Module 01 Student Workshop: Introduction

Please note:
- These training materials were developed and tested in Ansys Release 2023 R1. Although they are expected to behave similarly in later releases, this has not been tested and is not guaranteed.
- The screen images included with these training materials may vary from the visual appearance of a local software session.
- Although some workshop files may open successfully in previous releases, backward compatibility is somewhat unlikely and is not guaranteed.
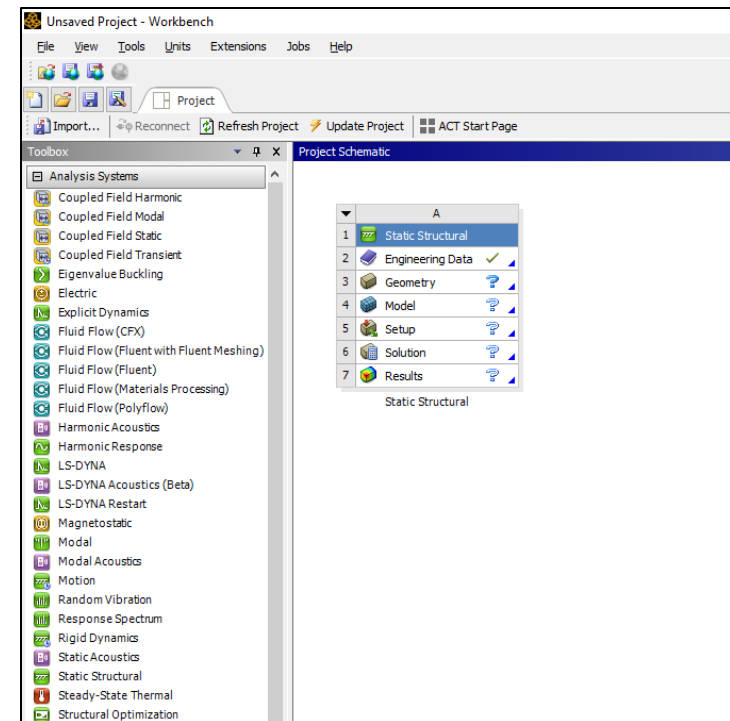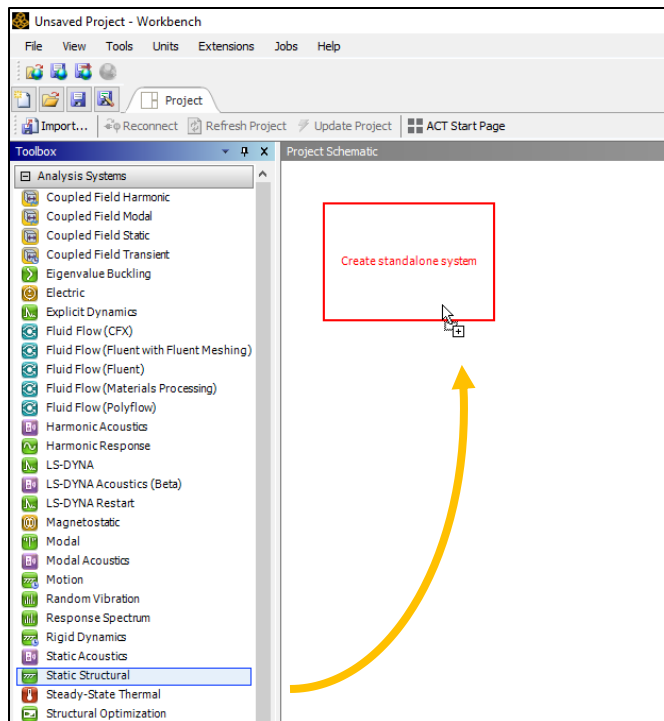
Workshop 01: Introduction

Use this guide to work on the Journal Bearing model. In this module, you'll set up a basic Static Structural Analysis and become familiar with the end-to-end simulation workflow. (The Journal Bearing model will be described more fully in subsequent modules.)
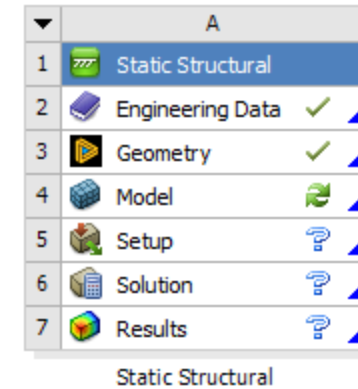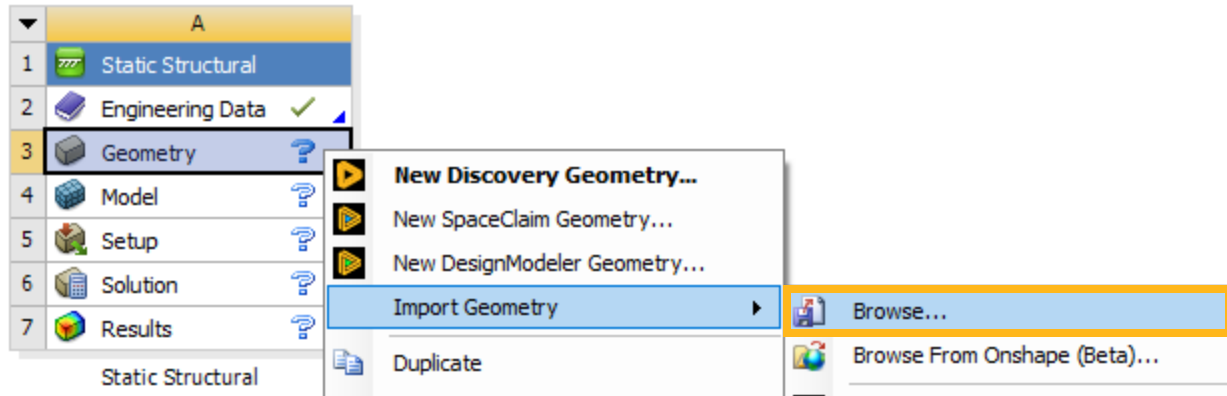
Workshop 01: Introduction

- Open Ansys Workbench: Windows Start Menu button → Ansys [release] → Workbench [release]

- Under the Toolbox Analysis Systems group, click and drag analysis system Static Structural onto the Project Schematic, drop it on target Create standalone system (this will be the only target available)

Workshop 01: Introduction

- RMB—Geometry → Import Geometry → Browse… → bearing_housing.scdoc → Open
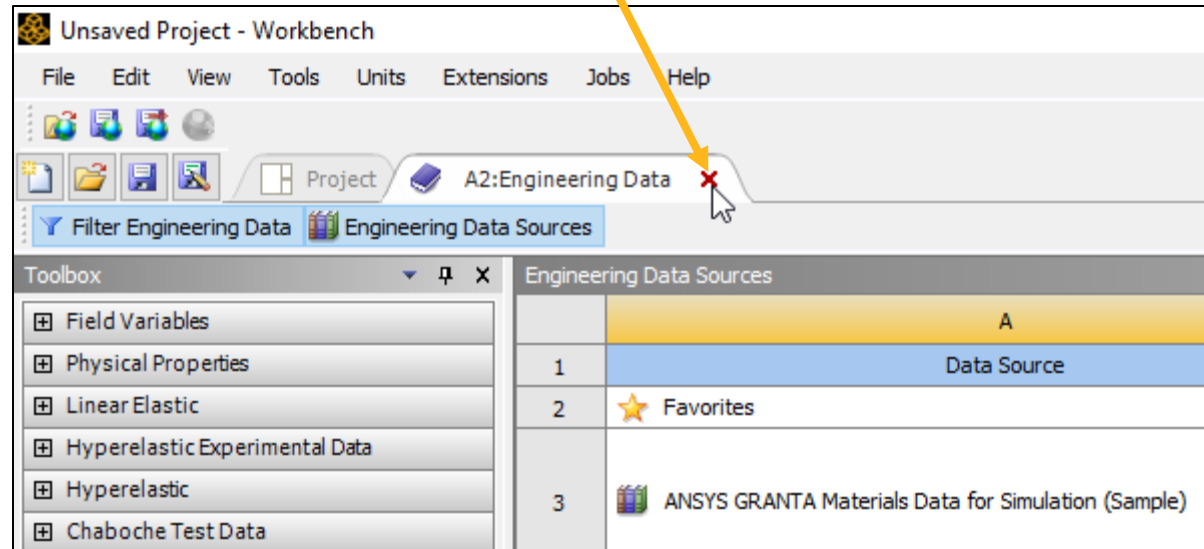
Workshop 01: Introduction

- RMB—Engineering Data → Edit…

- Toggle toolbar button Engineering Data Sources to "on"

- Click Data Source General Materials

- In table Outline, click the + icon next to Stainless Steel

Workshop 01: Introduction

- Toggle toolbar button Engineering Data Sources to "off"
- Click the small x on tab A2:Engineering Data to close the Engineering Data application

Workshop 01: Introduction

- RMB—Model → Edit... to open Ansys Mechanical

- Expand the Geometry branch

- Select Part branch Base\Base

- In view Details, set Material—Assignment to Stainless Steel using drop-down menu
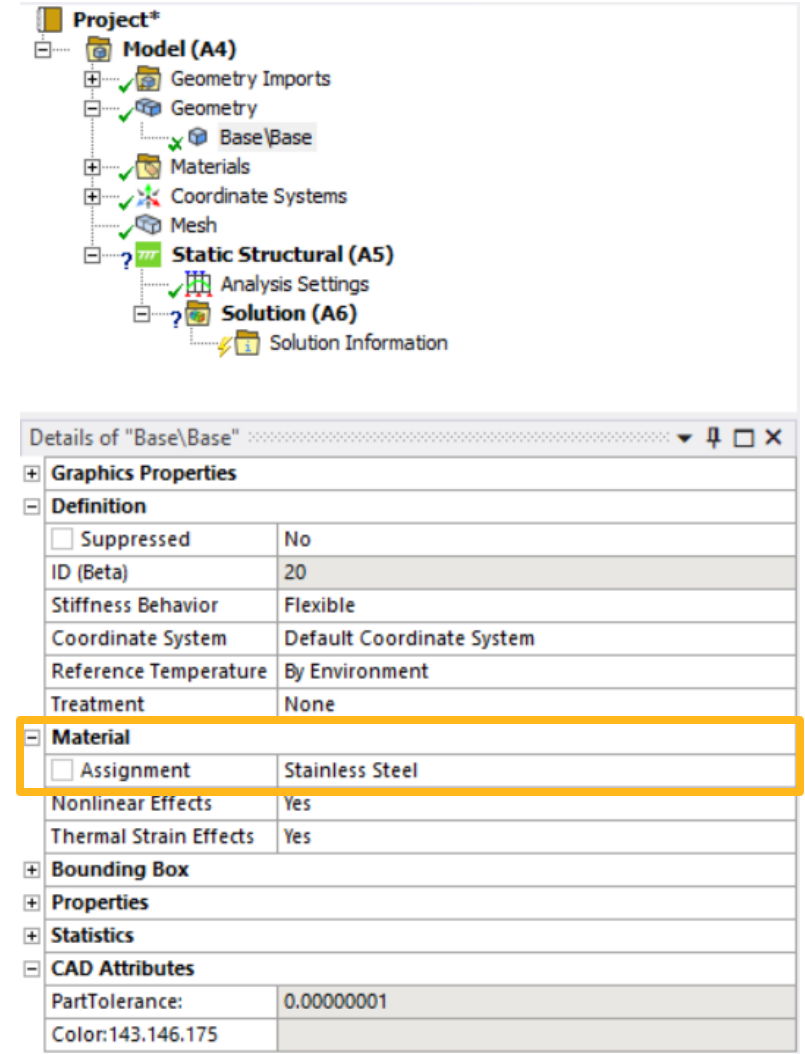
Workshop 01: Introduction

- Shortcuts for graphics view control:

Click middle mouse button for rotate

Shift-click middle mouse button for zoom

Ctrl-click middle mouse button for pan

Function key F7 for zoom-to-fit

Click and hold right mouse button to drag a zoom window

Workshop 01: Introduction

- Select branch Static Structural (A5)

- Change the selection mode to the Face Selection filter

- Select the two faces on the bottom of the part (use the Ctrl key for multiple selection)

- RMB—Static Structural → Insert → Frictionless Support

- Select the two internal faces of the bolt holes

- RMB—Static Structural → Insert → Frictionless Support

A: Static Structural
Frictionless Support
Time: 1. s

☐ Frictionless Support

A: Static Structural
Frictionless Support 2
Time: 1. s

☐ Frictionless Support 2
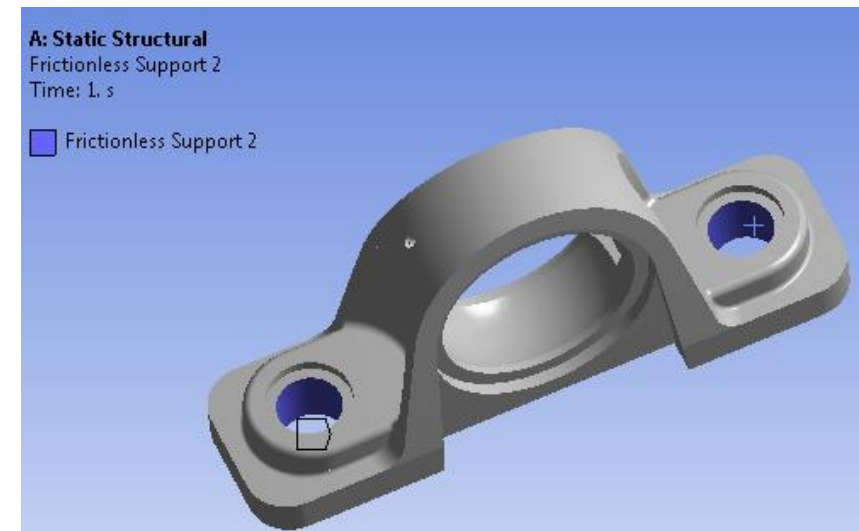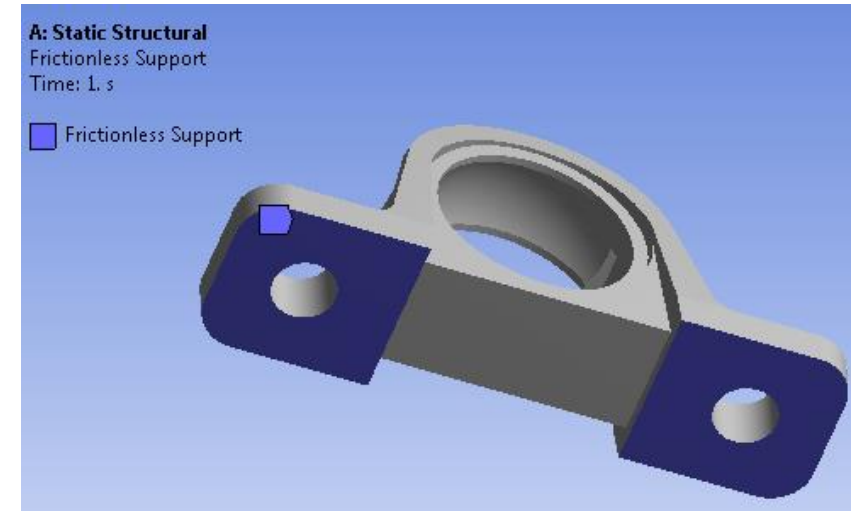
Workshop 01: Introduction

- Select the internal cylindrical surface shown below

- RMB—Static Structural → Insert → Force

- In the Details panel, set the Define By to 'Components'

- Set Y Component to 500 N and Z Component to 1000 N (if necessary, set Units to Metric (mm, t, N, s, mV, mA) using the Units menu in the Home ribbon)
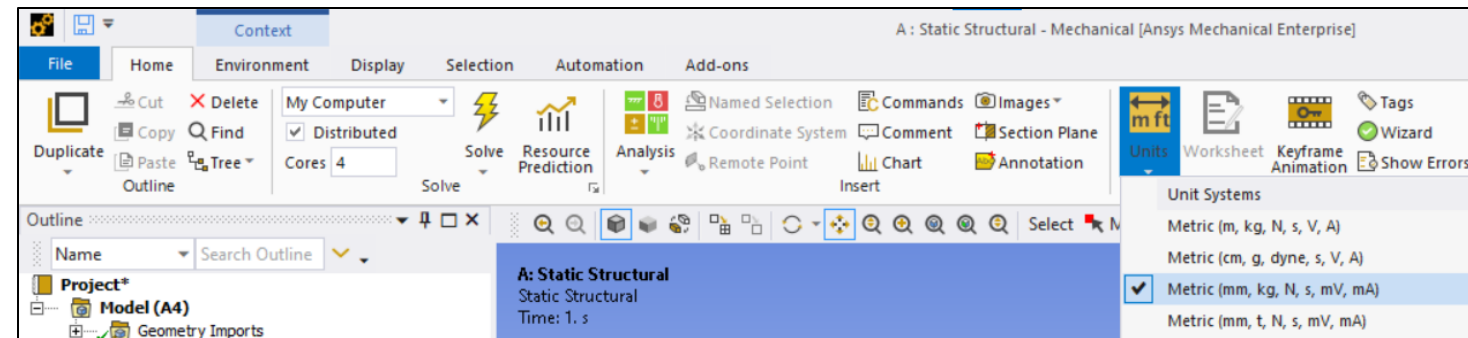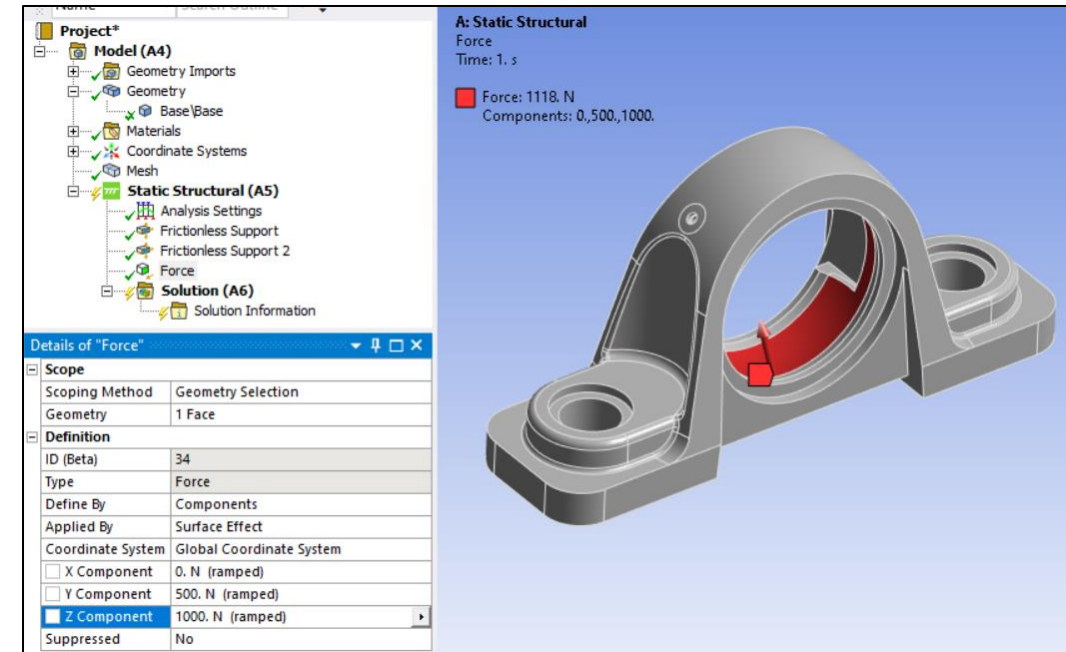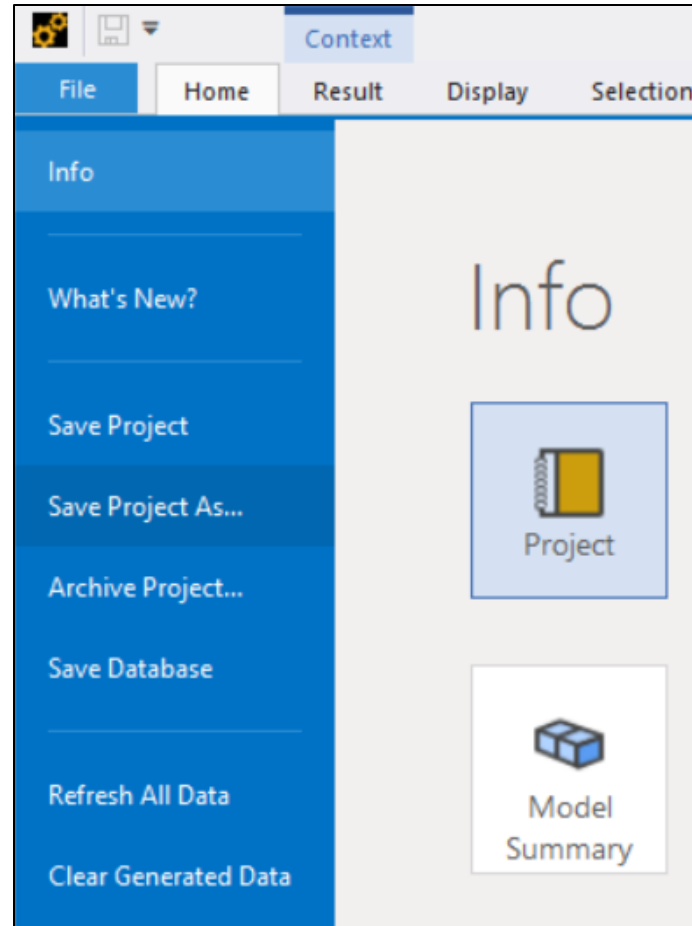
Workshop 01: Introduction

- RMB—Solution → Insert → Stress → Equivalent (von-Mises)

- RMB—Solution → Insert → Deformation → Total

- Solve

- Select branch Equivalent Stress or Total Deformation to review results

©2023 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.
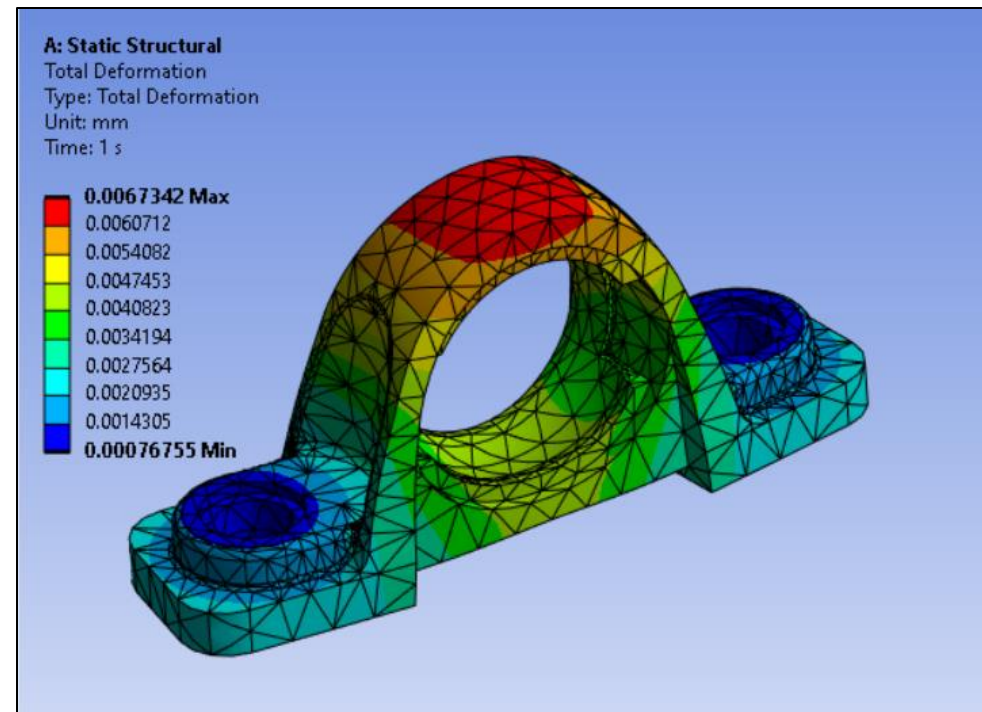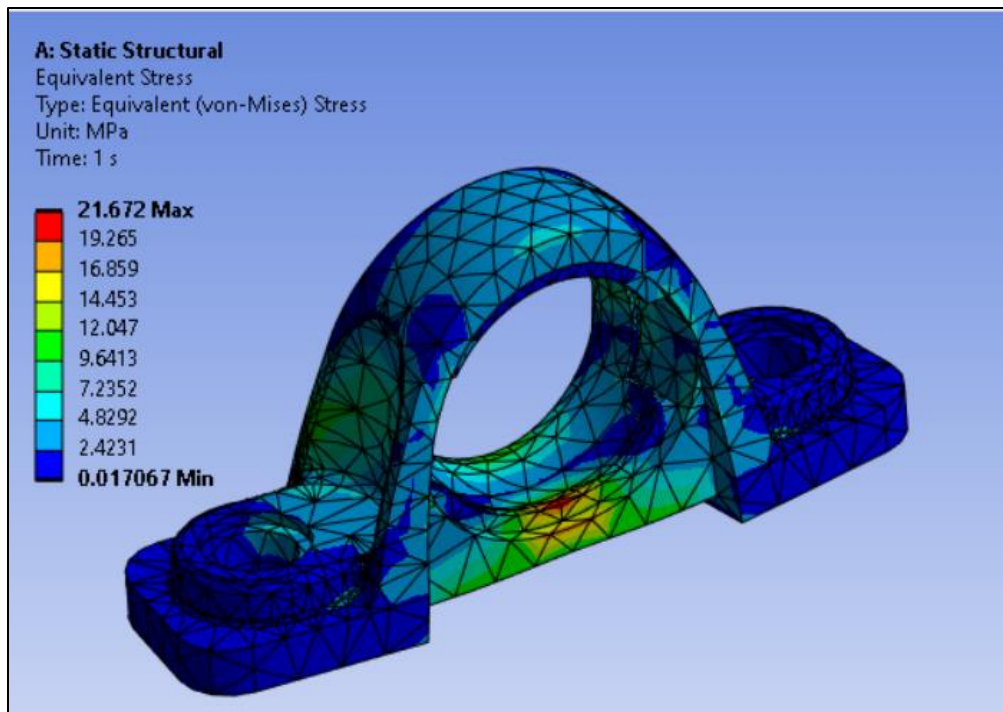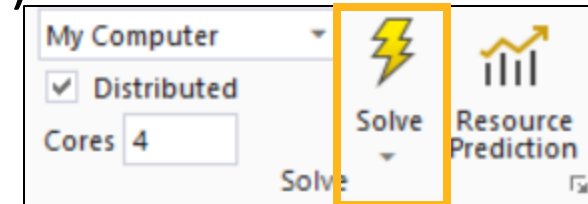
Workshop 01: Introduction

- Save the Project: File → Save Project As... → "WS01.wbpj"

End of presentation