

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 nonlinear 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 two parts:

- [Part 1: Basics of Nonlinear Analysis](#)
- [Part 2: Solving Contact Problems](#)

Part 1: Basics of Nonlinear Analysis

1. Introduction

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:

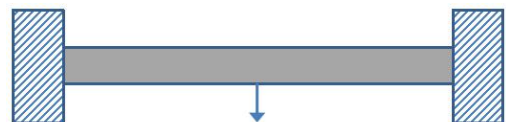
- Linear vs Nonlinear response
- Sources of nonlinearities
 1. Geometric nonlinearity
 - I. Large strain
 - II. Small strains but large displacements and/or rotations
 - III. Snap-Through
 - IV. Buckling
 2. Material nonlinearity
 - I. Nonlinear elasticity
 - II. Plasticity
 3. Force nonlinearity
 4. Boundary nonlinearity
- Solution algorithms
 1. Newton's method
 2. Arc-Length method

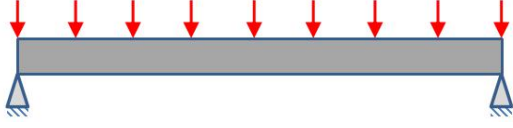
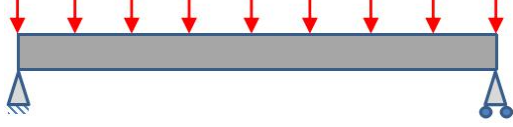
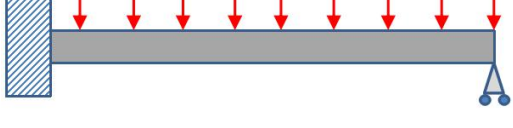
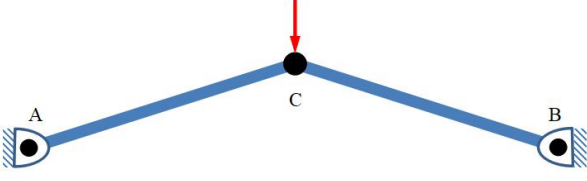
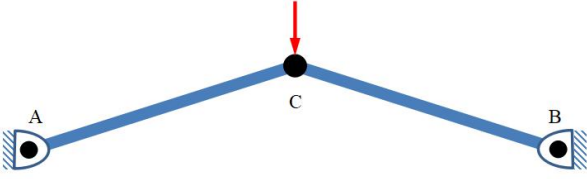
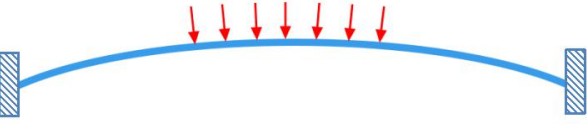
2. Exercises

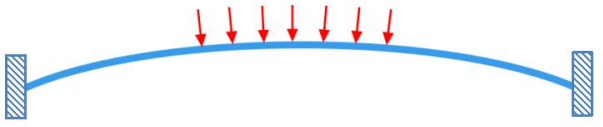
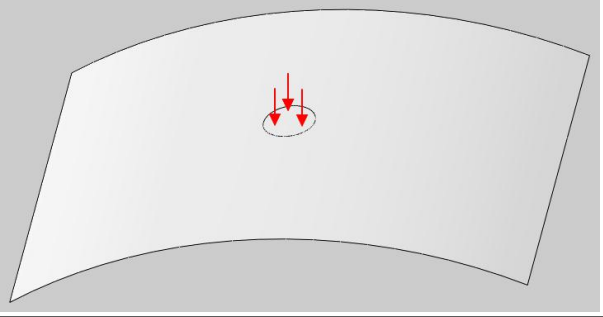
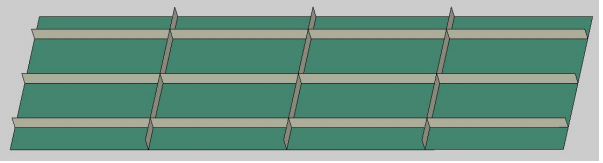

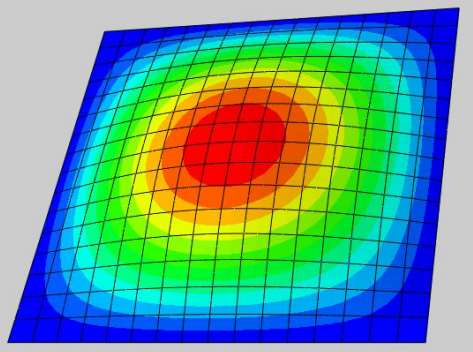
This course adopts Problem-Based Learning approach. It consists of tutorials providing intensive instructions to perform analysis of nonlinear problems. Details of topics covered in exercises is given below.

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

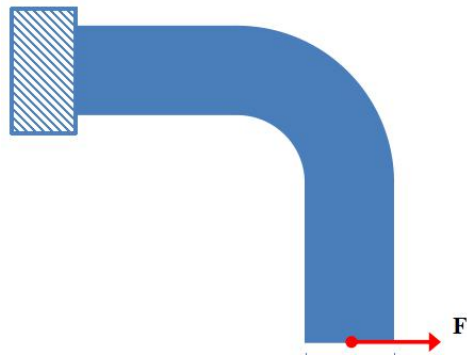


<p>Exercise 2</p> <ul style="list-style-type: none"> Analyzing a beam pinned on both ends When does geometric nonlinearity has significant influence on solution even for small deformation Plotting membrane forces Plotting reaction forces in axial direction at supports 	
<p>Exercise 3</p> <ul style="list-style-type: none"> Analyzing a simply supported beam Influence of boundary conditions on the geometric nonlinearity Influence of boundary conditions on membrane forces Plotting membrane forces 	
<p>Exercise 4</p> <ul style="list-style-type: none"> 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 	
<p>Exercise 5</p> <ul style="list-style-type: none"> 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 	
<p>Exercise 6</p> <ul style="list-style-type: none"> Analyzing a two bar truss using force-control Obtaining solution using force-control method Comparing displacement-control and force-control results Introducing stabilization 	
<p>Exercise 7</p> <ul style="list-style-type: none"> 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 	

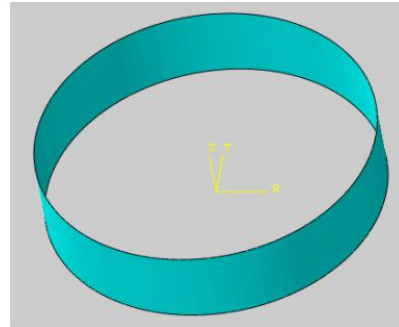
<p>Exercise 8</p> <ul style="list-style-type: none"> Analyzing a circular arch modeled using shell elements Simulating snap-through using Riks method Modeling arch using shell elements Specifying analysis stopping criteria How to choose magnitude of LPF for stopping criteria Requesting Pressure loads as output 	
<p>Exercise 9</p> <ul style="list-style-type: none"> Analyzing a cylindrical roof Simulating snap-through Comparison of results using <ul style="list-style-type: none"> ✓ Riks analysis ✓ General static Analysis 	
<p>Exercise 10</p> <ul style="list-style-type: none"> Analyze a structure undergoing localized buckling Defining MPC constraint Introducing stabilization 	
<p>Exercise 11</p> <ul style="list-style-type: none"> 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 	
<p>Exercise 12</p> <ul style="list-style-type: none"> 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 	

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

**Exercise 14**

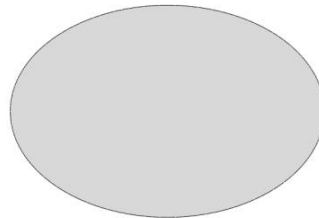
- Performing buckling analysis of a cylindrical shell subjected to compressive axial load
- Using Lanczos eigensolver
- Using pairs of repeated eigenmodes to introduce imperfection
- Using Lanczos eigensolver
- Investigating the post-buckling response using Riks procedure
- Seeding the imperfection using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

**Exercise 15**

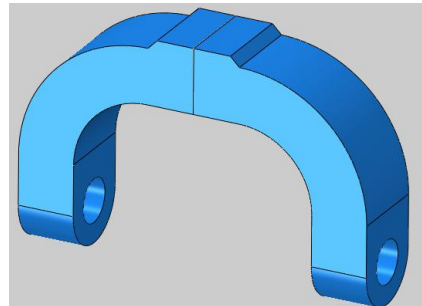
- Specifying follower and non-follower forces
- Comparing solution for follower and non-follower force

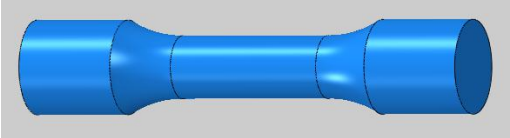
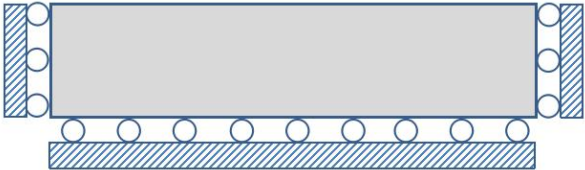
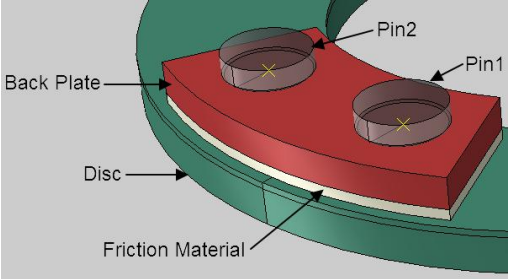
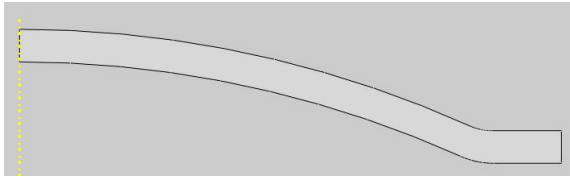
**Exercise 16**

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

**Exercise 17**

- 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



<p>Exercise 18</p> <ul style="list-style-type: none"> 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 	
<p>Exercise 19</p> <ul style="list-style-type: none"> Analyzing a bar subjected to a cyclic temperature load Applying a cyclic temperature load Investigating the residual stresses Temperature dependent plasticity model 	
<p>Exercise 20</p> <ul style="list-style-type: none"> 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 	
<p>Exercise 21</p> <ul style="list-style-type: none"> 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 	

Part 2: Solving Contact Problems

1. Introduction

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. In this section, the necessary concepts to perform a contact analysis are explained with detail. Furthermore the solution algorithm used for solving contact problems is described. Topics covered are as follow:

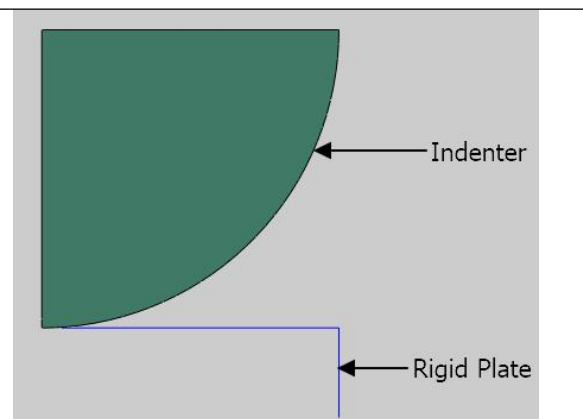
- Contact interaction
- Contact property models
 1. Hard Contact
 2. Soft Contact
 3. Friction models
- Contact constraint enforcement methods
 1. The direct method
 2. The penalty method
 3. The augmented Lagrange method
- Relative sliding of surfaces
- Slave and master surfaces
- Discretization of contact pair surfaces
- General contact and Contact pairs
- Solution algorithm for contact problems
- Diagnosing convergence difficulties and taking the corrective action

2. Exercises

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.

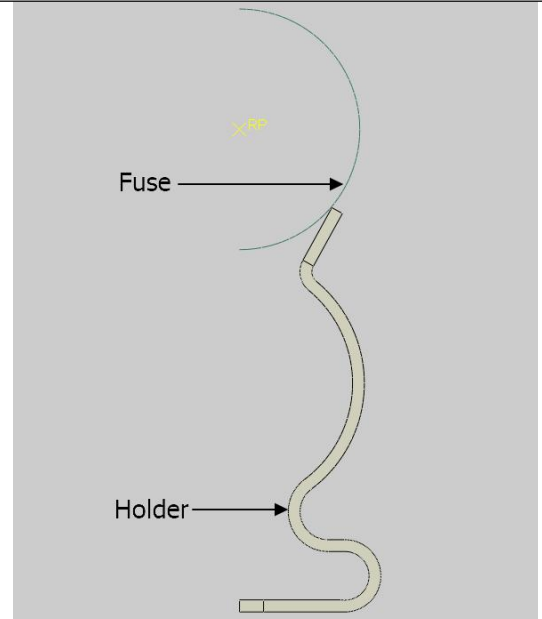
Exercise 1

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



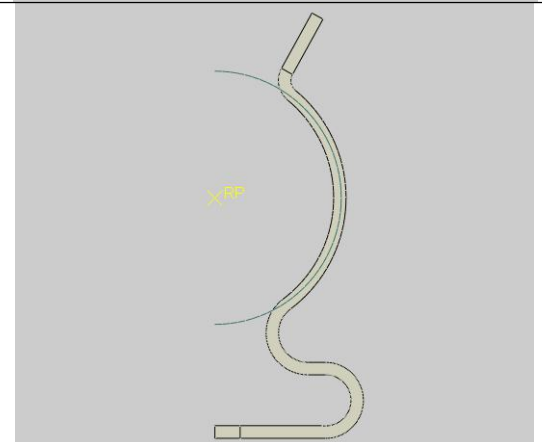
Exercise 2

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



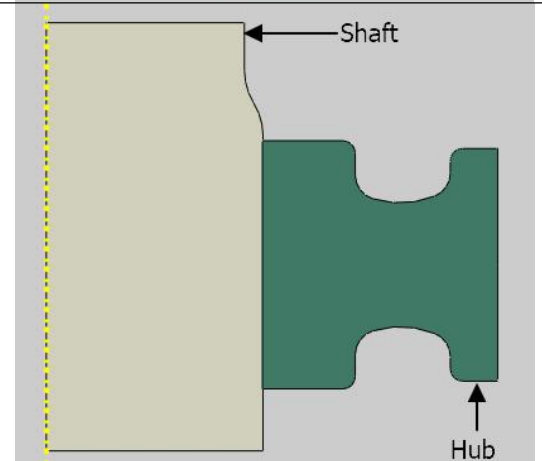
Exercise 3

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



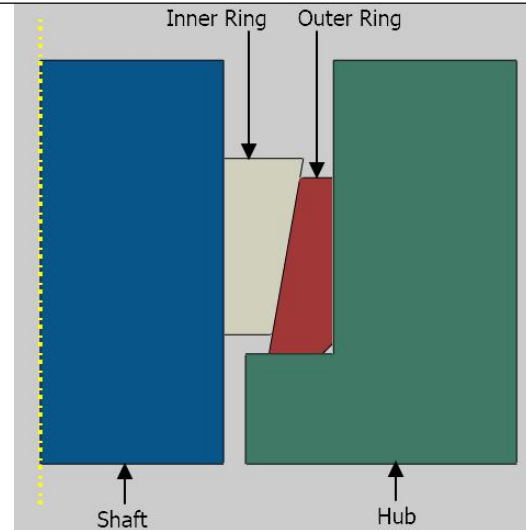
Exercise 4

- 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

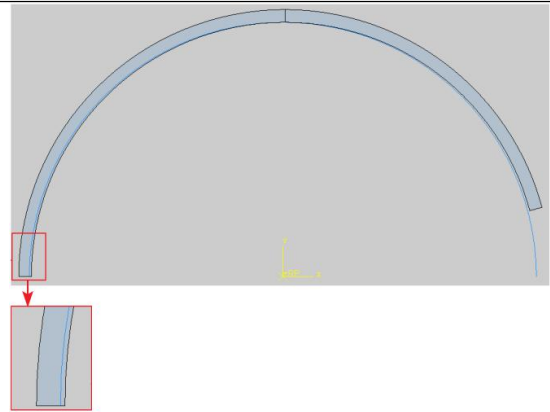


Exercise 5

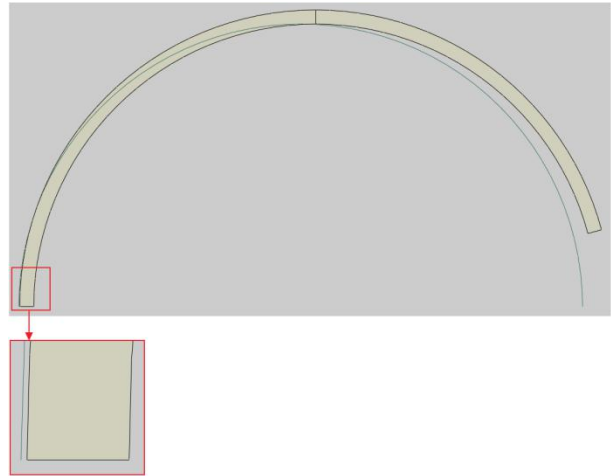
- Performing the stress analysis of shaft-hub assembly coupled by conical rings
- Choosing a master surface among two deformable surfaces.
- Diagnosing the error messages with Job Diagnostics dialog box
- Invoking unsymmetric solver

**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

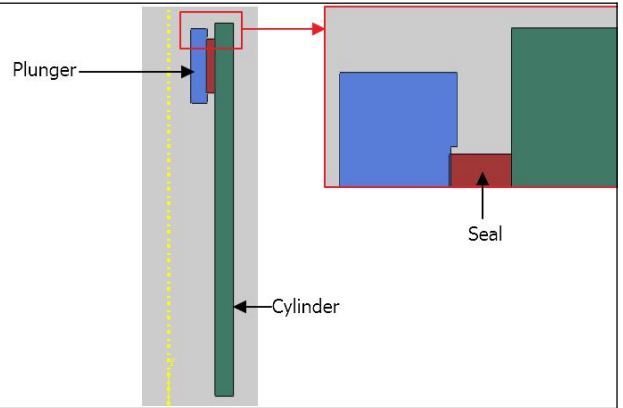
**Exercise 7**

- 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



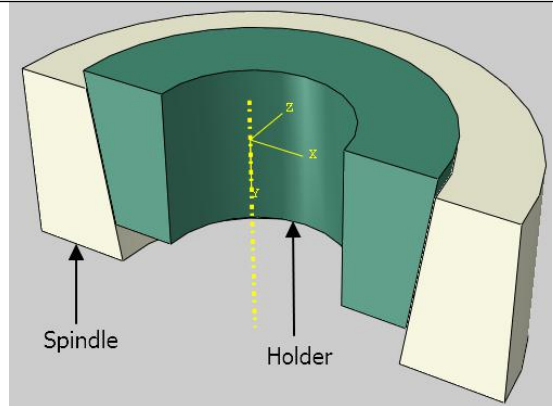
Exercise 8

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



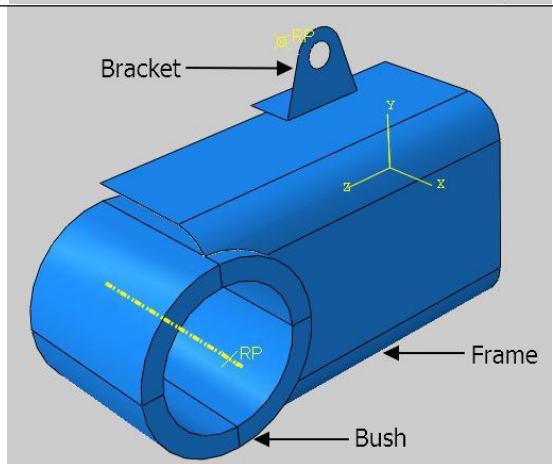
Exercise 9

- 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



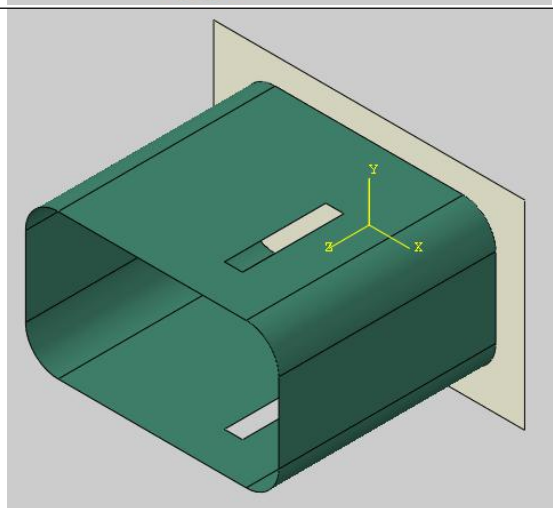
Exercise 10

- 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



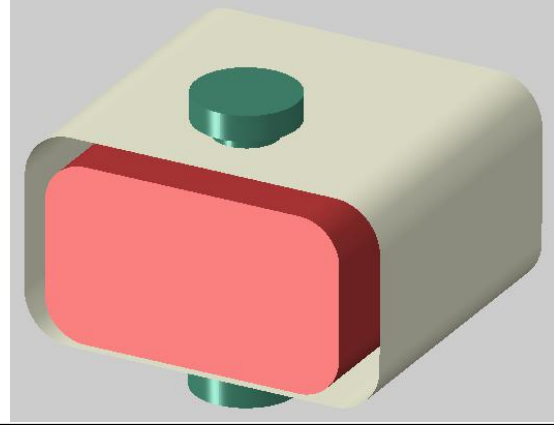
Exercise 11

- 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

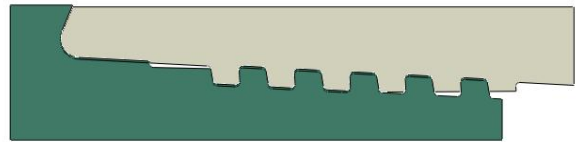


Exercise 12

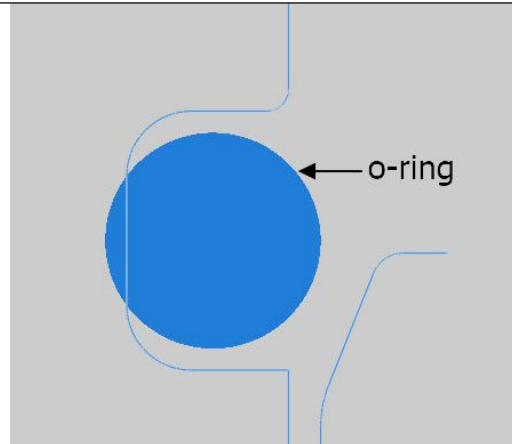
- Defining contact between the surfaces of solid and shell parts
- Accounting for shell thickness
- Using unsymmetric solver for better convergence
- Plotting surface normal to determine SPOS/SNEG faces of shell
- Choosing a master surface among two deformable surfaces.

**Exercise 13**

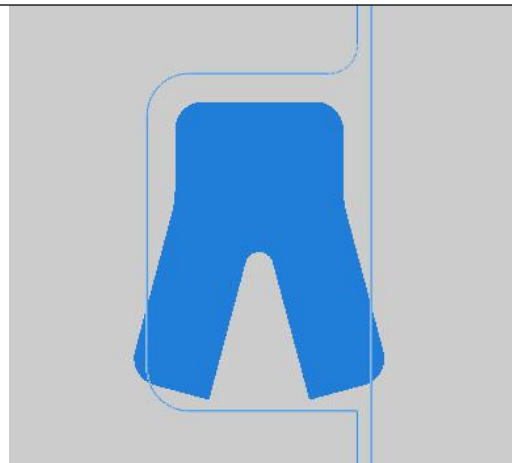
- Analyzing stresses in a VAM TOP connector due to internal fluid pressure
- Pressure penetration loading
- PPRESS output variable

**Exercise 14**

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

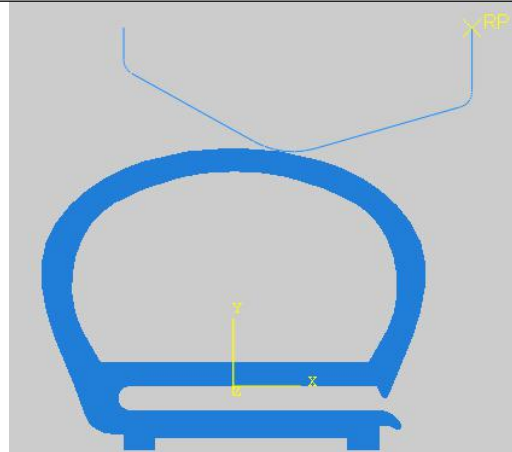
**Exercise 15**

- Analysis of an u-cup 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



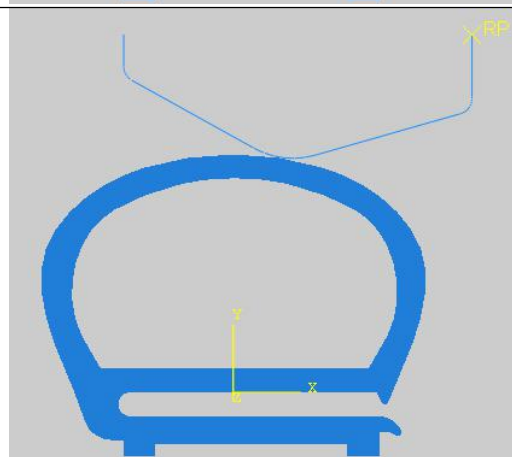
Exercise 16

- 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



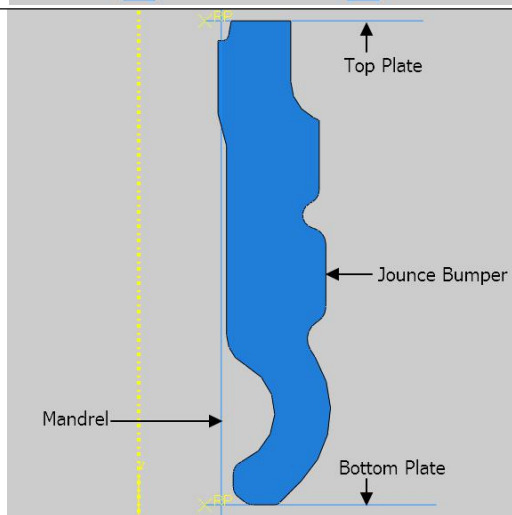
Exercise 17

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



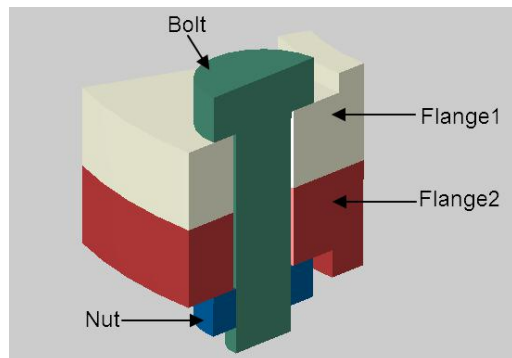
Exercise 18

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



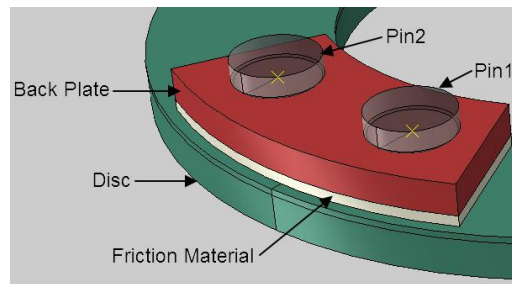
Exercise 19

- 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

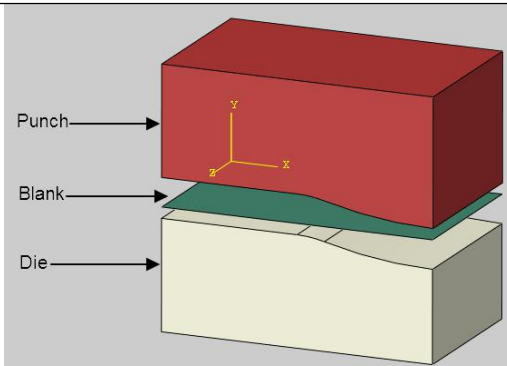


Exercise 20

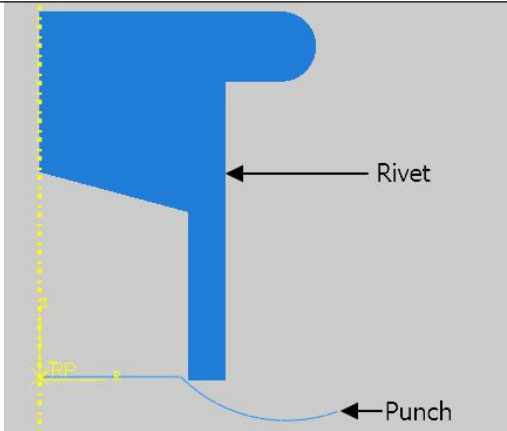
- Thermo-mechanical simulation of a disc brake system
- Defining temperature dependent coefficient of friction
- Defining thermal contact conductance as a function of gap clearance
- Defining an interaction to model heat transfer due to convection
- Specifying heat generation and its distribution
- Modifying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

**Exercise 21**

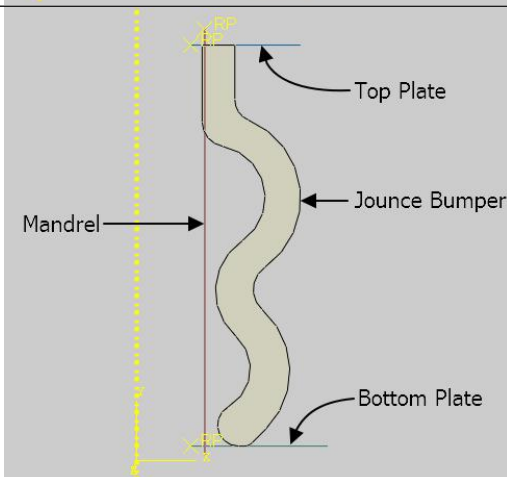
- Defining contact interaction for a punch-blank-die assembly
- Using general contact approach to define interaction
- Comparing contact pressure plots for general contact and contact pairs approach

**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

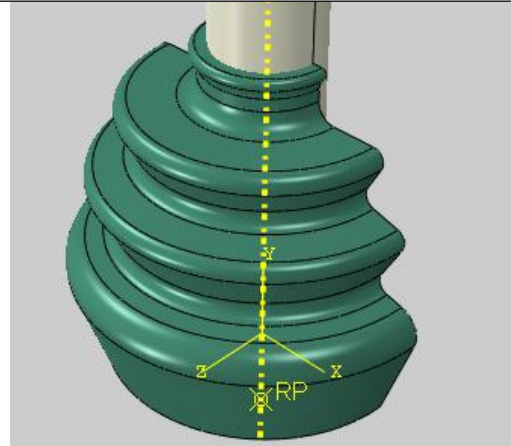
**Exercise 23**

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



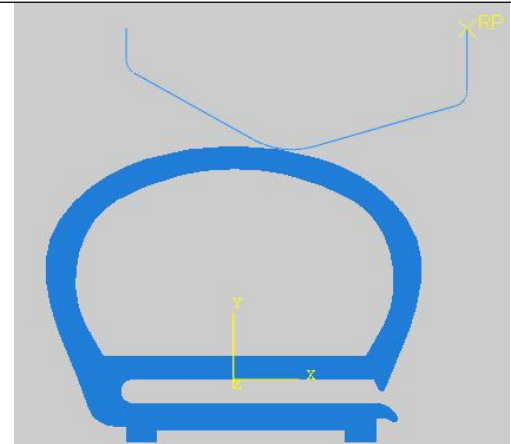
Exercise 24

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



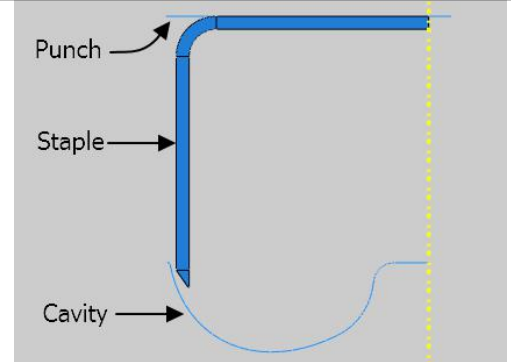
Exercise 25

- 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



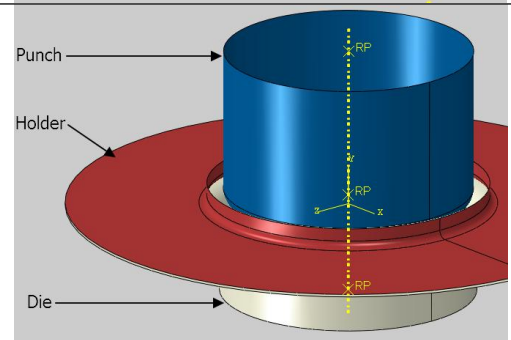
Exercise 26

- 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



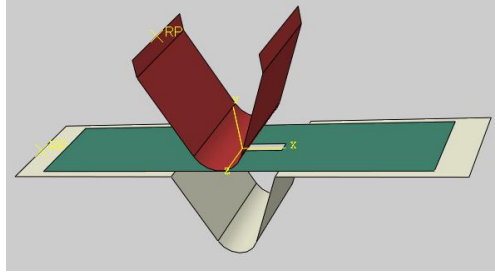
Exercise 27

- 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

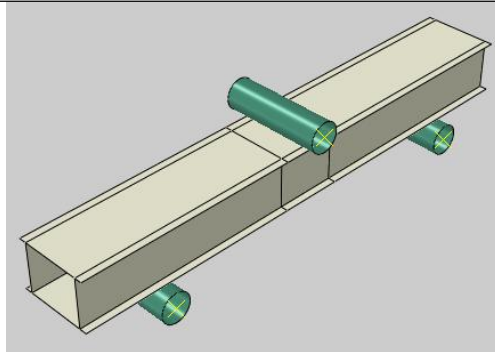


Exercise 28

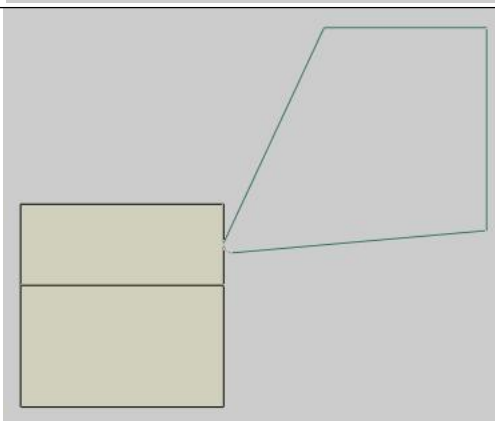
- Simulating the v-bending of a sheet metal blank
- Compare the results obtained with contact pair approach and general contact approach
- Rendering thickness of shell elements
- CTHICK output variable
- Contact controls to eliminate thickness reductions

**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

**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

