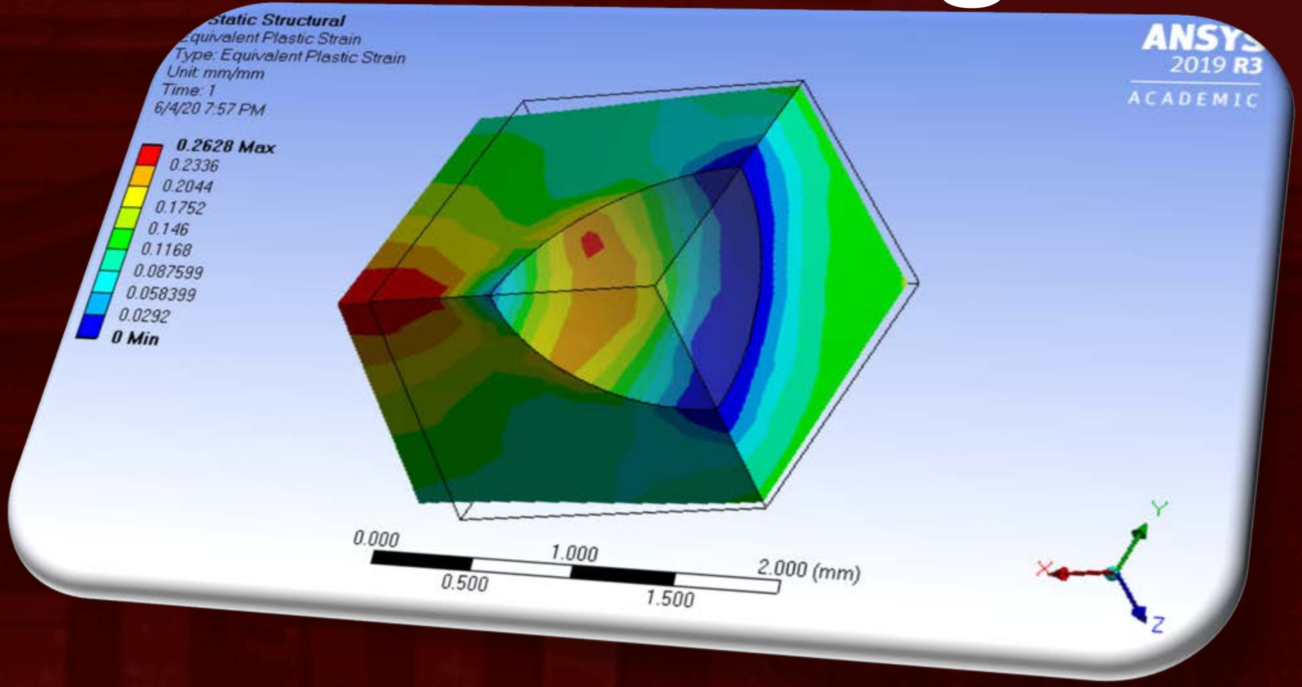
Introduction to FEA using Ansys-II



Umair Bin Asim
umair.asim@tamu.edu

Agenda

Discussion about yesterday’s session
Overview of FEA applications
Problem specification
Pre-analysis and start-up
Geometry
Mesh
Physics setup
Numerical solution and results
Verification and validation
Exercises

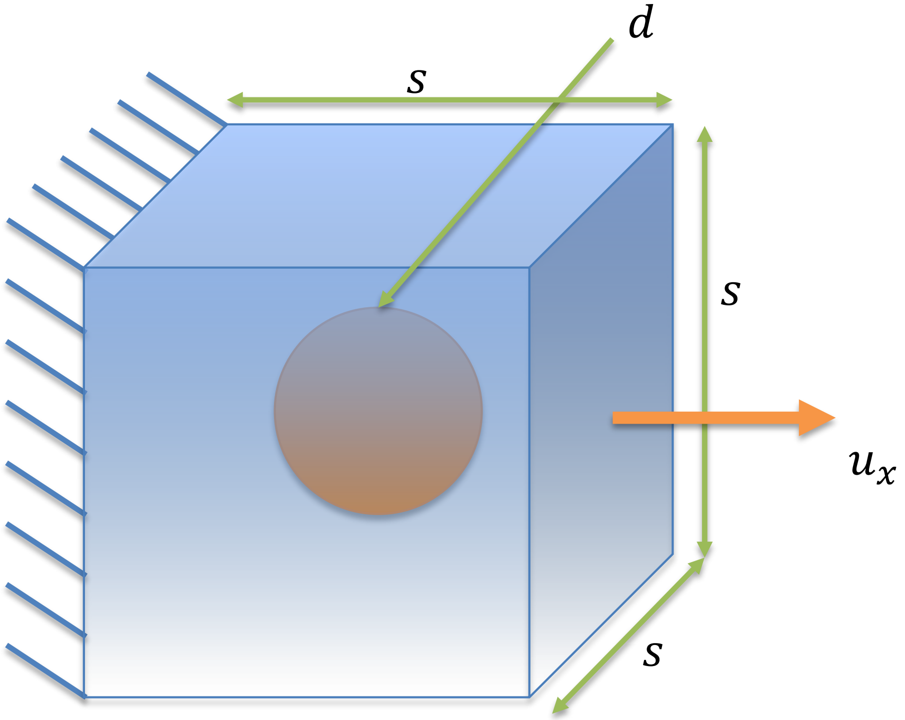
Yesterday's Exercises

1. Files of the Workbench project we worked on today's session are available as 'intro_fea_ansys_day1.zip' at /scratch/training/intro_ansys/ folder, you can unzip them in your scratch directory and use it for reference.
2. Try refining the mesh and comparing the results
3. Model quarter of the model and apply 'Symmetry' on the new edges using the procedure given at https://confluence.cornell.edu/display/SIMULATION/Plate+With+a+Hole+-+Physics+Setup and compare the results
4. Attempt the exercises given at https://confluence.cornell.edu/display/SIMULATION/Plate+With+a+Hole+-+Exercises

Problem specification

A cube (RVE) of ‘Titanium NL’ with a spherical inclusion of ‘Tungsten Carbide’ to simulate the behavior of a fictitious metal matrix composite under uniaxial tension:

- $r = 1\,mm,\ d = 2mm$
- $f = 0.1 = \frac{\frac{4}{3}\pi r^3}{s^3}$
- $s = \sqrt[3]{\frac{\frac{4}{3}\pi r^3}{f}} \cong 3.5mm$
- $u_x = 0.4mm$



Pre-Analysis

- This is a classical Eshelby's inclusion problem (sort of) and its analytical solution exists but…

  http://micro.stanford.edu/~caiwei/me340b/content/me340b-lecture02-v03.pdf

- Another way is to do simple homogenization using:

$$E_{hom} = E_1 v_1 + E_2 v_2$$

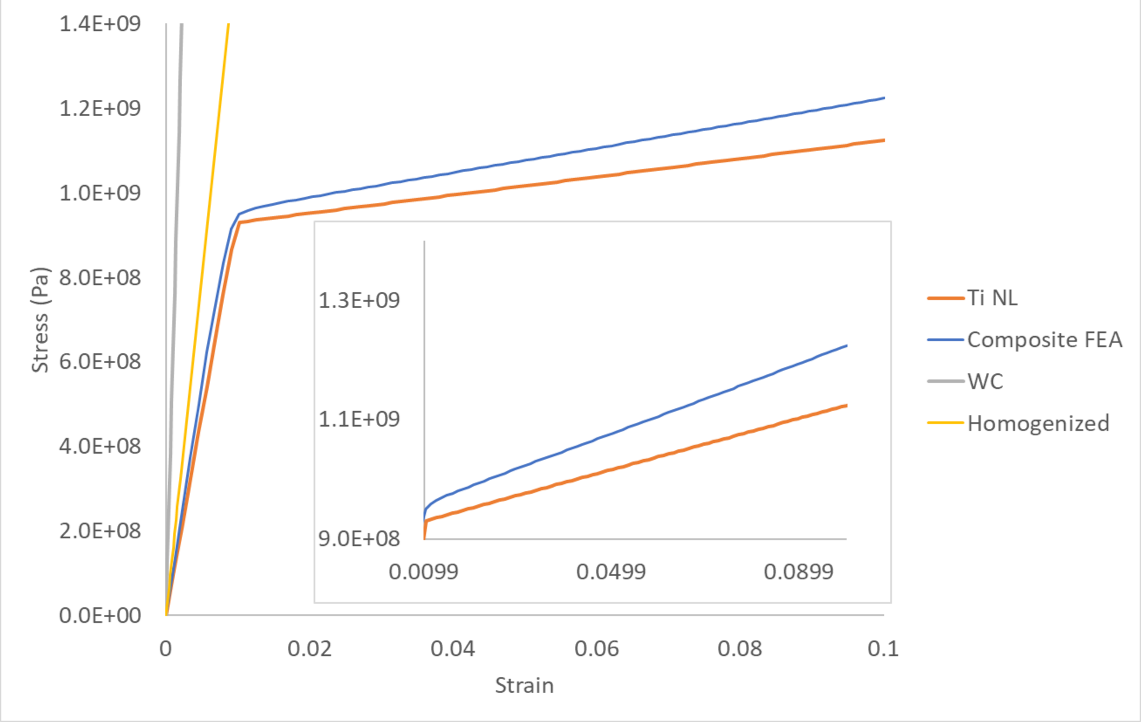


Fig: Stress-strain response of a titanium alloy, tungsten carbide and the FEA and homogenized results of the composite of Ti with 10% WC

Start-Up

- Make sure you have an HPRC account and VPN service installed
- Go to https://portal.hprc.tamu.edu/
- Select ‘Ada OnDemand Portal’
- Use your NetID and password to login
- In the toolbar at the top, select ‘Interactive Apps’ and select ‘Ansys Workbench’
- Select/enter these values from/in drop-down/textbox
  - Ansys version: ANSYS/2019R3, Number of hours: 4, Number of cores: 1, Memory per core (GB): 2, Node type: GPU, Email (optional): (Enter your email address to get the status of your session).
- Click the ‘Launch’ button
- A new page opens, wait if it says so. Once a session has been setup click on ‘Launch noVNC in New Tab’ button
- Ansys Workbench will open in a new tab.
- **Make sure not to close the tab and keep on saving the progress in ‘/scratch/user/*netid*/ansys_second_tutorial’.**
- And if you have then do not panic, go to the previous tab and click ‘Launch noVNC in New Tab’ button again

Start-Up

- Workbench will start in a new tab of your browser
  - make sure to click on ‘maximize’ button at the right top corner so that you see the complete window
- Select ‘Static Structural’ from the toolbox
- Select ‘Metric(tonne, mm…’ from toolbar>Units
- Import ‘Tungsten Carbide’ from ‘Engineering Data Sources’ tab and ‘Granta Design Sample Materials’ under ‘Engineering Data Sources’ field
- Import ‘Titanium NL’ from ‘General Non-linear Materials’

Geometry

- Double click on ‘Geometry’ option under ‘Static Structural’ system in ‘Project Schematic’ (those who are using their own computers, right-click on ‘Geometry’ and select ‘New DesignModeler Geometry’)
- Again make sure to ‘Maximize’ it and acquaint yourself with the interface.
- Update the units once again
- Follow in class instructions to create 3D geometry of a 3.5mmx3.5x3.5mm cube with a sphere of 2mm diameter in the center
- Then apply the symmetry condition across XY,YZ and XZ planes to only solve quarter of the model

Mesh

- Double click on ‘Model’ option under ‘Static Structural’
- ‘Maximize’ the new window
- Make sure the units are set to ‘Metric (mm, kg,…)’ by clicking on ‘Metric…’ text on the lower right hand-side on the status bar.
- Follow in class instruction to discretize the geometry into with auto settings

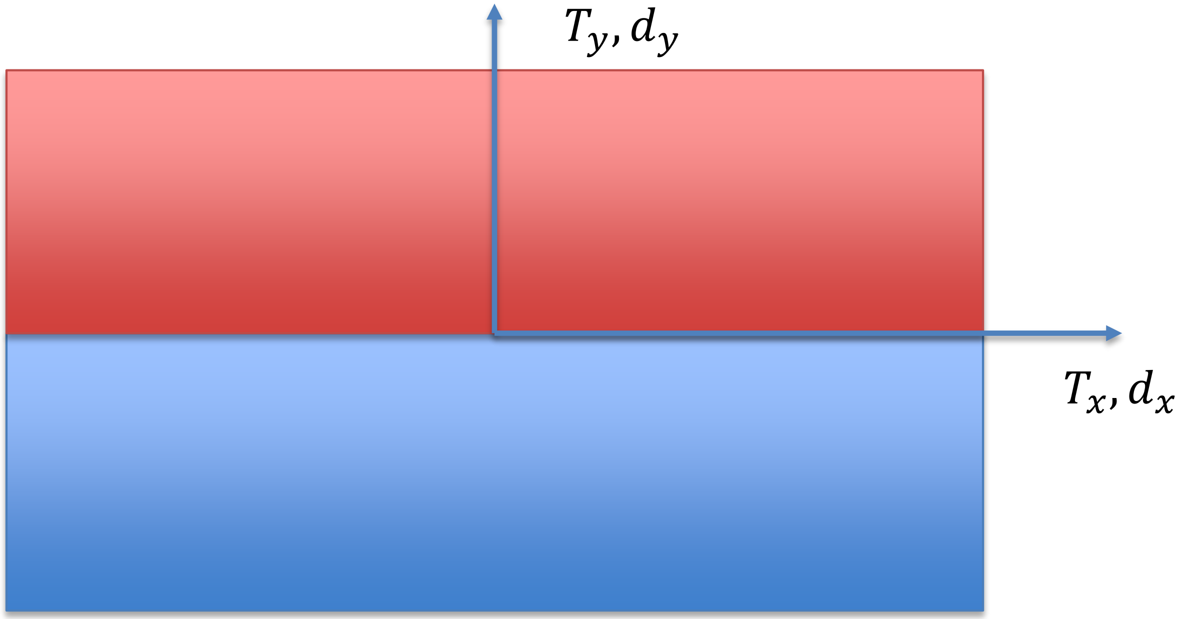
Physics Setup

- Continue in the ‘Mechanical’ interface
- Assign the material properties created during start-up to the meshed geometry
- Apply ‘Displacement’ of 0.2 mm on the face perpendicular to x-axis and make sure the symmetries are applied right.
- Make sure that the ‘Large Deflection’ option is ‘ON’ in ‘Analysis Settings’ details

Interaction/Contact

- Bonded
- No separation
- Frictionless
- Rough
- Frictional

Details of each of them can be found here (make sure you are connected through VPN) on page number 701 and 702: https://hprc.tamu.edu/softwareDocs/ansys/v182/ANSYS%20Mechanical%20Users%20Guide.pdf

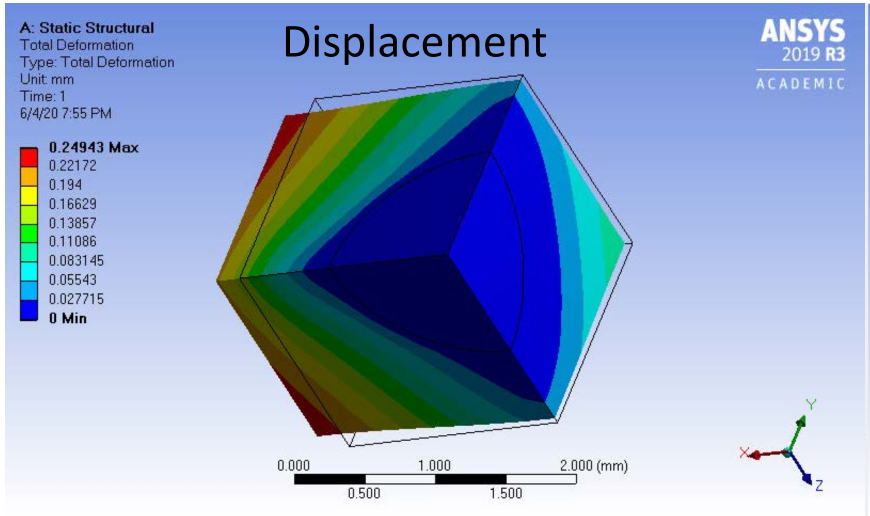


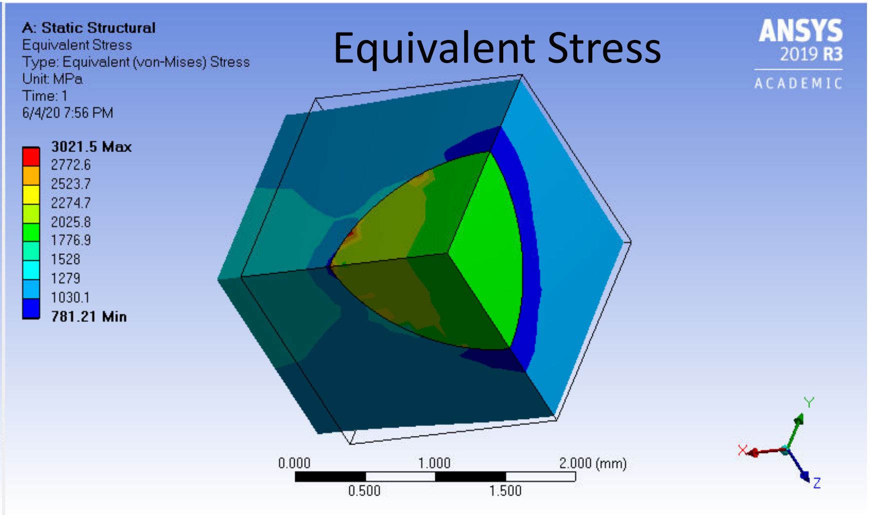
Numerical Solution and Results

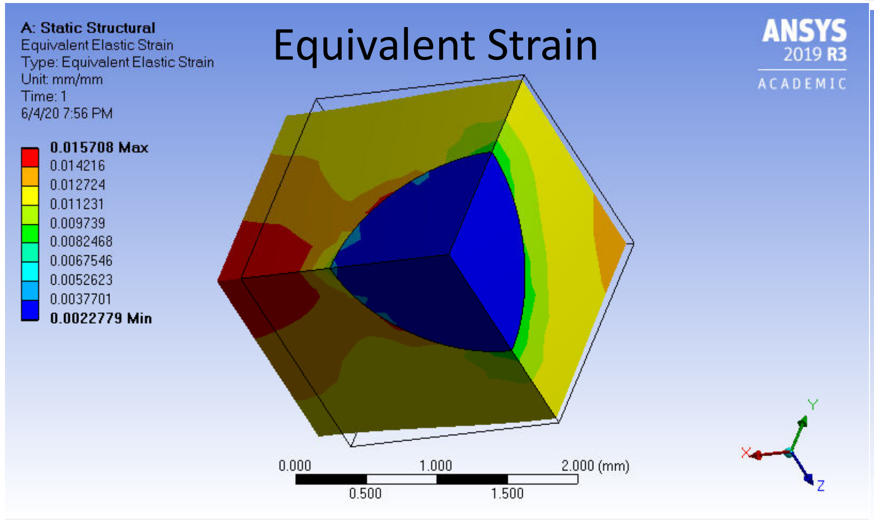
TEXAS A&M UNIVERSITY

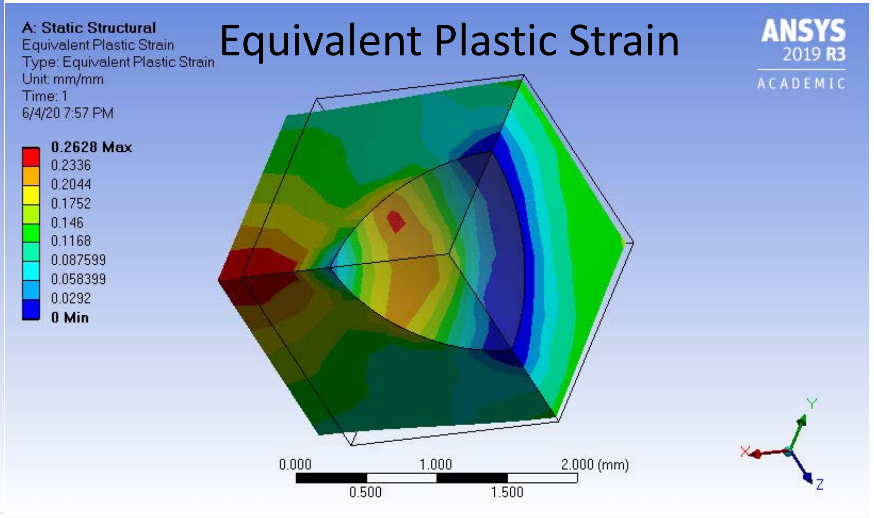
- Continue in the ‘Mechanical’ interface
- Call for the results of deformation and equivalent stress and strain and equivalent plastic strain
- Also create ‘Probes’ for values of reaction force on the face perpendicular to x-axis and deformations in x, y and z direction of corner vertex.
- Solve for these results
- Look at the solutions for deformation and stress distribution in the plate

Results


Displacement
A: Static Structural
Total Deformation
Type: Total Deformation
Unit: mm
Time: 1
6/4/20 7:55 PM
0.24943 Max
0.22172
0.194
0.16629
0.13857
0.11086
0.083145
0.05543
0.027715
0 Min
ANSYS
2019 R3
ACADEMIC
0.000
1.000
2.000 (mm)
0.500
1.500

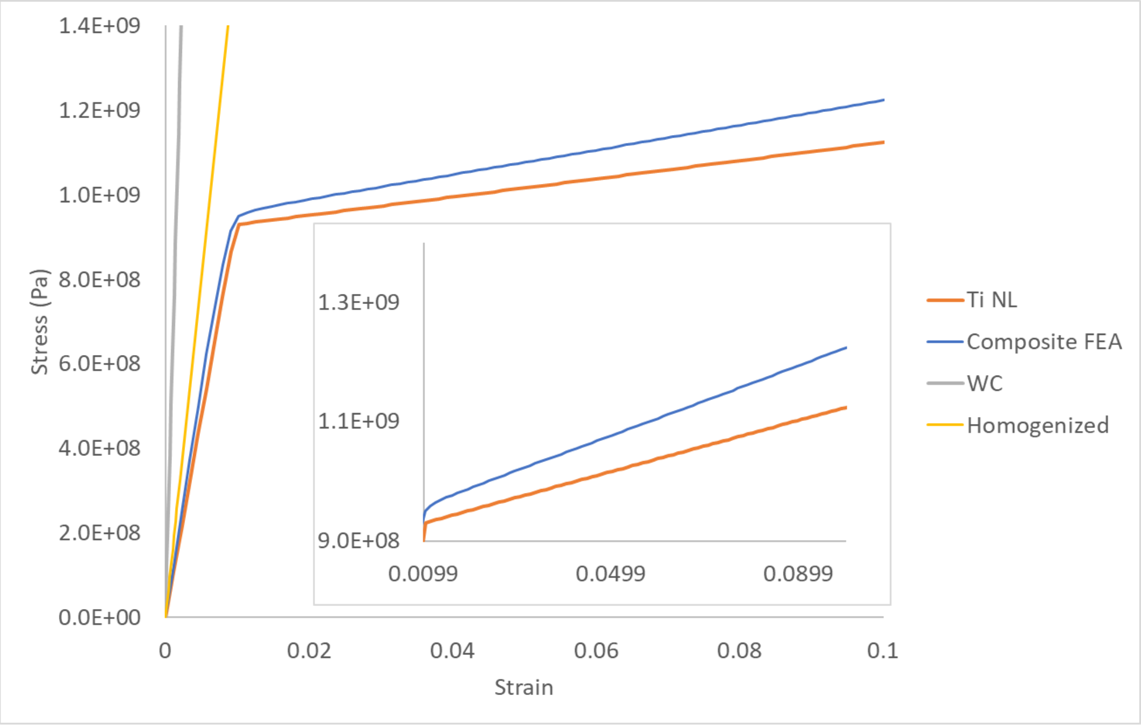

Equivalent Stress
A: Static Structural
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: MPa
Time: 1
6/4/20 7:56 PM
3021.5 Max
2772.6
2523.7
2274.7
2025.8
1776.9
1528
1279
1030.1
781.21 Min
ANSYS
2019 R3
ACADEMIC
0.000
1.000
2.000 (mm)
0.500
1.500


Equivalent Strain
A: Static Structural
Equivalent Elastic Strain
Type: Equivalent Elastic Strain
Unit: mm/mm
Time: 1
6/4/20 7:56 PM
0.015708 Max
0.014216
0.012724
0.011231
0.009739
0.0082468
0.0067546
0.0052623
0.0037701
0.0022779 Min
ANSYS
2019 R3
ACADEMIC
0.000
1.000
2.000 (mm)
0.500
1.500


Equivalent Plastic Strain
A: Static Structural
Equivalent Plastic Strain
Type: Equivalent Plastic Strain
Unit: mm/mm
Time: 1
6/4/20 7:57 PM
0.2628 Max
0.2336
0.2044
0.1752
0.146
0.1168
0.087599
0.058399
0.0292
0 Min
ANSYS
2019 R3
ACADEMIC
0.000
1.000
2.000 (mm)
0.500
1.500

Verification and validation

- Comparison of the numerical results with analytical approximate solutions:
- Do not forget to save the project.



Exercises

1. Files of the Workbench project we worked on today's session are available as 'intro_fea_ansys_day2.zip' at /scratch/training/intro_ansys/ folder, you can unzip them in your scratch directory and use it for reference.
2. Try refining the mesh and comparing the results
3. Try reducing the volume fraction to 0.05 and increase to 0.2 and see how it affects the stress-strain response
4. Try to add materials of your choice for the matrix and inclusion and see the effect
5. Try a cylindrical inclusion rather than spherical with same volume fractions of 0.05, 0.1 and 0.2 and compare them with spherical inclusion results and also with spherical void of different volume fractions


Thank you