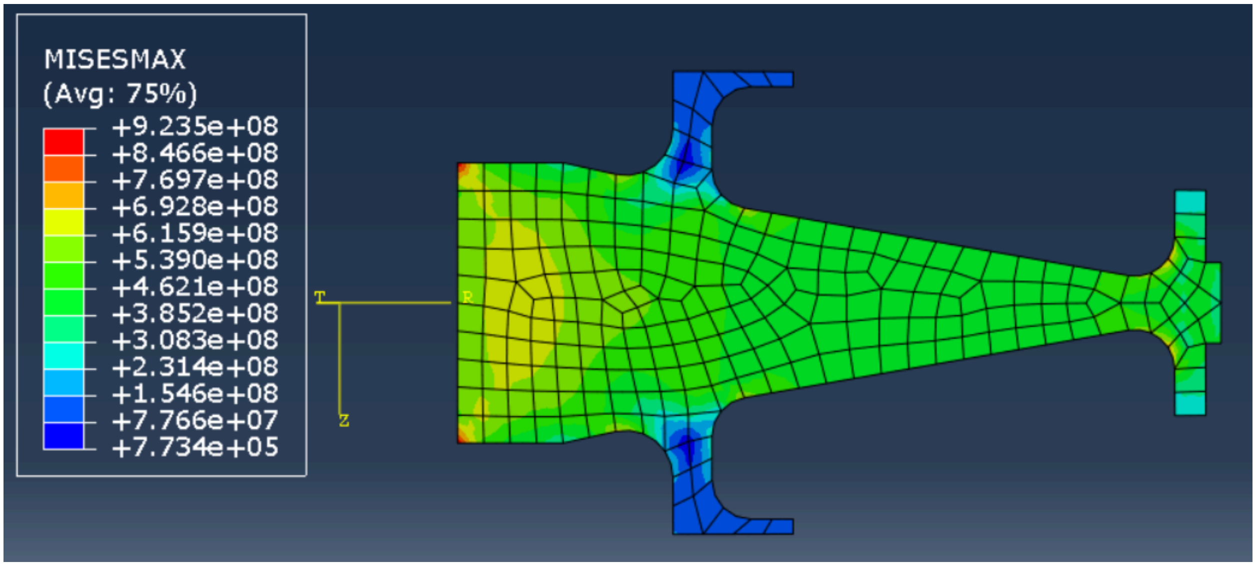
Post-processing

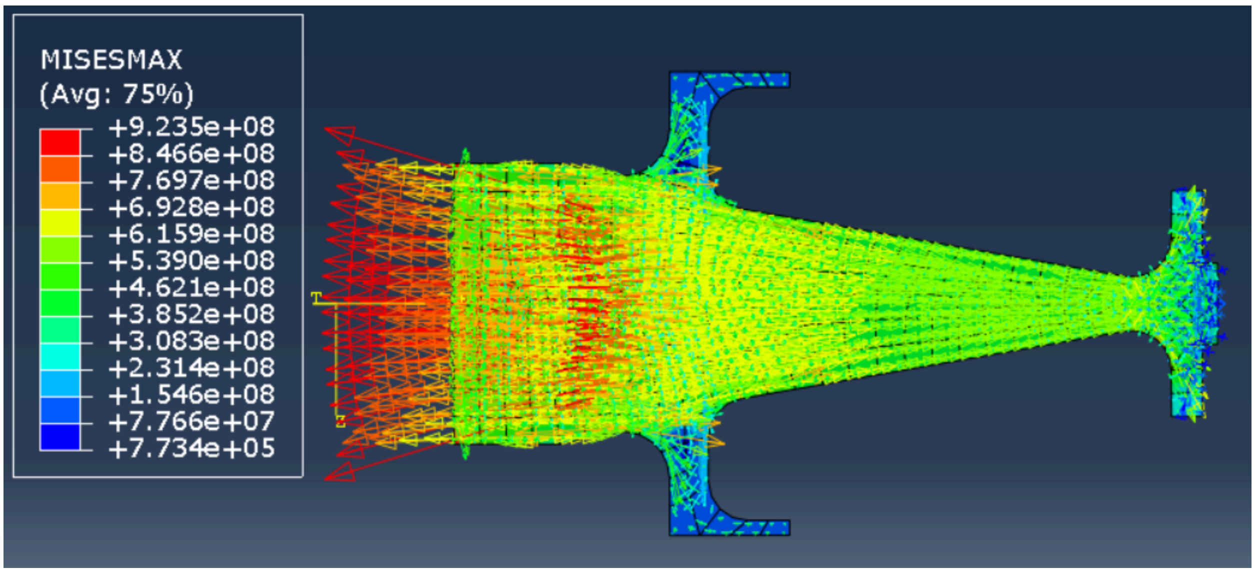
maximum von Mises stress	9.235E+08
maximum hoop stress	4.126E+08
maximum radial stress	1.042E+09
maximum deformation	1.062E-02
maximum heat flux vector	6.324E+04

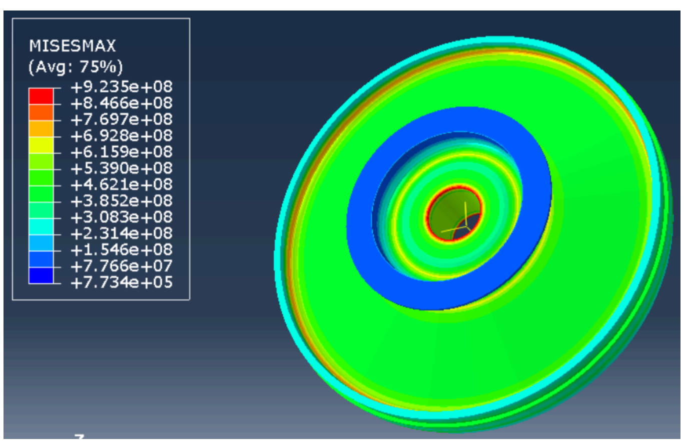
Impact of each on the failure of the model:

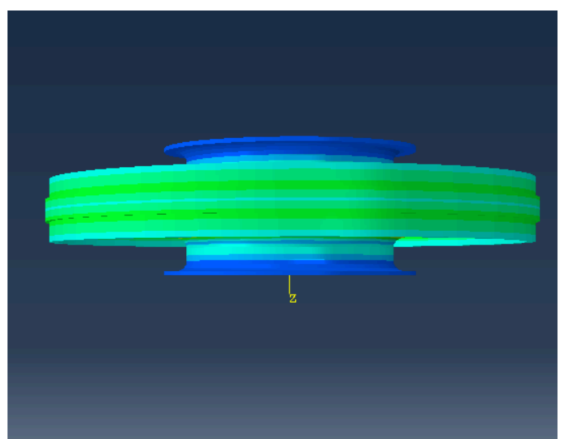
maximum von Mises stress	Exceeding the yield strength can lead to plastic deformation and eventual structural failure of the disk.
maximum hoop stress	High hoop stresses can cause radial cracking and catastrophic disk burst failure.
maximum radial stress	Excessive radial stresses can lead to axial cracks and potential disk separation.
maximum deformation	Excessive deformation can cause blade-to-casing rubs, leading to blade damage and potential disk imbalance.
maximum heat flux vector	High heat fluxes can cause material degradation, thermal stresses, and, ultimately, thermal failure of the disk.

Maximum von Mises stress:

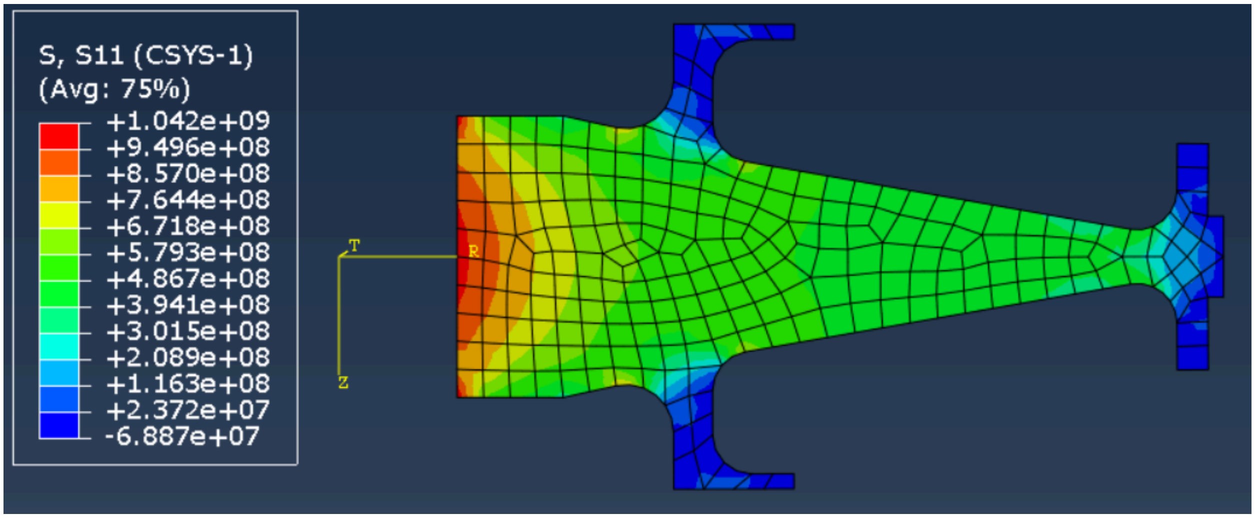


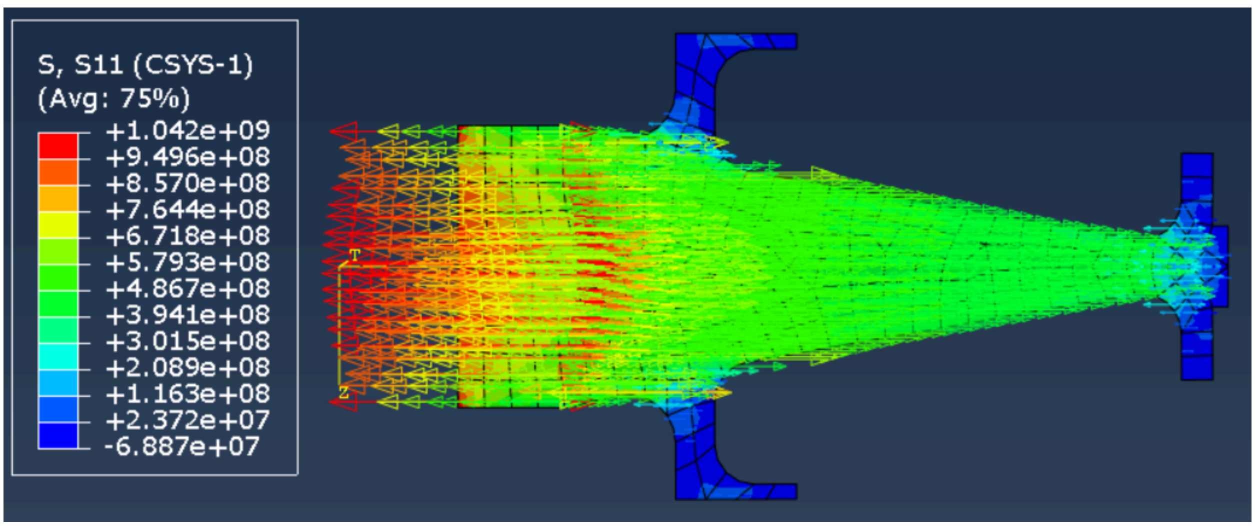


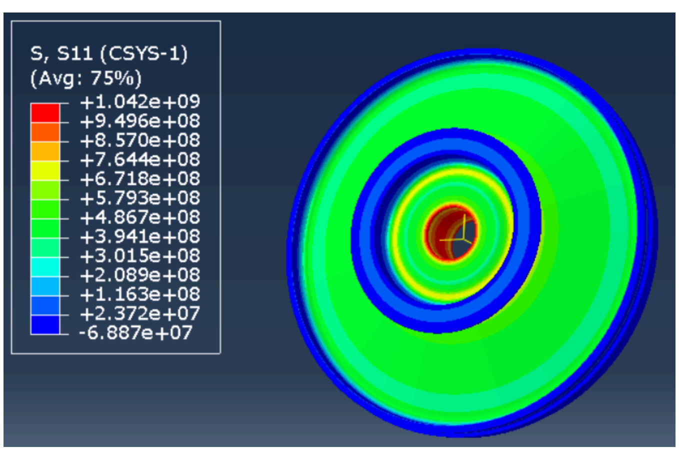


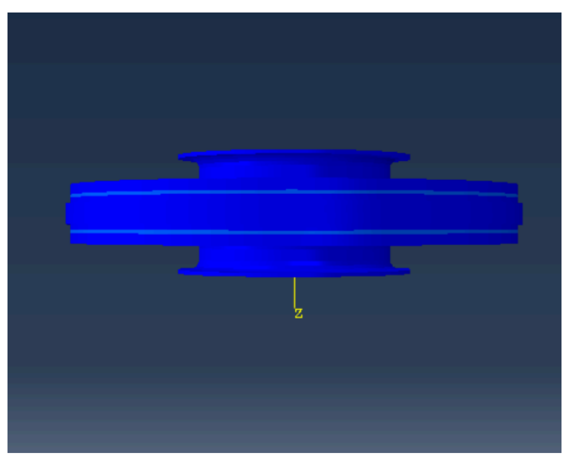


Maximum radial stress:

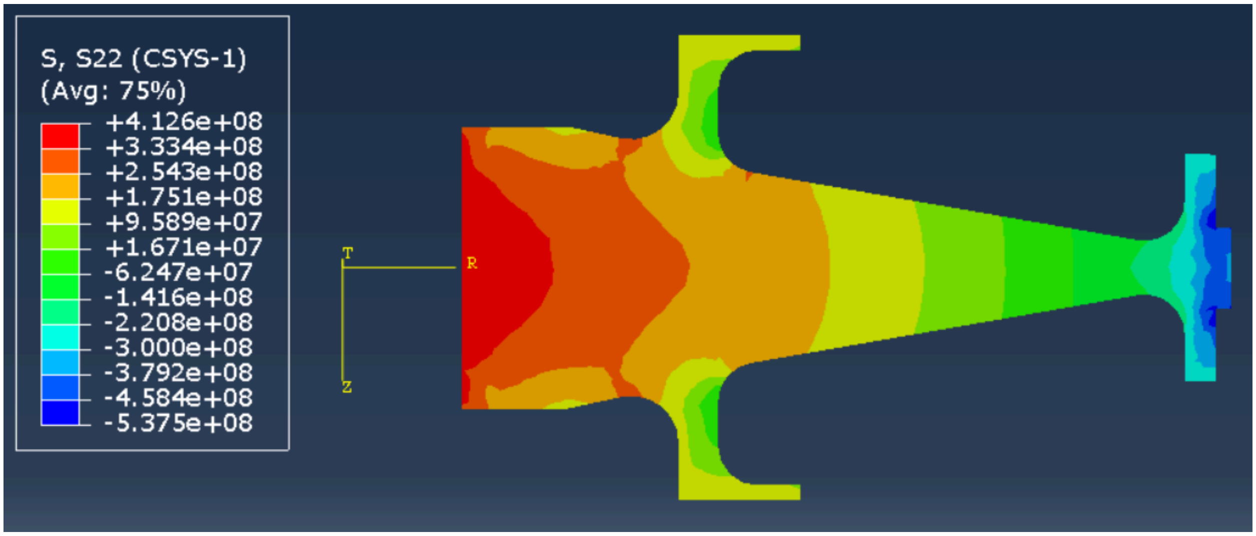


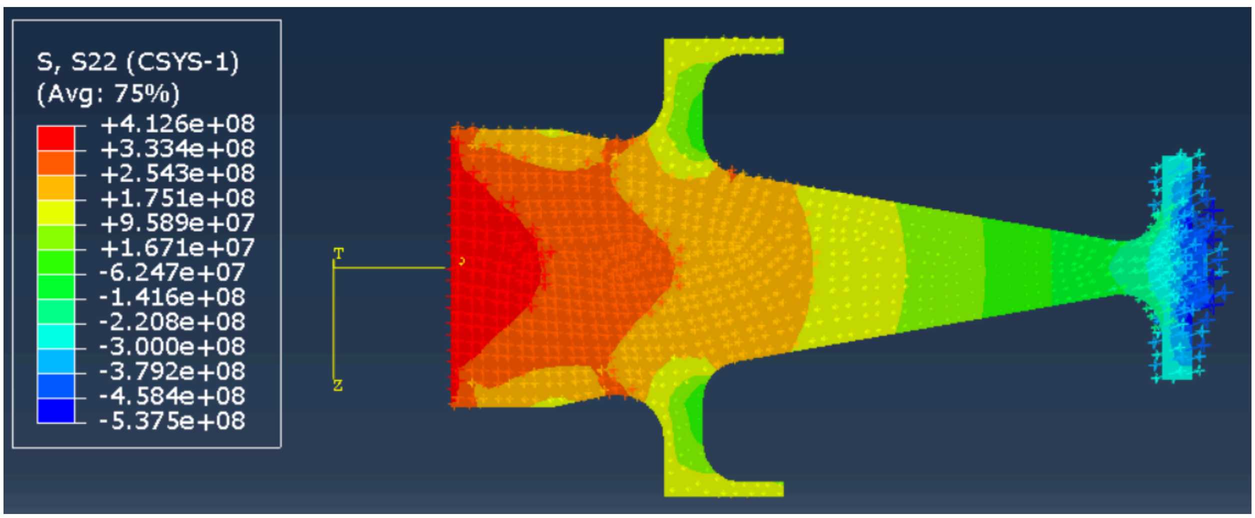


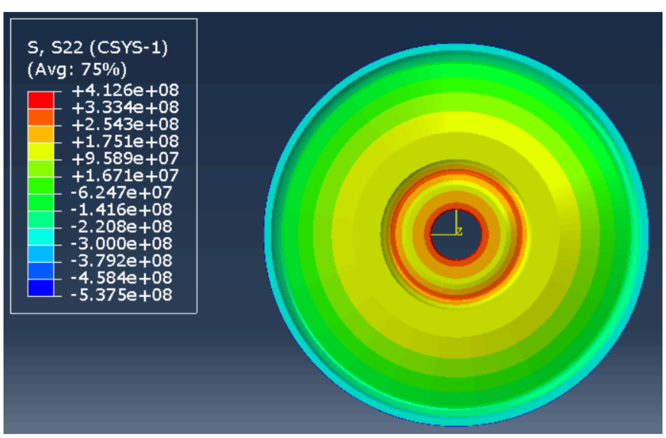


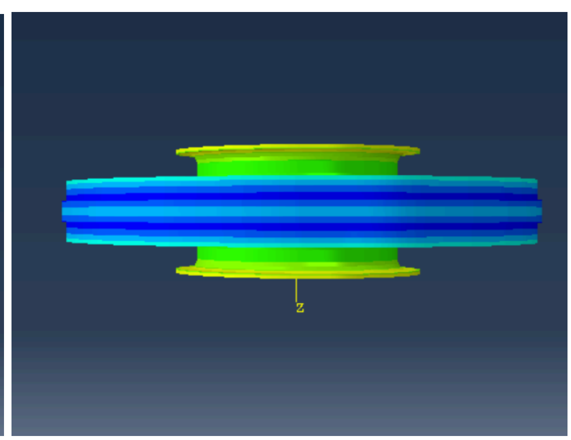


Maximum Hoops stress:

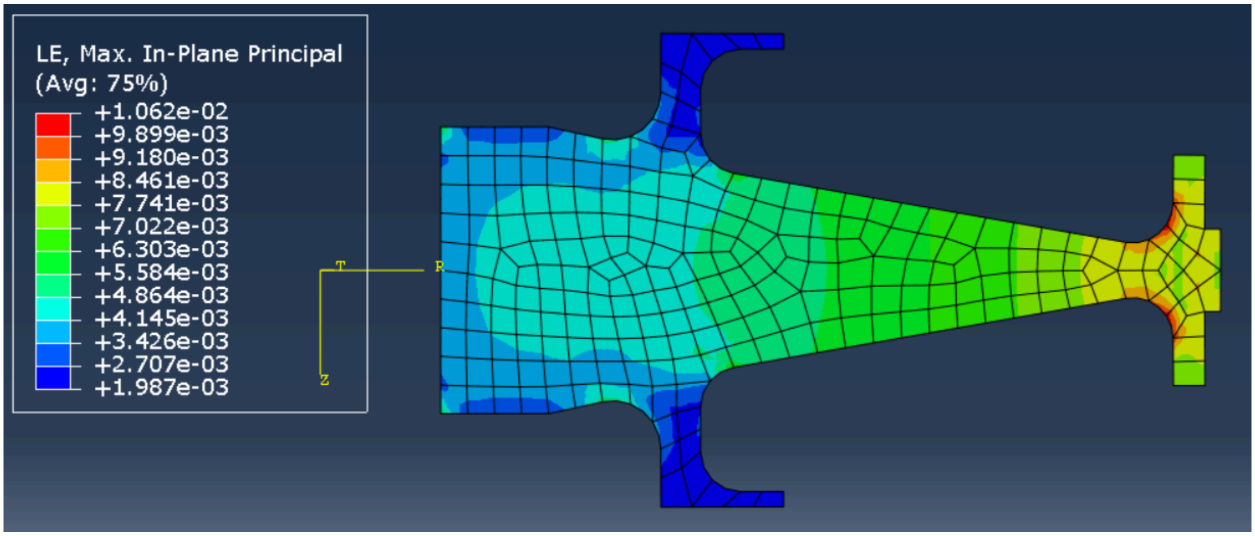


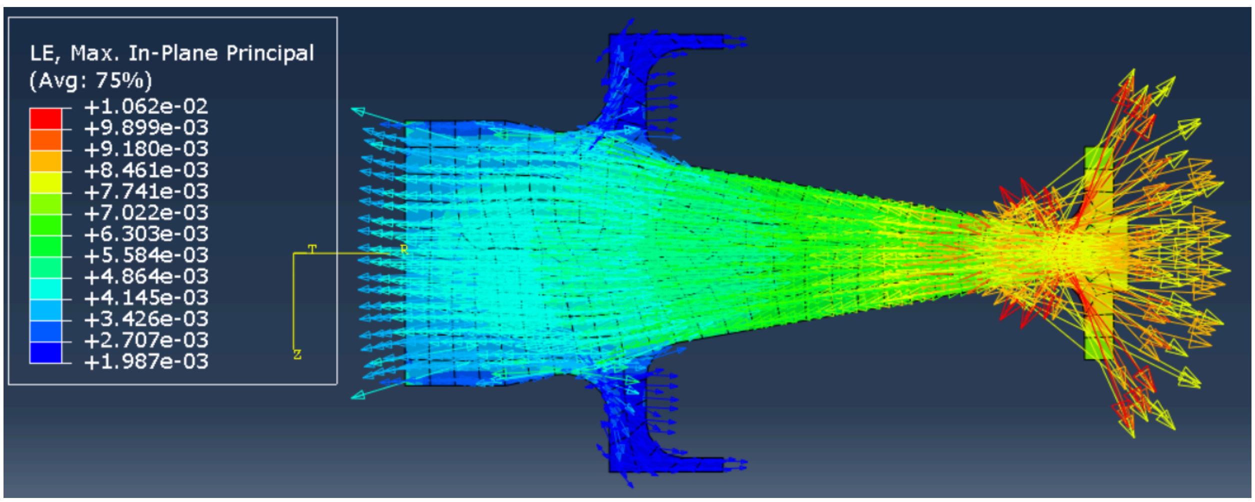


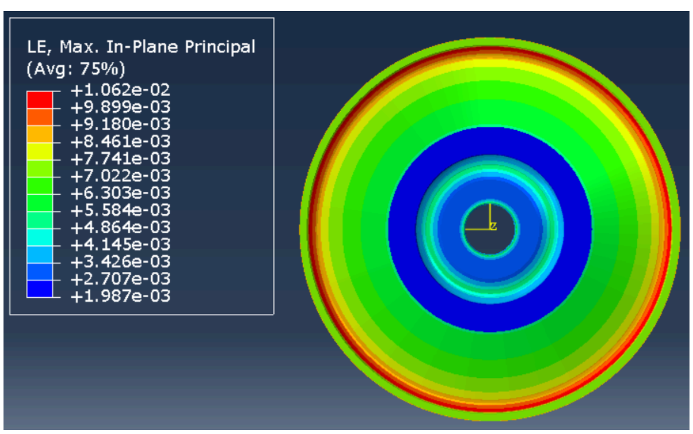


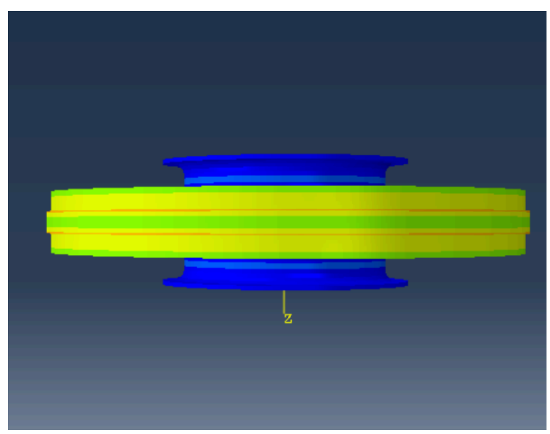


Maximum deformation

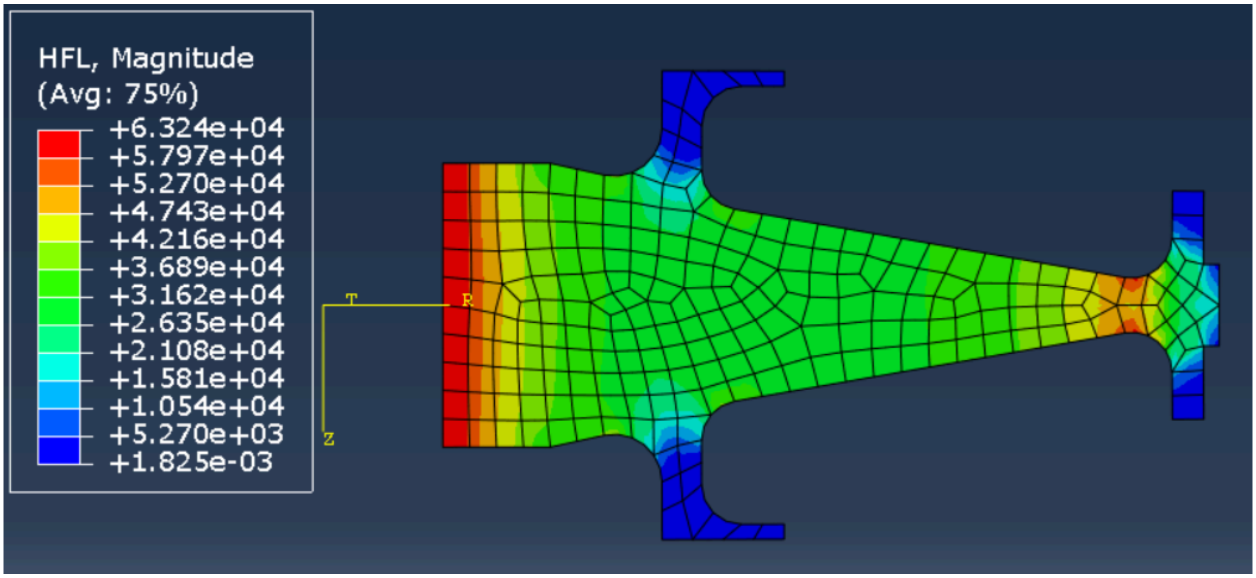


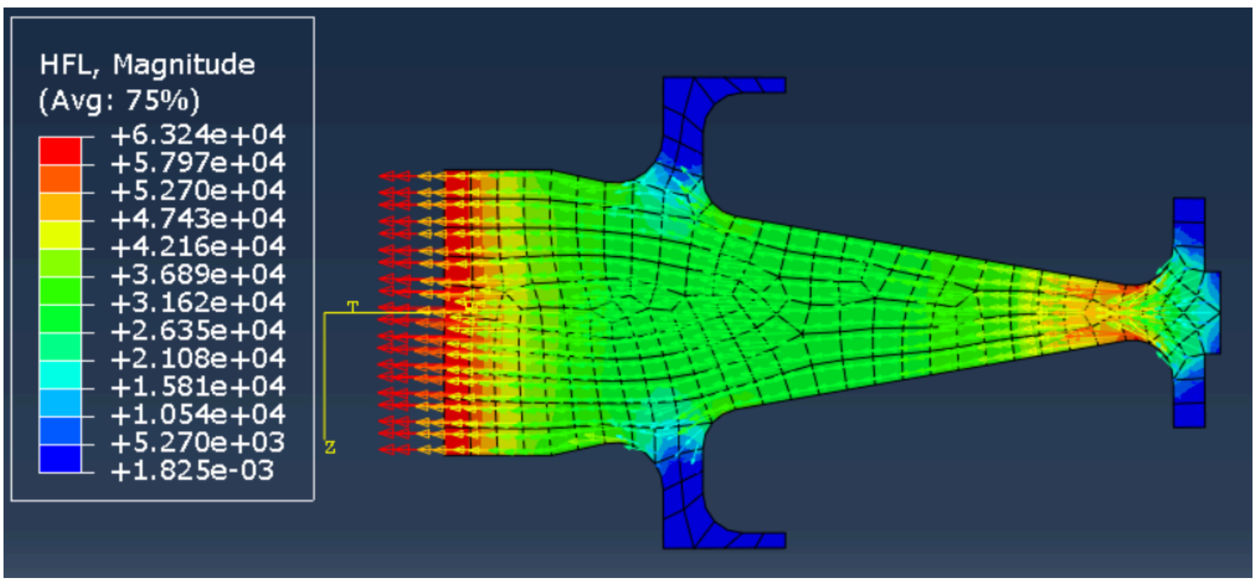


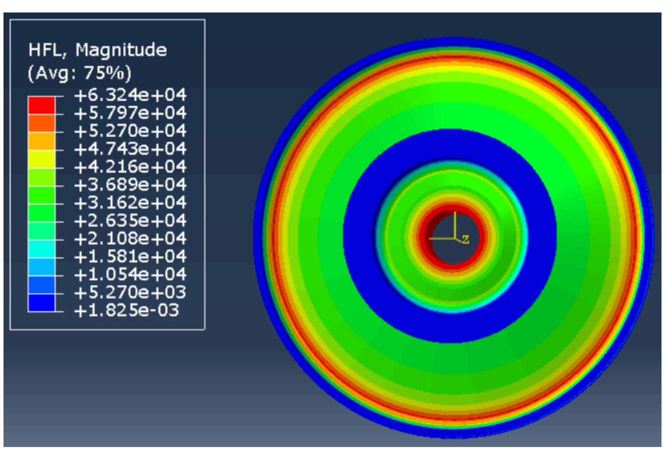


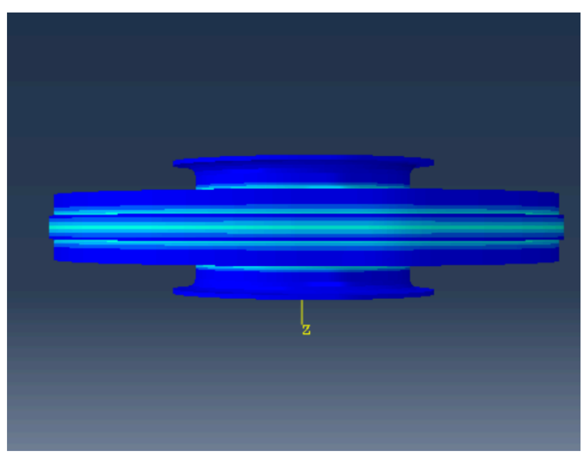


Maximum heat flux:









Integration Method:

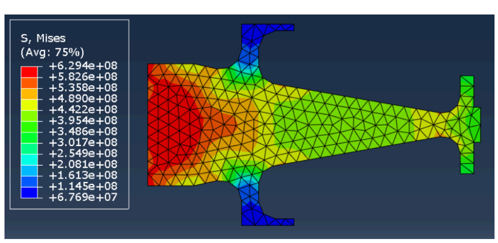
We came across reduced integration during element selection and briefly discussed reduced integration over there.

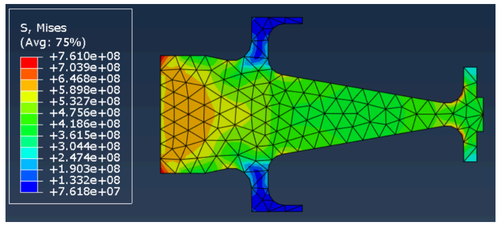
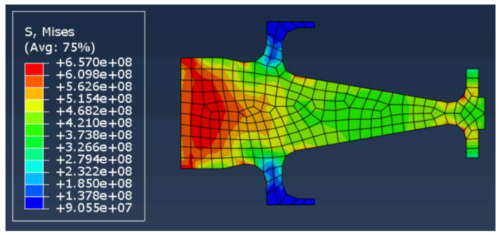
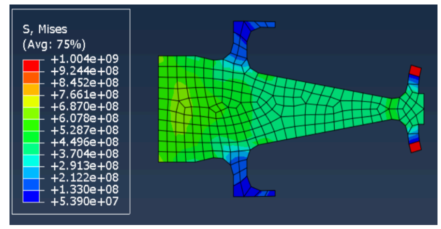
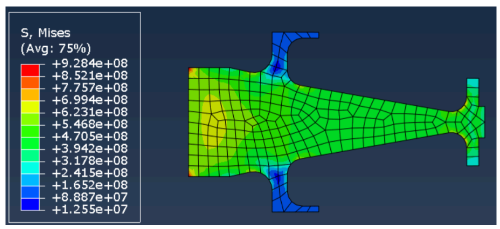
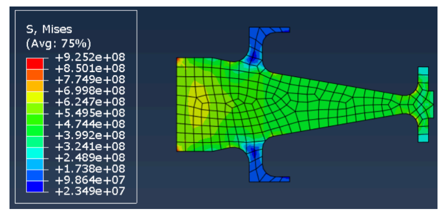
Here, the integration method refers to the numerical technique employed to evaluate the integrals that arise in the formulation of the finite element method. There are two main types of integration methods used in finite element analysis:

1. Full Integration:
    a. In full integration, the integrals are evaluated using a higher-order numerical integration scheme, such as Gaussian quadrature.
    b. It typically provides more accurate results, especially for problems with higher-order elements and complex stress/strain distributions.
    c. However, It can be computationally expensive, particularly for problems with many elements.
2. Reduced Integration:
    a. Reduced or under-integration integration uses a lower-order numerical integration scheme to evaluate the integrals.
    b. This can significantly reduce computational cost as the number of integration points is reduced.
    c. It can be particularly beneficial for problems with simple stress/strain distributions, where the accuracy requirements are not as stringent.
    d. However, reduced integration can also lead to issues such as shear locking and hourglassing, which must be addressed through appropriate element selection and formulations.

The Reduced integration method uses an hourglass control feature, which helps address potential instabilities from reduced integration.

Comparison:

Element Type	Full Integration Method	Reduced Integration Method
Linear Triangular		No reduced integration option

<table>
<tr><td>Quadratic Triangular</td><td></td><td>No reduced integration option</td></tr>
<tr><td>Linear Quad</td><td></td><td></td></tr>
<tr><td>Quadratic Quad</td><td></td><td></td></tr>
</table>

Observations:

Both methods gave similar results for the quadratic quad, whereas the linear quad has a floor ceil value difference. Linear Quad elements have a lower-order polynomial shape function, which can lead to issues such as shear locking and hourglassing when using reduced integration.

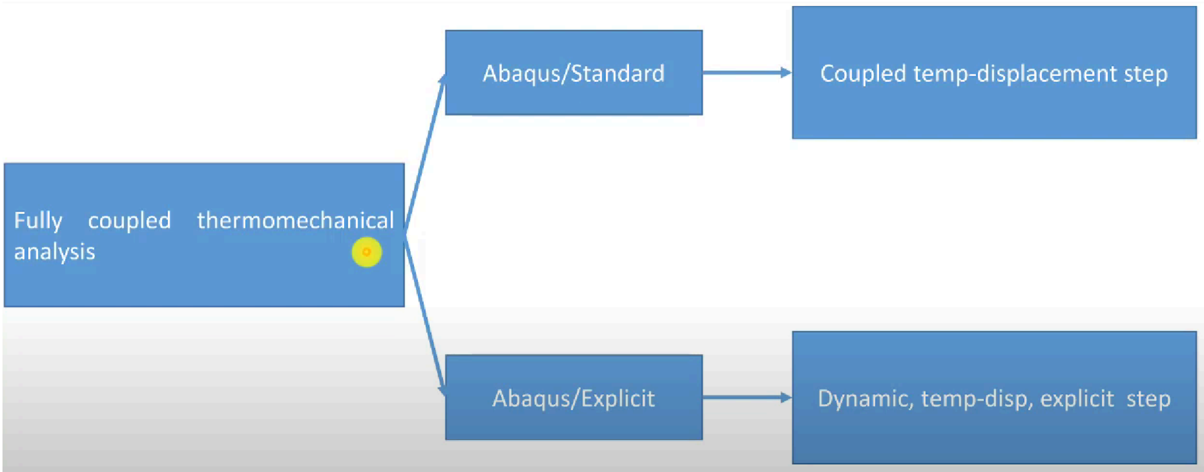
Solver:

There are two main solvers in Abaqus:

- Standard - a group of solvers for implicit analyses (linear and nonlinear static and dynamic)
- Explicit - explicit solver for dynamic analyses

Of course, apart from these two general solver types, there are also solvers specific to different kinds of simulation. For example, the following eigensolvers can be found in Abaqus:

- Lanczos - for eigenfrequency extraction and buckling simulations
- subspace iteration - same uses as Lanczos
- AMS - for eigenfrequency extraction



Standard solver is being used as we're using a Coupled temp-displacement step.

The standard solver's implicit time integration scheme and nonlinear analysis capabilities allow for the accurate modeling of the disk's complex stress and deformation behavior, including the coupled thermal-structural interactions.

In ABAQUS/Standard, the temperatures are integrated using a backward-difference scheme, and the nonlinear coupled system is solved using Newton's method. ABAQUS/Standard offers an exact and approximate implementation of Newton's fully coupled temperature-displacement analysis method.

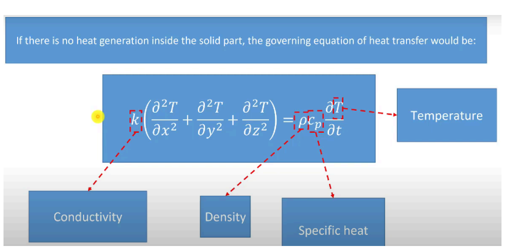
An exact implementation of Newton's method involves a nonsymmetric Jacobian matrix, as is illustrated in the following matrix representation of the coupled equations:

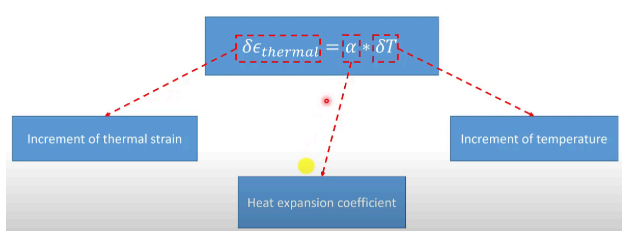
$$\begin{bmatrix} K_{uu} & K_{u\theta} \\ K_{\theta u} & K_{\theta\theta} \end{bmatrix} \begin{Bmatrix} \Delta u \\ \Delta \theta \end{Bmatrix} = \begin{Bmatrix} R_u \\ R_\theta \end{Bmatrix},$$

where Δu and $\Delta \theta$ are the respective corrections to the incremental displacement and temperature, K_{ij} are submatrices of the fully coupled Jacobian matrix, and R_u and R_θ are the mechanical and thermal residual vectors, respectively.

Solving this system of equations requires using the unsymmetric matrix storage and solution scheme. Furthermore, the mechanical and thermal equations must be solved simultaneously. The method provides quadratic convergence when the solution estimate is within the algorithm's radius of convergence. The exact implementation is used by default.

Governing Equations (Extra information):





Optimization

There are three main types of Optimization processes in the abacus:

1. **Topology Optimization**
    a. **Purpose**: It determines the optimal material layout within a given design space for a specified set of loads, boundary conditions, and constraints.
    b. **Key Features**:
        i. It is ideal for early-stage design to explore the best structural layout.
        ii. Results in a conceptual design that maximizes stiffness or minimizes weight by removing unnecessary material.
2. **Shape Optimization**
    a. **Purpose**: It modifies the geometry of a structure to improve performance metrics such as stress distribution, displacement, or natural frequencies.
    b. **Key Features**:
        i. It fine-tunes the shape of specific features or boundaries of a structure.
        ii. Typically used in later design stages to refine and enhance existing designs based on detailed stress analysis.
3. **Sizing Optimization**
    a. **Purpose**: It optimizes structural members' thickness/ cross-sectional dimensions to achieve desired performance objectives.
    b. **Key Features**:
        i. It adjusts dimensions such as thickness, beam widths, or shell thicknesses to minimize weight or maximize strength.
        ii. It is suitable for designs where maintaining the overall shape is critical, but internal dimensions can vary for optimization.

We're trying to perform Shape optimization with the given constraints:

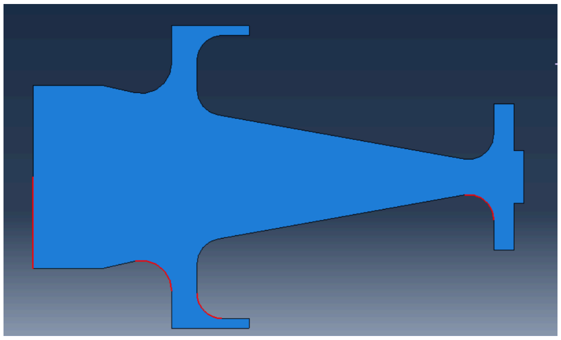
$$\begin{aligned} \min_{D.V} \quad & \sigma_{vonMises} \\ \text{s.t.} \quad & d \leq 1.5, \\ & H_f \leq 0.05, \\ & \sigma_{radial} \leq 460, \\ & \sigma_{hoop} \leq 200 \end{aligned}$$

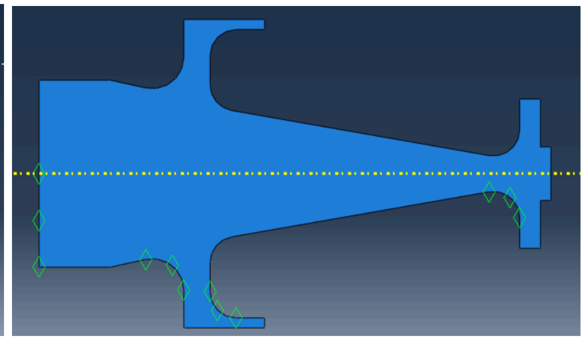
Table 2: Description of design variables for Turbine Disk example

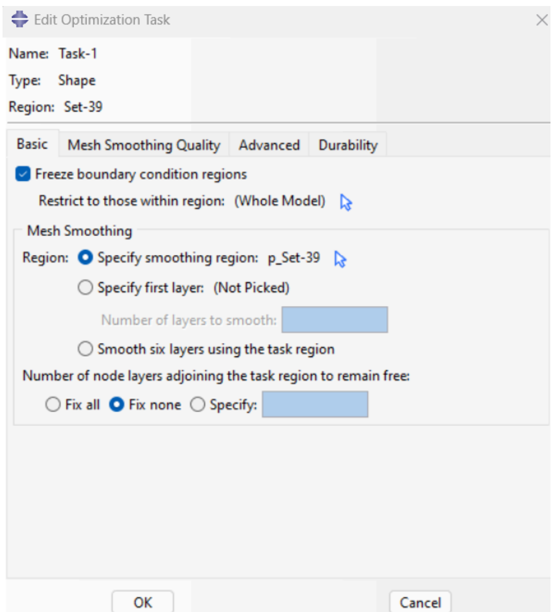
S.No.	Design variables	Lower bound (mm)	Upper bound (mm)
1	P1 - Bore width	151	185
2	P2 - Rib filet radius	21	78
3	P3 - Snap filet radius	21	78
4	P4 - Snap undercut filet radius	31	88

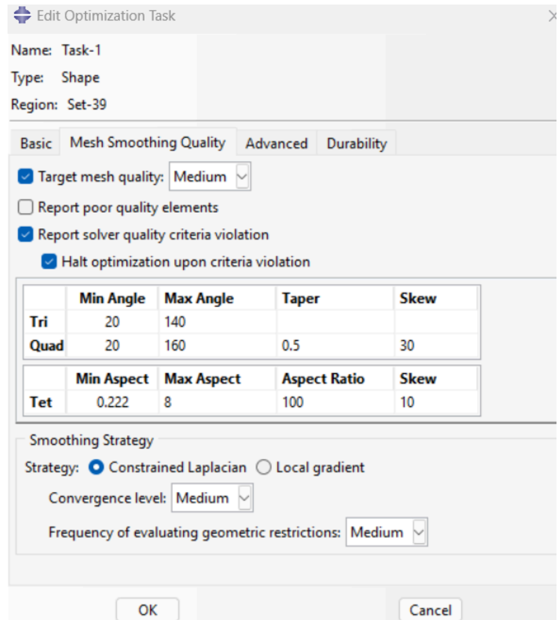
Brief steps:

1. Create a shape optimization task where we define the elements/nodes of consideration that should be optimized

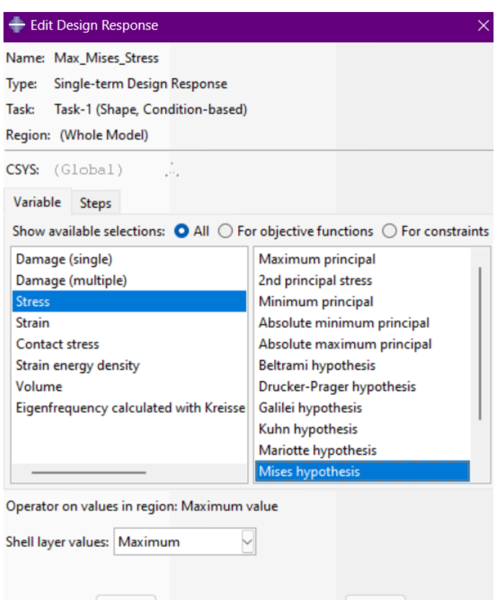


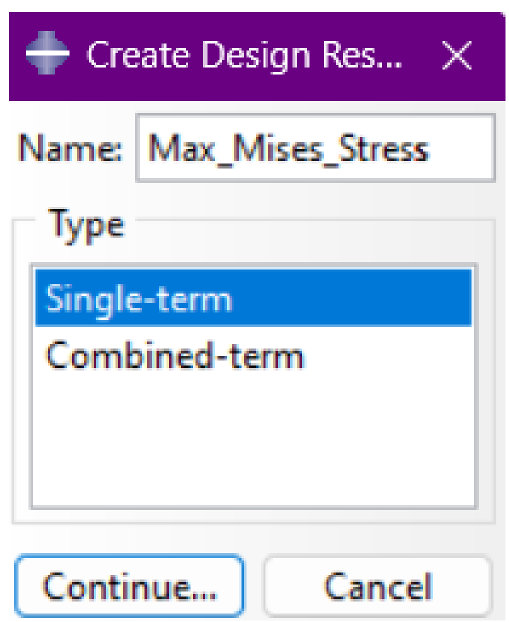




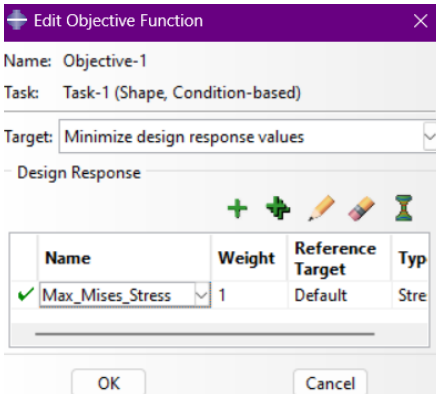


2. Create a Design Response:

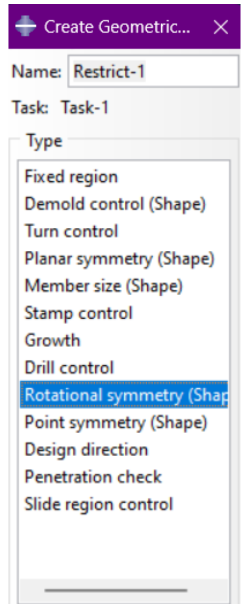


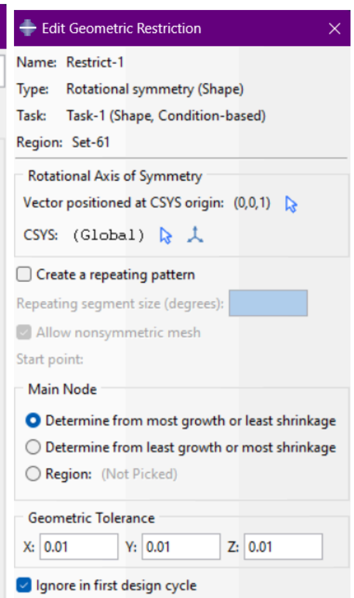


3. Create Objective Function:

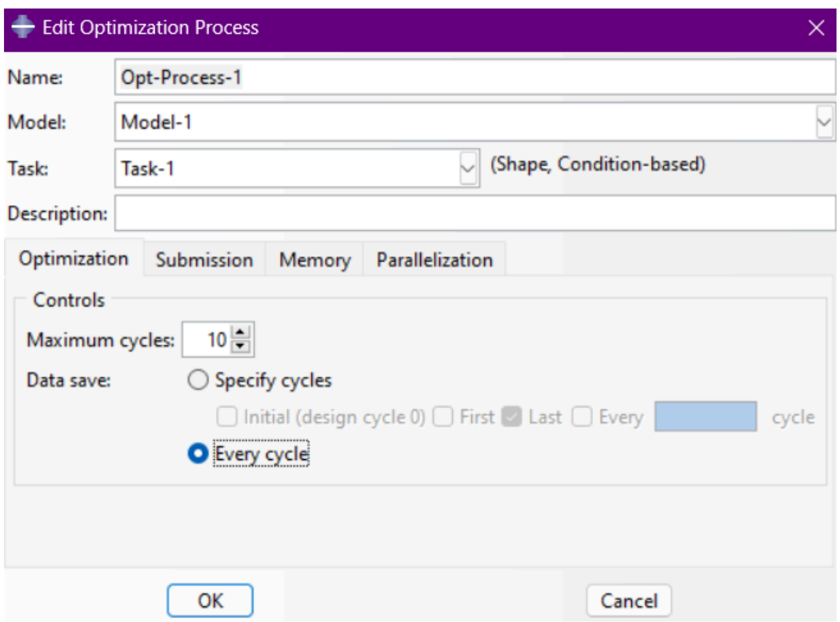


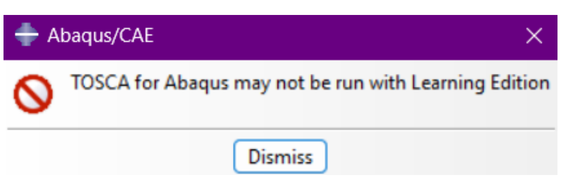
4. Create Geometric Restrictions:





5. Create an optimization process and then submit it and visualize the results:





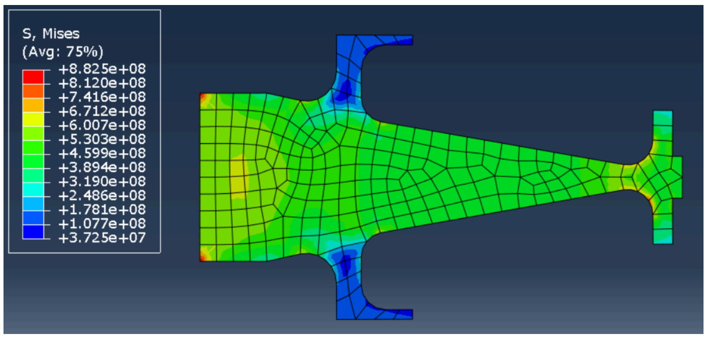
We can use TOSCA software: Abaqus extension to optimize the shape and weight. But it's not available. So, I'm following the approach mentioned below.

Manual Optimization Method:

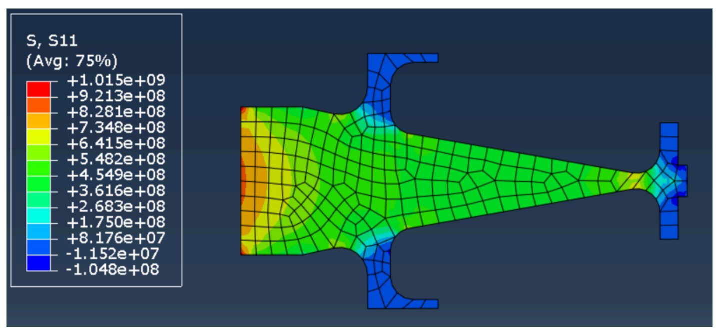
The basic idea is to consider the mid-value of each design variable, perform the analysis, and check if the maximum von Mises stress is minimized or not, as well as all other constraints. If it's not minimized, we take the mid value of the mid value (Calculated before) and the upper bound. It's similar to binary search optimization.

Optimising P1:

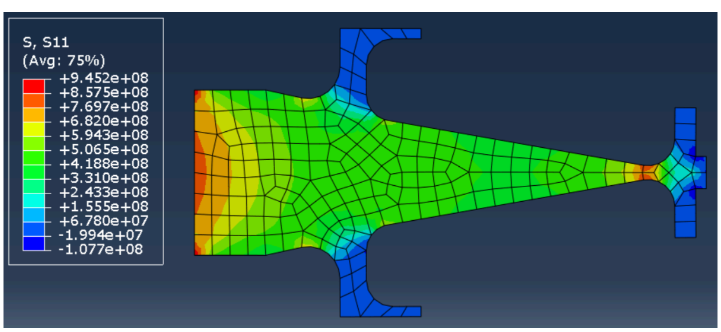
P1 = 168mm



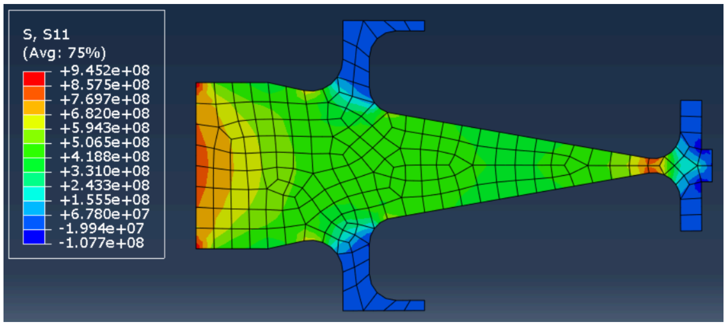
P1 = 159.5mm



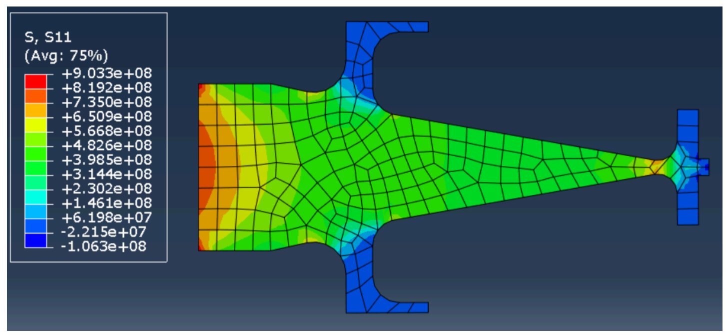
P1 = 155.25



Optimising P2: P2 = 49.5mm



P2 = 35.25



P2 = 28.125

How do you reduce the number of function calls in the optimization run?

Several strategies can be employed to reduce the number of function calls during an optimization process. Using ***surrogate models***, which are computationally cheaper approximations of the original objective function, can significantly decrease the number of calls to the expensive original function. ***Gradient-based optimization*** methods that leverage gradient information can also require fewer function evaluations than derivative-free methods. ***Adaptive sampling*** techniques, such as expected improvement or knowledge gradient, can focus the sampling on promising regions of the search space, further reducing the overall number of function calls. ***Parallel computation, approximate gradients, early stopping criteria, warm-starting, and reduced-order models*** are additional approaches that can help minimize the number of function evaluations required to reach the optimal solution.

I could not complete the optimization using the method I thought was best because of a lack of time. I want to learn more about optimization and do some projects, so please share the optimization solution with us.

Reference:

- https://www.researchgate.net/figure/Different-stages-of-disk-burst-simulation_fig3_313938109
- Elhefny, A., & Liang, G. (2013, March 1). *Stress and deformation of rocket gas turbine disc under different loads using finite element modelling*. Propulsion and Power Research. https://doi.org/10.1016/j.jppr.2013.01.002.
- Taamneh, Y. (2017, September 1). *Thermal analysis of gas turbine disk integrated with rotating heat pipes*. Case Studies in Thermal Engineering. https://doi.org/10.1016/j.csite.2017.09.002.
- I. (n.d.). *Design and Finite Element Analysis (FEA) of Gas Turbine Rotor Disc*. IJRASET. https://www.ijraset.com/research-paper/finite-element-analysis-fea-of-gas-turbine-rotor-disc
- https://www.bestech.com.au/wp-content/uploads/Modulus-of-Elasticity.pdf
- Axisymmetric analysis tutorial for beginners | ABAQUS CAE
- ABAQUS Tutorial for Heat Transfer Analysis | Part 1 (Steady State)
- Thermo-mechanical simulation in ABAQUS: Part 1
- Thermomechanical Analysis in Abaqus: How to Define Material Properties
- Abacus Documentation