Solving Nonlinear Problems with Abaqus

Solving Nonlinear Problems with Abaqus is an extensive course which provides practical information to perform nonlinear FEA analysis in Abaqus. This course takes step-by-step approach and presents from introductory to advanced technique in a gradual way. In a real life problem, there are many nonlinearities present in a system. Solving such problems is a really challenging task. This course begins with simple problems having a single non-linearity and presents the appropriate techniques to solve it. Later in the course more complex problems are presented. The course is divided in the following eight sections:

1. Basics of Nonlinear Analysis
2. Geometric nonlinearity
3. Force nonlinearity
4. Material nonlinearity
5. Boundary nonlinearity (Contact Problems)
6. Finite element discretization
7. Miscellaneous (Multiple nonlinearities)
8. User exercises

Section 1: Basics of Nonlinear Analysis

In this section, an overview of sources of nonlinearities in structures is given. Furthermore solution algorithms for nonlinear problems are described in detail. The objective of this section is to give an overview of the physics involved in nonlinear problems and how to choose the best solution strategy. Topics covered are as follow:

- Linear vs Nonlinear response
- Sources of nonlinearities
    1. Geometric nonlinearity
        a) Large strain
        b) Small strains but large displacements and/or rotations
        c) Stress stiffening
        d) Snap-Through
        e) Buckling
    2. Material nonlinearity
        a) Plasticity
        b) Nonlinear elasticity
        c) Viscoelasticity
    3. Force nonlinearity
    4. Boundary nonlinearity
- Solution algorithms
    1. Newton's method
    2. Arc-Length method
- Solving contact problems
    1. Contact property model
        a) Hard Contact
        b) Soft Contact
        c) Friction models
    2. Contact constraint enforcement methods
        a) The direct method
        b) The penalty method

        c) The augmented Lagrange method
    3. Relative sliding of surfaces
    4. Slave and master surfaces
    5. Discretization of contact pair surfaces
    6. General contact and Contact pairs
    7. Solution Algorithm for contact problems

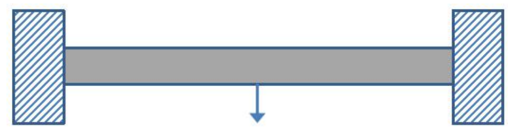
- Finite Element Discretization
    1. Lagrangian mesh
    2. Eulerian mesh
    3. Mesh Convergence
        a) Mesh refinement metrics
        b) Mesh refinement techniques
    4. Adaptivity techniques
        a) ALE Adaptive meshing
        b) Adaptive remeshing
        c) Mesh-to-mesh solution mapping

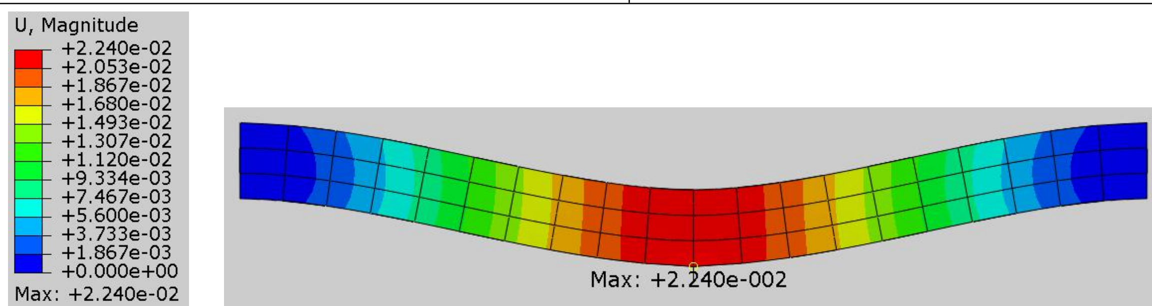
Section 2: Geometric nonlinearity

In this section, a large number of exercises are presented providing intensive instructions to perform analysis of geometric nonlinear problems. Details of topics covered in exercises are given below.

Exercise 1

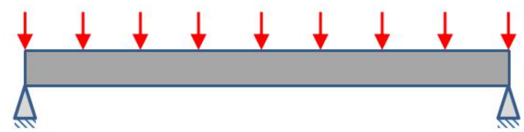
- Analyzing a fixed beam
- How to determine if nonlinear analysis is required
- Turning on the NLgeom option
- Understanding Job monitor for a nonlinear analysis

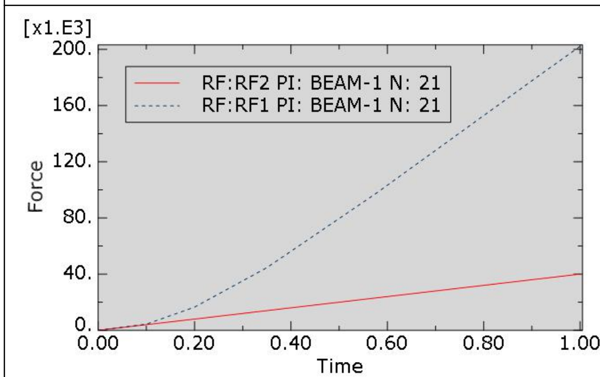




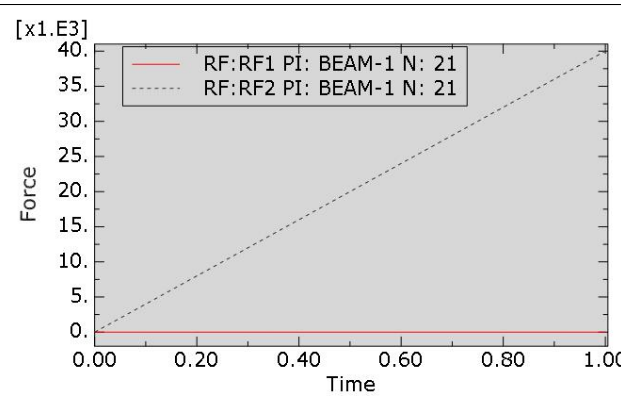
Exercise 2

- Analyzing a beam pinned on both ends
- When does geometric nonlinearity has significant influence on solution even for small deformation
- Plotting membrane forces
- Plotting reaction forces in axial direction at supports

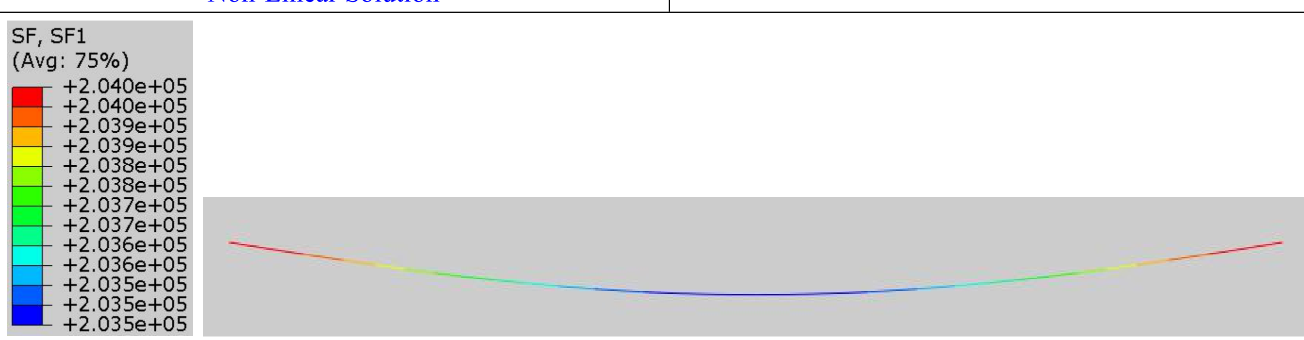




Non-Linear Solution

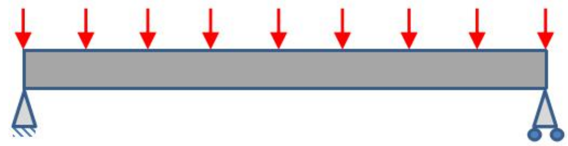


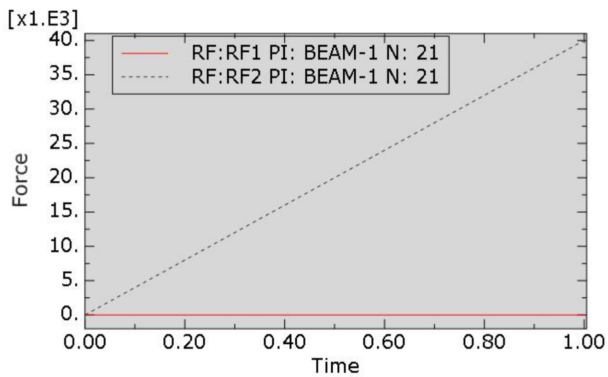
Linear Solution



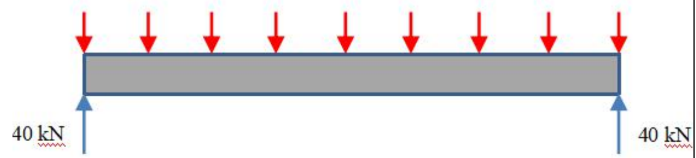
Exercise 3

- Analyzing a simply supported beam
- Influence of boundary conditions on the geometric nonlinearity
- Influence of boundary conditions on membrane forces
- Plotting membrane forces





Nonlinear Solution

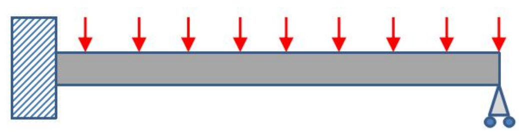


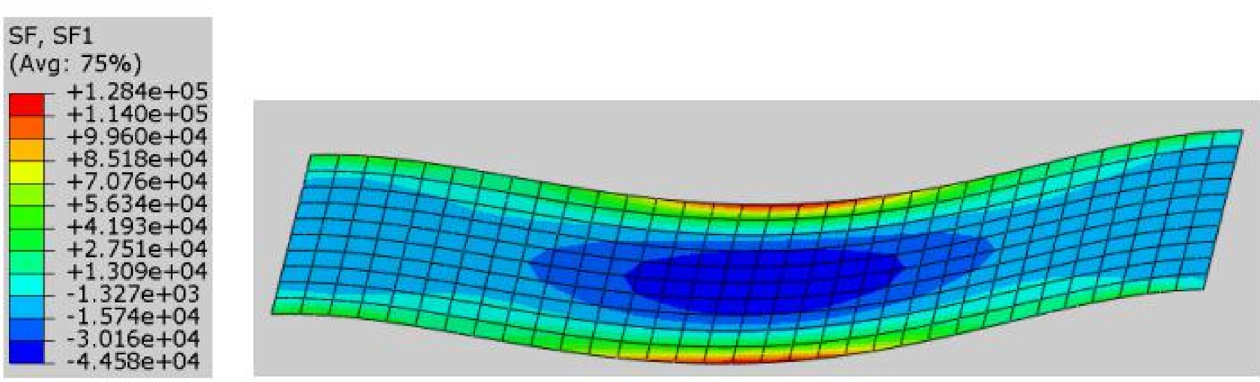
$$Deflection = \frac{5}{384}\frac{pL^4}{EI} = 0.0483$$

Analytical Solution

Exercise 4

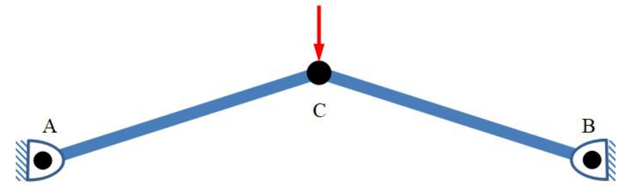
- Analyzing a plate
- Influence of geometric nonlinearity on solution for small strain problem
- Plotting the axial strain
- Investigating membrane effects
- Symbol plot of reaction forces

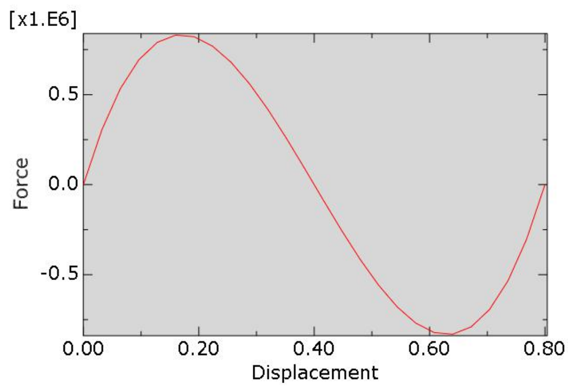


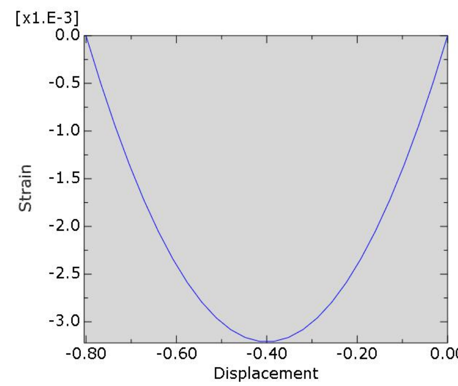


Exercise 5

- Analyzing a two bar truss
- Simulating snap through using General static analysis
- Solving small strain, large rotation problem
- Obtaining solution using displacement control
- Comparing analytical and numerical solution

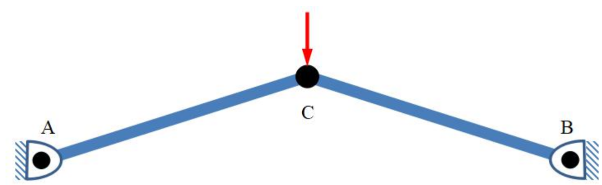


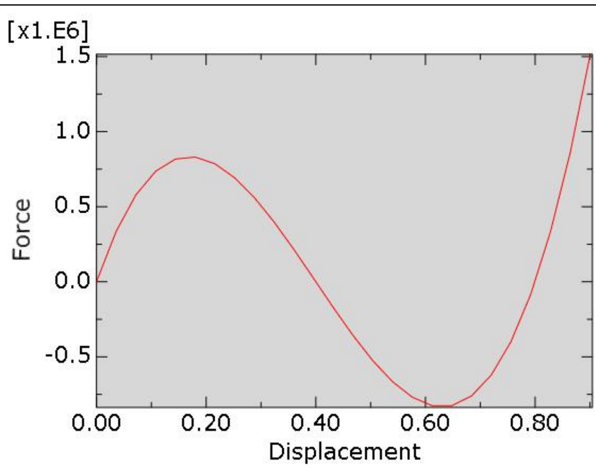




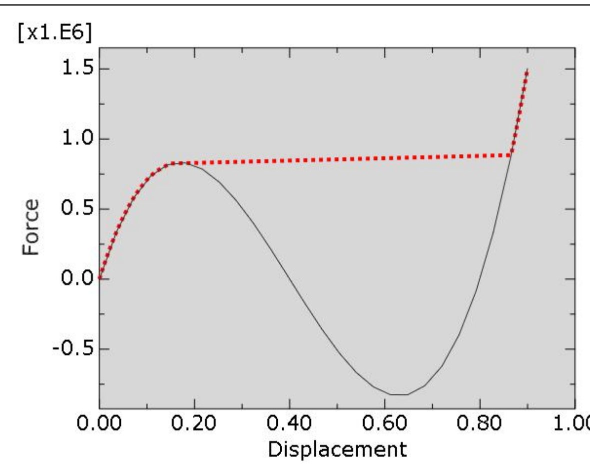
Exercise 6:

- Analyzing a two bar truss using force-control
- Investigating the snap-through
- Comparing displacement-control and force-control results
- Introducing stabilization





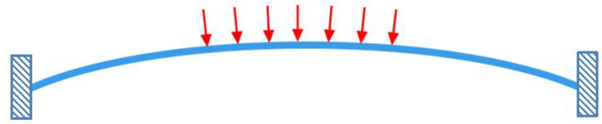
Displacement-control analysis

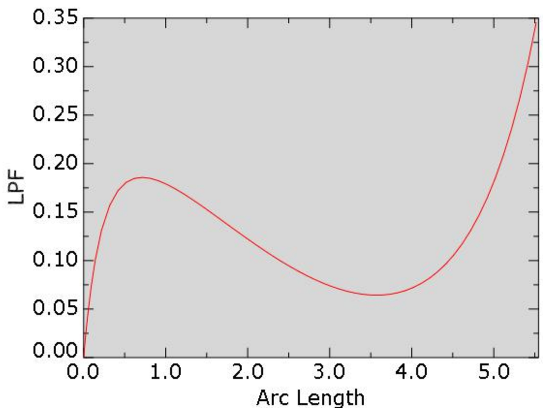


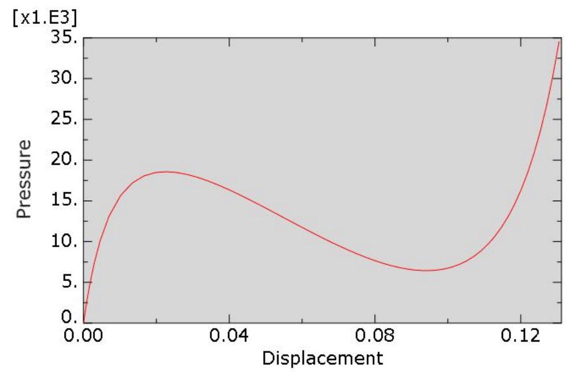
The dotted line represents the load-control analysis and solid line represents the displacement-control analysis.

Exercise 7

- Analyzing a circular arch
- Simulating snap-through using Riks method
- Modeling arch using beam elements
- Specifying analysis stopping criteria
- Understanding Job monitor for a Riks analysis
- Comparing job monitor and LPF graph

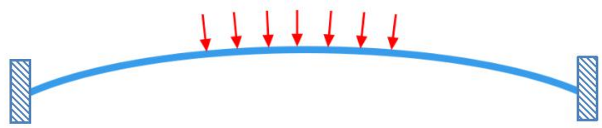


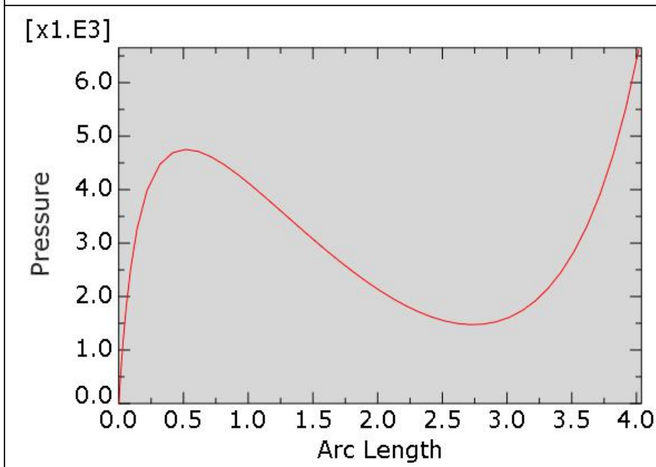


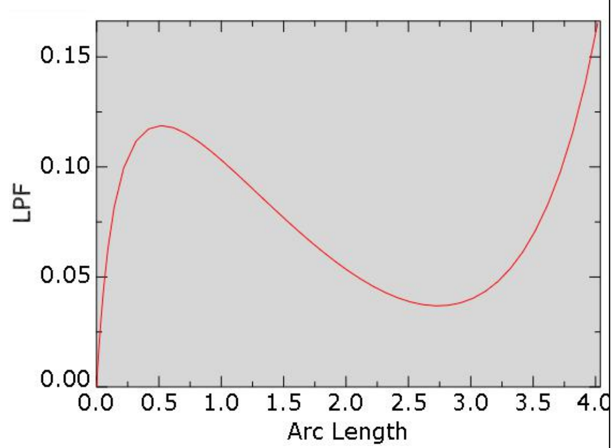


Exercise 8

- Analyzing a circular arch modeled using shell elements
- Simulating snap-through using Riks method
- Specifying analysis stopping criteria
- How to choose magnitude of LPF for stopping criteria
- Requesting Pressure loads as output

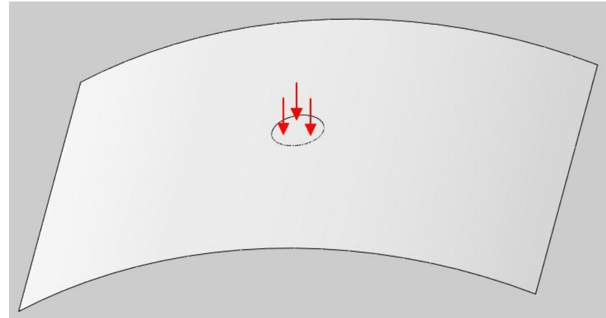


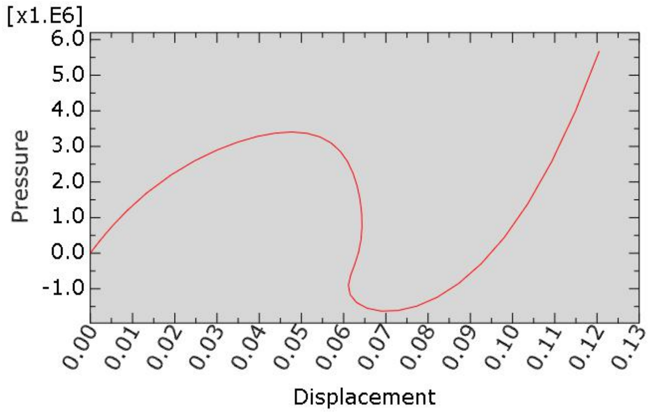




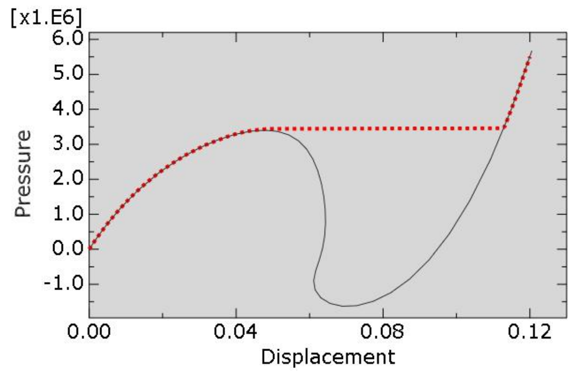
Exercise 9

- Analyzing a cylindrical roof
- Simulating snap-through behavior
- Comparison of results using
  - ✓ Riks analysis
  - ✓ General static Analysis





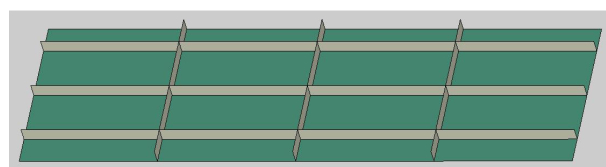
Pressure versus the displacement of center node

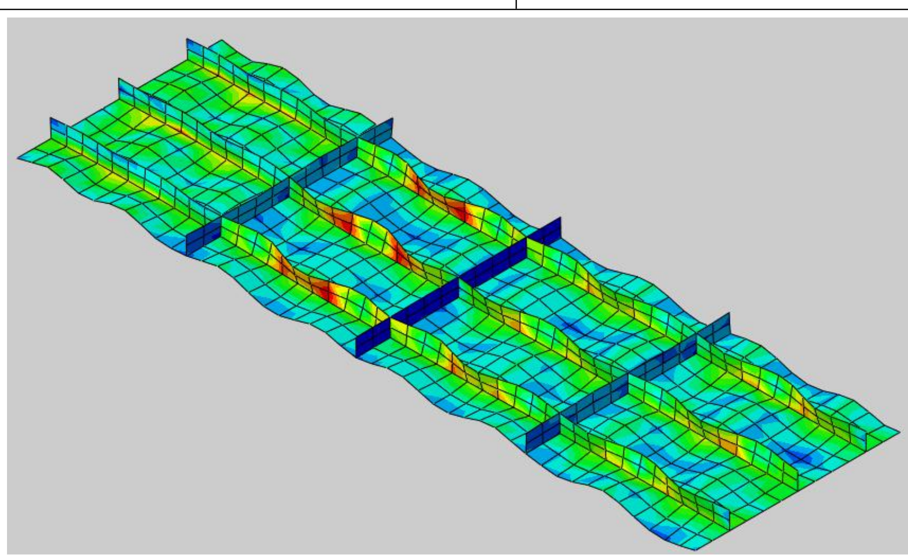


The dotted line represents the static, general analysis and solid line represents the Riks analysis.

Exercise 10

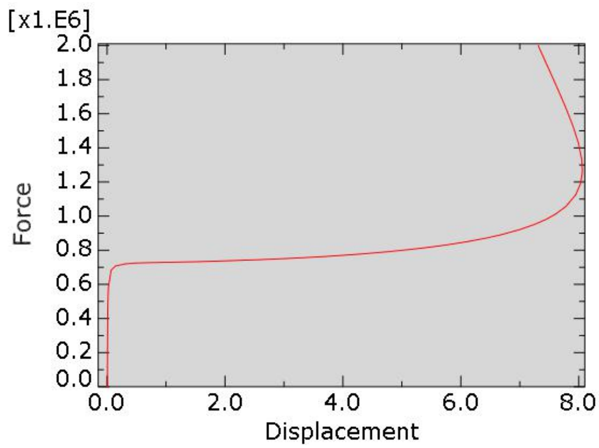
- Analyze a structure undergoing localized buckling
- Defining MPC constraint
- Introducing stabilization

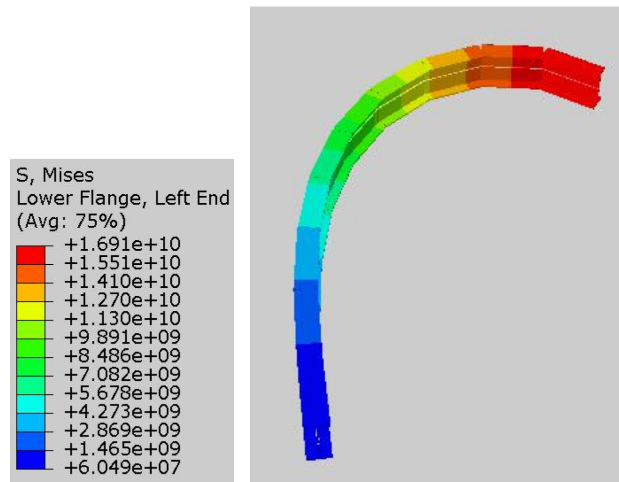




Exercise 11

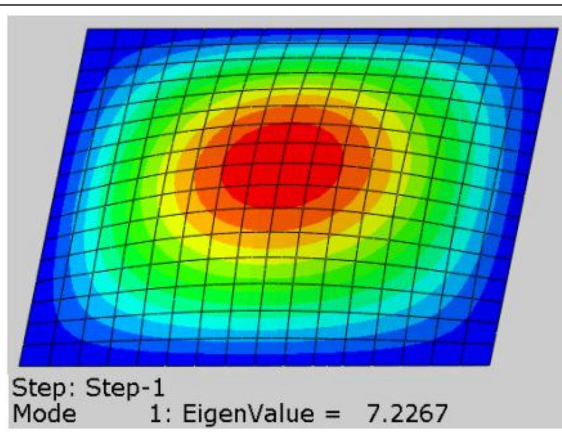
- Performing buckling analysis of a column
- Estimating buckling modes using eigenvalue analysis
- Investigating the post-buckling response using General static analysis
- Seeding the imperfection by introducing trigger load



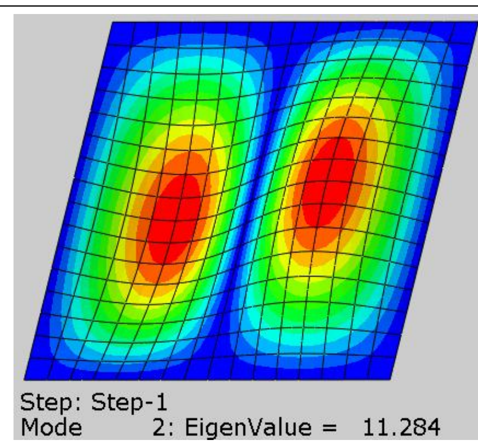


Exercise 12

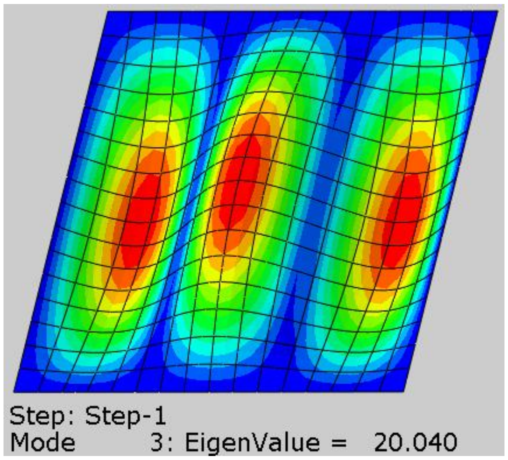
- Performing buckling analysis of a thin plate
- Estimating buckling modes using eigenvalue analysis
- Investigating the post-buckling response using General static analysis
- Seeding the imperfection by introducing trigger load



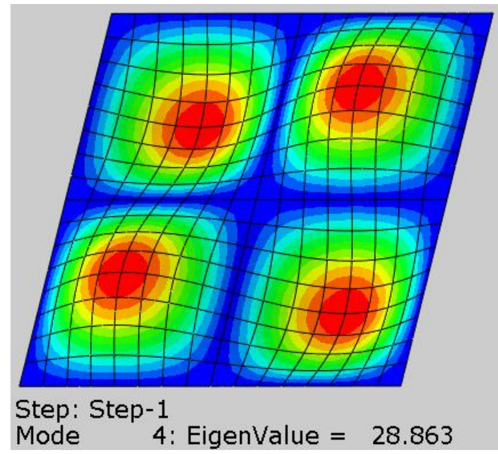
First mode



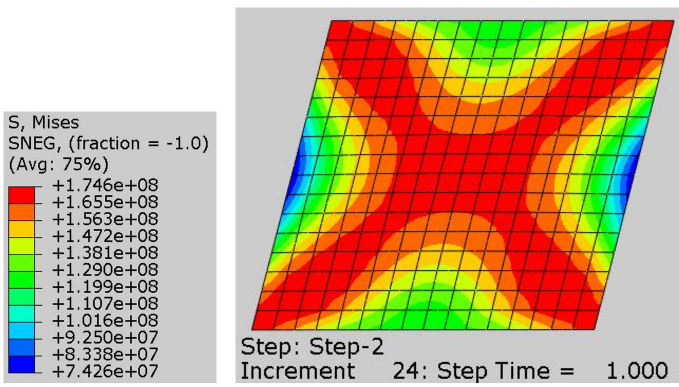
Second mode



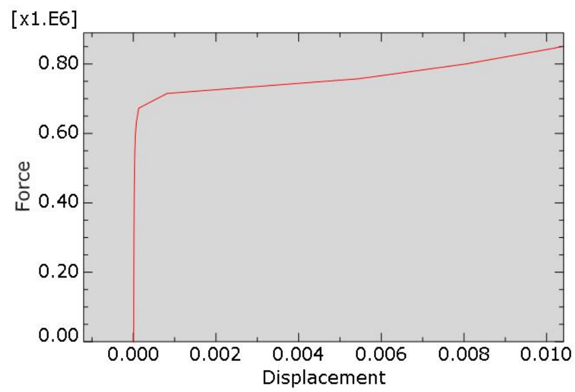
Third mode



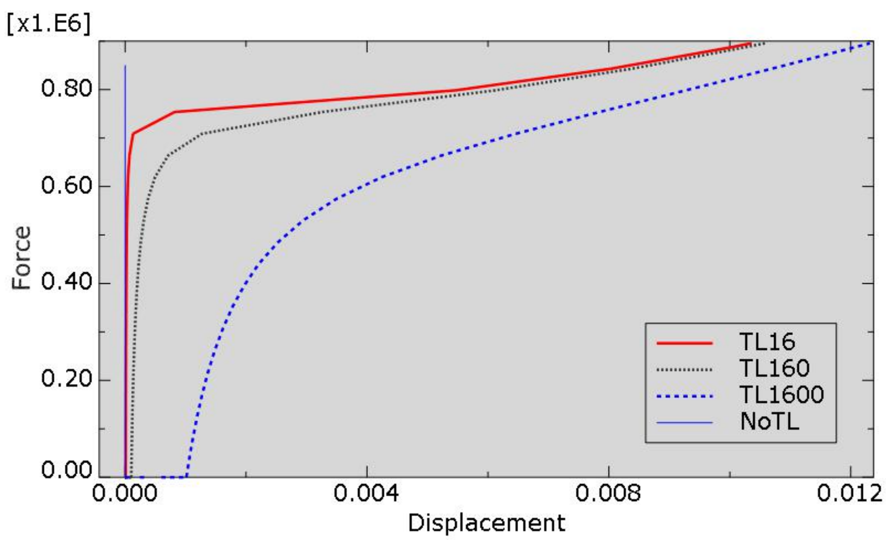
Fourth mode



Load-Displacement analysis



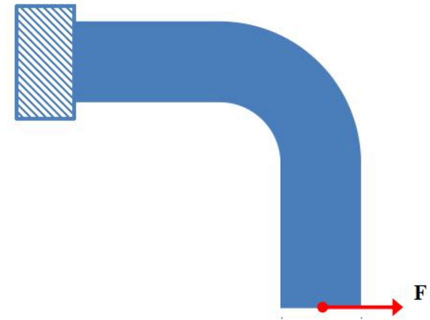
Load versus out of plane displacement

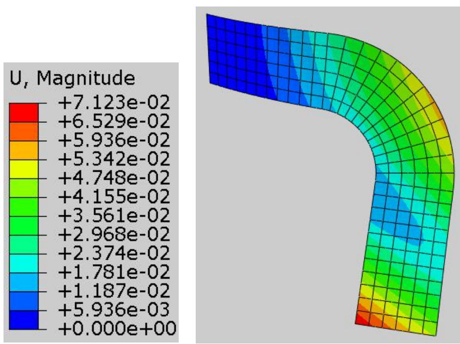


Effect of imperfection magnitude on response

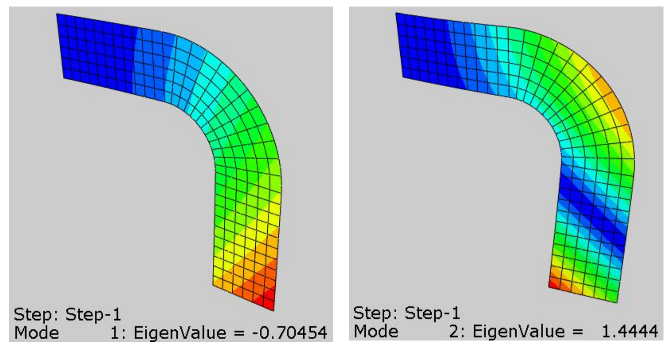
Exercise 13

- Performing buckling analysis of a bracket
- Understanding negative eigenvalues
- Seeding the imperfection using eigenmode shape
- Importance of increment size to capture the buckling response
- Investigating the effect of imperfection magnitude on response of the structure

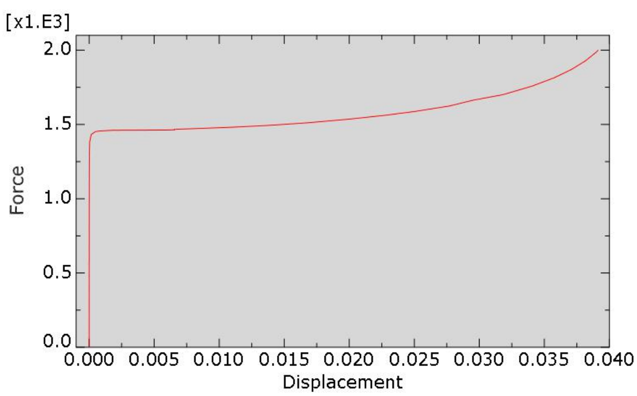




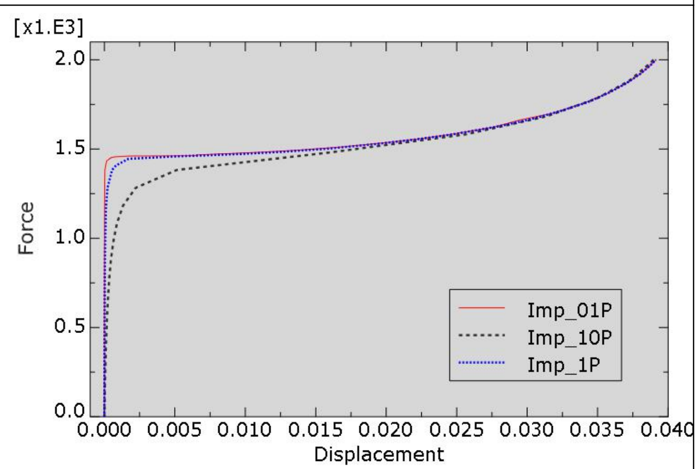
Buckled shape



First and second mode



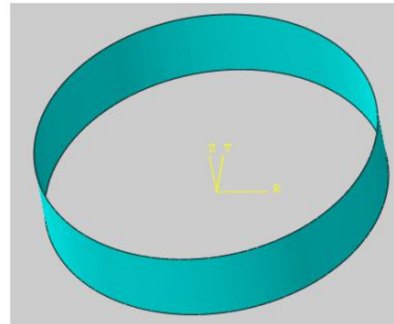
Force versus out of plane displacement of center node

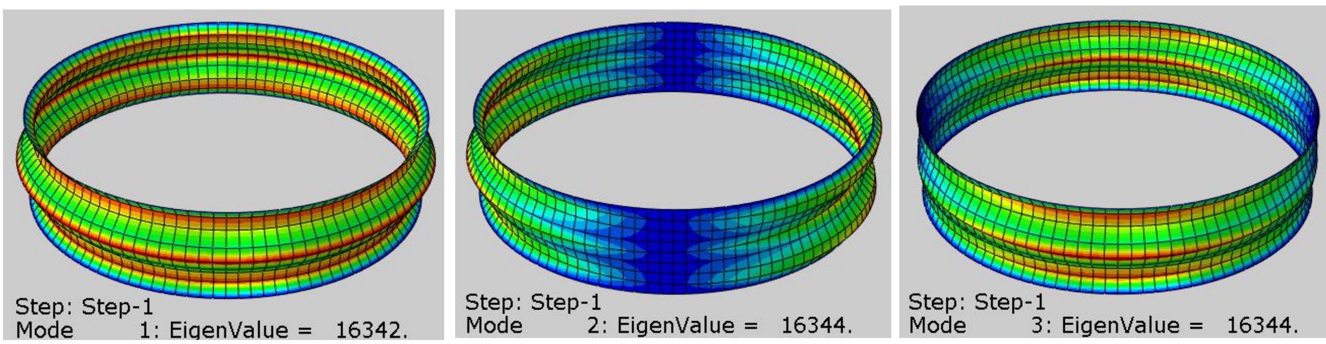


Effect of imperfection magnitude on response

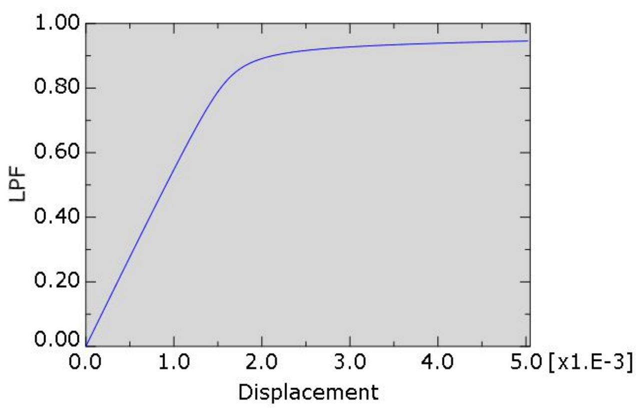
Exercise 14

- Performing buckling analysis of a a cylindrical shell subjected to compressive axial load
- Using Lanczos eigensolver
- Using pairs of repeated eigenmodes to introduce imperfection
- Using Lanczos eigensolver
- Investigating the post-buckling response using Riks procedure
- Seeding the imperfection using multiple eigenmodes

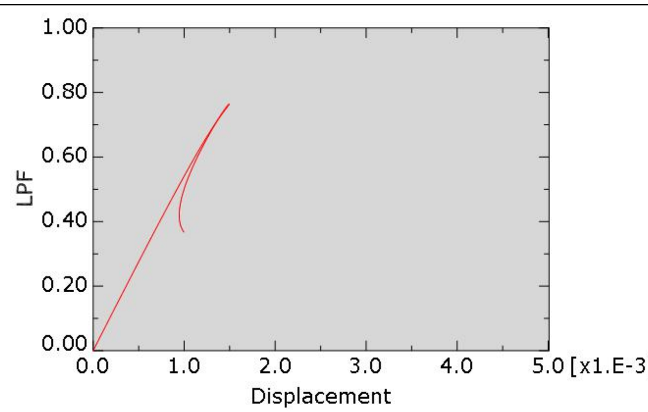




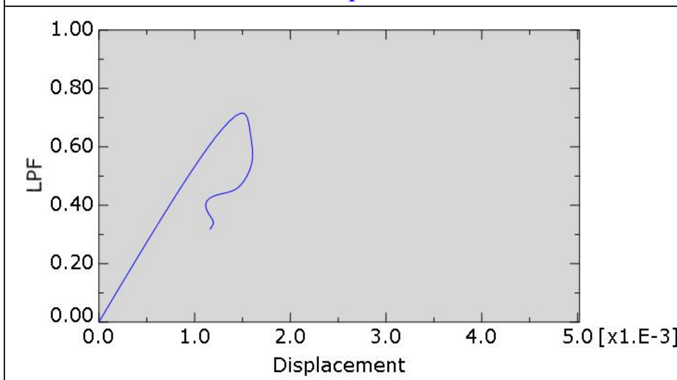
First three eigenmodes



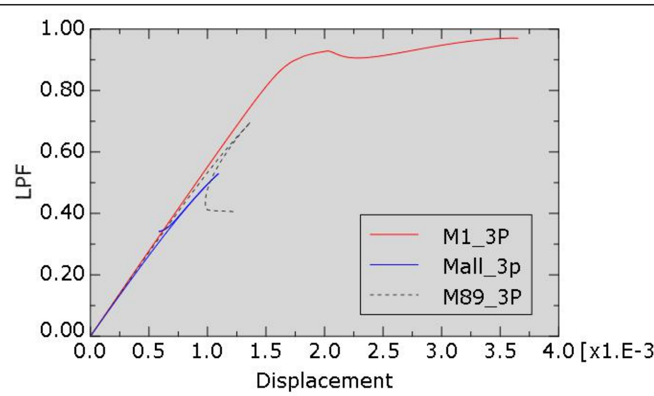
Response of the cylinder when one eigenmode is used to seed the imperfection.



Response of the cylinder when two eigenmodes are used to seed the imperfection.



Response of the cylinder when first fifteen eigenmodes are used to seed the imperfection.



Comparison of response when different combinations of eigenmodes are used to seed the imperfection.

Section 3: Force nonlinearity

In this section exercises are presented providing instructions to solve problems involving force nonlinearity. Details of topics covered in exercises are given below.

Exercise 1

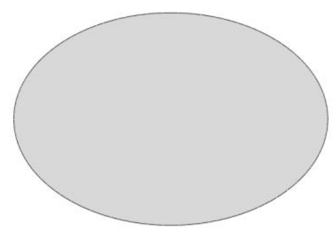
- Specifying follower and non-follower forces
- Comparing solution for follower and non-follower force

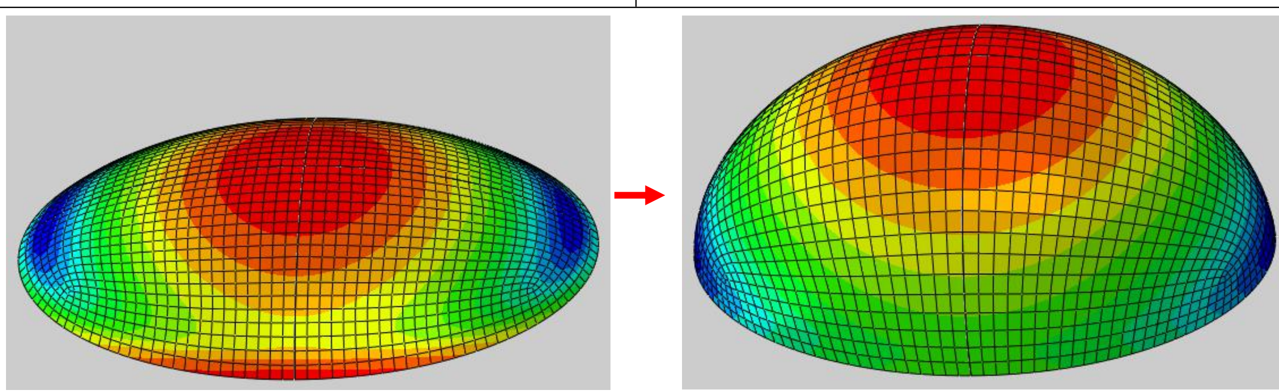
Follower force

Non-Follower force

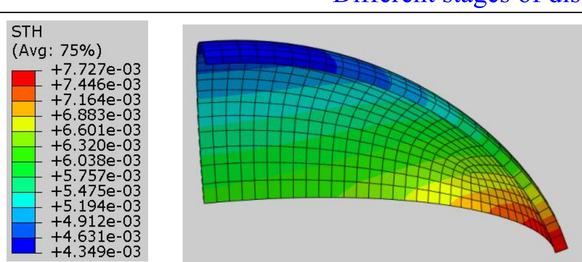
Exercise 2

- Simulating rubber disc subjected to pressure loading
- Plotting thickness for the shell elements
- Plotting the thickness strain
- Visualizing the shell thickness on screen

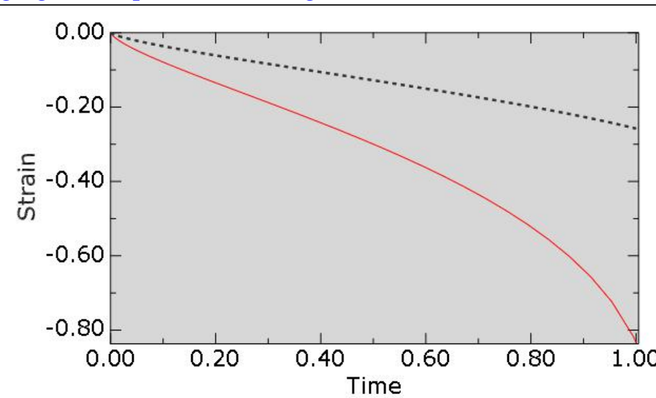




Different stages of disc bulging due to pressure loading



Visualizing the shell thickness



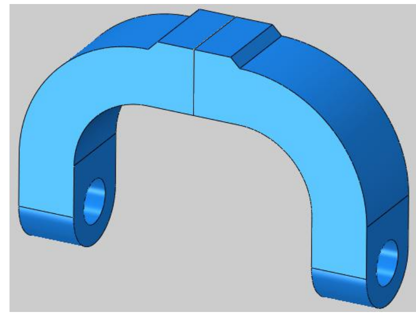
Thickness strain at two different locations on the disc

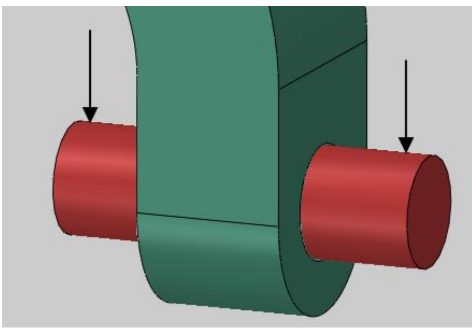
Section 4: Material nonlinearity

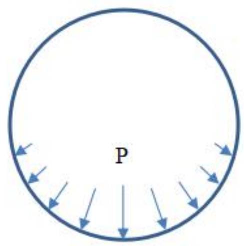
In this section exercises are presented providing instructions to solve problems involving material nonlinearity. Details of topics covered in exercises are given below.

Exercise 1

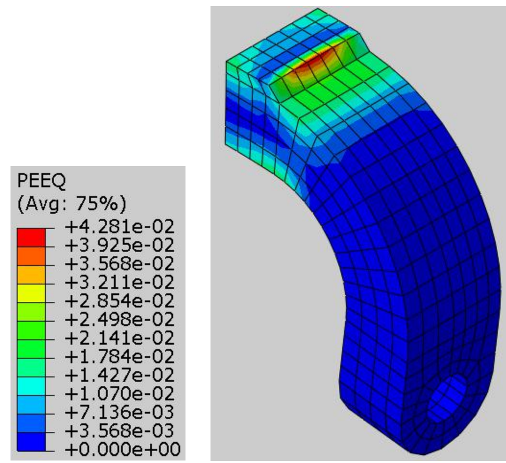
- Analyze a yoke undergoing plastic deformation
- Applying a pressure load with a sinusoidal distribution
- Defining plasticity from given stress-strain curve
- Plotting the equivalent plastic strain





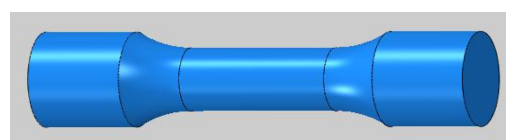


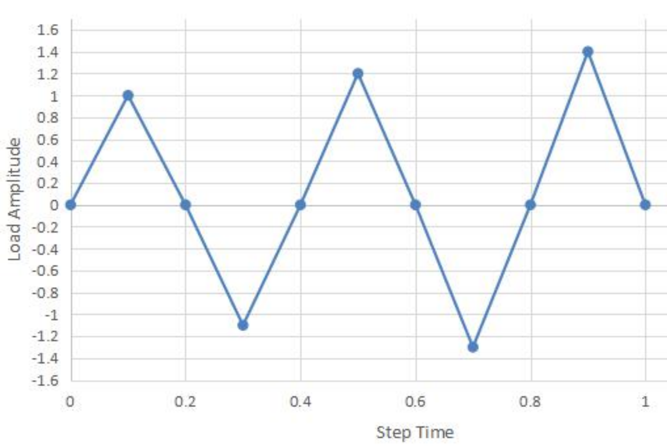
Sinusoidal distribution



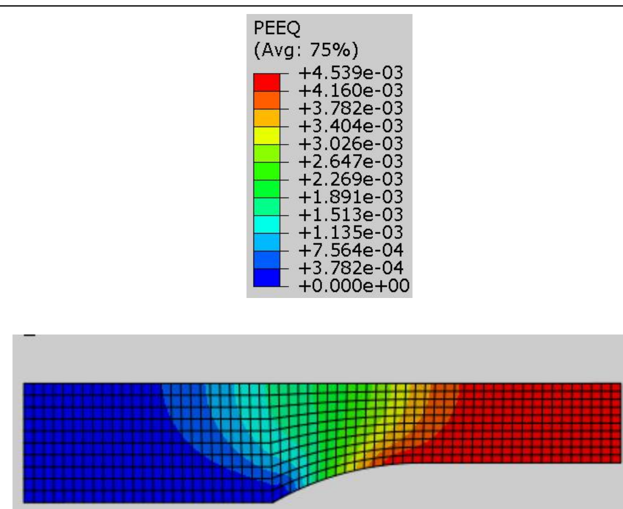
Exercise 2

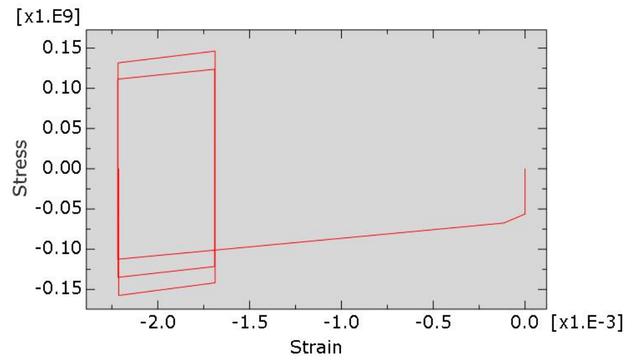
- Analyzing a test rod subjected to cyclic loading
- Applying a cyclic load
- Performing analysis assuming kinematic hardening
- Performing analysis assuming isotropic hardening
- Accumulated strain measures: PEEQ vs PEMAG



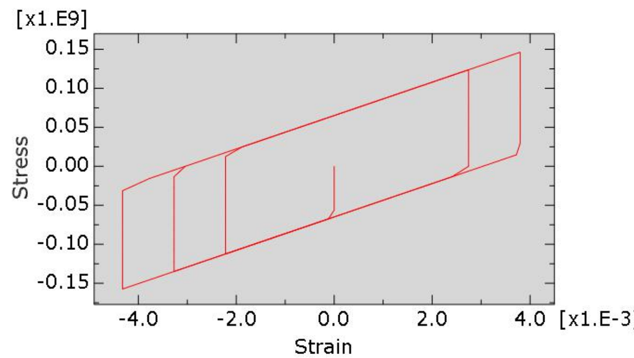


Variation of load





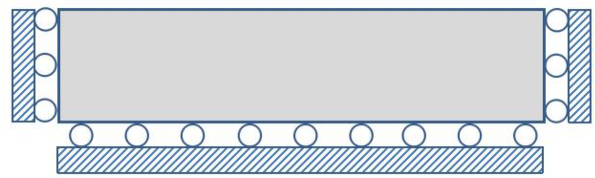
Isotropic hardening

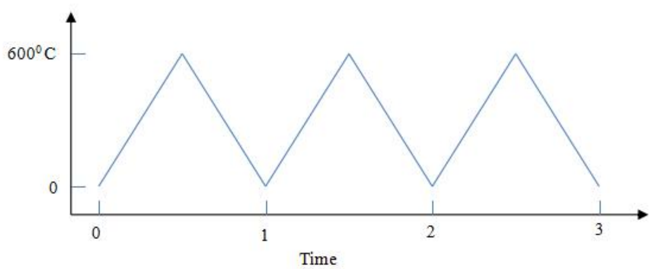


Kinematic hardening

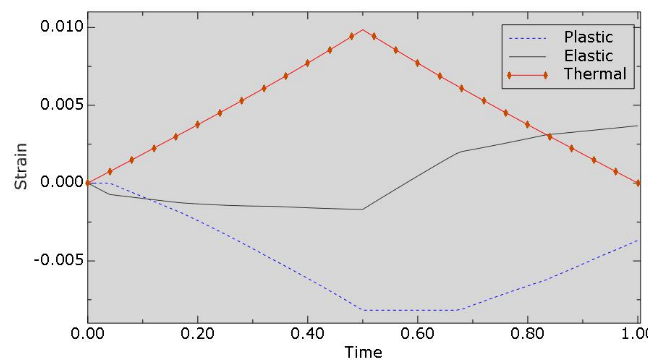
Exercise 3

- Analyzing a bar subjected to a cyclic temperature load
- Applying a cyclic temperature load
- Investigating the residual stresses
- Temperature dependent plasticity model

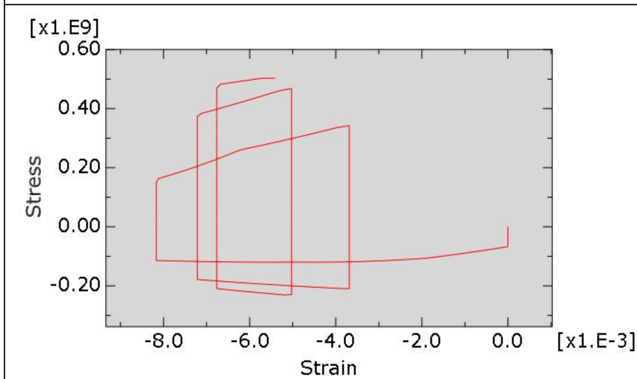


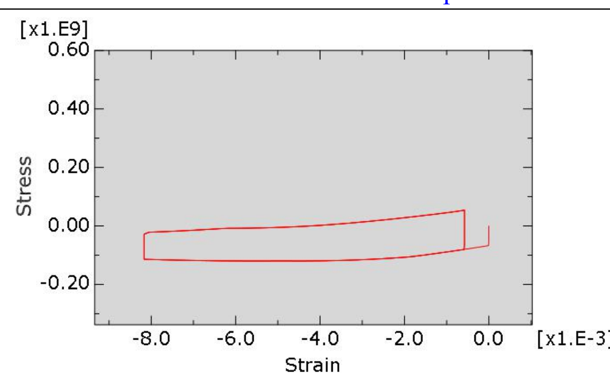


Evolution of temperature load with time



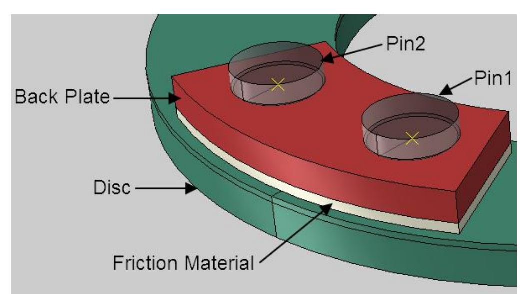
Evolution of strains in the part



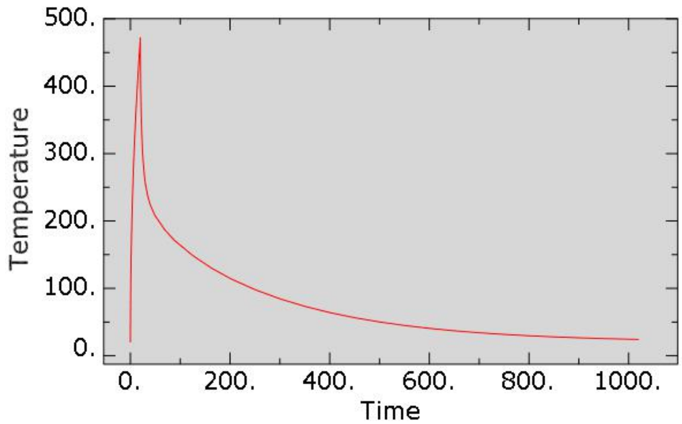


Exercise 4

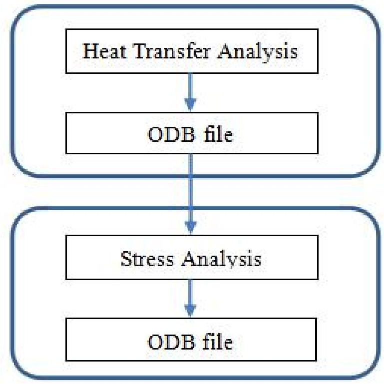
- Analyzing a brake disc subjected to cyclic temperature load
- Performing sequential stress analysis
- Cyclic loading due to repeated braking
- Importing temperature history
- Temperature dependent kinematic hardening model
- Investigating the residual stresses
- Temperature dependent material model



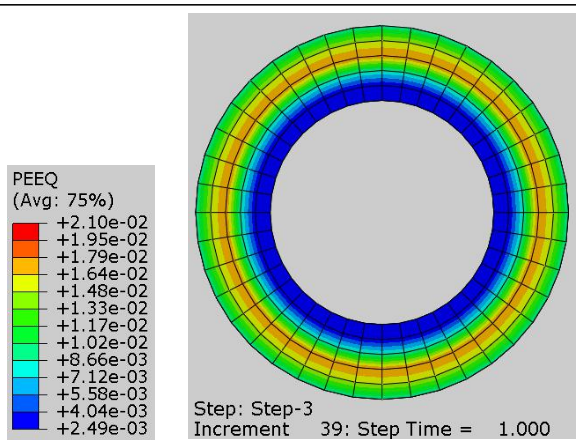
- Transforming results to cylindrical coordinate system
- Plotting circumferential stress against circumferential plastic strain



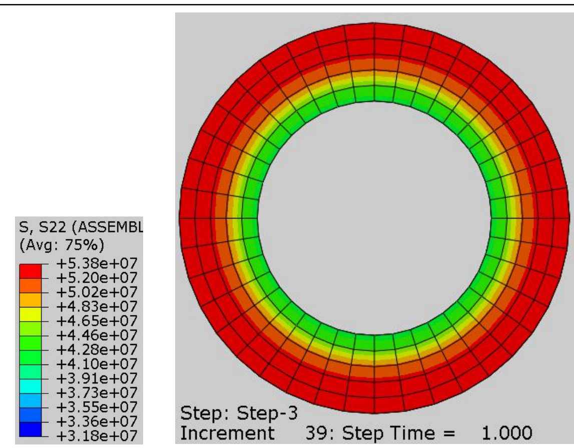
The temperature history for a node on the disc surface for one brake cycle.



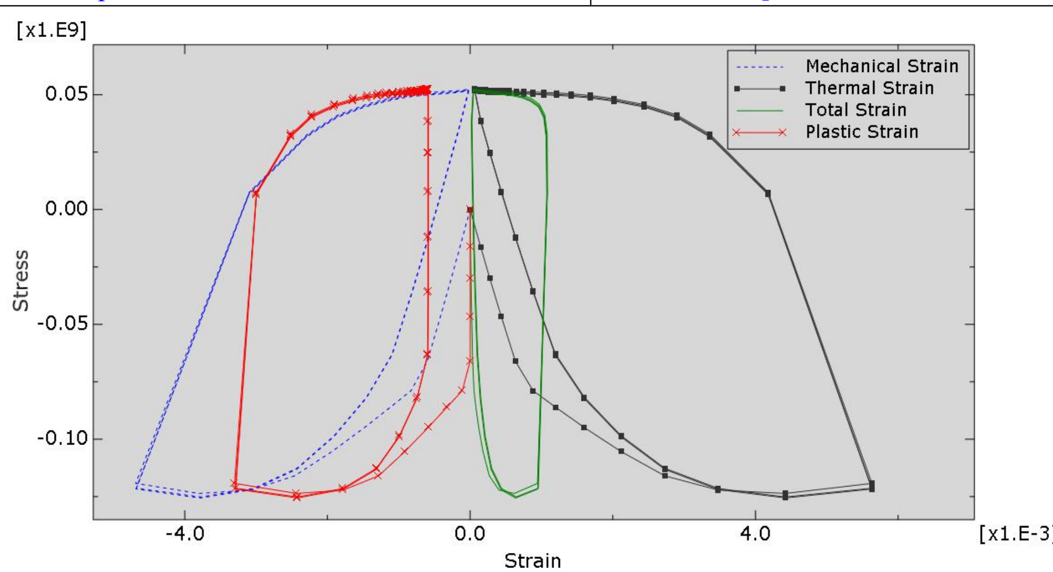
Workflow of this sequential approach



Residual plastic strain on the disc surface



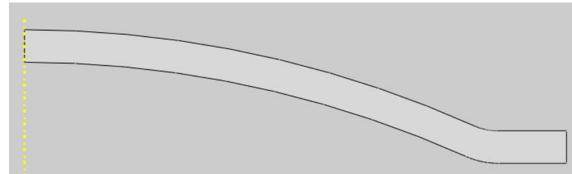
Contour plot of the circumferential stresses

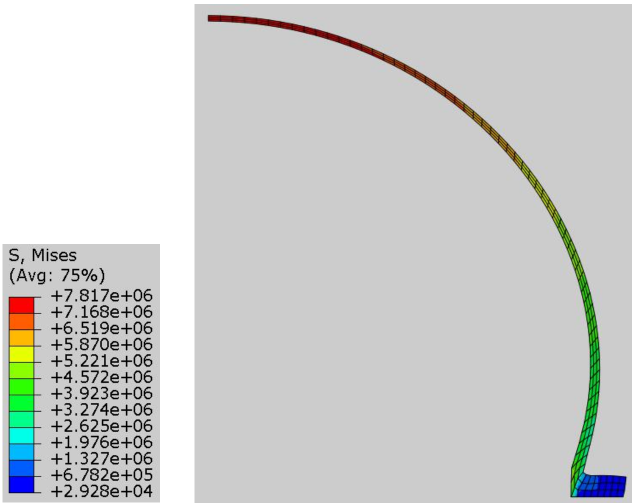


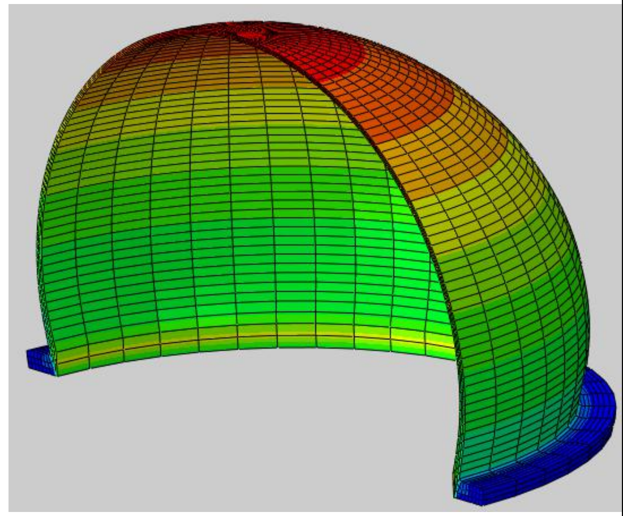
A graph of the circumferential stresses against different measures of the strain in circumferential direction for the repeated braking

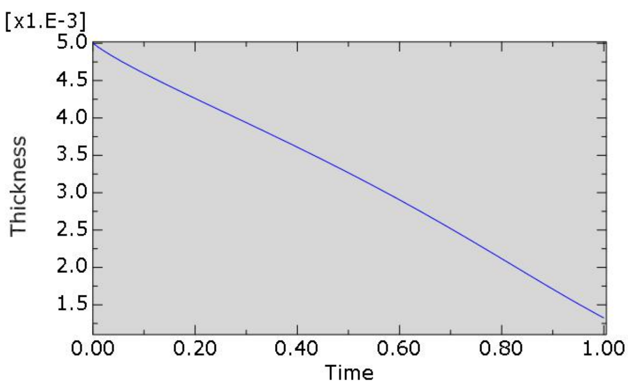
Exercise 5

- Simulate bulging of a rubber disc due to applied pressure
- Defining a Mooney-Rivlin hyperelastic model
- Plotting the history of thickness change of the disc
- Plotting the history of thickness strain

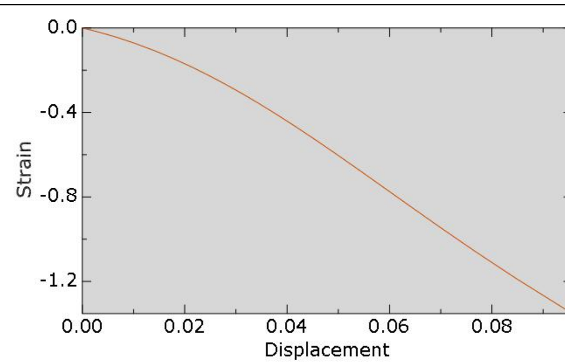








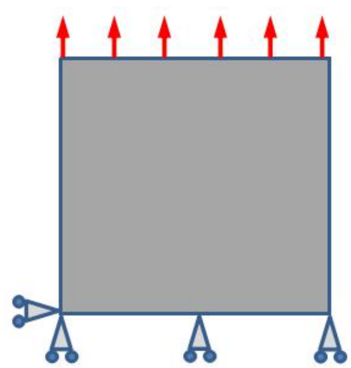
The thickness of the disc at the centerline

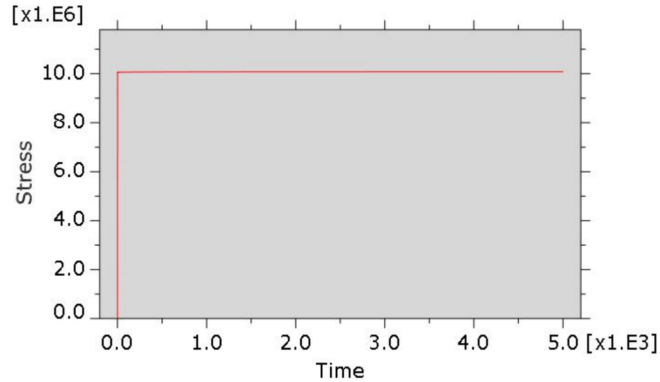


Thickness strain at the center of the disc versus the displacement of the center node

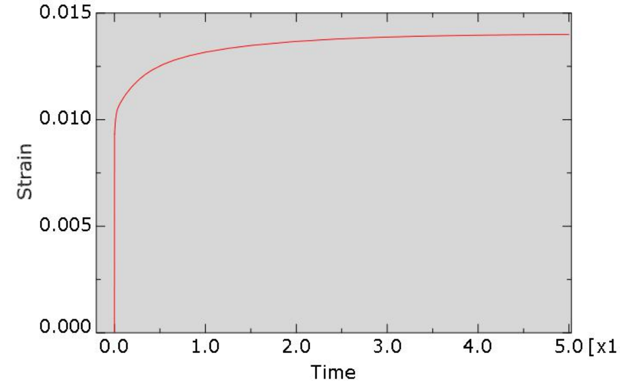
Exercise 6

- Analyze a viscoelastic sheet subjected to constant pressure load
- Defining a linear viscoelastic model for a linear elastic material.
- Defining visco procedure to simulate time-dependent behavior
- Plotting the creep strain.





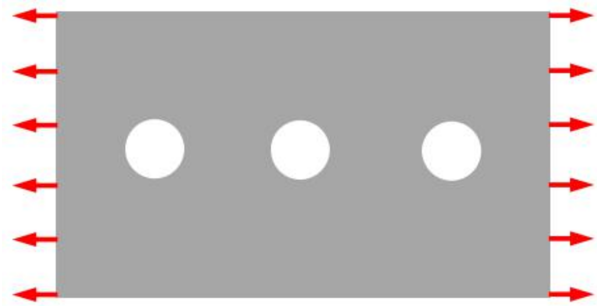
Stress along the y-direction

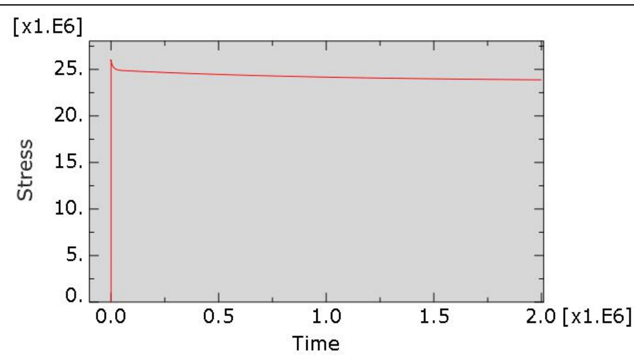


Creep in the sheet

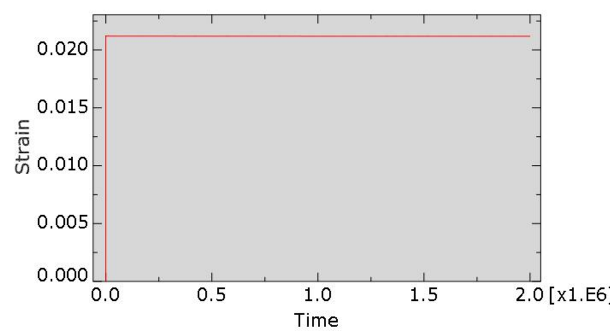
Exercise 7

- Investigating stress relaxation in a viscoelastic sheet subjected to constant strain.
- Defining a linear viscoelastic model for a hyperelastic material.
- Plotting the stress relaxation.





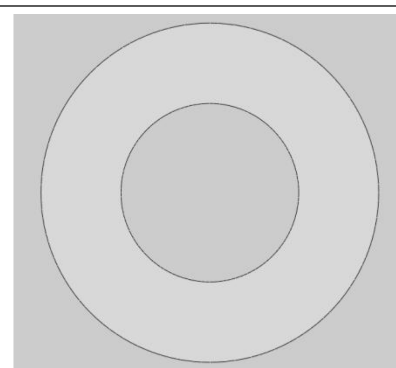
Stress relaxation in the sheet

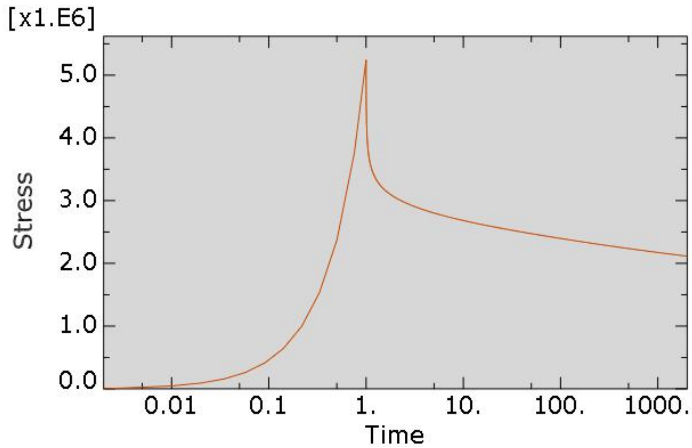


Strain along the x-direction

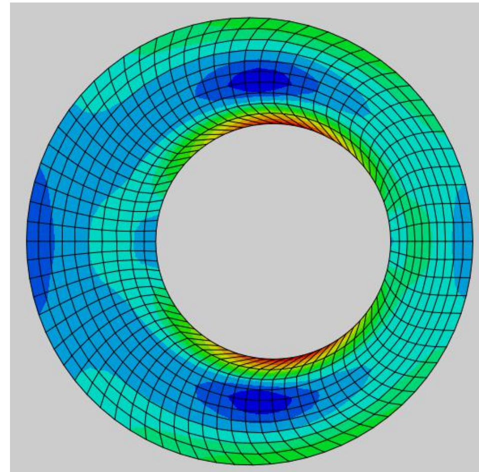
Exercise 8

- Investigating stress relaxation in a viscoelastic bushing subjected to constant strain.
- Defining a nonlinear viscoelastic model for a hyperelastic material.
- Plotting the stress relaxation.





Stress relaxation in the bushing

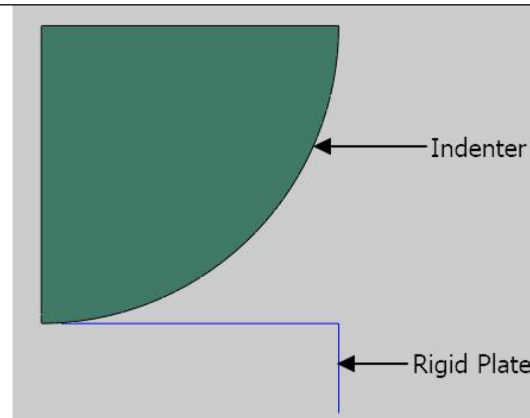


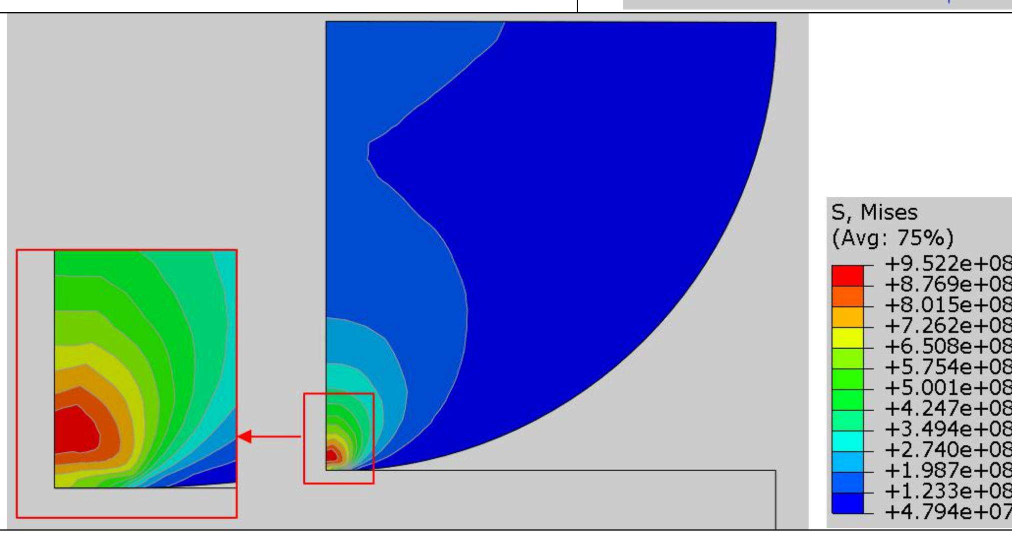
Section 5: Contact Problems

Contact problems are considered to be highly nonlinear and are one of the most difficult ones to solve. This section aims to provide practical information to perform contact analysis in Abaqus. A large number of exercises are presented providing intensive instructions to perform analysis of contact problems. Details of topics covered in exercises are given below. During such analysis it is very common to face convergence difficulties. Quite a few tutorials are devoted to diagnose such difficulties and take the corrective action.

Exercise 1

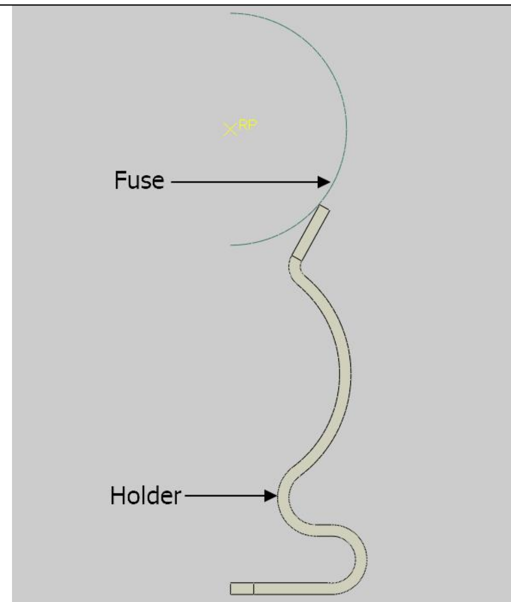
- Analyzing an elastic cylindrical indenter contacting a rigid surface
- Defining a contact pair interaction
- Plotting contact pressure along a path



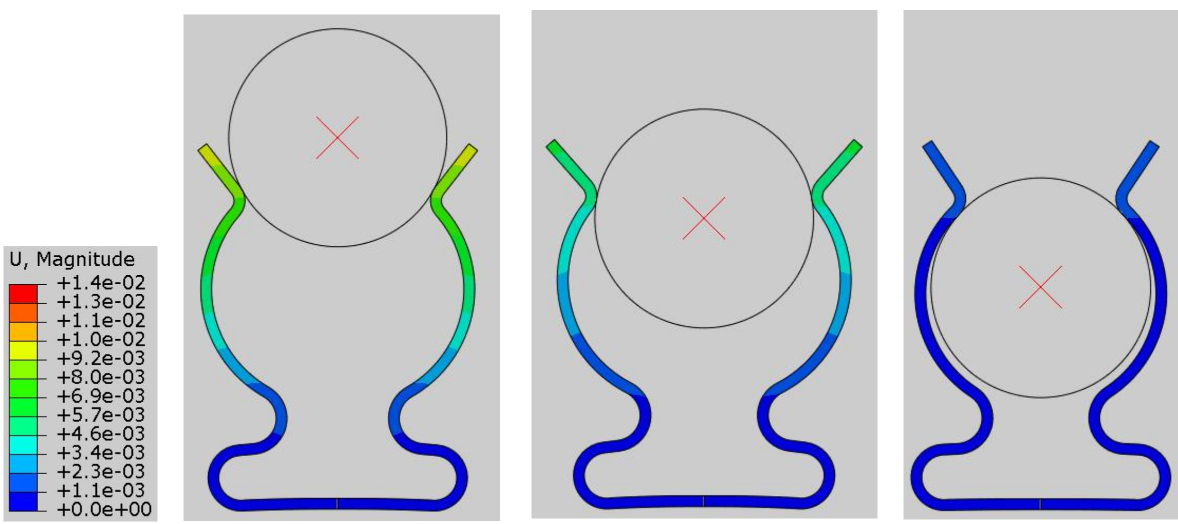


Exercise 2

- Simulation of a fuse being pushed into its holder
- Using boundary conditions to establish contact
- Extruding and mirroring simulation results



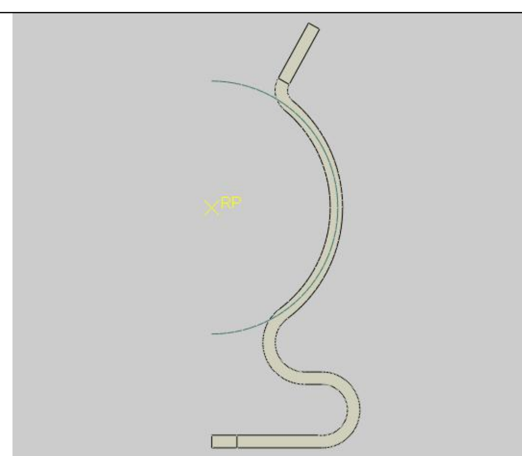
Initial positions of components



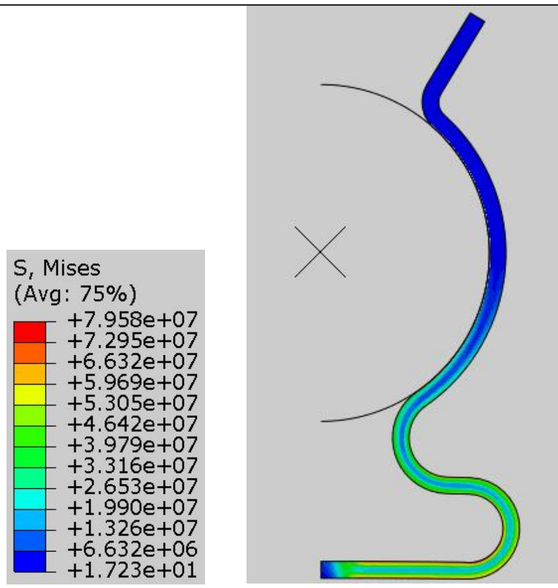
Different stages of fuse moving into holder

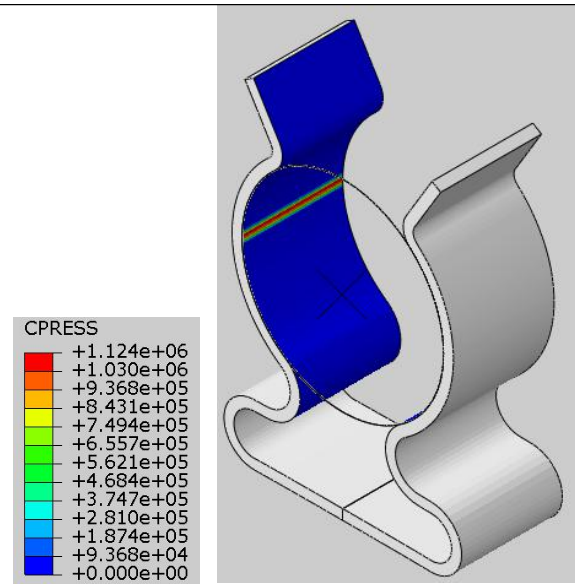
Exercise 3

- Simulation of a fuse placement into its holder using interference resolution technique
- Resolving interference to establish contact



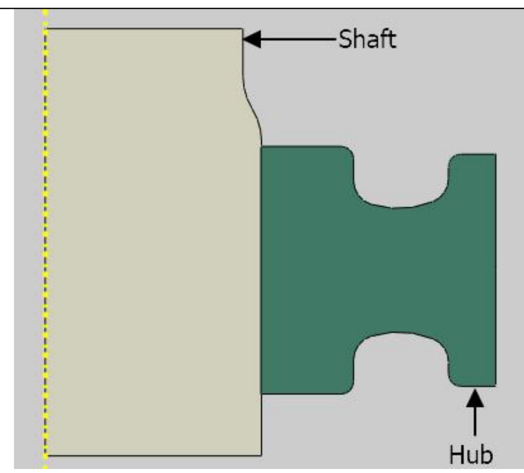
Initial positions of components

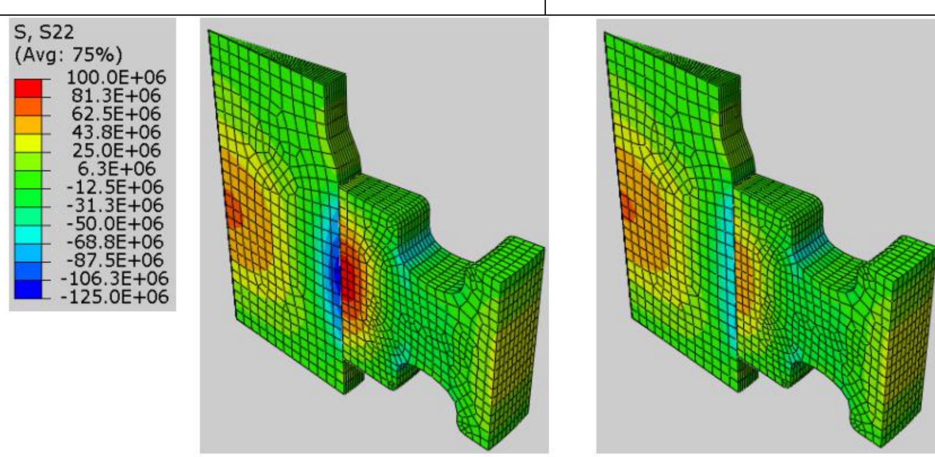




Exercise 4

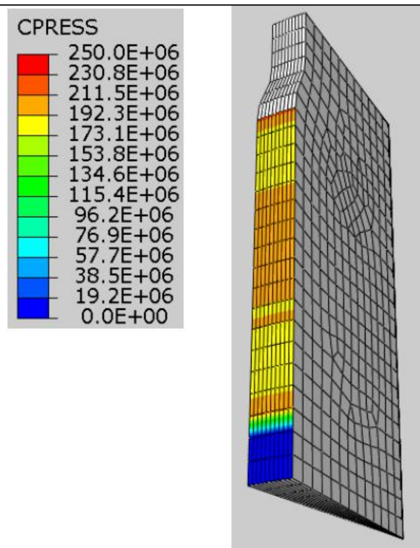
- Performing the stress analysis of shaft-hub assembly coupled by interference fit.
- Obtaining a solution by interference resolution
- Obtaining a solution by thermal expansion and later contraction of hub
- Comparing solution by interference resolution vs. solution by thermal expansion
- Maximum torque that surfaces in contact can transmit



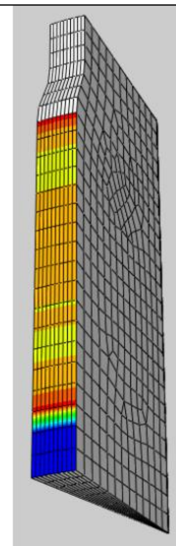


Solution by thermal expansion

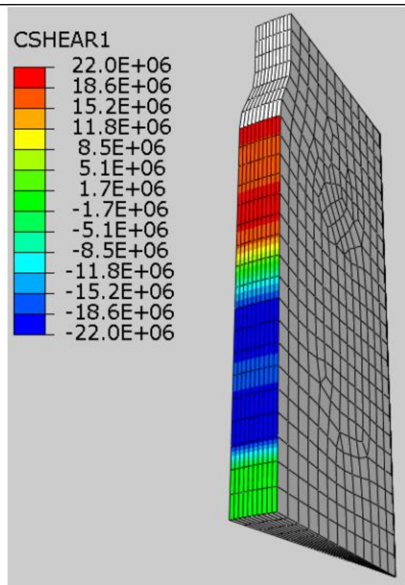
Solution by interference resolution



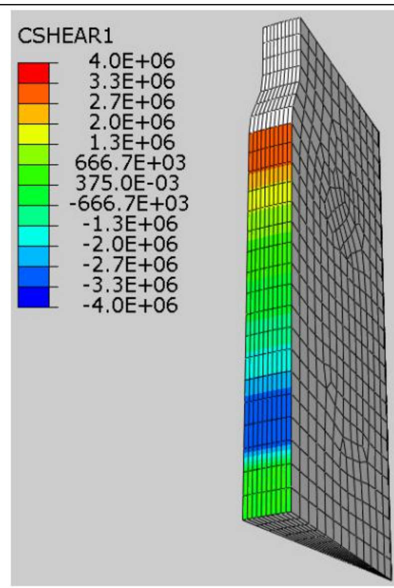
Solution by thermal expansion



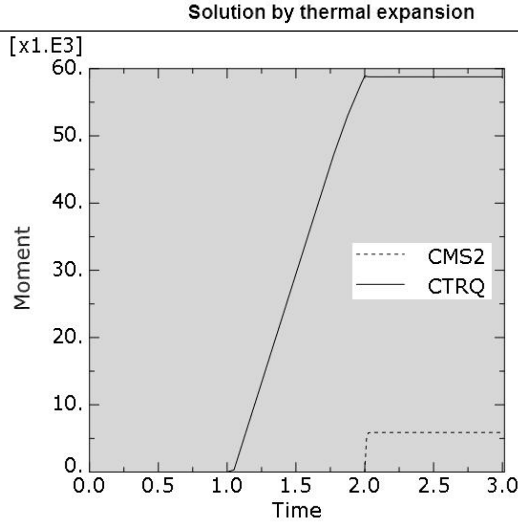
Solution by interference resolution



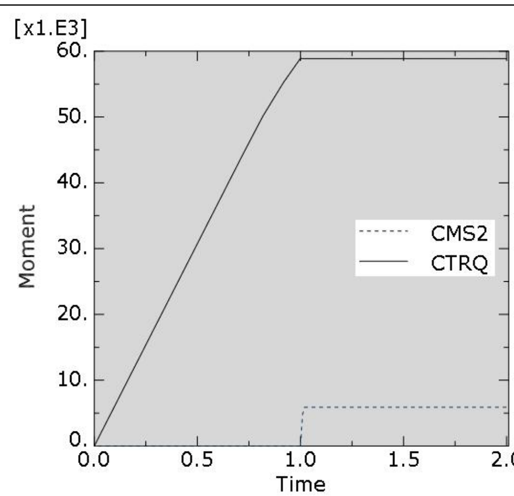
Solution by thermal expansion



Solution by interference resolution



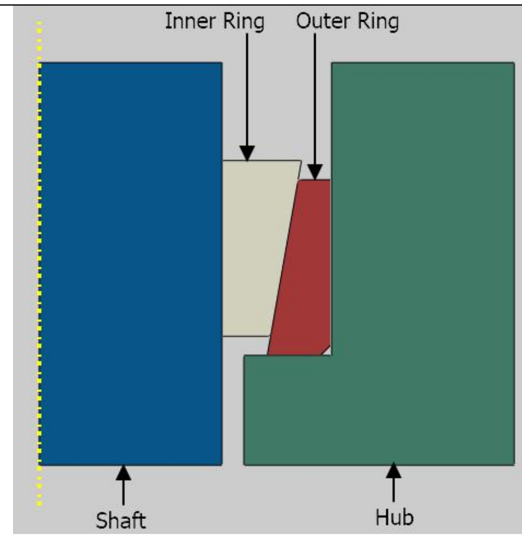
Solution by thermal expansion

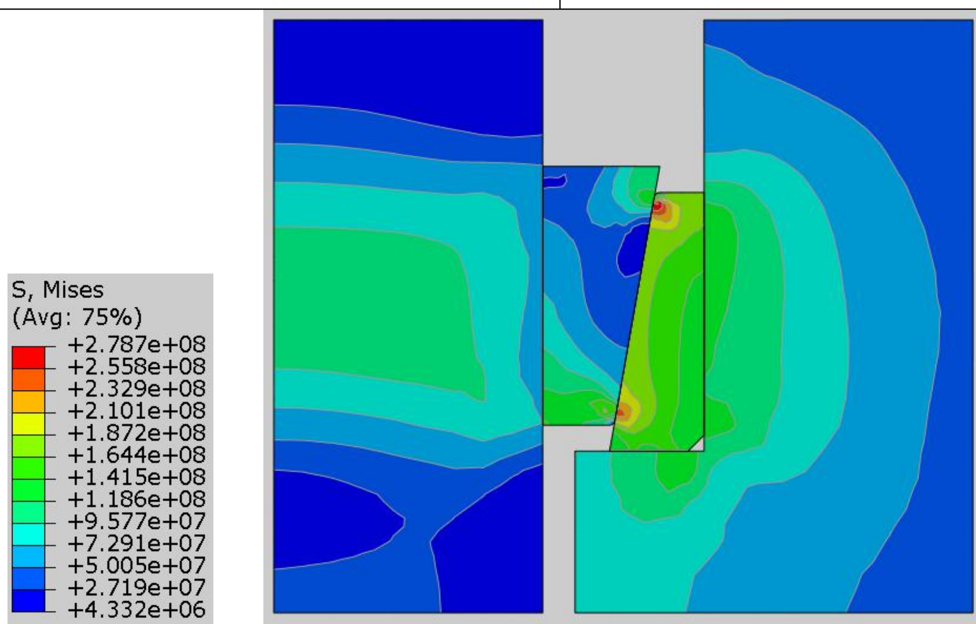


Solution by interference resolution

Exercise 5

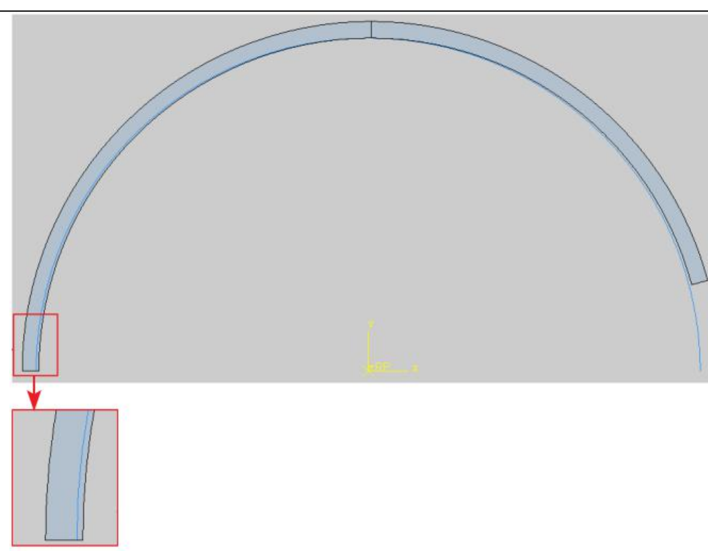
- Performing the stress analysis of shaft-hub assembly coupled by conical rings
- Choosing a master surface among two deformable surfaces.
- Diagnosing the error messages with Job Diagnostics dialog box
- Invoking unsymmetric solver

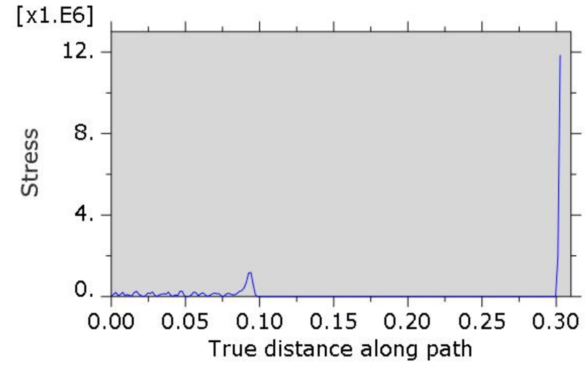




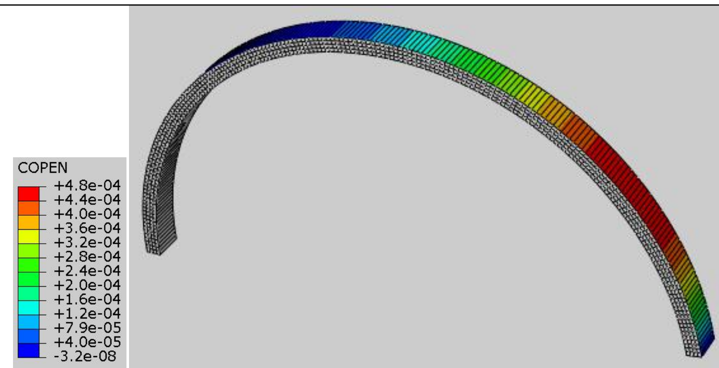
Exercise 6

- Performing the stress analysis of a piston ring fitted inside a rigid housing.
- Using interference resolution capability to resolve initial overclosure
- Comparing the results obtained with small sliding vs. finite sliding
- Introducing the contact stabilization to alleviate the convergence difficulties
- Improving refinement level of displayed results
- CSTATUS output variable





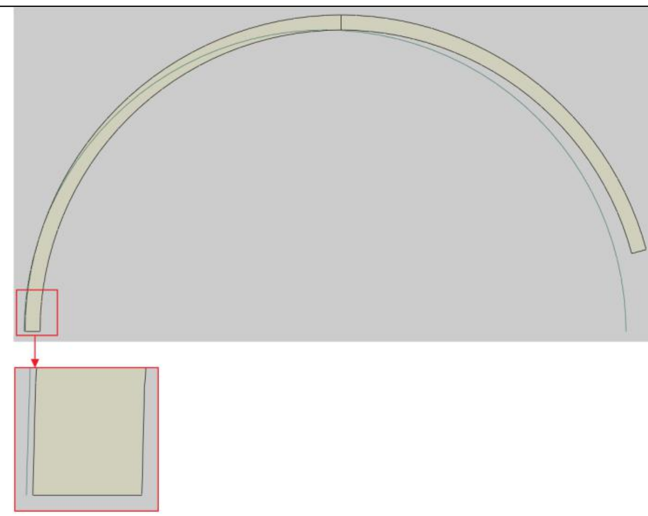
Contact stress along the boundary of the ring

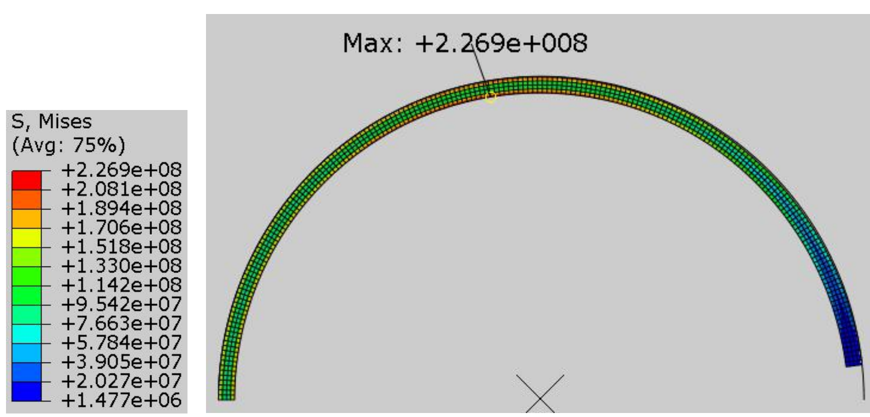


Contour plot of COPEN(contact opening)

Exercise 7

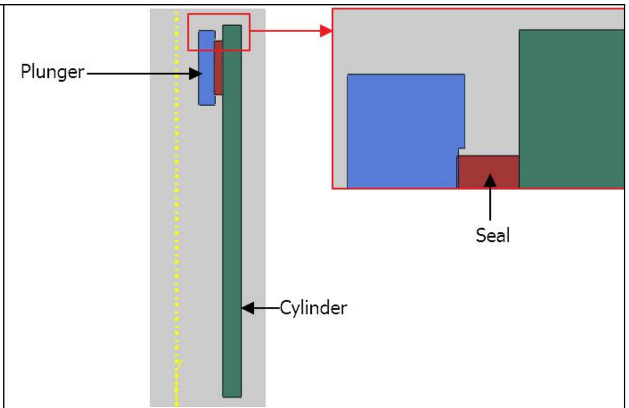
- Performing the stress analysis of a piston ring fitted inside a rigid housing by using multi-step approach instead of interference resolution to solve the contact problem

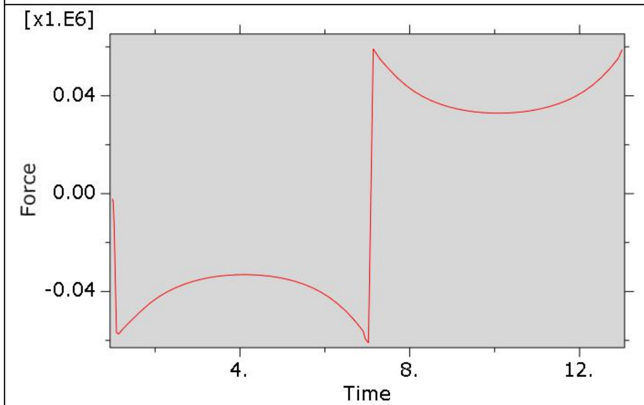




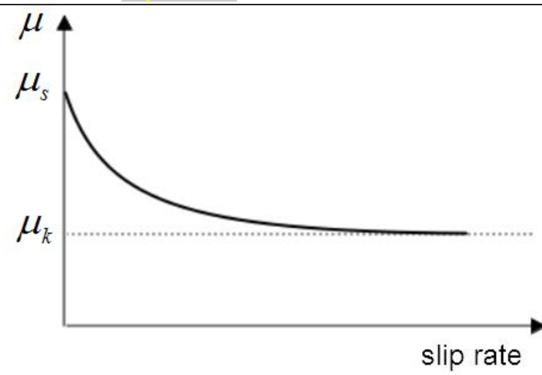
Exercise 8

- Simulation of a reciprocating plunger
- Slip-rate dependent friction model
- Exponential decay friction model
- Rough friction
- Stick-slip behavior





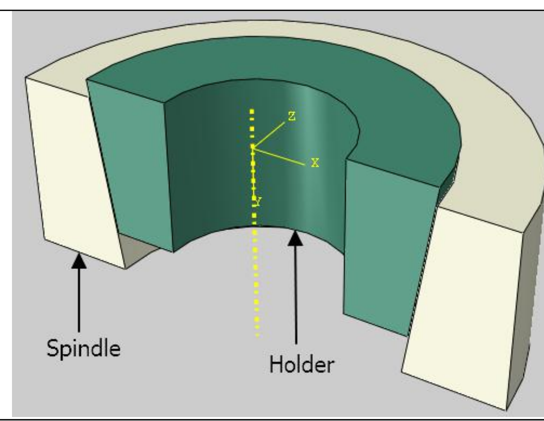
Force due to frictional stress at the contact interface

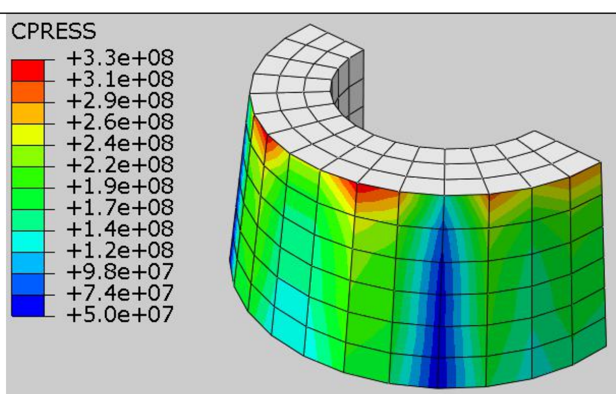


Exponential decay friction model

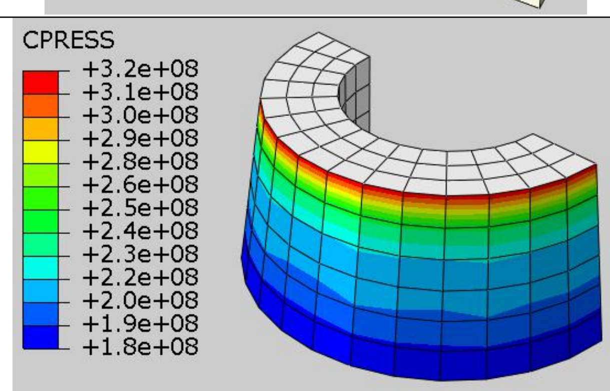
Exercise 9

- Simulating contact stresses at the interface of a spindle-holder assembly
- Smoothing contact surfaces to overcome the contact stress inaccuracy arising due to faceted surface geometry





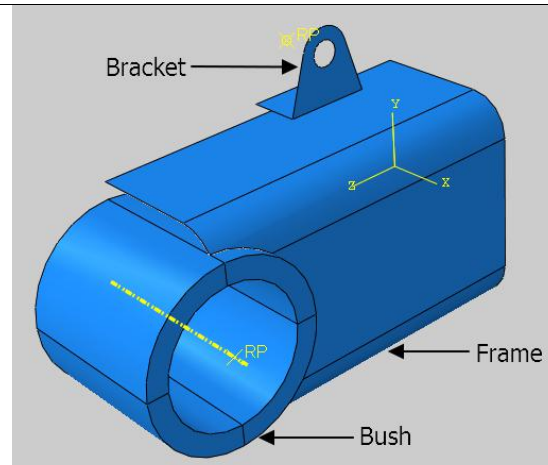
Stress inaccuracy arising due to faceted surface geometry

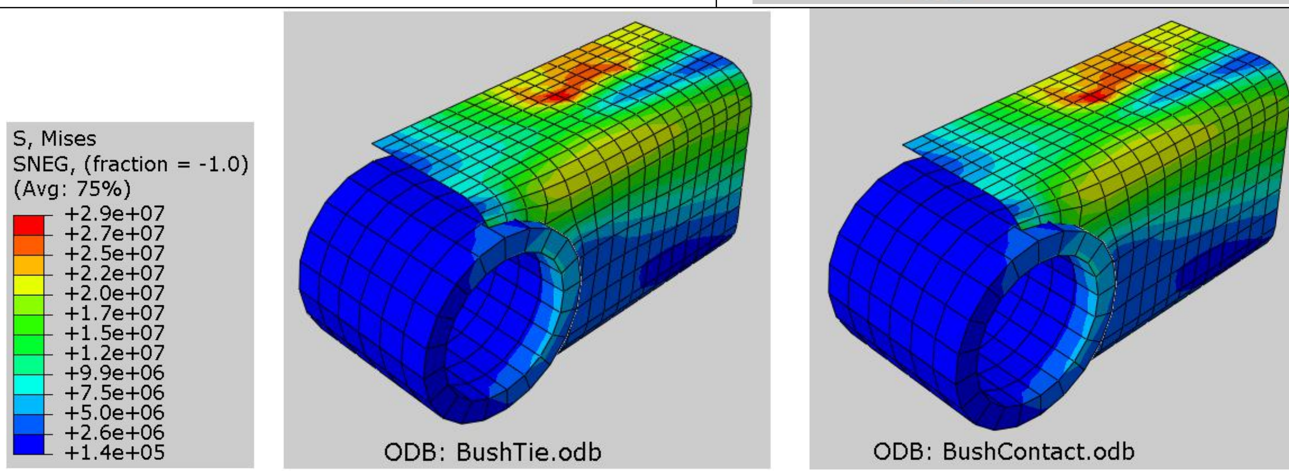


Corrected solution obtained be smoothing contact surfaces

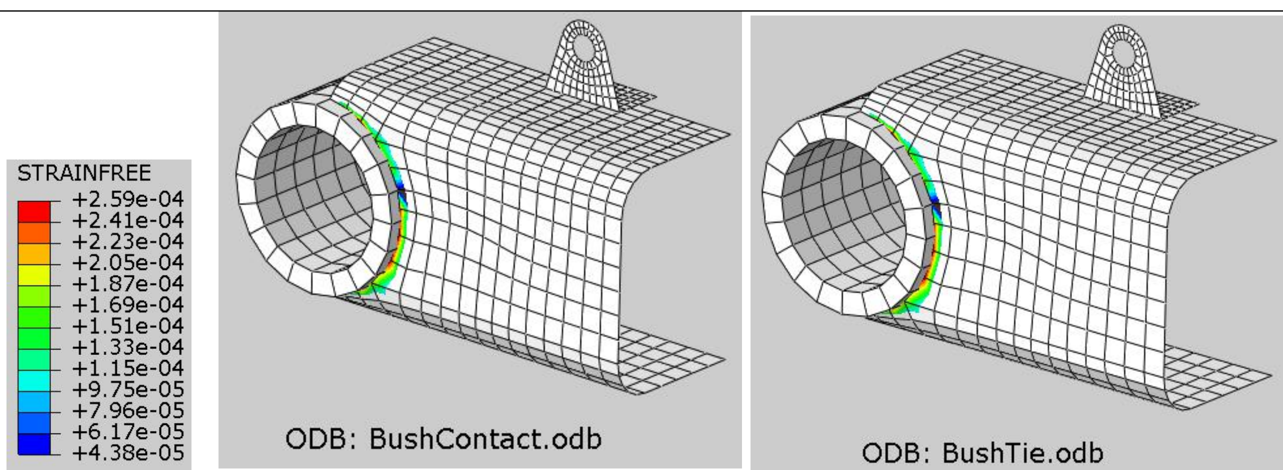
Exercise 10

- Welding a shell part to a solid part using
  1) Tied contact
  2) Tie constraint
- Slave node adjustment to establish contact
- STRAINFREE output variable
- Diagnosing the error messages with Job Monitor dialog box





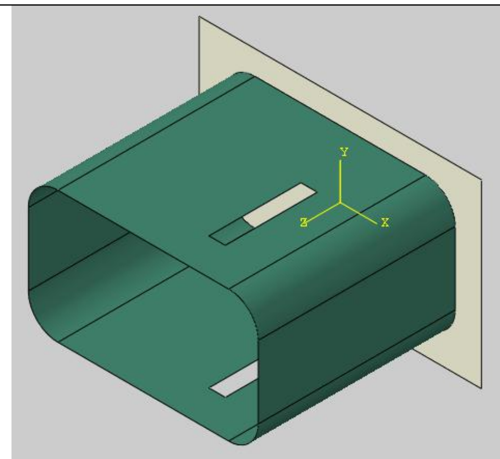
Comparing the 'tied contact' to 'tie constraint'

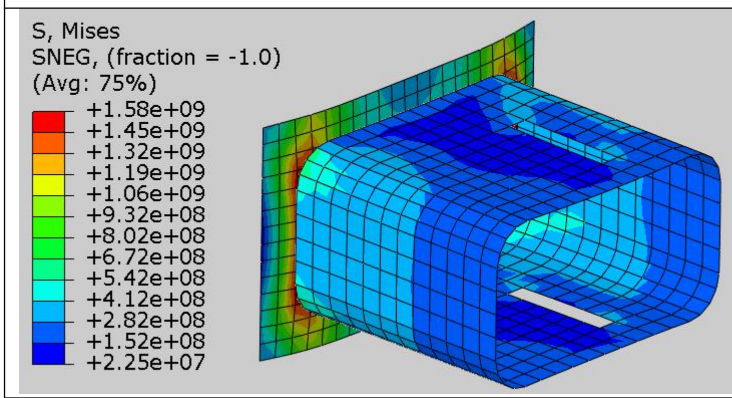


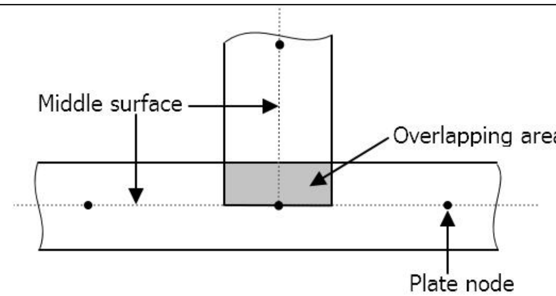
Comparing the strain-free adjustments to initial positions

Exercise 11

- Using tied contact to "weld" two shell parts together
- Using shell offset to avoid overlapping
- Using query tool to determine SPOS/SNEG faces of shell

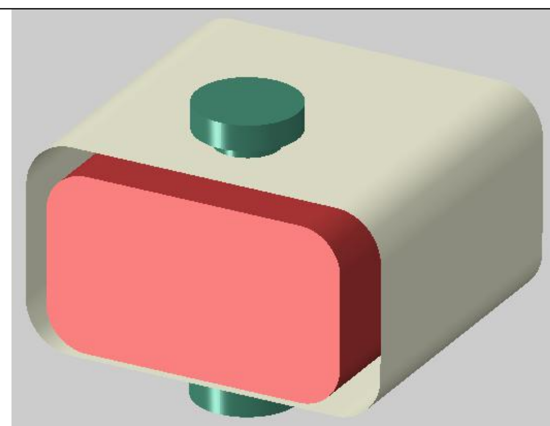


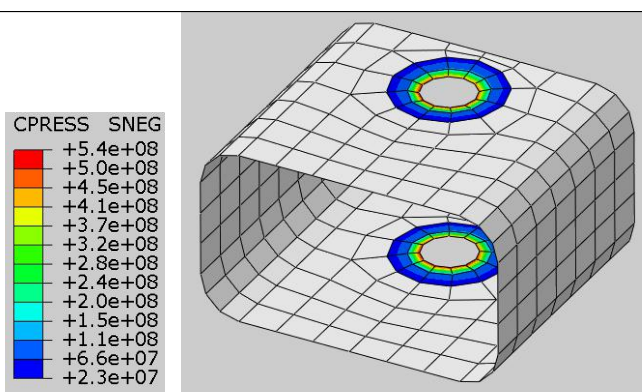


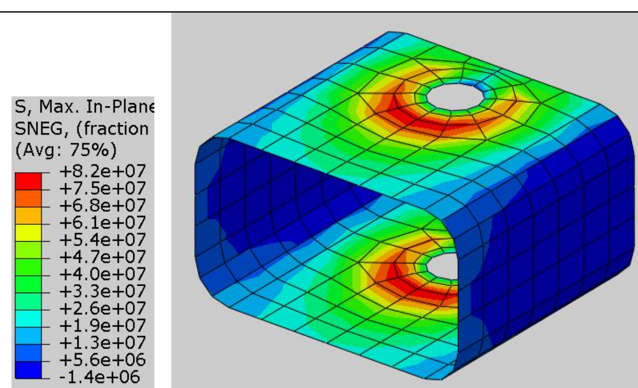


Exercise 12

- Defining contact between the surfaces of solid and shell parts
- Accounting for shell thickness
- Using unsymmetric solver for better convergence
- Plotting surface normal to determine SPOS/SNEG faces of shell
- Choosing a master surface among two deformable surfaces.

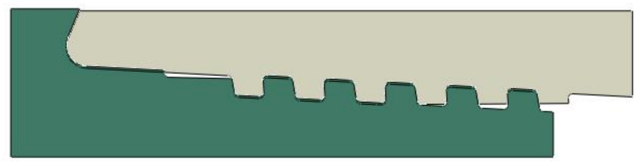


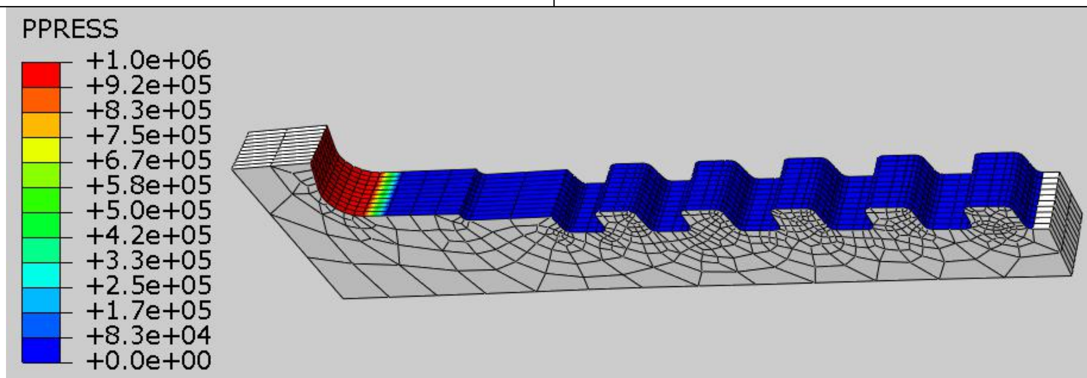




Exercise 13

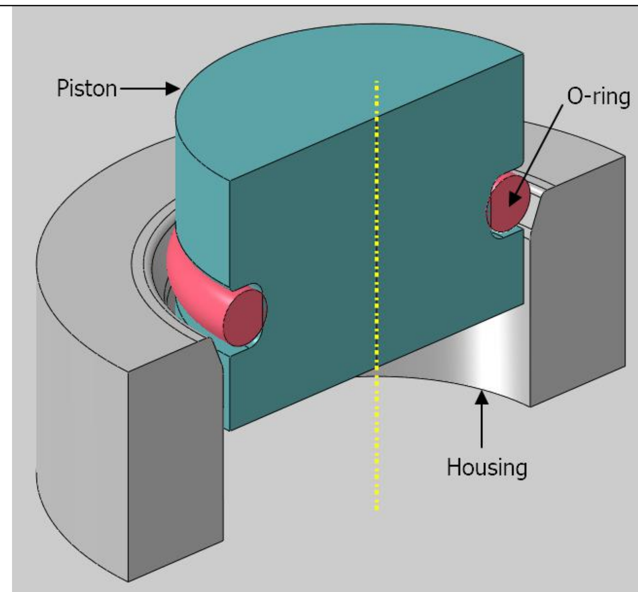
- Analyzing stresses in a VAM TOP connector due to internal fluid pressure
- Pressure penetration loading
- PPRESS output variable

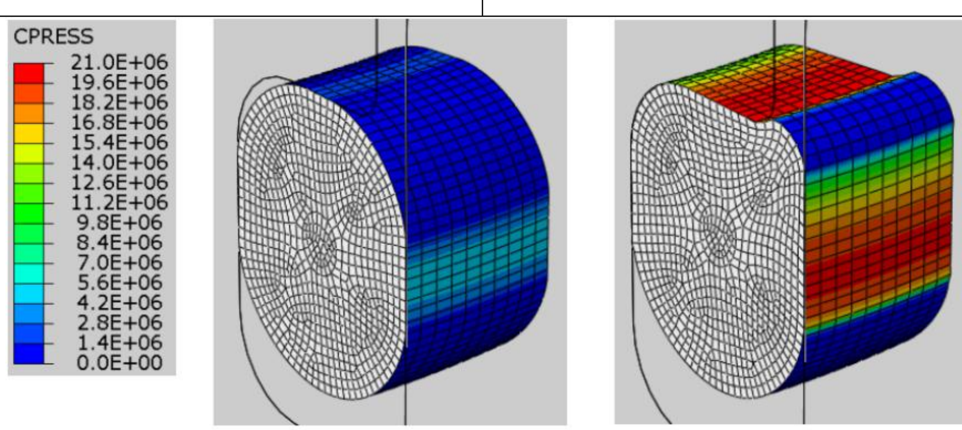




Exercise 14

- Analysis of an o-ring seal under the pressure of a fluid
- Pressure penetration interaction
- Diagnosing the convergence problems

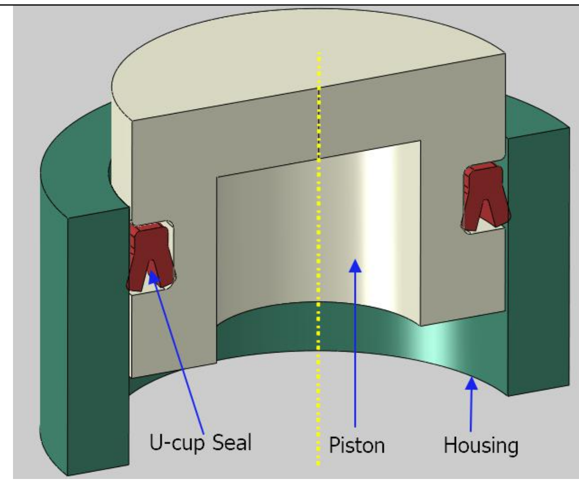


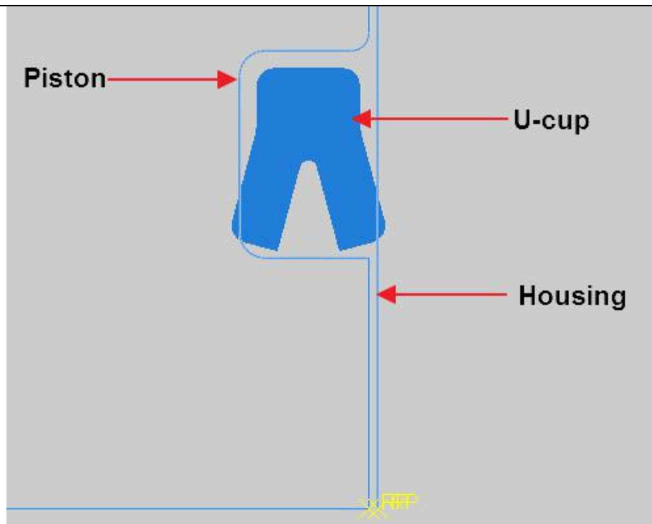


before applying fluid pressure after applying fluid pressure

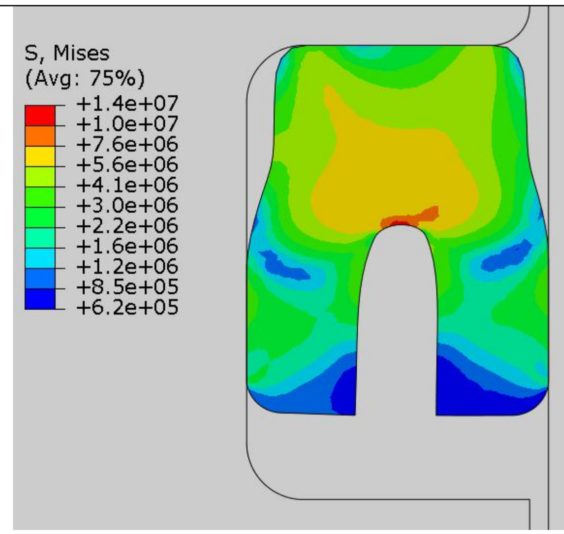
Exercise 15

- Analysis of an u-cup seal under the pressure of a fluid
- Node-to-surface discretization for interference resolution
- Diagnosing the cause of error message using Job Diagnostics dialog box
- Introducing the contact stabilization to alleviate the convergence difficulties

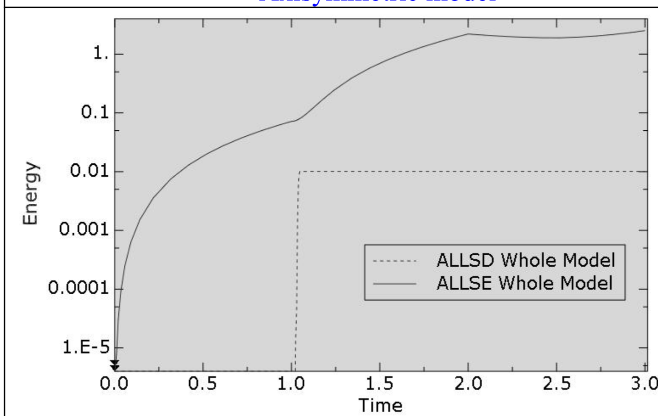




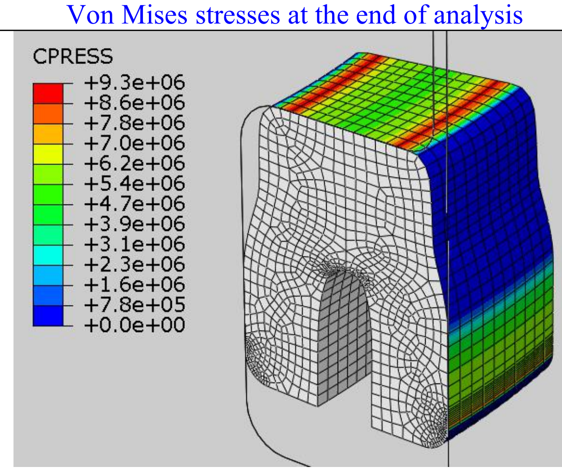
Axisymmetric model



Von Mises stresses at the end of analysis



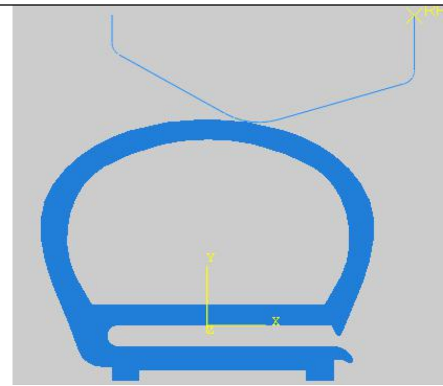
Comparing the energy dissipation due to stabilization (ALLSD) to the elastic strain energy of the model (ALLSE)

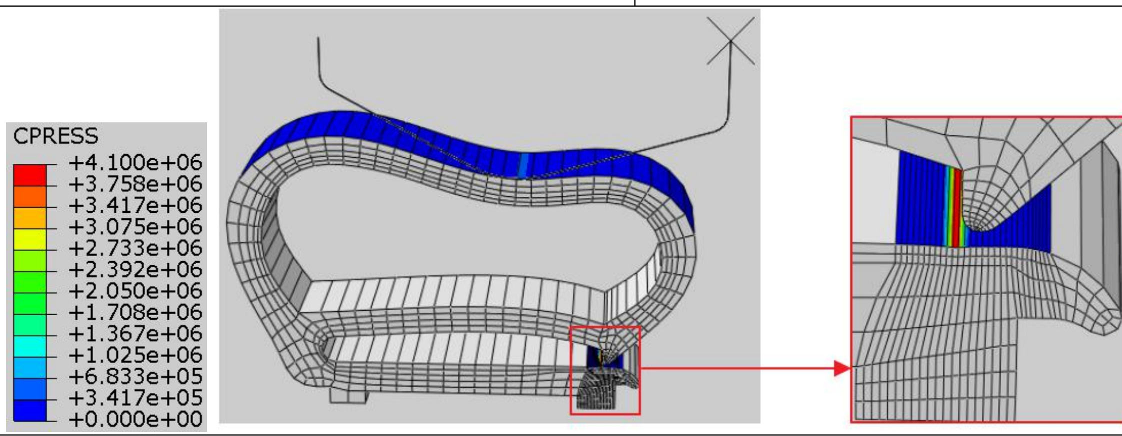


The contact pressure distribution

Exercise 16

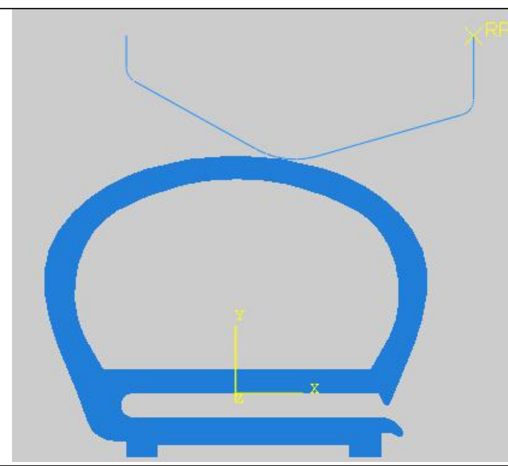
- Simulation of a hyperelastic door seal
- Defining self contact
- Detecting chattering in the contact using Job Diagnostics dialog box
- Understanding "Negative eigenvalues" error message
- Introducing the contact stabilization
- Fraction of damping at end of step

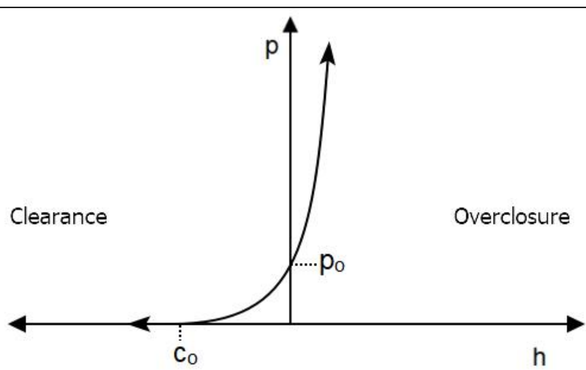


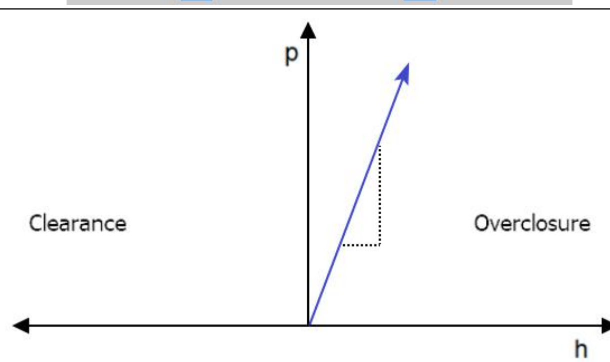


Exercise 17

- Contact analysis of the door seal using "soft" contact
- Defining the following softened contact models.
  I. Exponential
  II. Linear
  III. Tabular

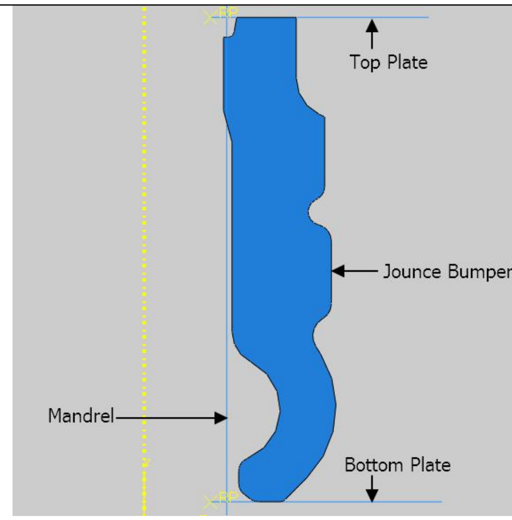


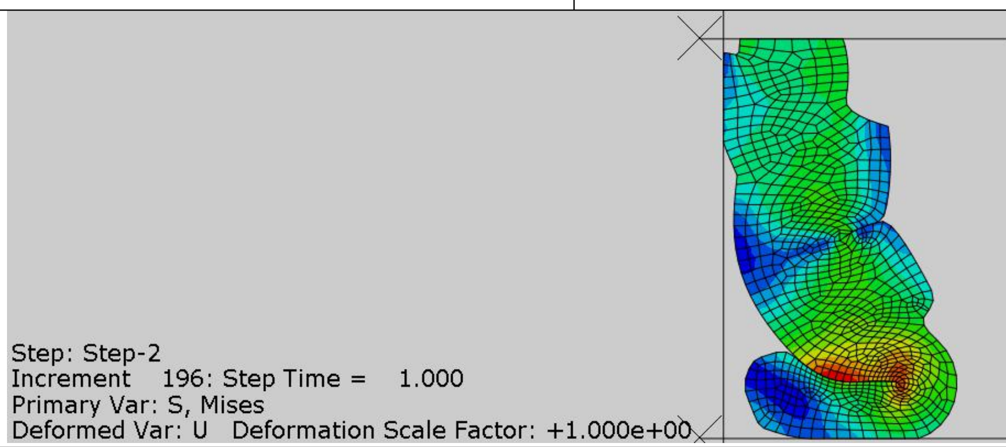




Exercise 18

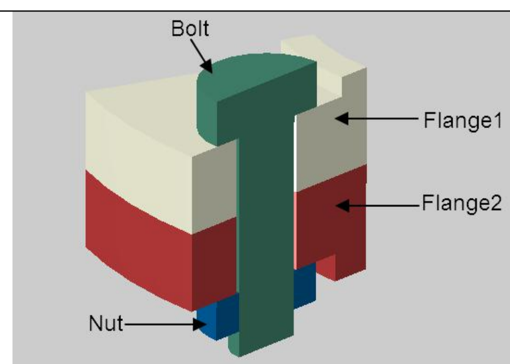
- Simulating a jounce bumper.
- Defining self contact
- Using Line Search algorithm to prevent divergence

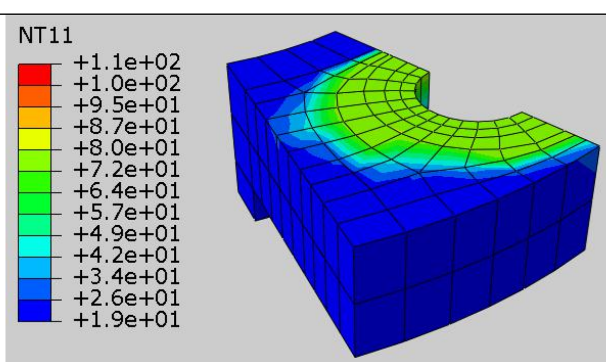


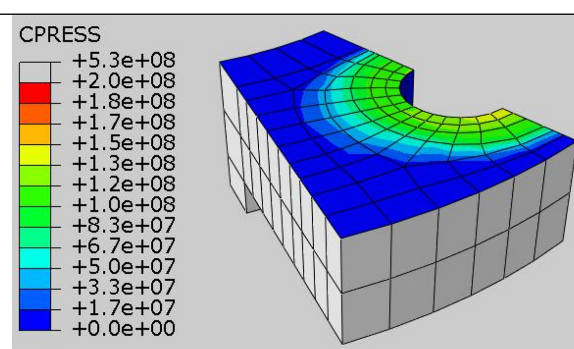


Exercise 19

- Performing the heat transfer analysis of a bolted flange joint
- Defining thermal contact conductance as a function of contact pressure
- Finding contact pairs using contact detection tool

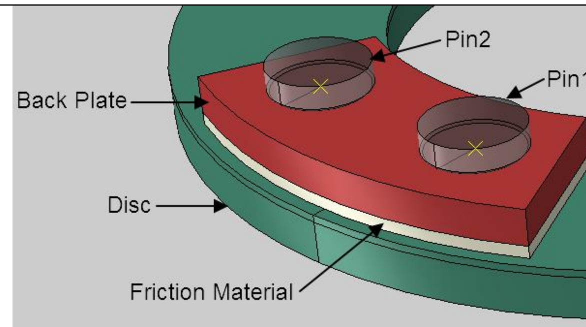


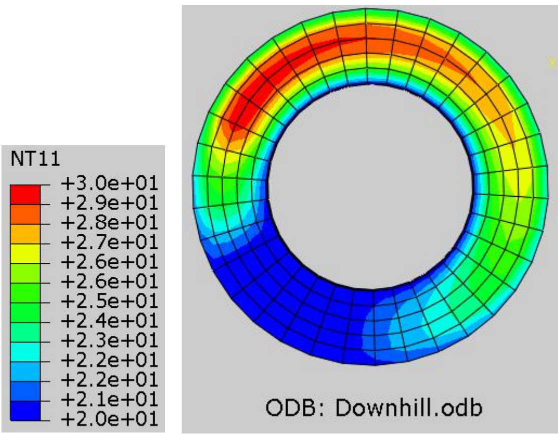


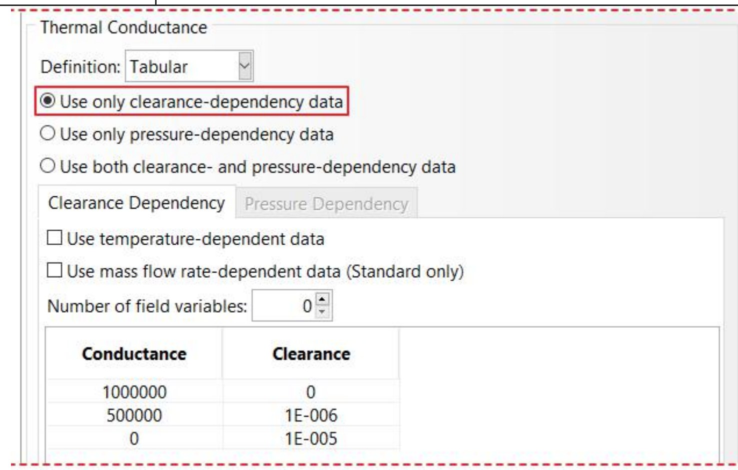


Exercise 20

- Thermo-mechanical simulation of a disc brake system
- Defining temperature dependent coefficient of friction
- Defining thermal contact conductance is a function of gap clearance
- Defining an interaction to model heat transfer due to convection
- Specifying heat generation and its distribution
- Modifying solution controls to reduce computational cost

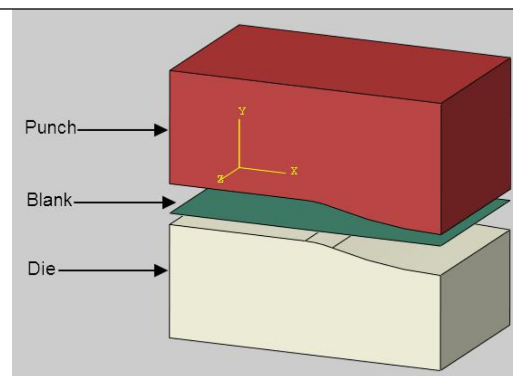


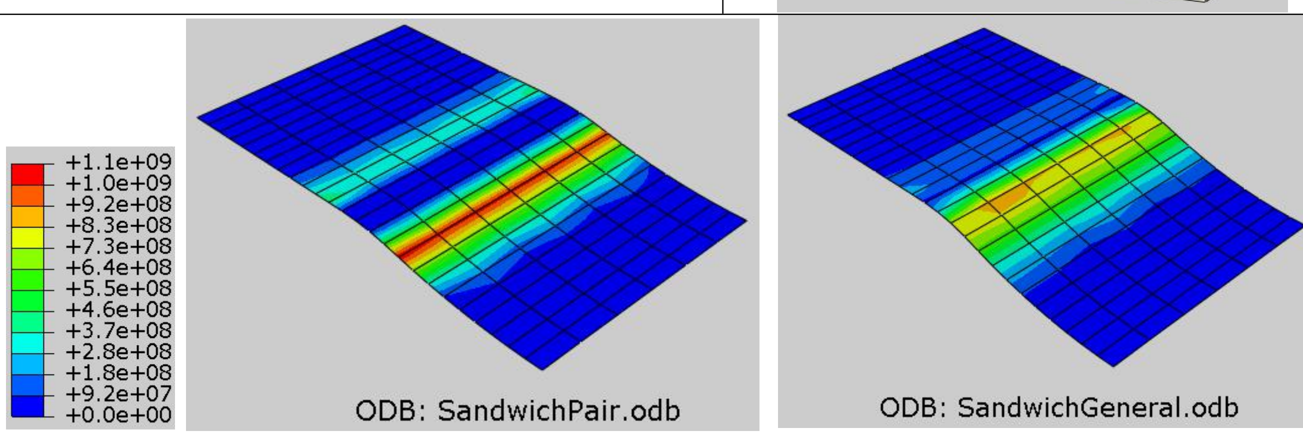




Exercise 21

- Defining contact interaction for a punch-blank-die assembly
- Using general contact approach to define interaction
- Comparing contact pressure plots for general contact and contact pairs approach

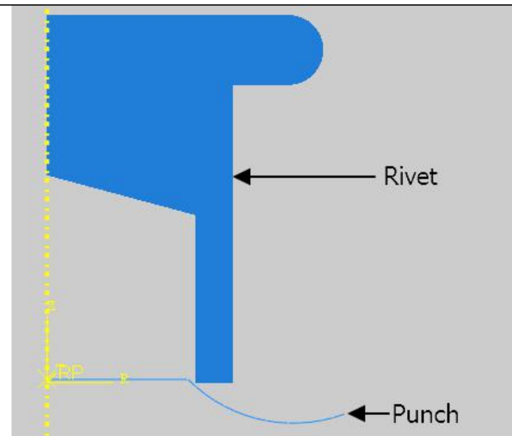


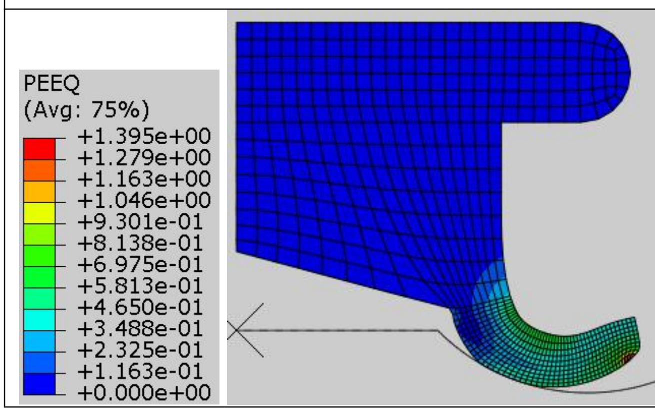


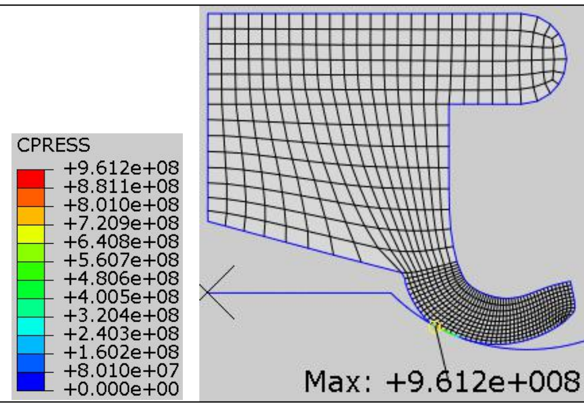
Comparison of contact pressure distribution for general contact and contact pairs approach

Exercise 22

- Simulation of the upsetting of a tubular rivet
- Visualizing default master-slave assignment in general contact
- Overriding the default master-slave assignment in general contact
- Automatic surface smoothing in general contact
- Comparing contact normal force for the general contact approach with contact pairs approach

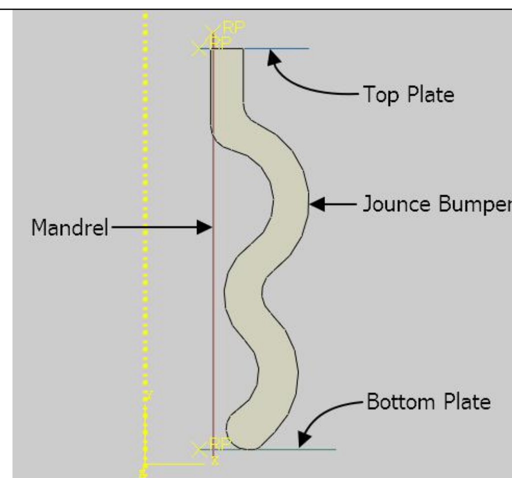


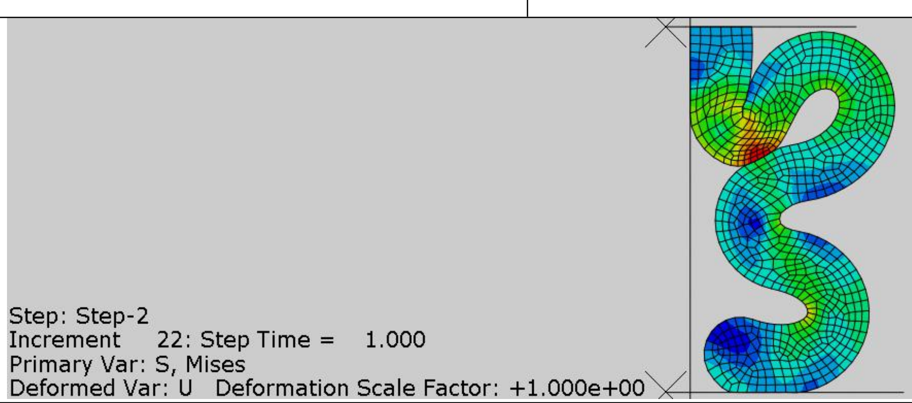




Exercise 23

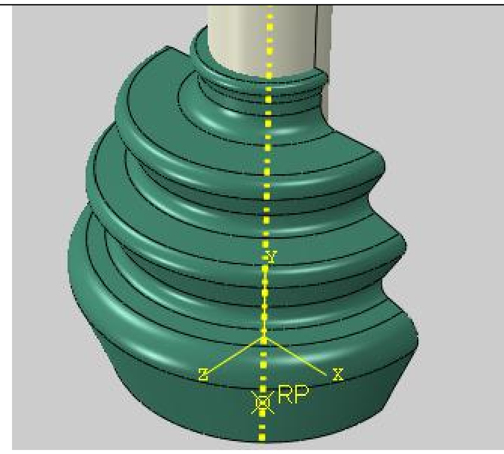
- Simulating a jounce bumper.
- Defining a contact initialization to resolve initial overclosures in a general contact interaction

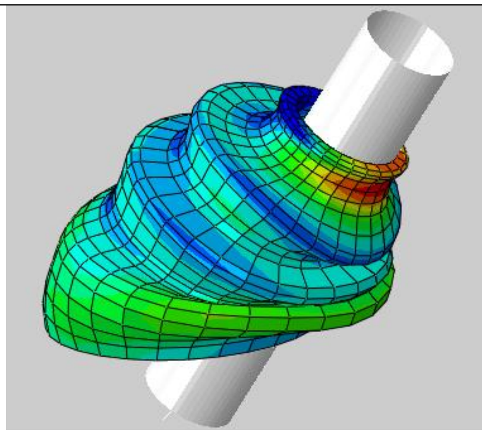


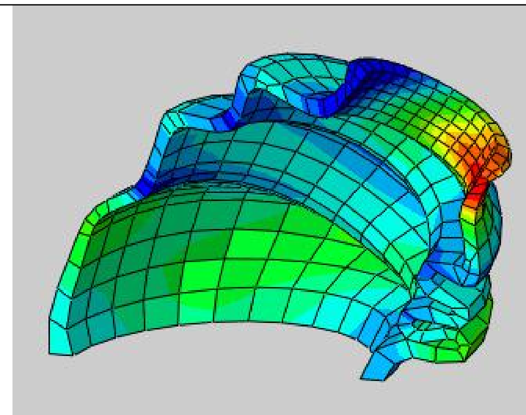


Exercise 24

- Simulating the angular movement of the shaft of a boot seal
- Defining a contact pairs and a general contact interaction together in a model

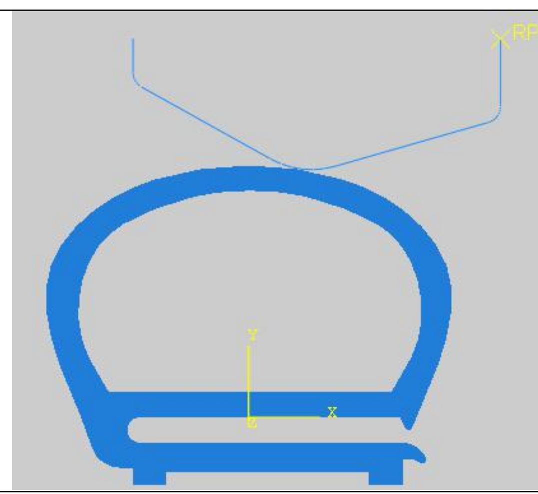


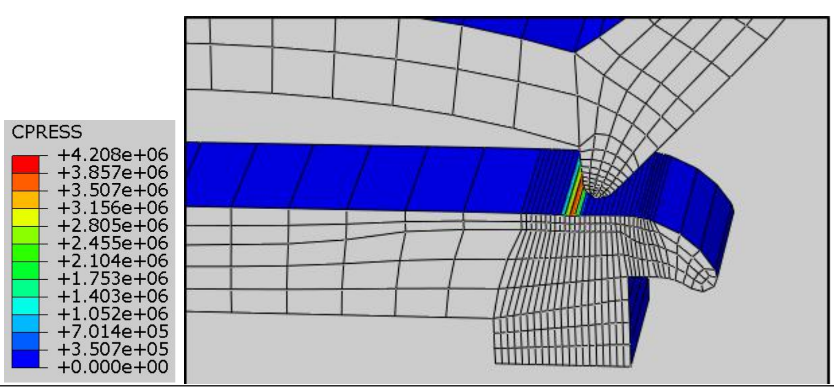




Exercise 25

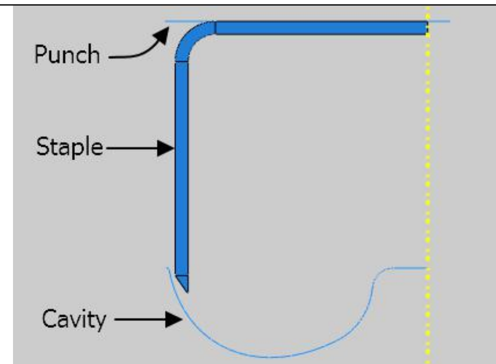
- Performing the stress analysis of a door seal.
- Introducing contact stabilization in a general contact interaction
- Applying stabilization in tangential direction.
- Reversing the orientation of individual faces during the surface definition

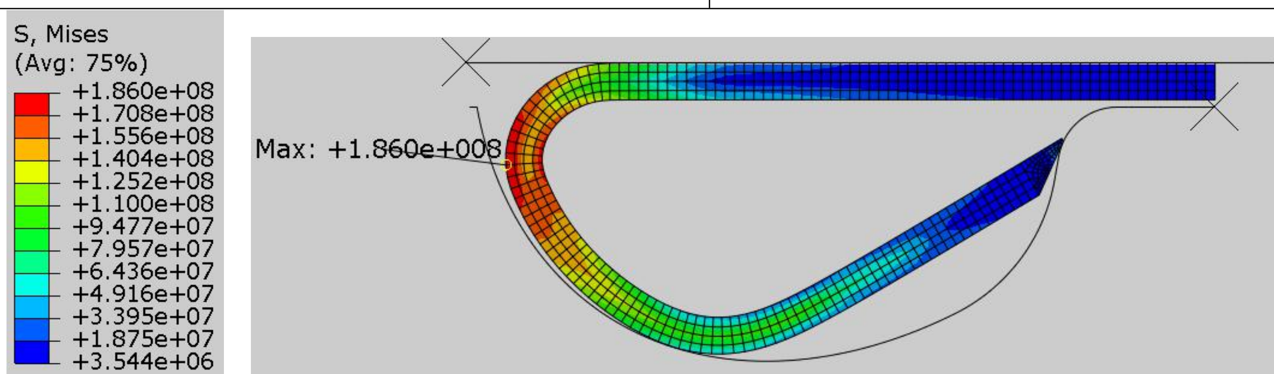




Exercise 26

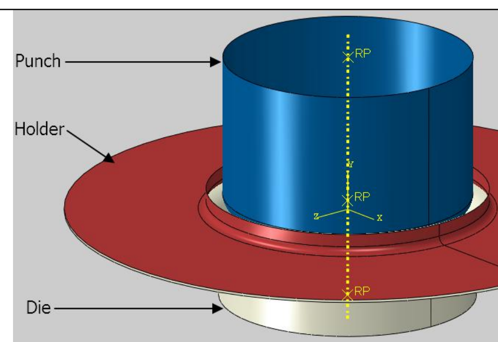
- Simulation of the bending of a staple
- Simulating the quasi-static problem using both Abaqus/Standard and Abaqus/Explicit
- Using the "softened" contact to overcome the convergence problems

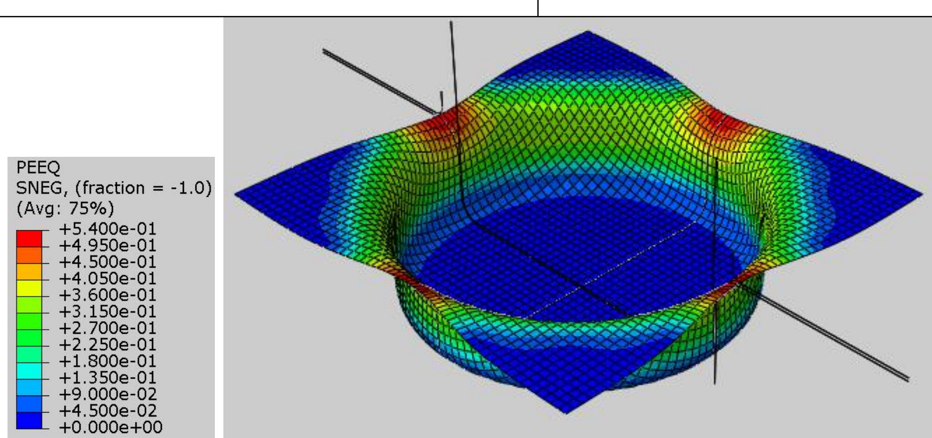




Exercise 27

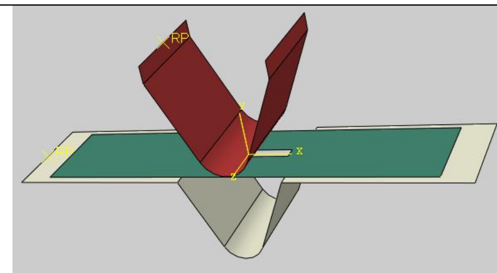
- Simulating the deep drawing of a cup using both Abaqus/Standard and Abaqus/Explicit.
- Monitoring the thickness of shell elements
- Considering thickness changes in a general contact interaction

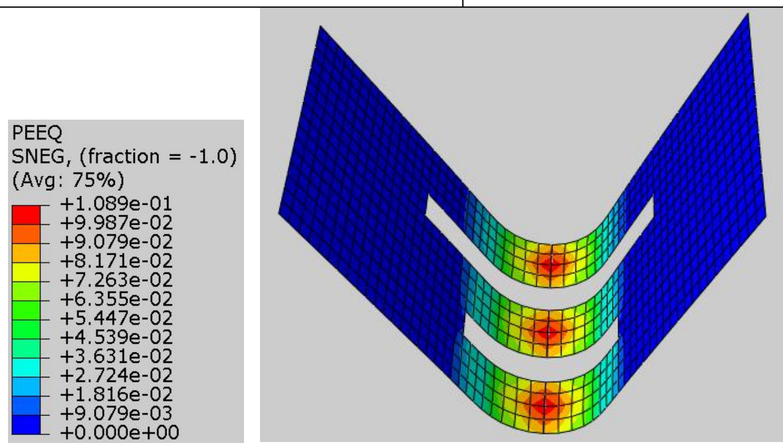




Exercise 28

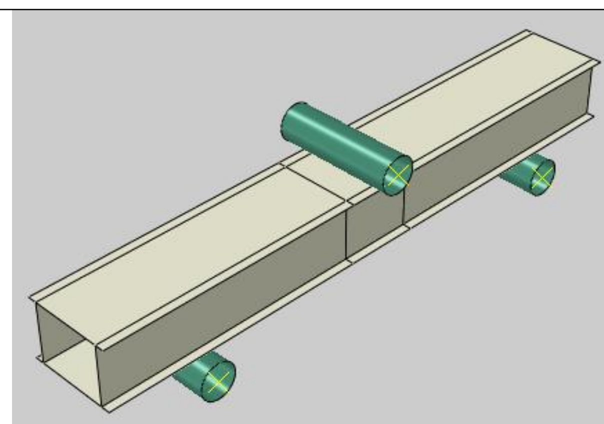
- Simulating the v-bending of a sheet metal blank
- Compare the results obtained with contact pair approach and general contact approach
- Rendering thickness of shell elements
- CTHICK output variable
- Contact controls to eliminate thickness reductions

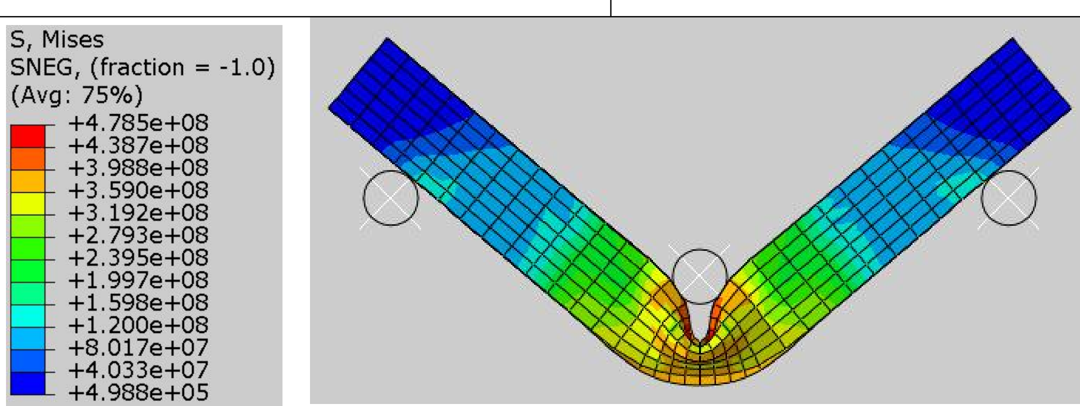




Exercise 29

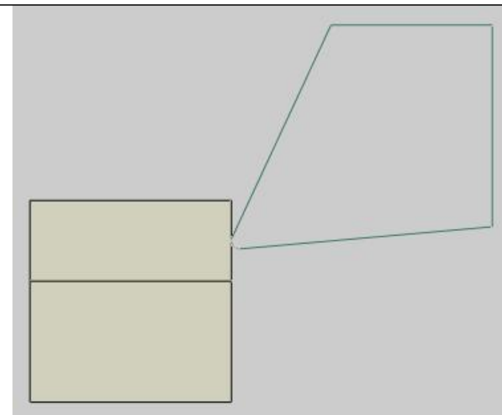
- Bending of an extrusion under quasi-static loading conditions
- Handling of T-intersections in shells in contact pair interactions
- Penalty vs Kinematic contact enforcement method

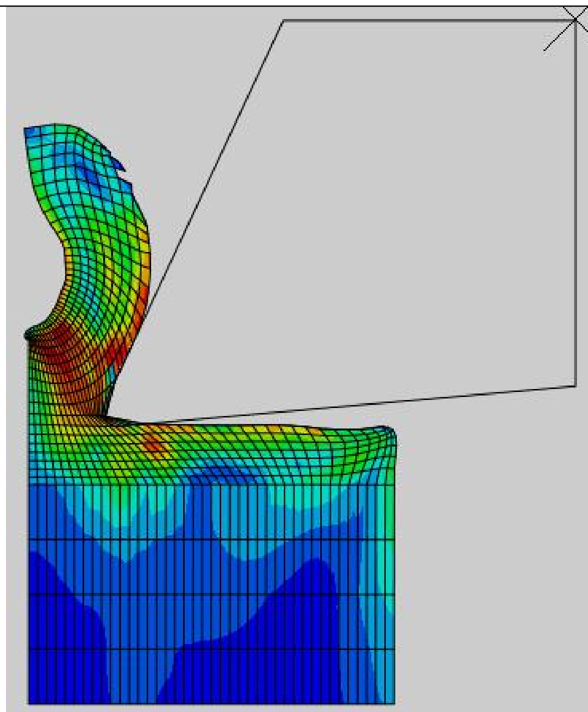




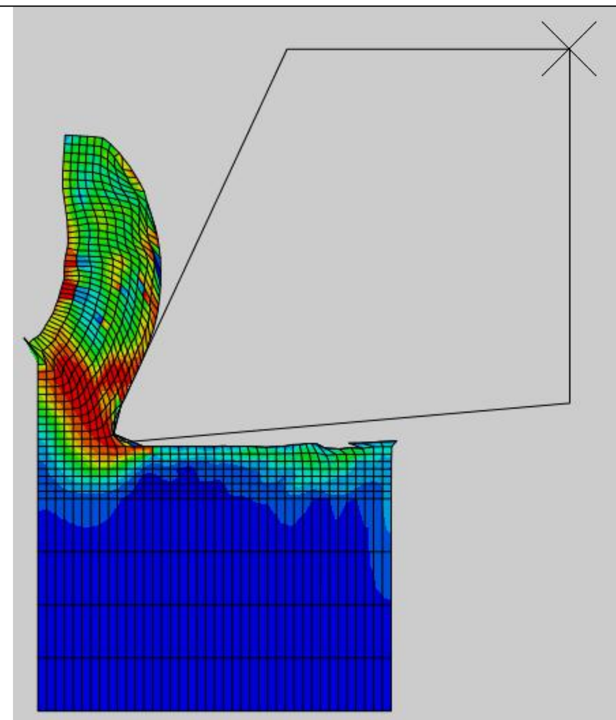
Exercise 30

- Simulating the metal cutting using the contact pair approach
- Element deletion functionality to model erosion due to material failure
- Using a node-based surface as slave to model surface erosion
- STATUS output variable to exclude failed elements
- ALE adaptive meshing

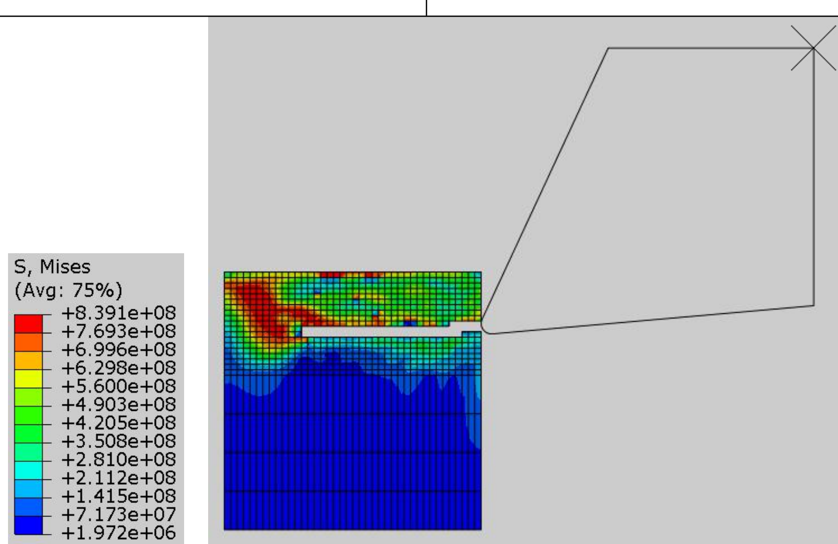




Results obtained with ALE adaptive meshing



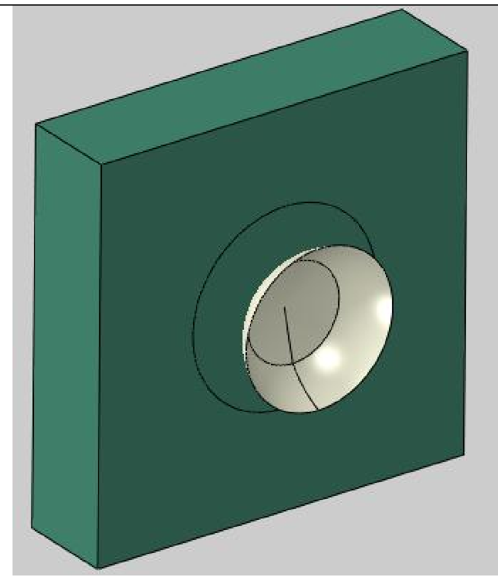
Results obtained without ALE adaptive meshing

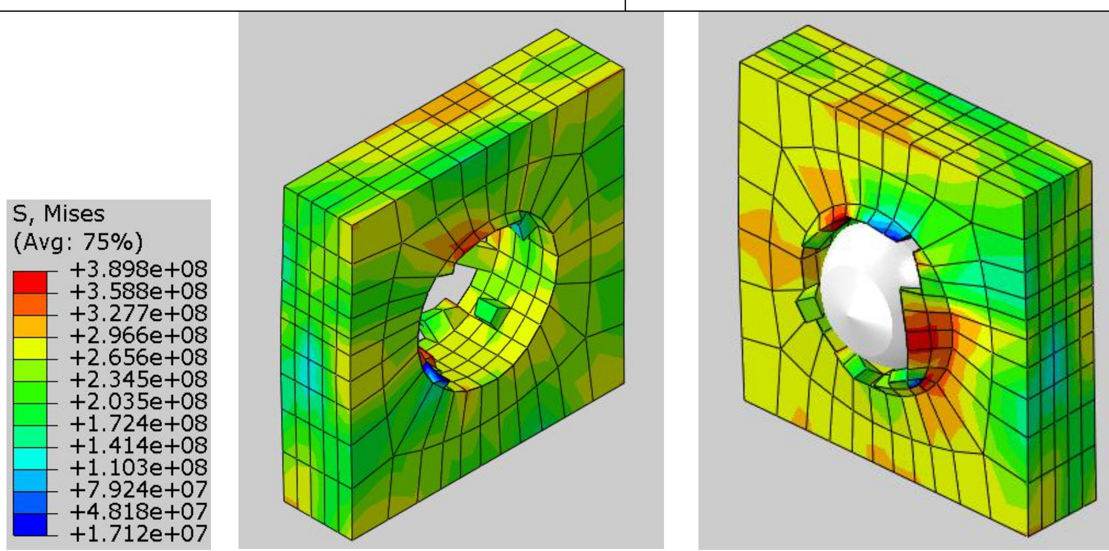


Distribution of von Mises stresses in the undeformed shape at the end of the simulation.

Exercise 31

- Simulating the erosion of a plate due to impacting projectile
- Creating a surface containing interior faces of an element set
- Surface erosion using element-based surface
- Editing the input file



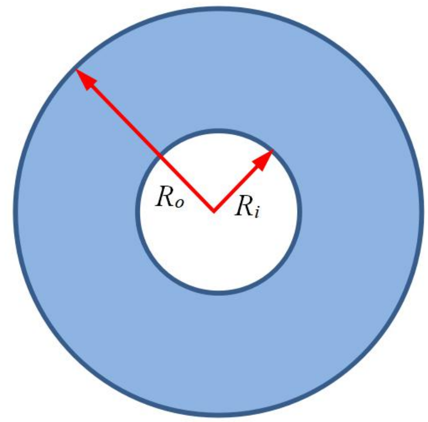


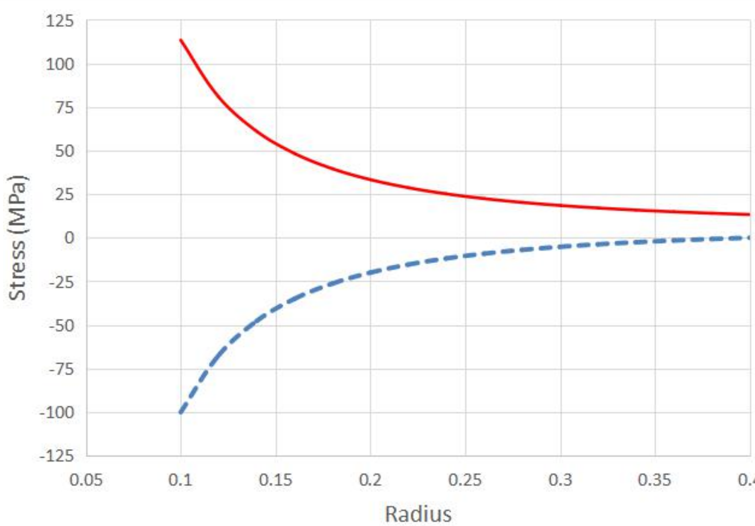
Section 6: Finite Element Discretization

In this section exercises related to the topic of finite element meshing e.g. accuracy of predictions made by a finite element analysis, adaptive remeshing and ALE adaptive meshing are presented. Details of topics covered in exercises are given below.

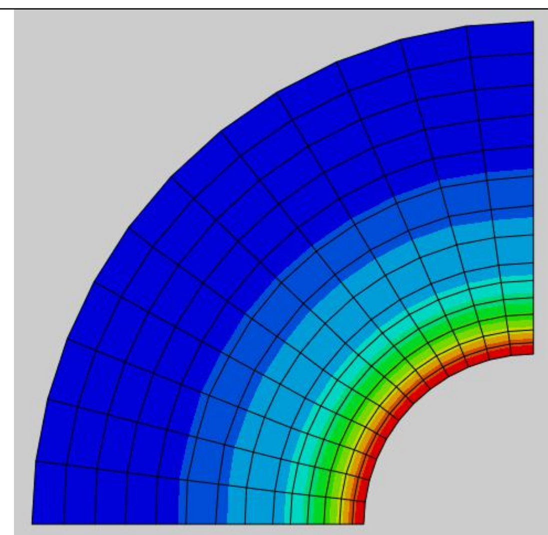
Exercise 1

- Evaluating accuracy of predictions made by a finite element analysis
- Requesting error indicator variables as field output
- Refining mesh in the region of highest error
- Comparing numerical and analytical results





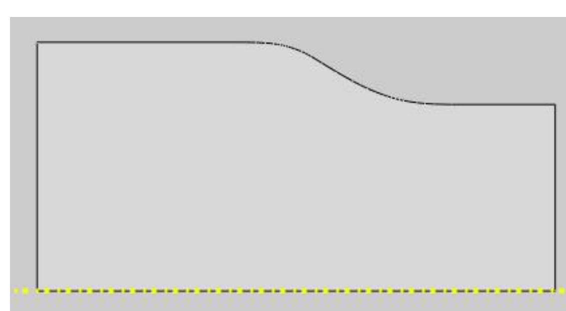
Radial and circumferential stresses in the thick cylinder

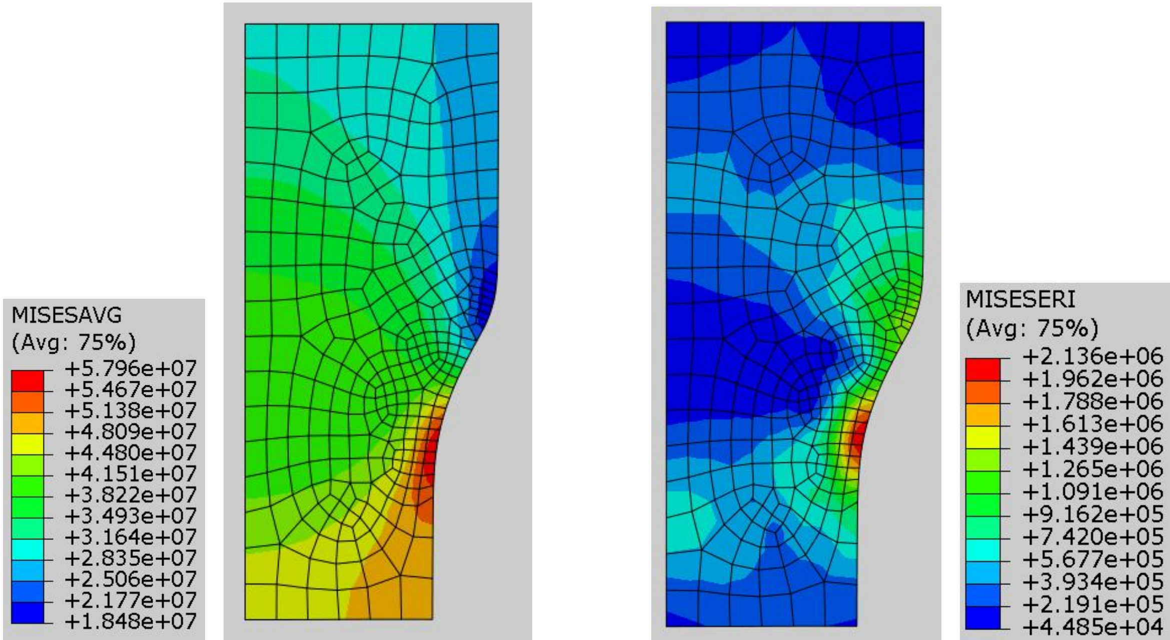


The contour plot of error indicator variable

Exercise 2

- Evaluating accuracy of predictions made by a finite element analysis
- Computing normalized error indicator
- Refining mesh to reduce normalized error indicator

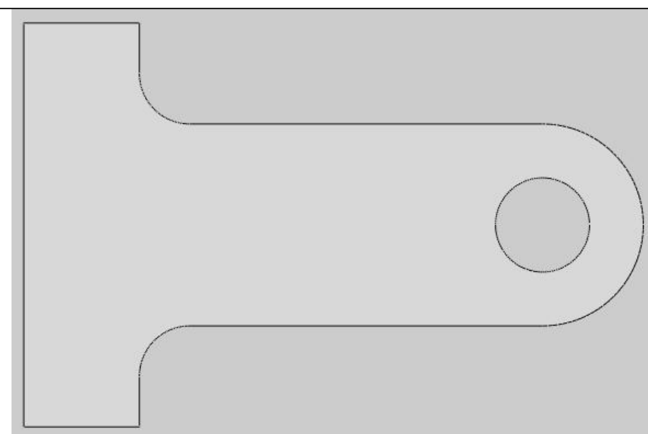


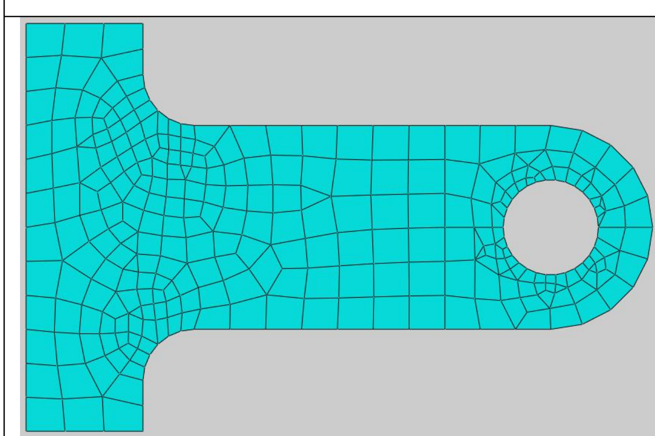


The contour plot of error indicator and base solution variables

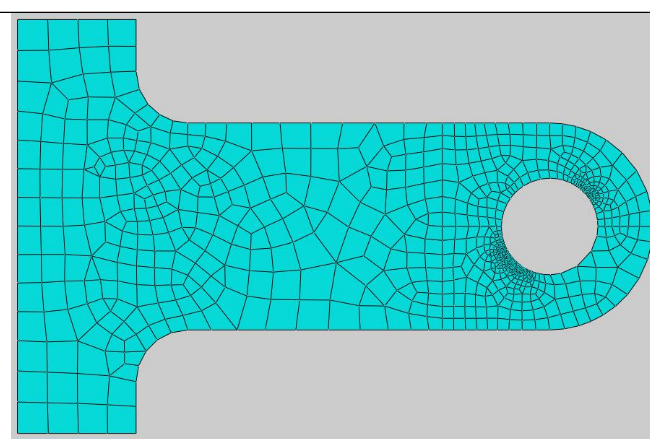
Exercise 3

- Performing adaptive remeshing of a bracket.
- Creating a remeshing rule
- Creating adaptivity process





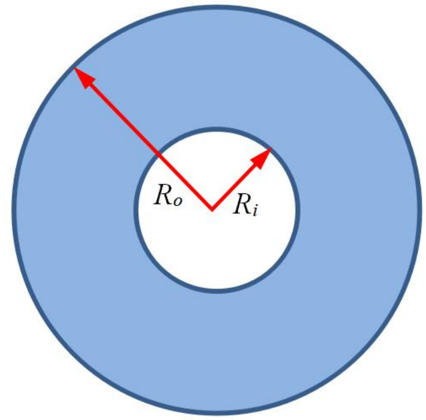
Initial mesh

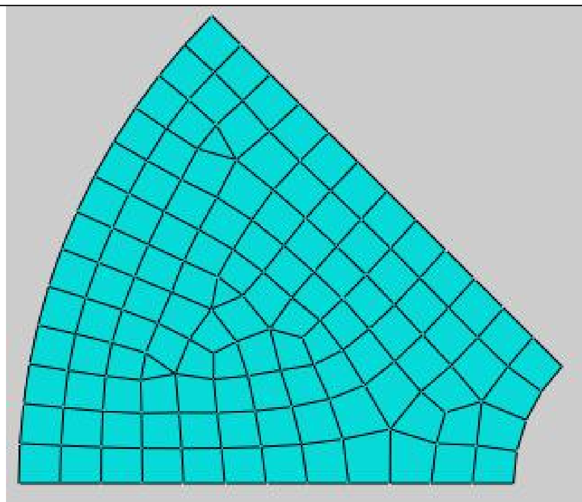


Mesh generated by adaptive remeshing process

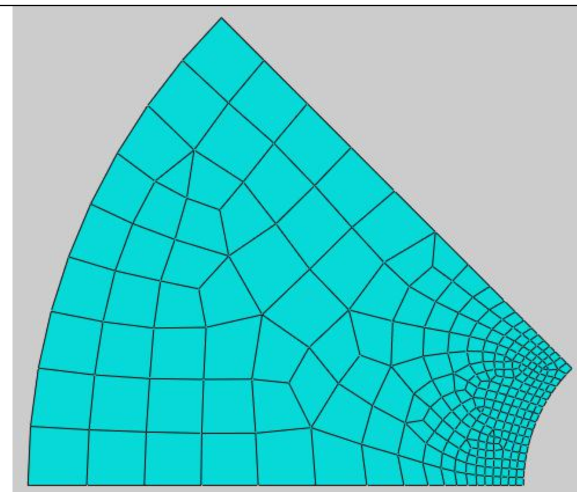
Exercise 4

- Performing adaptive remeshing of a thick cylinder.
- Creating a remeshing rule
- Creating adaptivity process

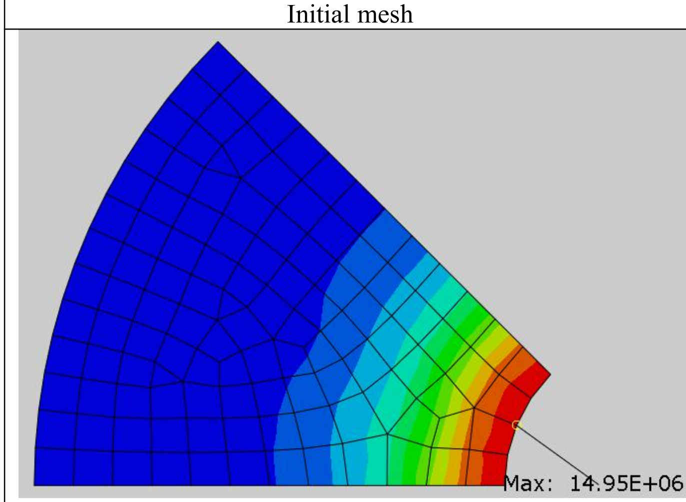




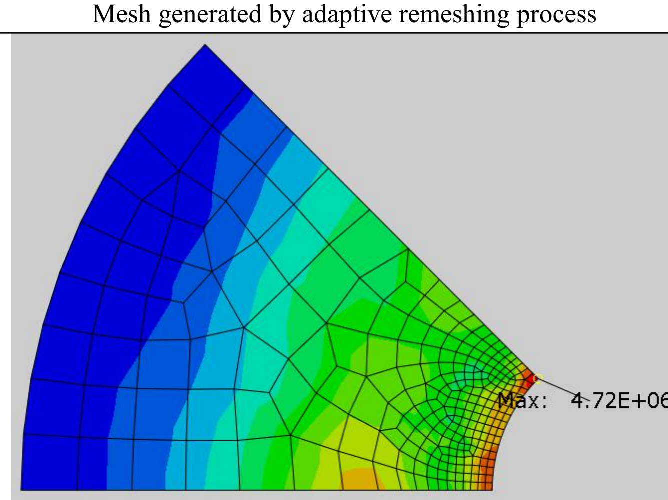
Initial mesh



Mesh generated by adaptive remeshing process



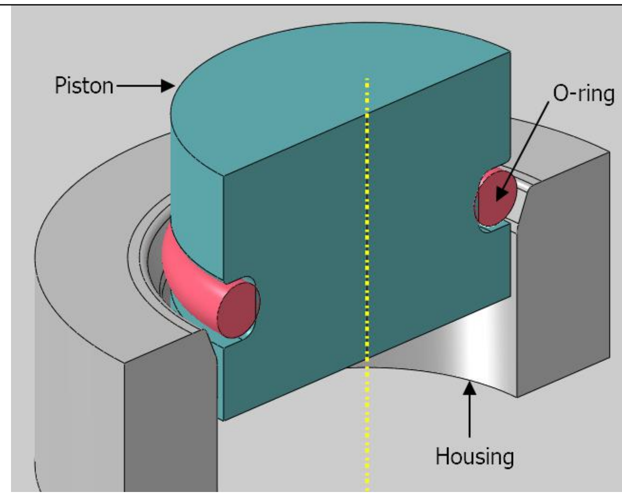
Contour plot of error indicator in initial mesh

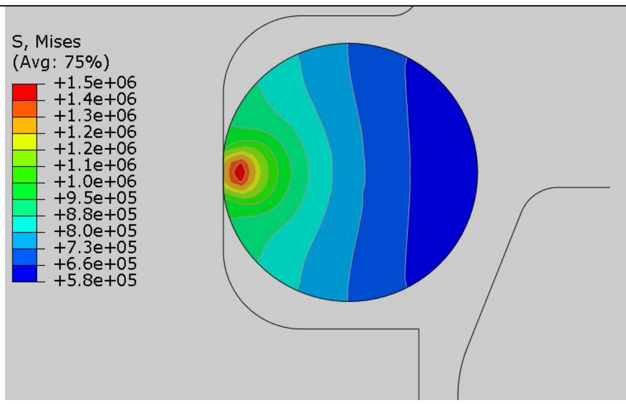


Contour plot of error indicator in the mesh generated with adaptive meshing

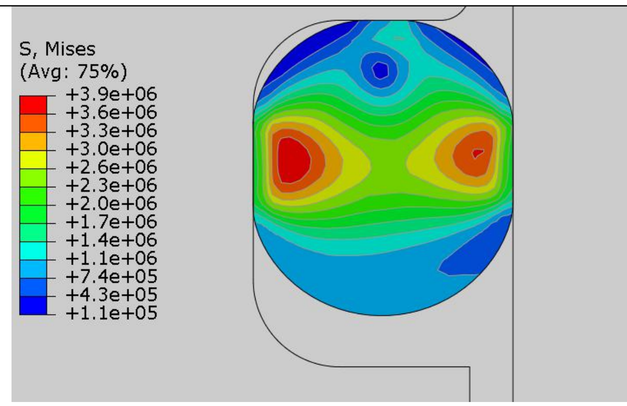
Exercise 5

- Performing adaptive remeshing of a rubber o-ring seal undergoing large strain
- Performing a provisional analysis for mesh refinement

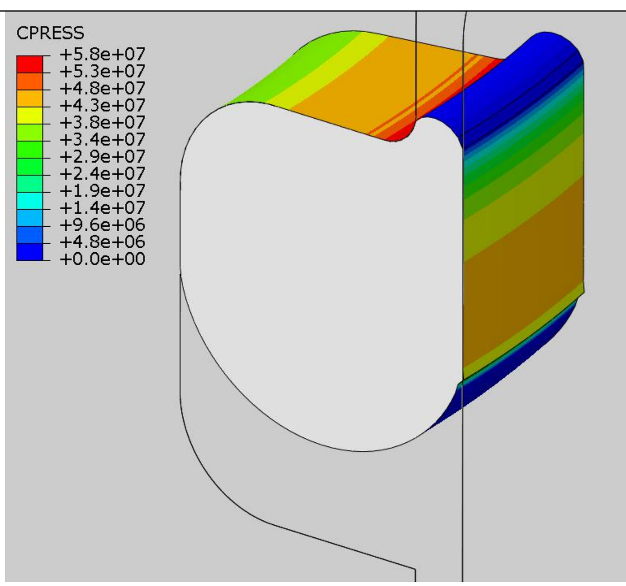




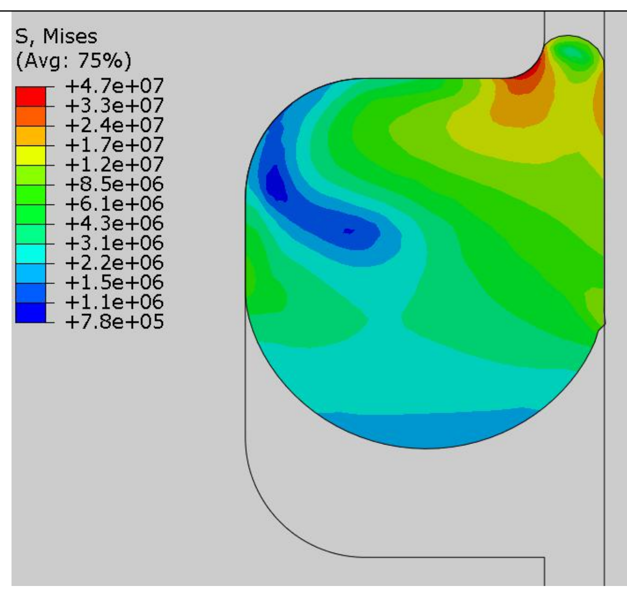
After resolution of interference



As the housing moves upward, it also pulls the seal upward



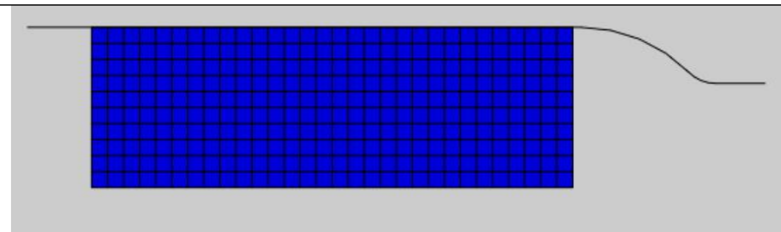
As the fluid pressure acts on the seal, the contact pressure increases significantly.



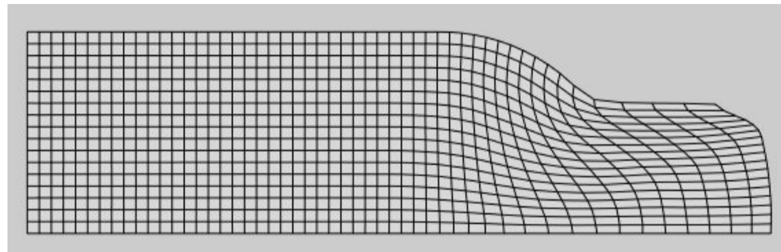
As the fluid pressure acts on the seal, it is pushed into the gap between piston and housing.

Exercise 6

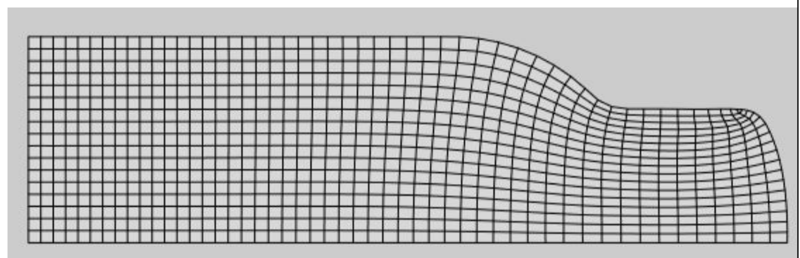
- Simulating the extrusion of a cylindrical metal bar
- Convergence issues due to excessive distortion
- Using ALE adaptive meshing to control mesh distortion



Part before extrusion



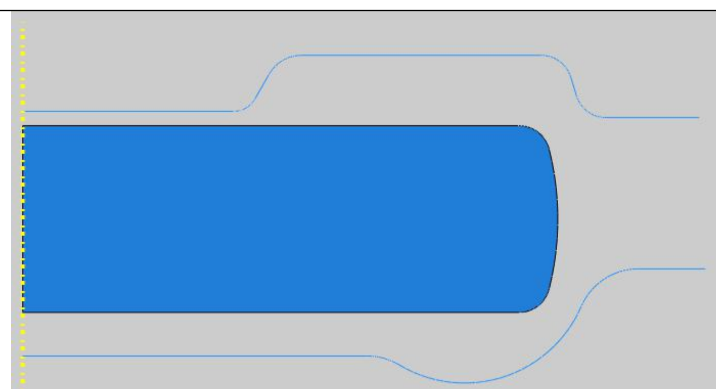
Without adaptive meshing

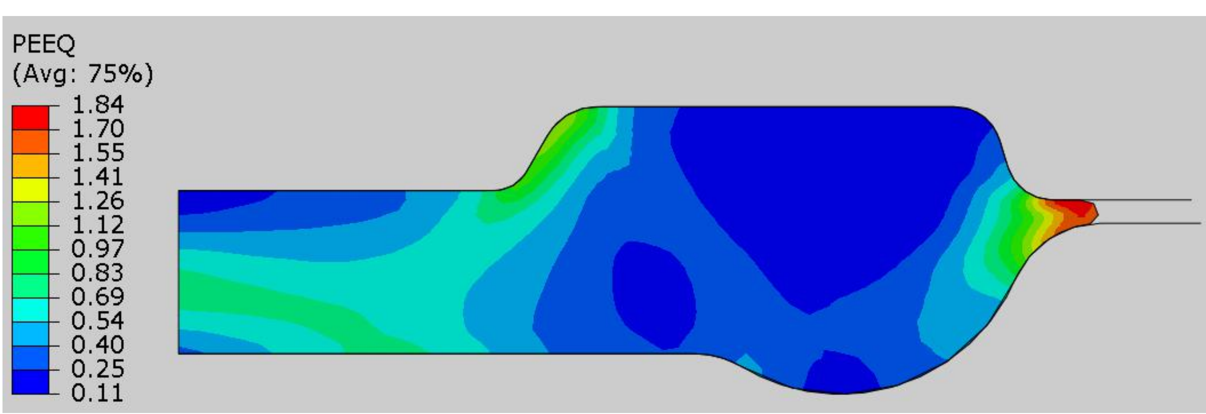


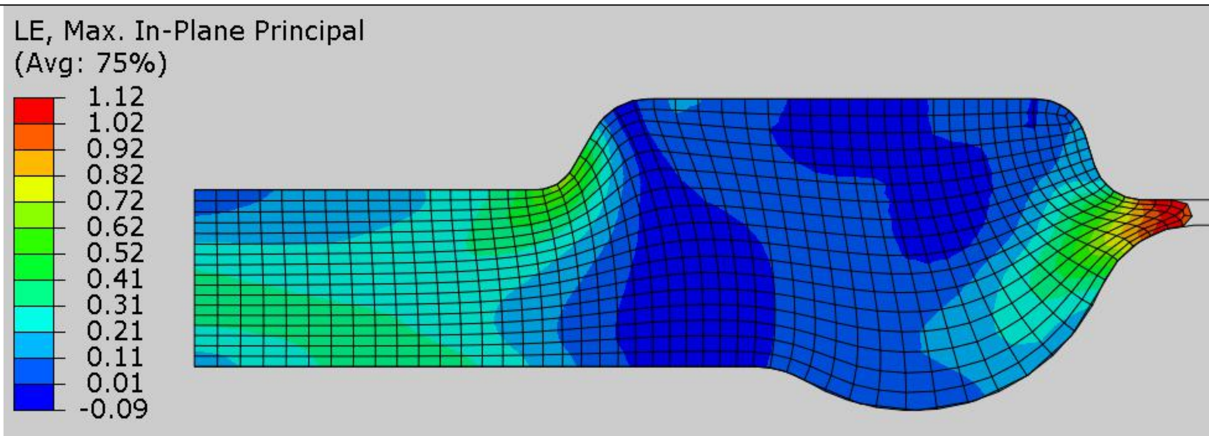
With adaptive meshing

Exercise 7

- Simulating the metal forging using the ALE adaptive meshing approach
- Comparing the kinetic energy history to the internal energy history

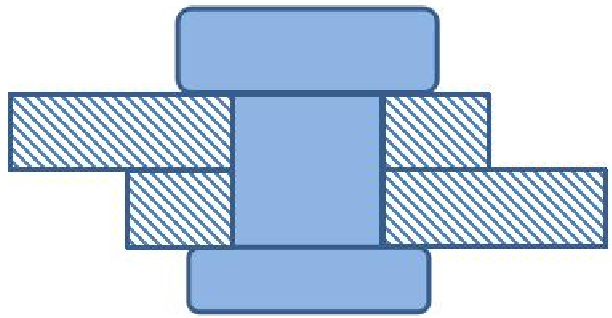


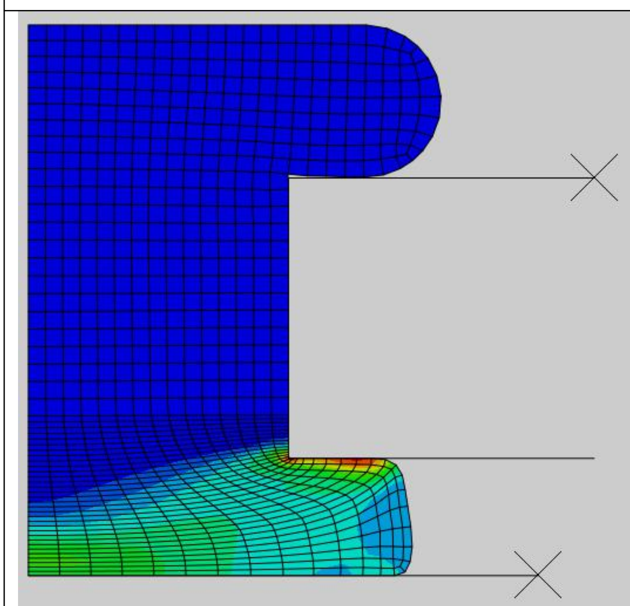




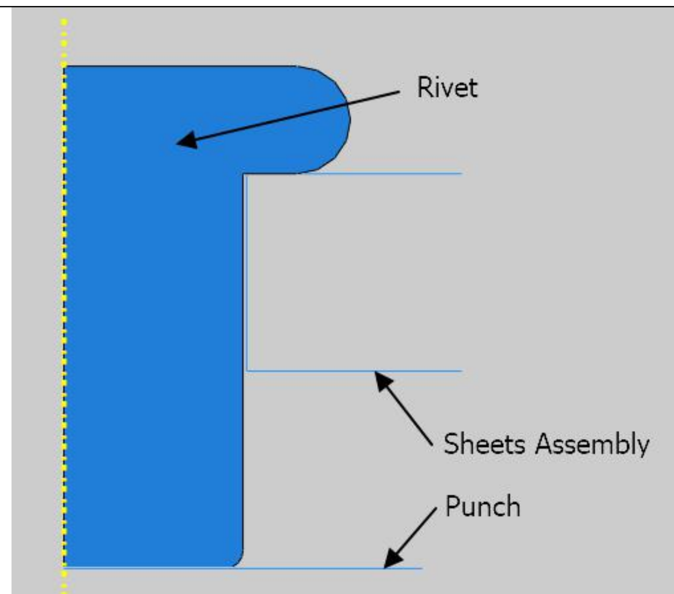
Exercise 8

- Simulating the rivet forming process
- Changing the intensity of the ALE adaptive meshing
- Changing the frequency of ALE adaptive meshing



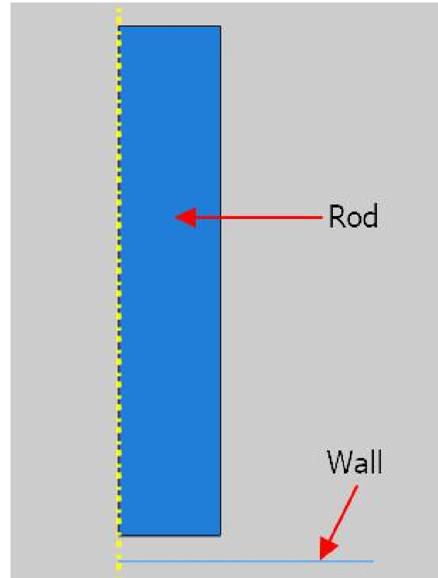


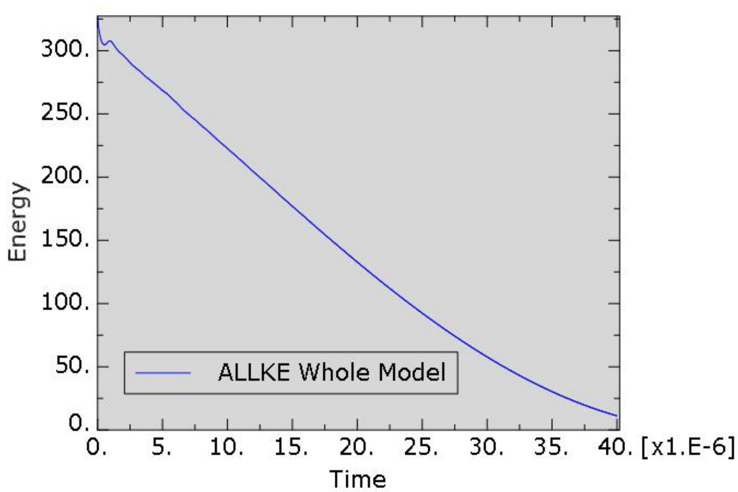
Distribution of plastic strain

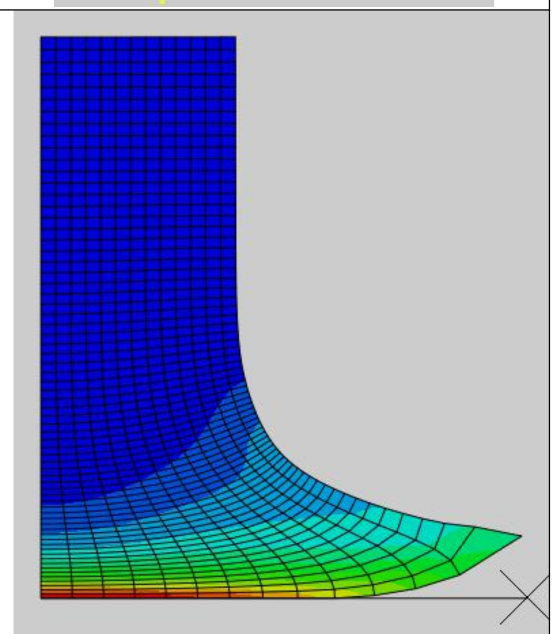


Exercise 9

- Simulating the high velocity impact of a metallic rod onto a rigid wall using the ALE adaptive meshing approach
- Choosing right frequency and intensity for impact problems



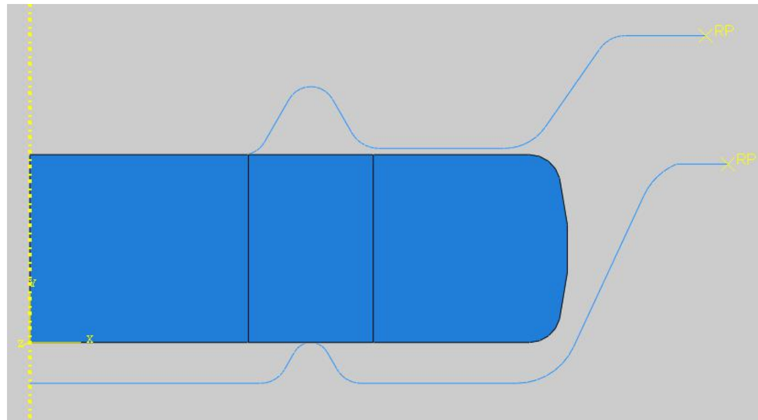


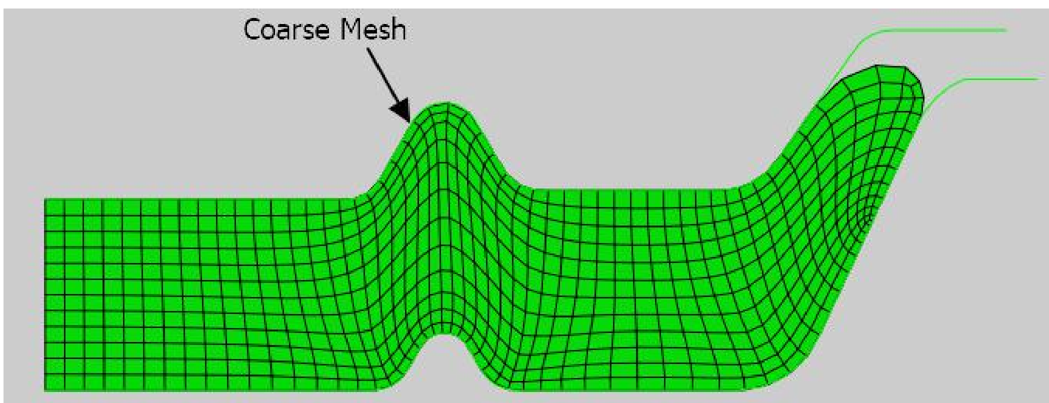


Distribution of plastic strain

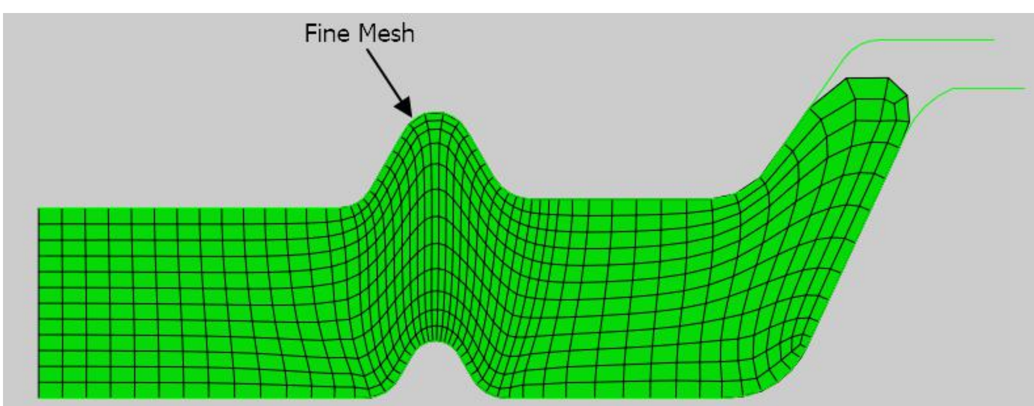
Exercise 10

- Simulating the metal forging using explicit solver
- How to preserve mesh grading during ALE adaptive meshing
- Specifying adaptive mesh control





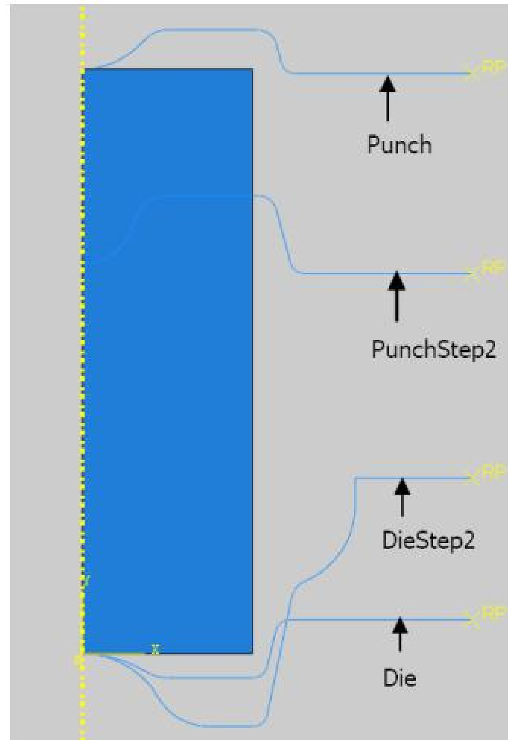
Without preservation of mesh grading

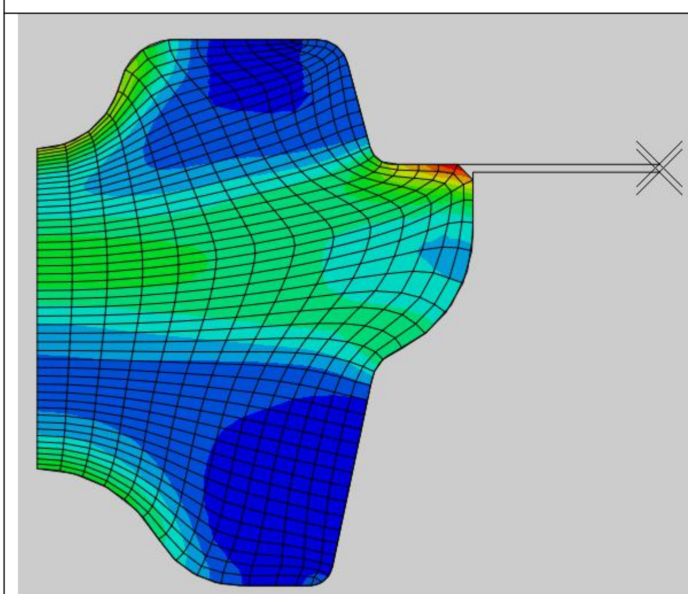


Preservation of mesh grading

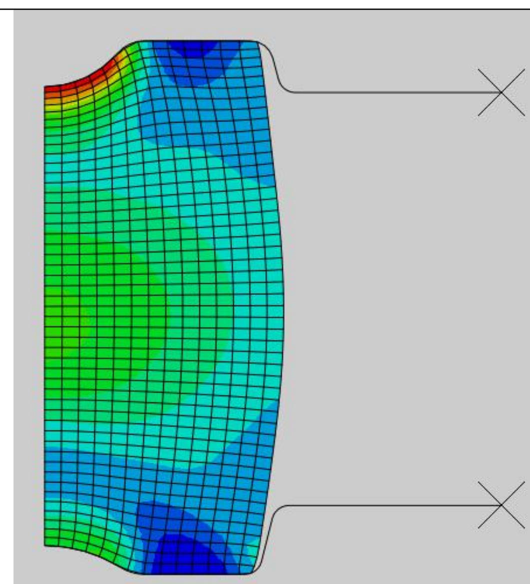
Exercise 11

- Simulating a two-step metal forging process
- Moving a workpiece through a series of cavities
- Using ALE adaptive meshing to maintain a high quality mesh





Distribution of equivalent plastic strain at the end of second step



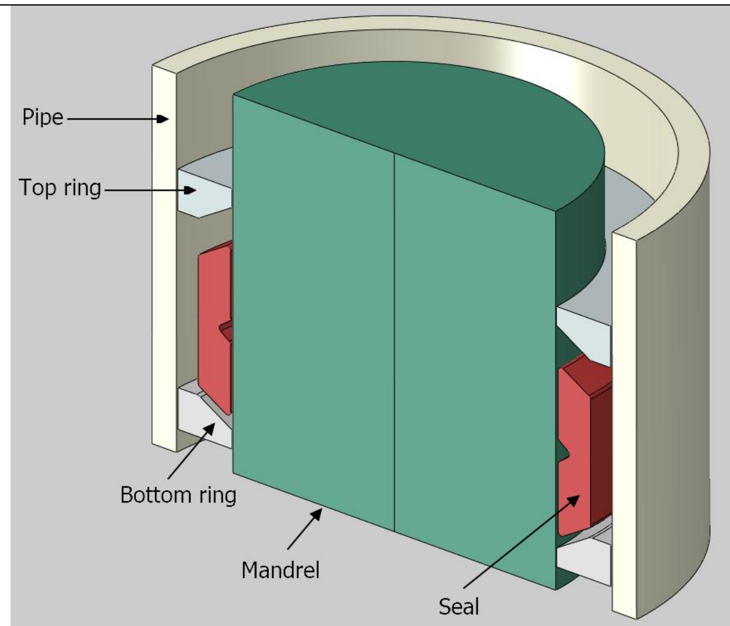
Distribution of equivalent plastic strain at the end of first step

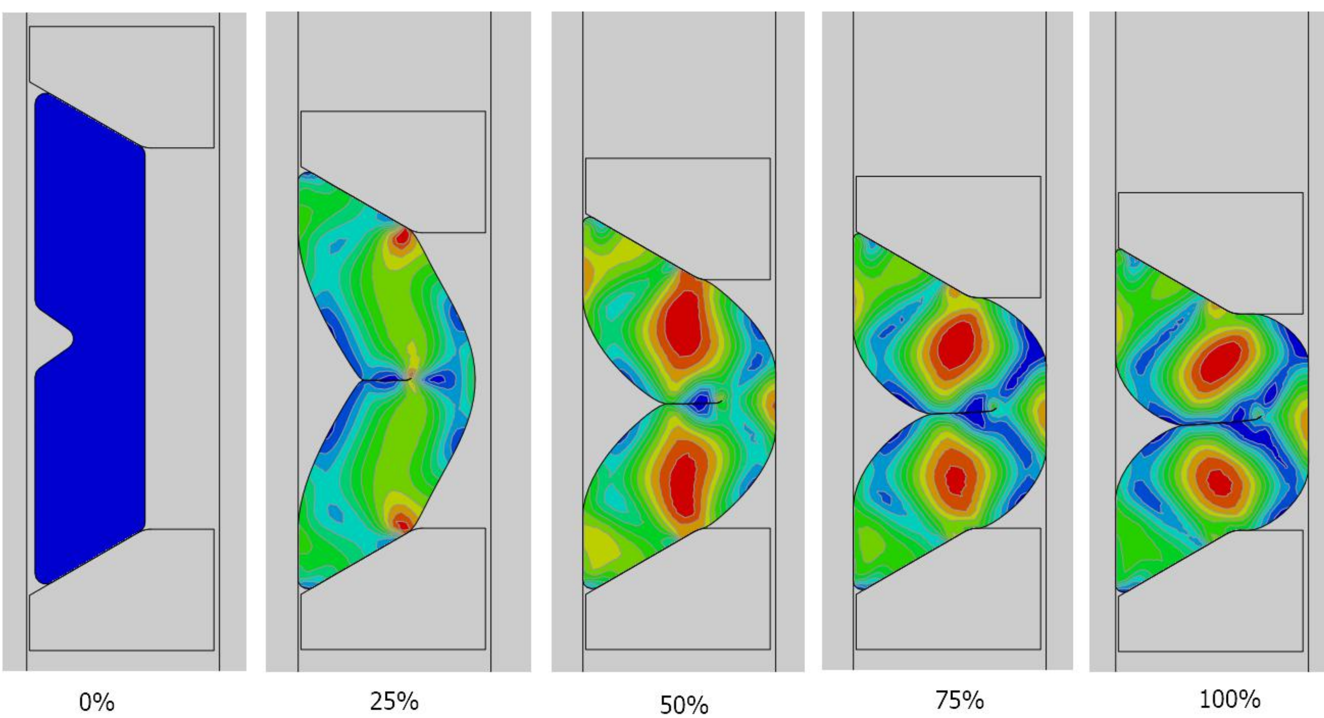
Section 7: Miscellaneous

In the previous sections, problems involving mostly single nonlinearities were presented. In engineering practice multiple nonlinearities are common. In this section problems involving multiple nonlinearities will be presented. Furthermore advanced techniques e.g. mesh adaptivity will also be discussed. Details of topics covered in exercises are given below.

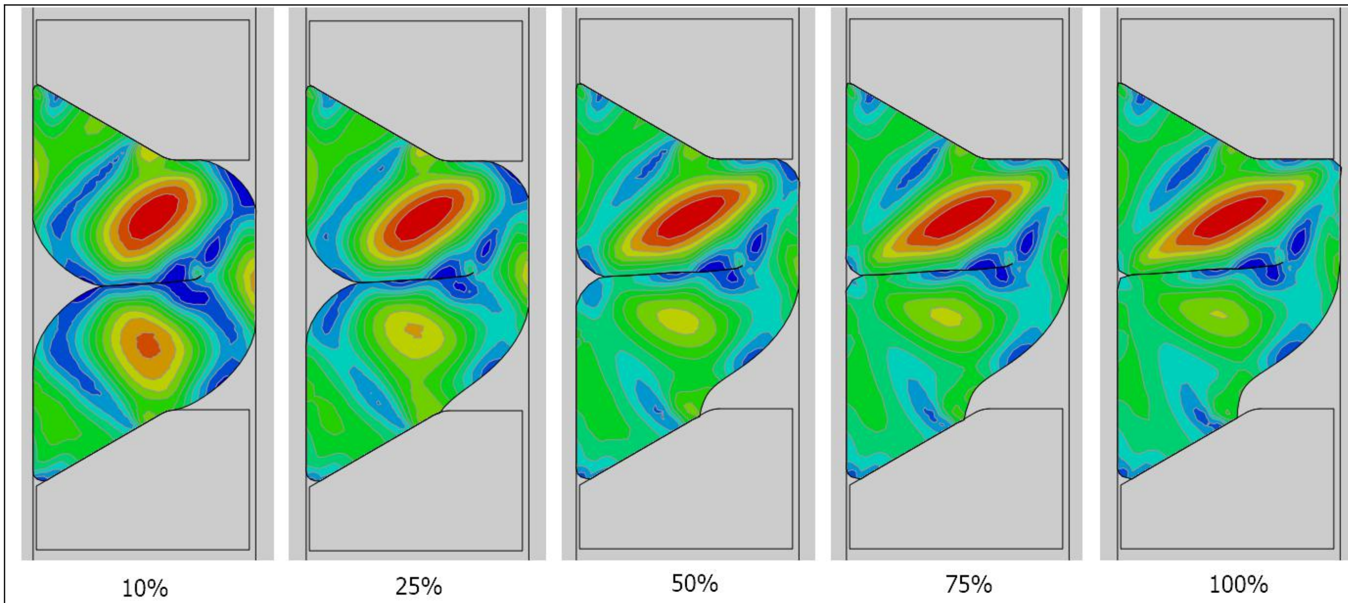
Exercise 1

- Analysis of a downhole packer seal subjected to the pressure of a fluid
- Pressure penetration interaction
- Self contact





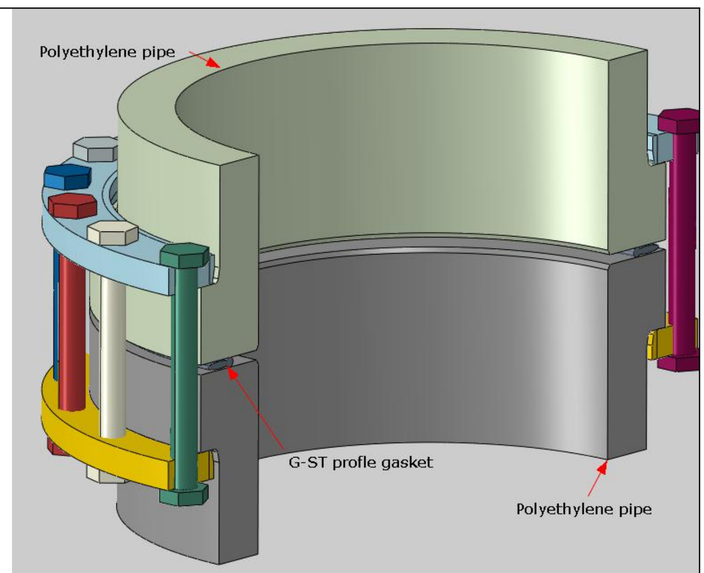
The contour plot of strain at different stages during first step



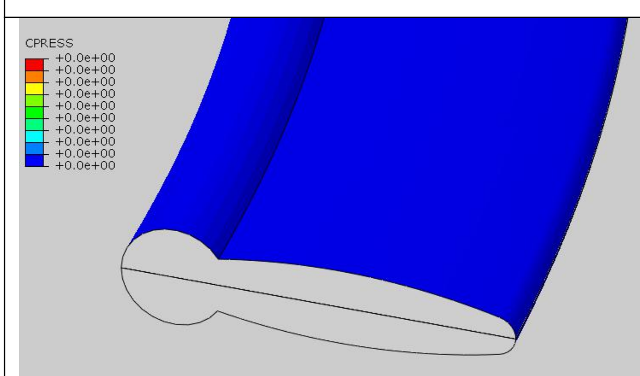
The contour plot of strain at different stages during second step

Exercise 2

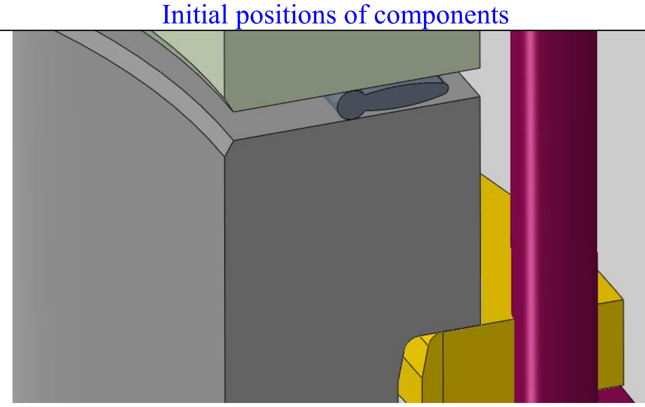
- Simulation of a flange joint for polyethylene pipes.
- Modeling time-dependent behaviors of pipe and gasket materials.
- Investigating the tightness of the flange joint over 100 years by taking into account the creep of pipe and gasket.
- Understanding "Unconnected regions in the model" warning message



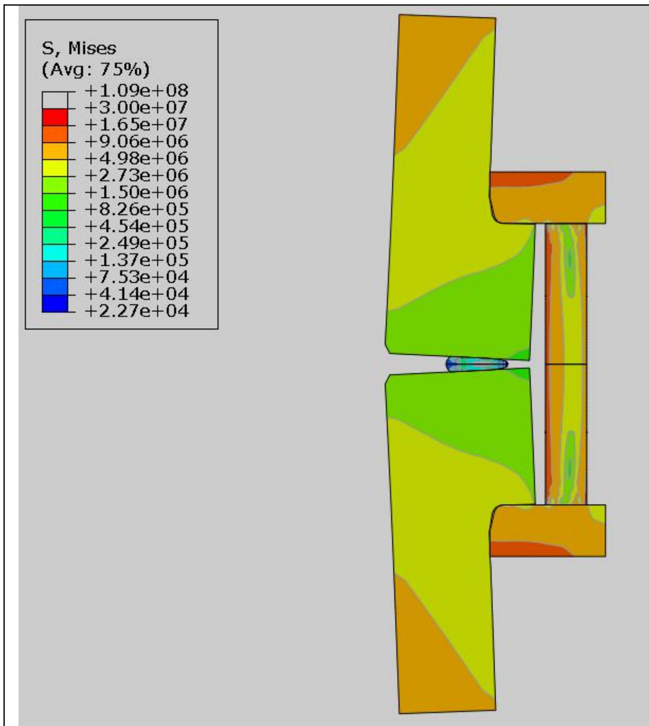
Initial positions of components



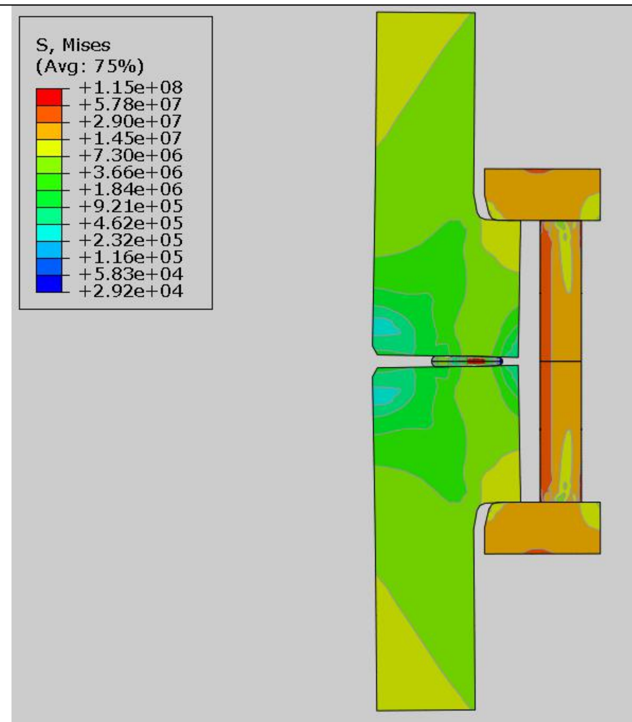
Contact pressure distribution on the surface of G-ST profile gasket before bolt tightening.



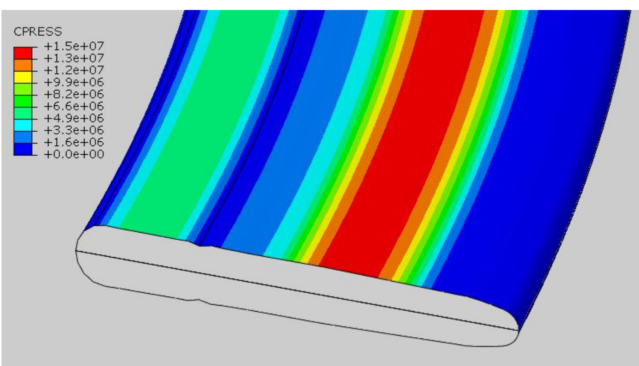
G-ST profile gasket in assembly



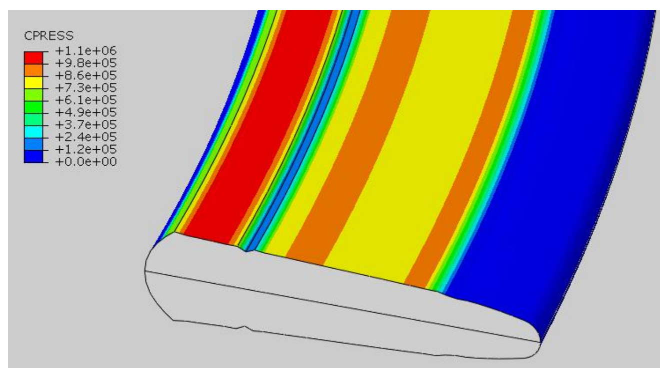
Stress distribution after the creep over 100 years



Stress distribution after the bolt tightening and fluid pressure application.



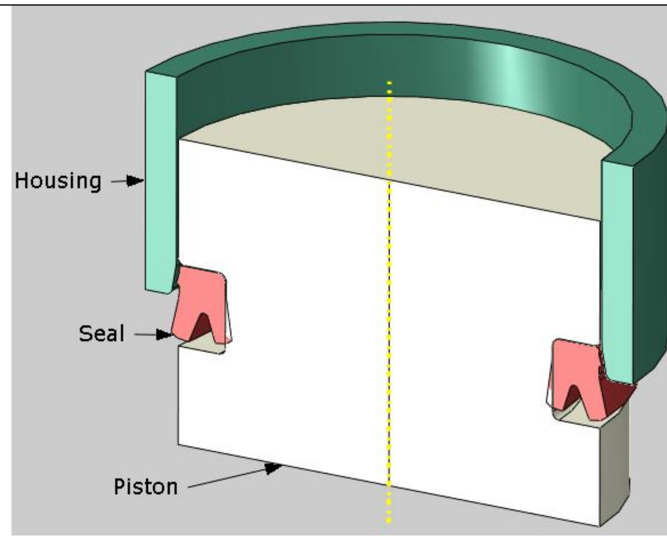
Contact pressure distribution on the surface of gasket after bolt tightening and fluid pressure application.

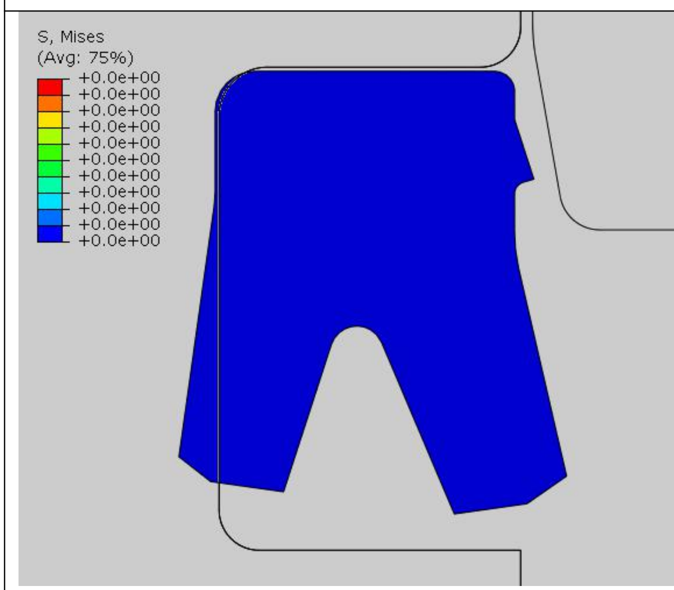


Contact pressure distribution on the surface of gasket after creep over 100 years.

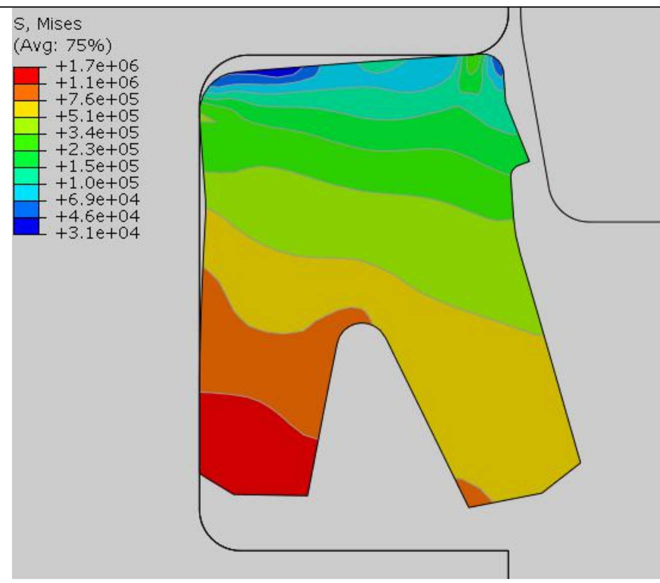
Exercise 3

- Analysis of an u-cup seal under the pressure of a fluid
- Pressure penetration interaction
- Self contact
- Understanding "Numerical singularities" warning message

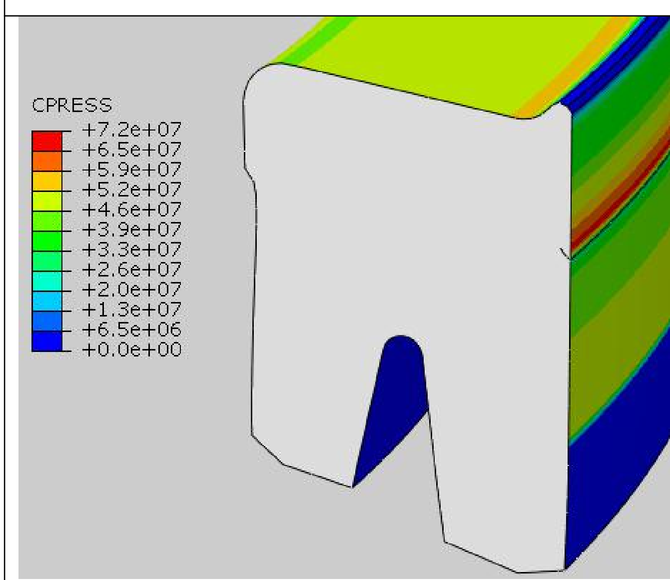




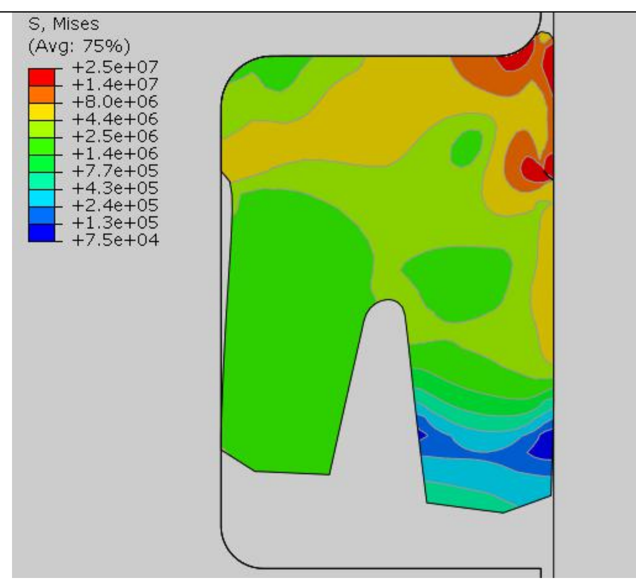
Before interference resolution



As interference is resolved, stresses develop in the seal.



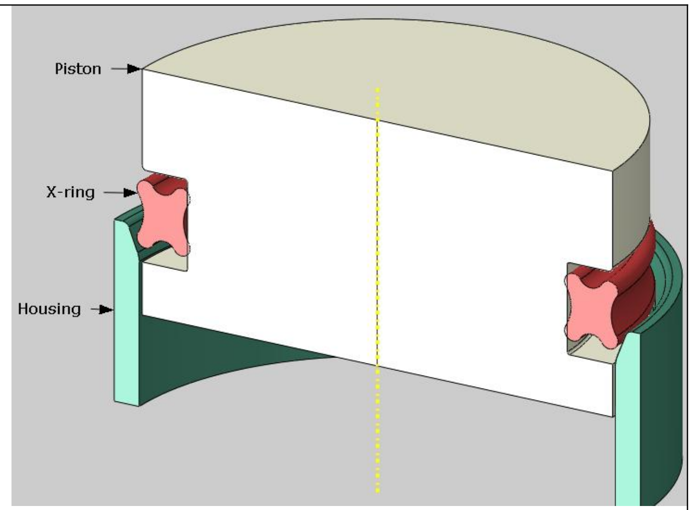
As the fluid pressure acts on the seal, the contact pressure increases significantly.

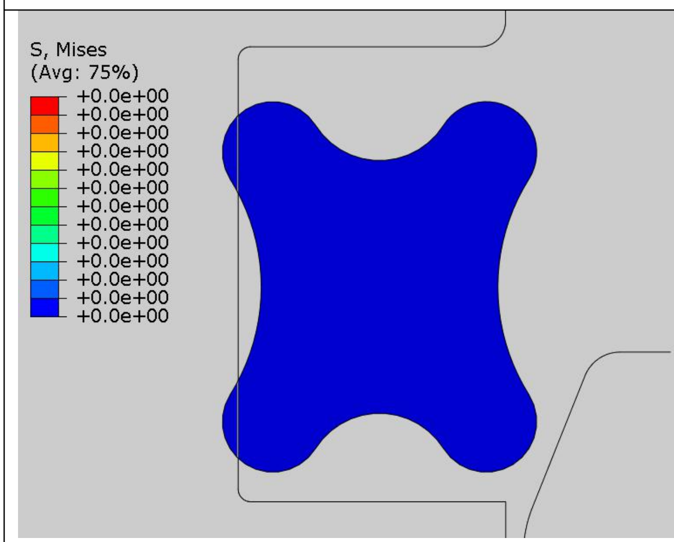


As the fluid pressure increases on the seal, it is pushed upwards.

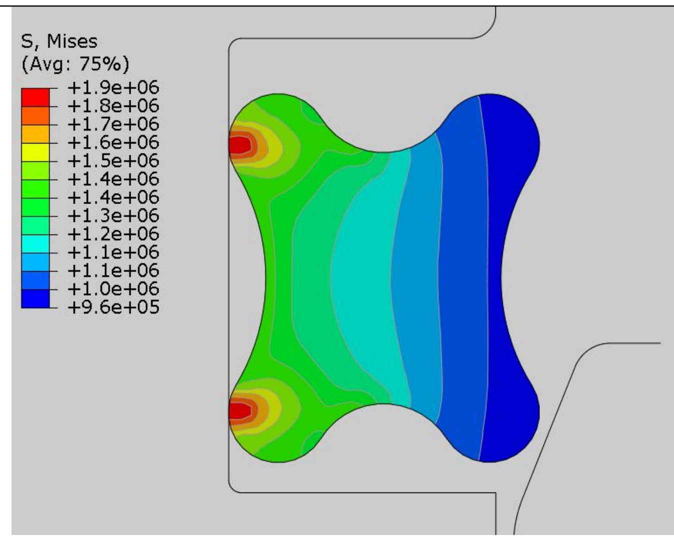
Exercise 4

- Analysis of an x-ring seal under the pressure of a fluid
- Modeling time-dependent behaviors of hyperelastic materials.
- Investigating the influence of creep on seal performance
- Improving the convergence rate by using unsymmetric solver

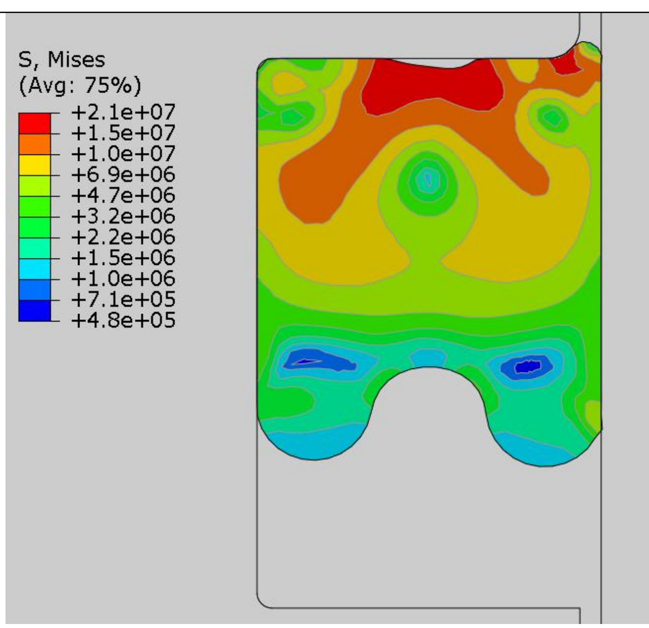




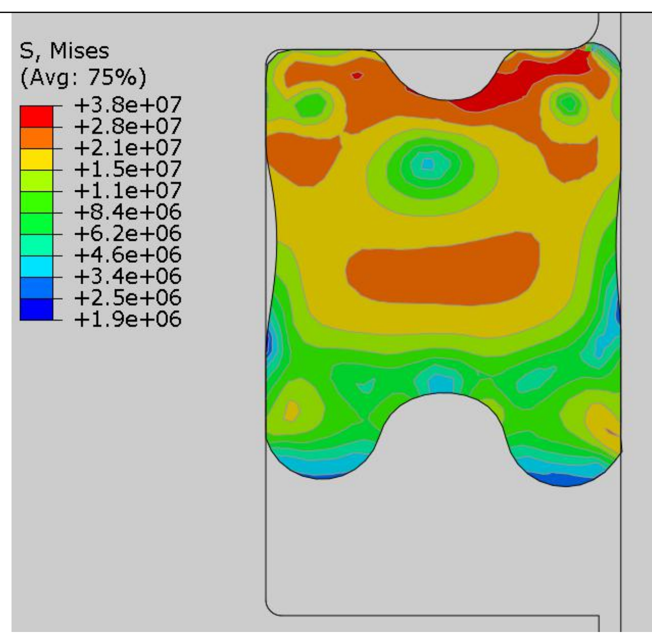
Before interference resolution



As interference is resolved, stresses develop in the seal.



As fluid pressure is held constant for 1000 minutes to study the creep, extrusion occurs.



As the fluid pressure increases on the seal, it is pushed upwards.

Section 8: User Exercises

This section contains models for the user practice. You can contact the instructor in case any help is required to solve these problems.