Engineering Institute of Technology.

Introduction to ANSYS Workbench

Software Instructions

Unit code	BSC105C		
Unit name	Mechanics of Machines		
Software used	ANSYS Workbench (Student)		
Version #	2020 R1		
Created by	Dr Subhankar Biswas	**Date**	30 July 2020
Reviewed by		**Date**	

Software Instructions: ANSYS Workbench 2020 R1 (Student)

Acknowledgments

We would like to acknowledge the following references used to compile these instructions for students:

- ANSYS Software Manual: Available via Software Help Menu
- Fundamentals of Finite Element Analysis, David V. Hutton, McGraw Hill (eBook)

COMMONWEALTH OF AUSTRALIA

Copyright Regulations 1969

WARNING

This material has been reproduced and communicated to you by or on behalf of The Engineering Institute of Technology pursuant to Part VB of the *Copyright Act 1968* (the Act).

The material in this communication may be subject to copyright under the Act. Any further reproduction or communication of this material by you may be the subject of copyright protection under the Act.

Do not remove this notice.

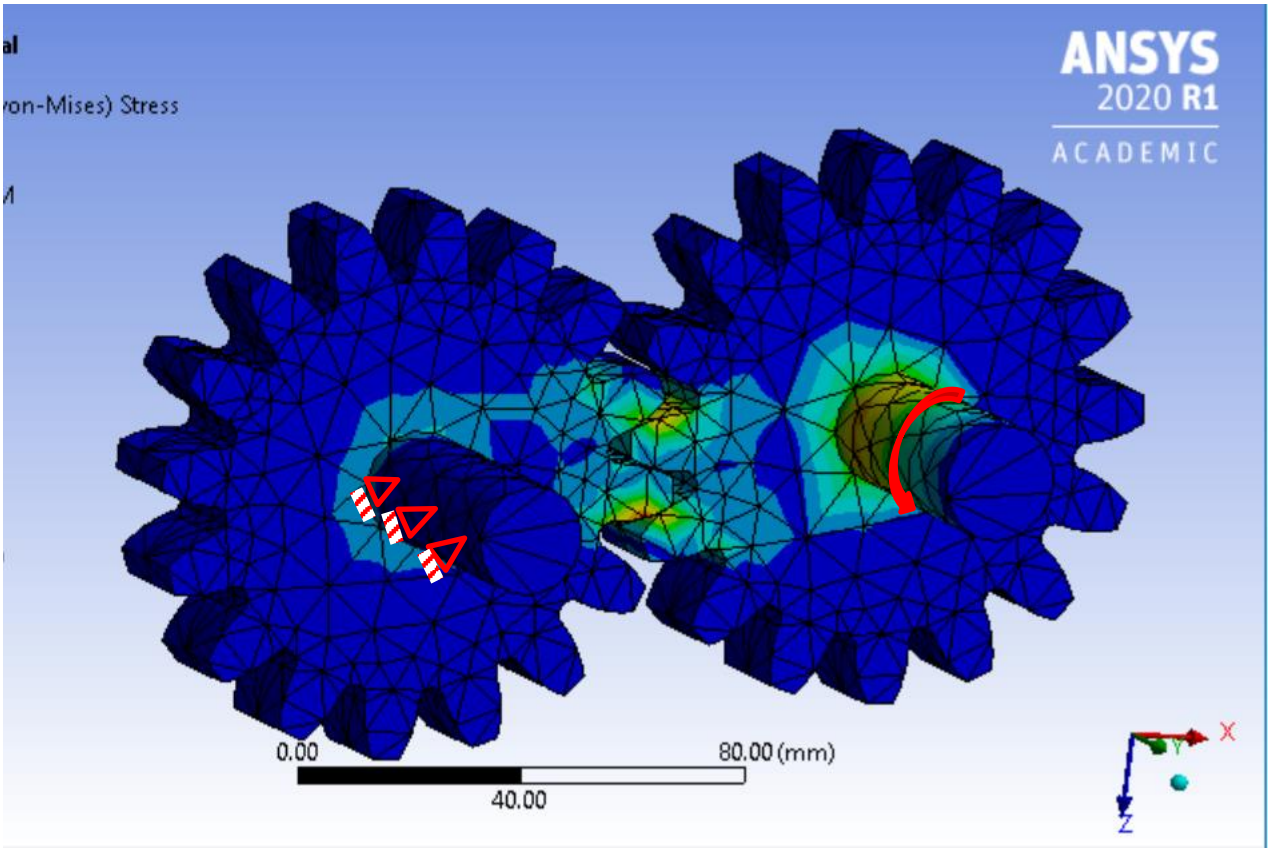
Software Instructions for ANSYS 2020 R1

Introduction to ANSYS Workbench

Objective:

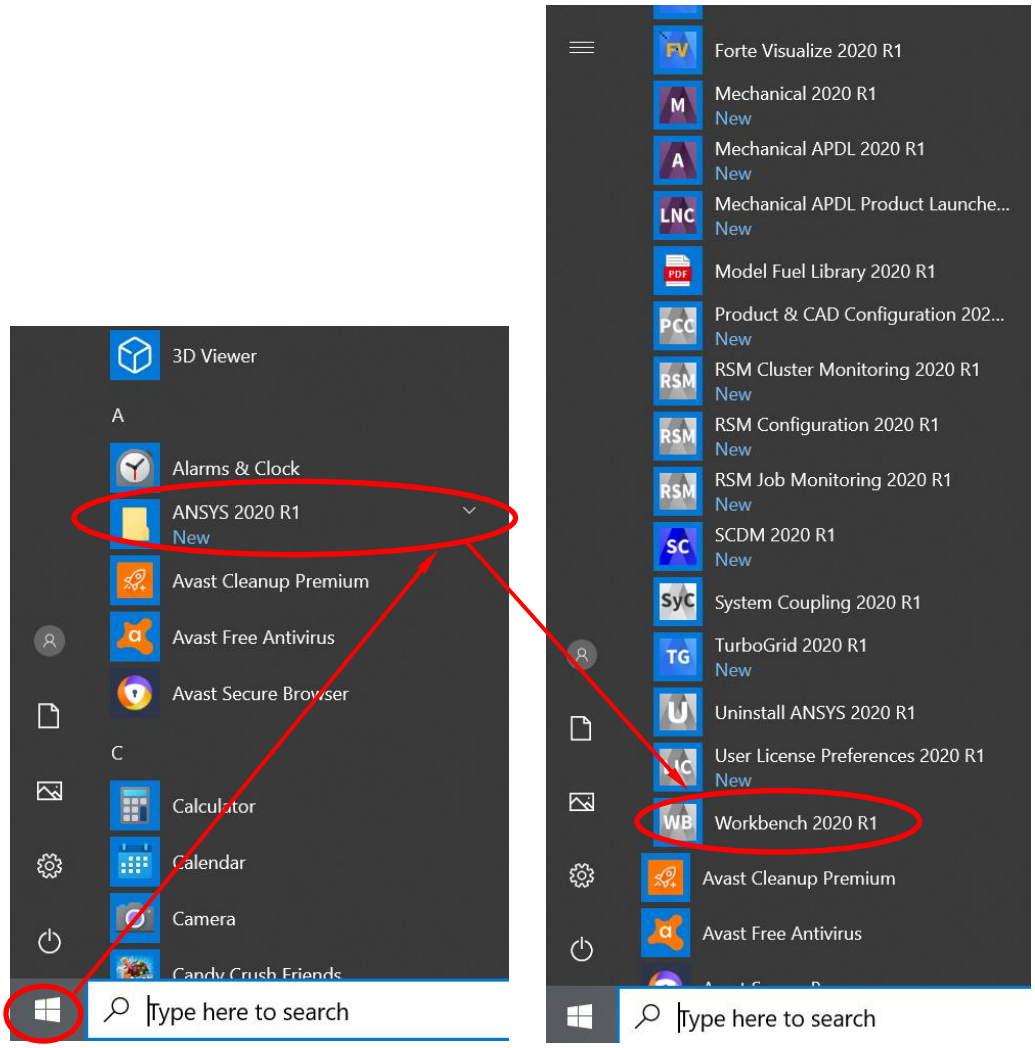
Our aim in this course is to get familiar with the workflow in the Finite Element Analysis (FEA) package in **ANSYS Workbench**. We will create a simple model of a spur gears that is fixed at one end and rotate at the other end. The analysis results allow us to assess the deflection, stress and strain distribution of the gears.

Structural Steel	
Density, ρ	7850 kg/m^3
Poisson's Ratio, ν	0.3
Young's Modulus, *E*	2.0 GPa
Tensile Yield Strength, σ_Y	250 MPa
Tensile Ultimate Strength, σ_t	460 MPa

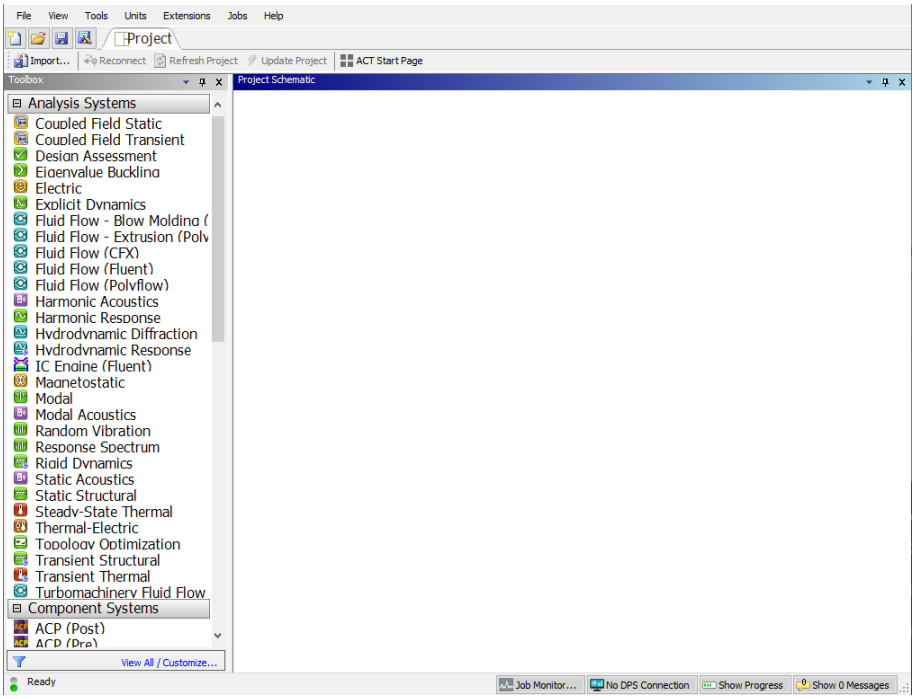


Installation and Launch ANSYS Workbench:

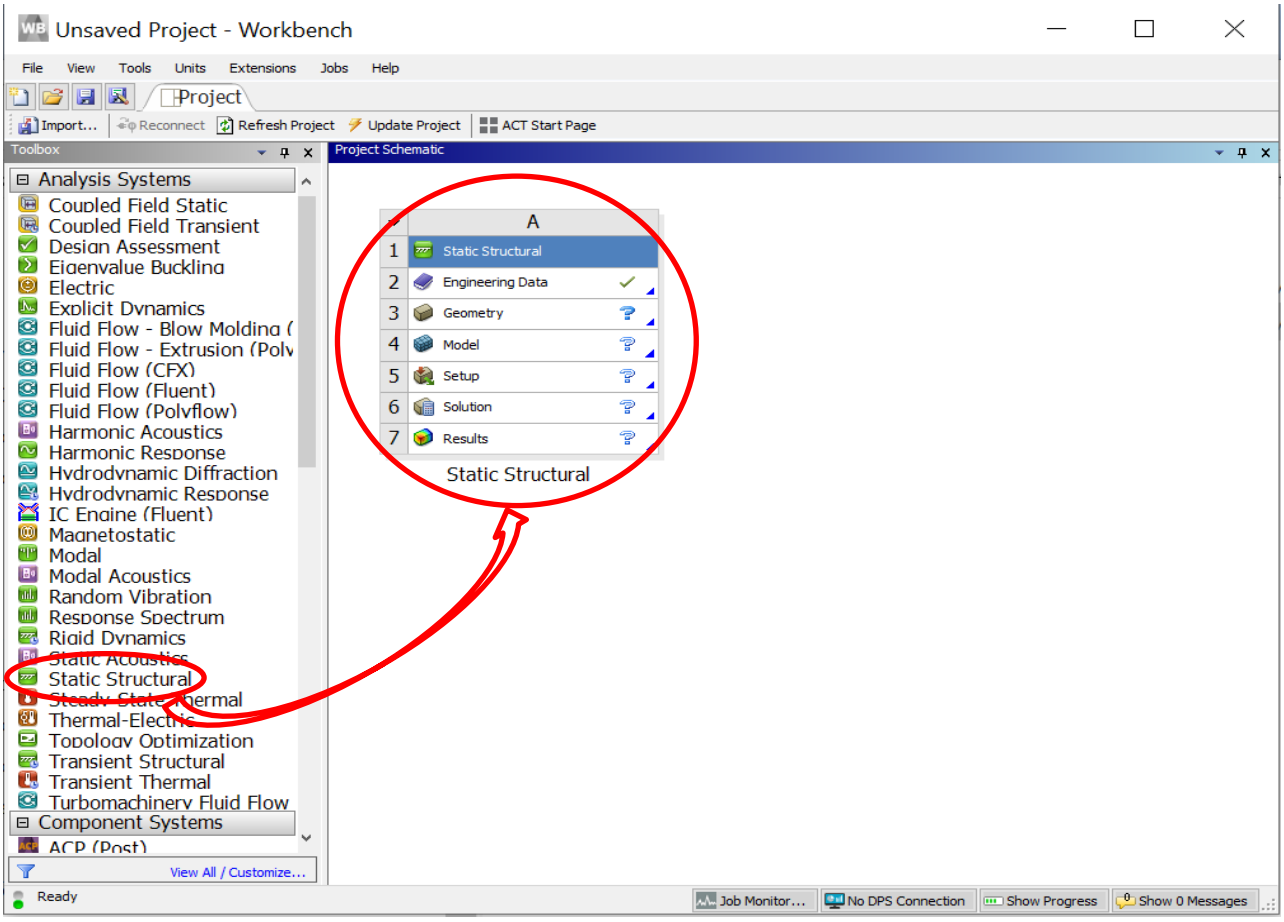
Step 1: Install ANSYS package available from https://www.ansys.com/academic/free-student-products (it's free!).
Step 2: Launch the ANSYS Workbench 2020 R1 from the local installation.



The ANSYS Workbench has now created a new project.



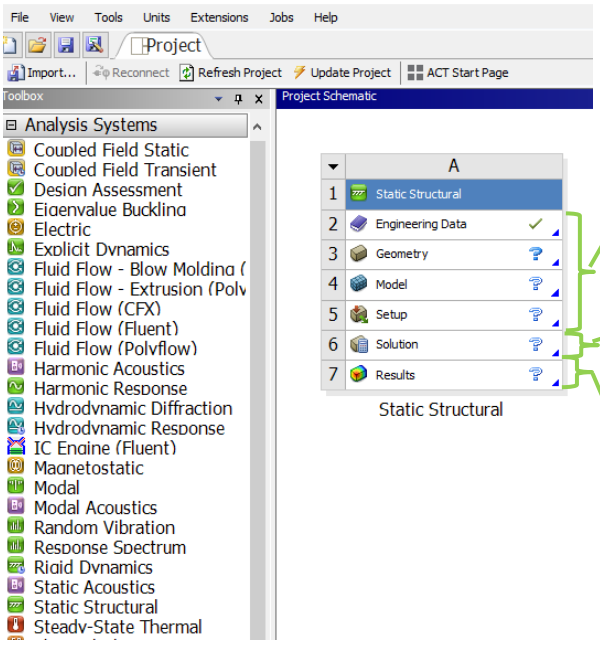
Step 3: Program schematic in Workbench: The program schematic allows to see and manipulate the Analysis systems. By dragging and dropping (or double clicking) different analysis systems can be inserted in the program schematic. We will create a standalone static structural system:



Save the project by hitting the **save as…** button and save it on your computer. Name your file Tute1_CantiBeam. Ansys will create a Workbench project file (*.wbpj) and a folder with the filename.

The Static Structural Analysis system:

In the "Static Structural" Analysis System we can perform quasistatic finite element analysis which includes all analysis steps – pre-processing, solving the model and post processing



Pre-processing:
During pre-processing the model is defined: Geometry, material data, loads and boundary conditions are specified in this step.

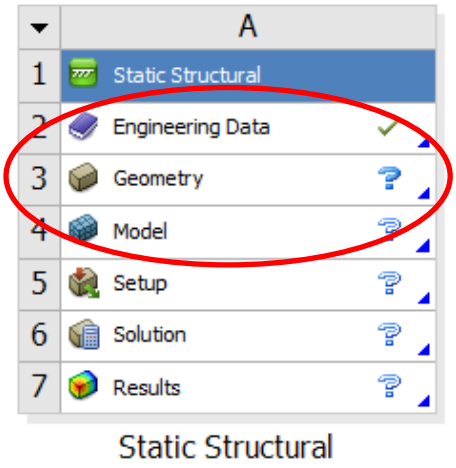
Solving:
Depending on what settings are chosen, Ansys is using different equations to solve the model.

Post-processing:
After solving the model, the results can be viewed and manipulated in the results step.

Pre-processing:

First we have to define the model. Each of the sub systems is fully defined when a green tick mark appears on the side. A question mark enotes that this field has incomplete input data. If upstream data has been modified and a field needs to be updated two green arrows appear. A yellow flash indicates that a field requires attention.

Engineering Data:
By default Ansys has preselected a material in the Engineering Data for us. This material is structural steel. How to modify the material data will be explained in a later seminar.



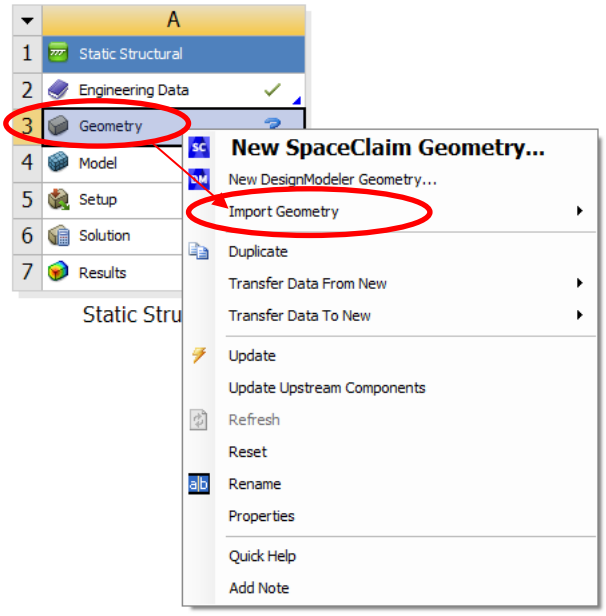
Geometry:
The geometry of our model is created and can be manipulated in the Ansys Design Modeller. This is a simplified CAD environment, tailored to work together with the modelling environments in Ansys Workbench.

Model:
Once the geometry has been specified it has to be meshed and loads, boundary conditions and other constraints must be assigned. For this task we will use the Ansys Mechanical modelling environment.

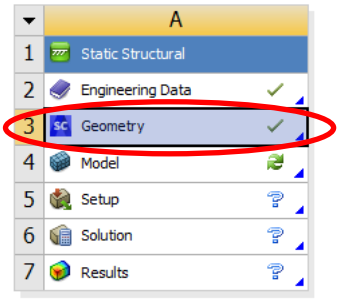
Defining the model geometry in Ansys SpaceClaim:

Step 4: Upload the given Spur gear geometry.

Right click on the Geometry from the project schematic and hit the Import Geometry option to upload the supplied geometry from the saved location.

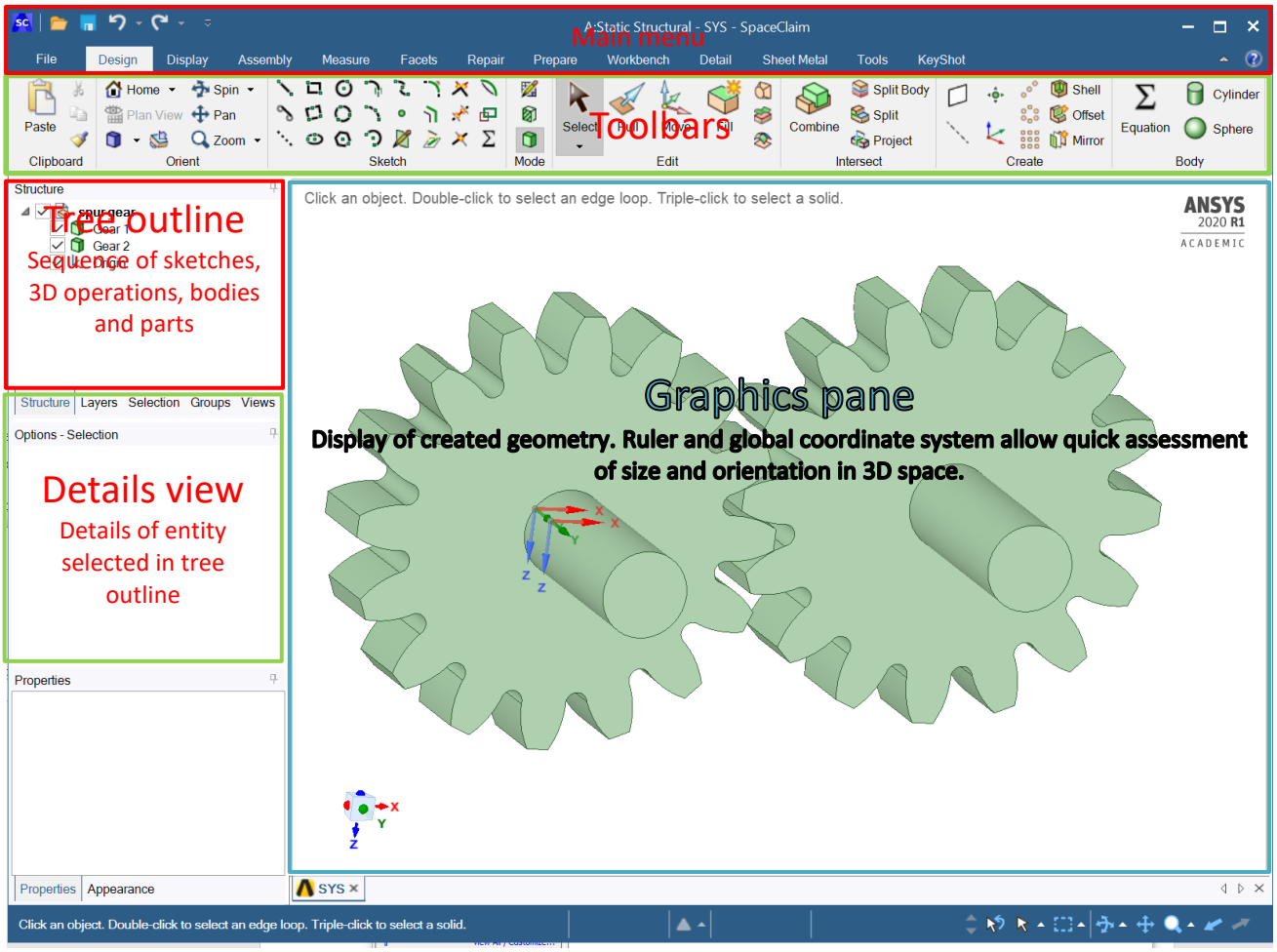


To see the uploaded geometry double-click on the Geometry.



Static Structural

It should appears below.

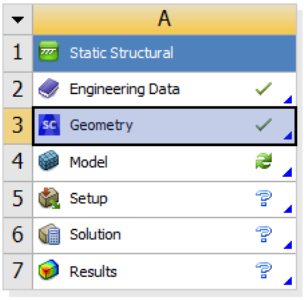


You can familiarise of the geometry views by using the computer mouse.

Function	keys	abbreviation
rotate	Middle mouse button	MMB
pan	Ctrl + Middle mouse button	Ctrl + MMB
zoom	Scroll wheel	SW

EIT Engineering Institute of Technology.

Return to the project schematic in Ansys Workbench. The green tick in the **Geometry** system cell indicates that the geometry is defined. The next step is to create a finite element model from this geometry. This is discretising the model into finite elements (i.e. meshing), applying loads and boundary conditions.

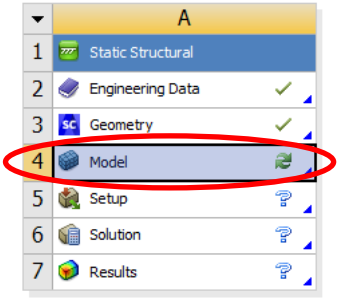


Static Structural

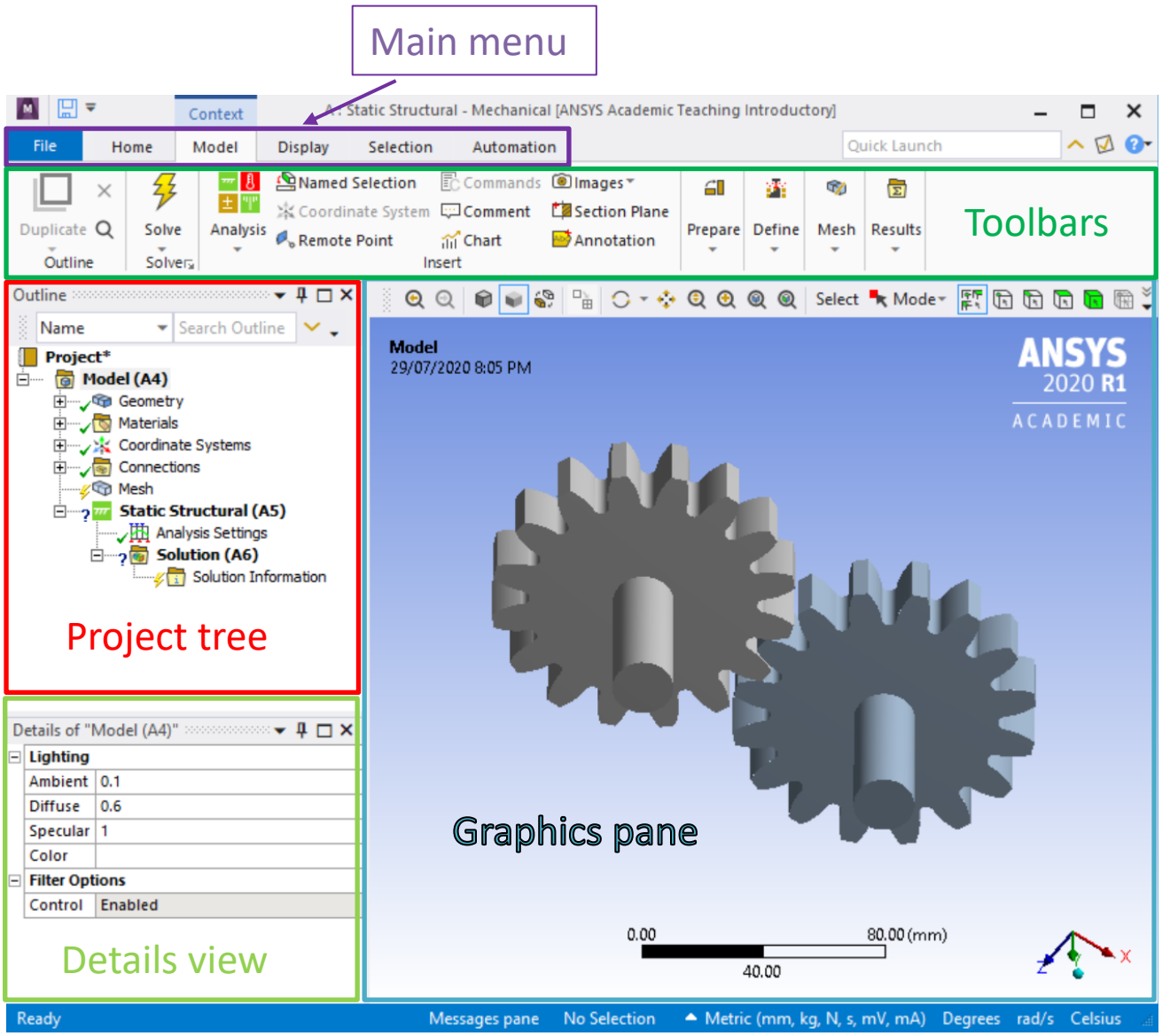
FEA Modelling

Step 5: Creating the FE-model

By double clicking the **Model** system cell, Ansys Mechanical is started. Ansys Mechanical is the modelling environment where we create the FE model and submit the analysis to the solver module. Main menu at the top, Toolbars underneath, on the left hand side we find the project tree and the details view and the main part is the graphics pane with ruler, global coordinate system and the geometry already loaded.

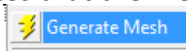


Static Structural

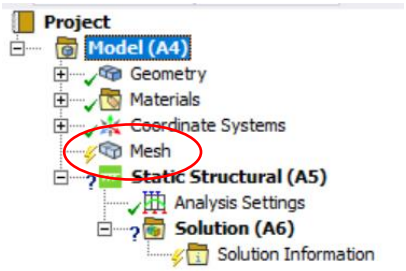


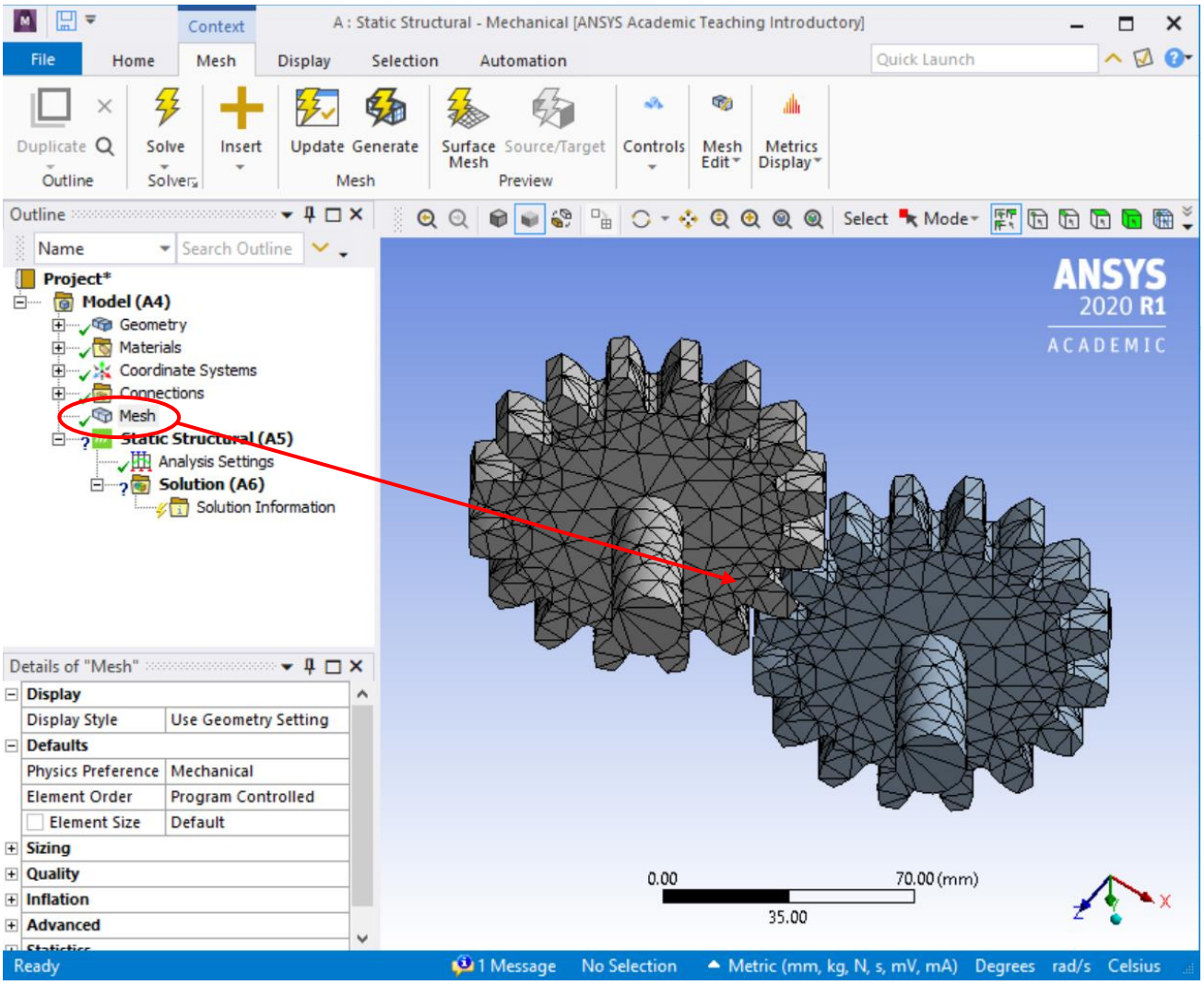
Step 5.1: *Meshing*

In Ansys Mechanical, the workflow is designed in a way that we can follow the project tree from the top to the bottom until all branches are marked with a green tick mark.

The yellow arrow at the **Mesh** branch indicates that the mesh is undefined. We generate the mesh by right clicking the **Mesh** branch and select 

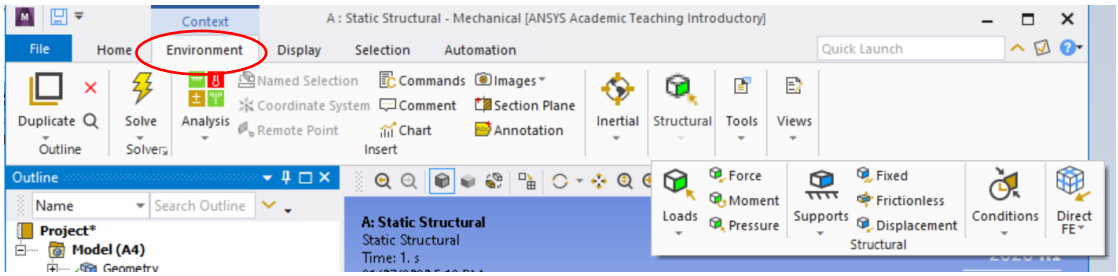
from the popup menu. In later tutorials we will learn more advanced ways of meshing.





Step 5.2: *Loads and boundary conditions*

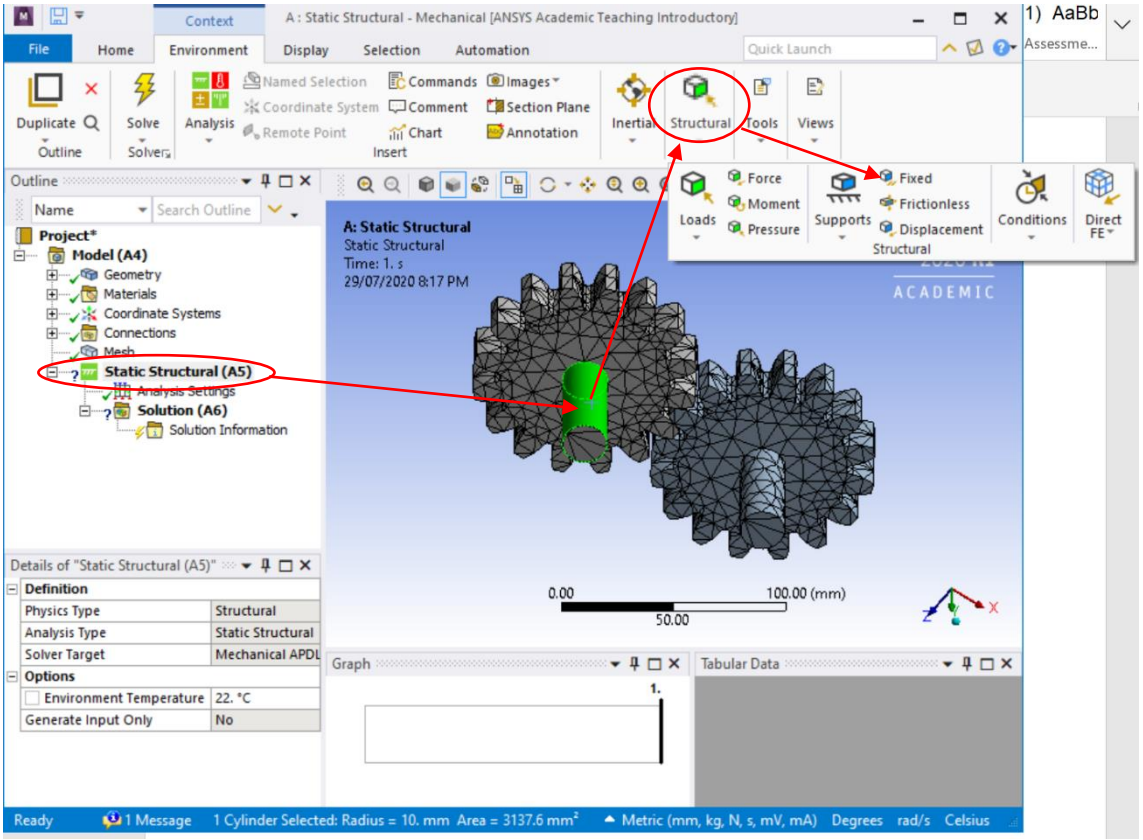
Next loads and boundary conditions have to be applied. As you select the **Static Structural (A5)** branch in the Project tree, the **Environment toolbox** appears.



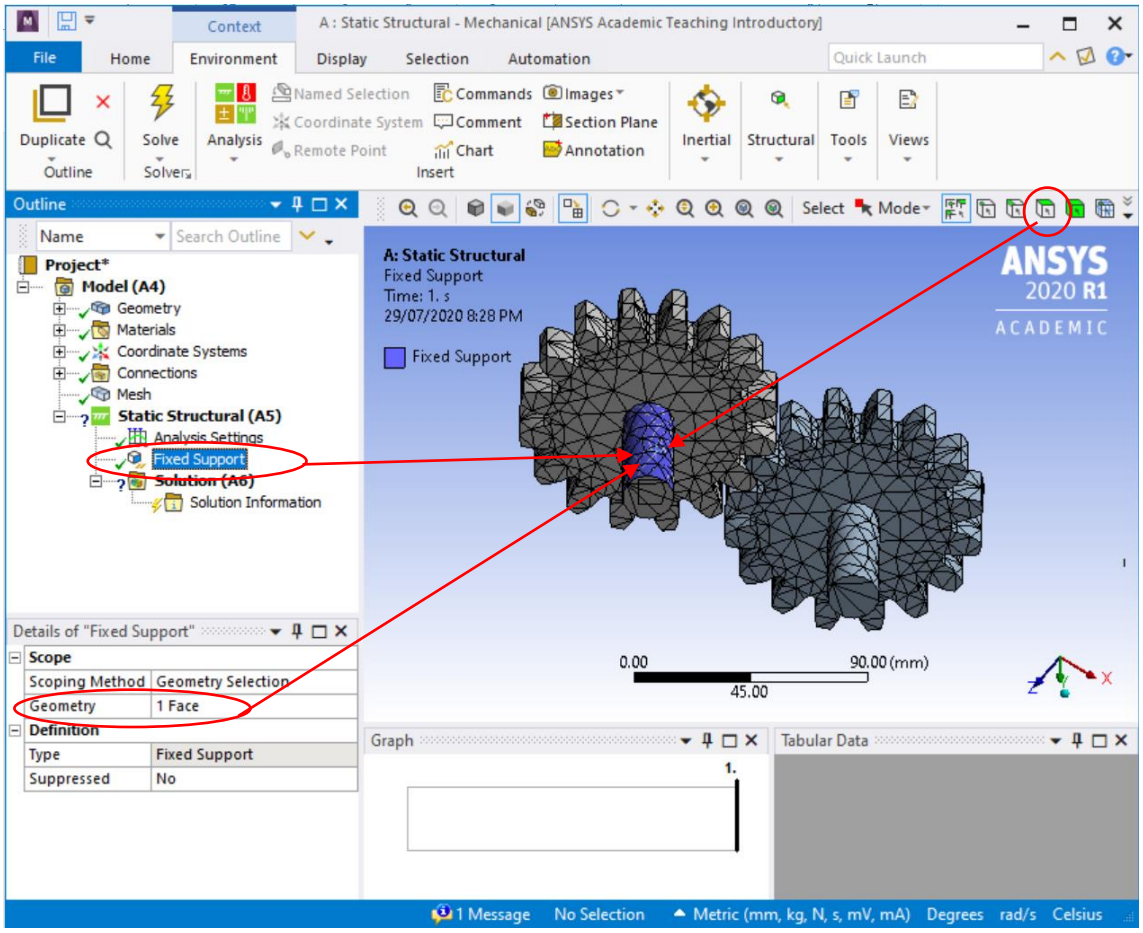
Take a moment to click through the toolboxes in the **Environment toolbox**. We want to apply a fixed support on one gear shaft and a moment (rotation) at the other gear shaft.

To fix one gear shaft , select a face of the one gear shaft in the graphics pane, and then select Structural>Supports > fixed (next to supports). If you cannot select a face, select the face toggle

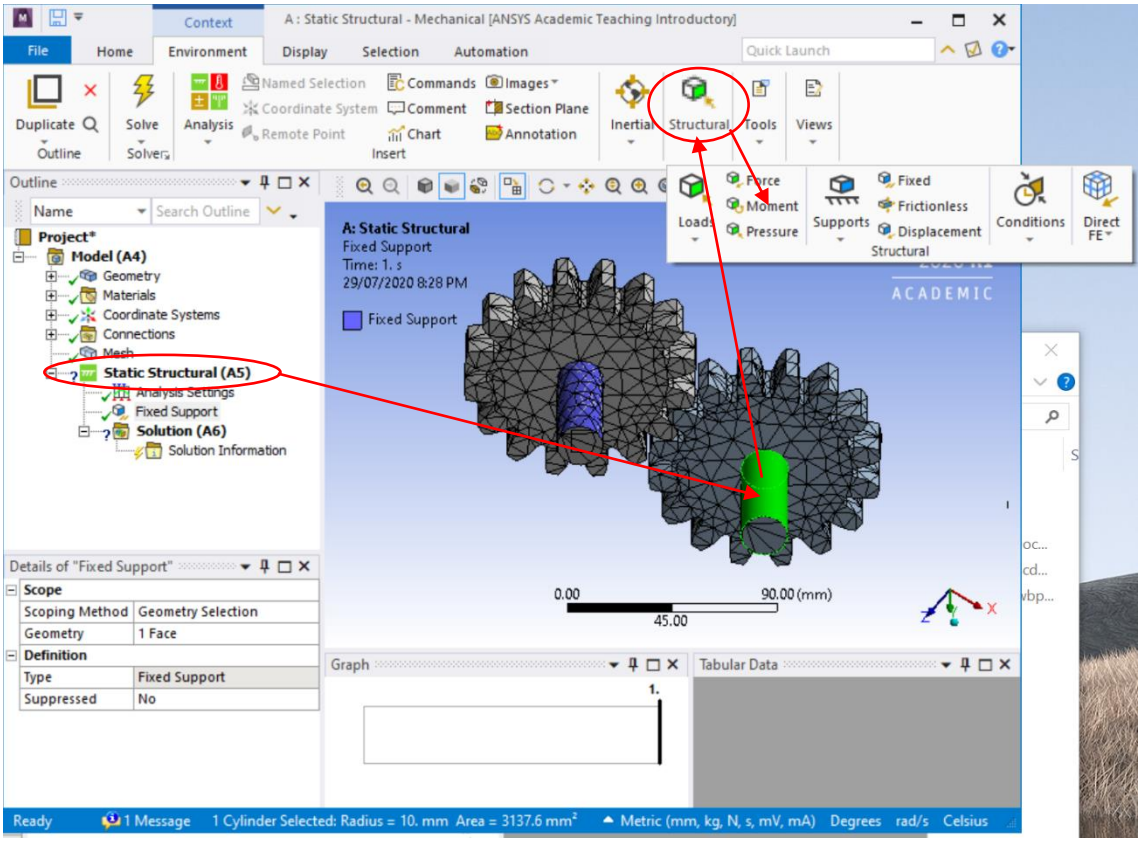
at the top right hand corner of the graphics pan.

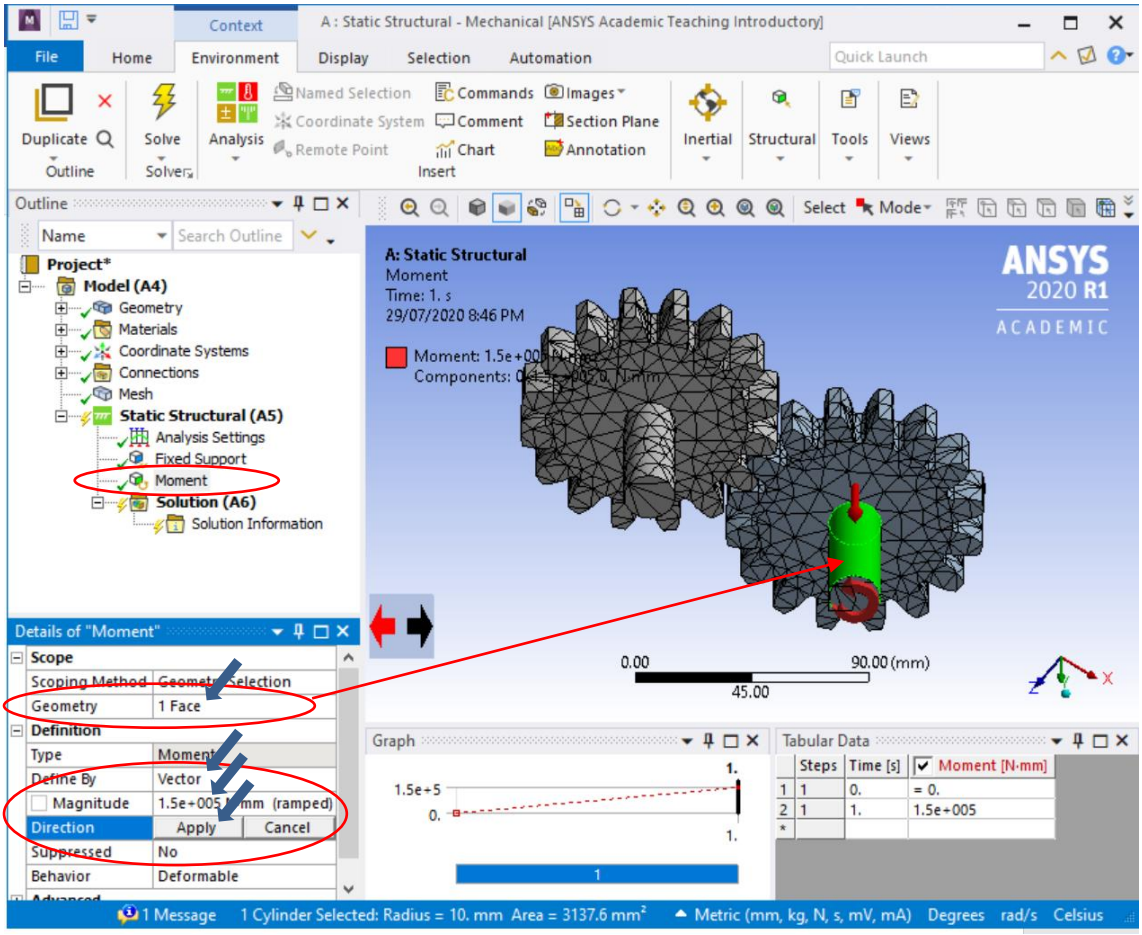


Engineering Institute of Technology.



Similarly, select the other end in the graphics pane, and then select 'Structural' in the **Environment Toolbox**>Structural>Loads>Moment (next to Force). In the Details view we are asked to specify moment value and direction. Define the moment by vector (not as a components), then specify the magnitude of the moment is 150,000 N-mm and apply the direction.

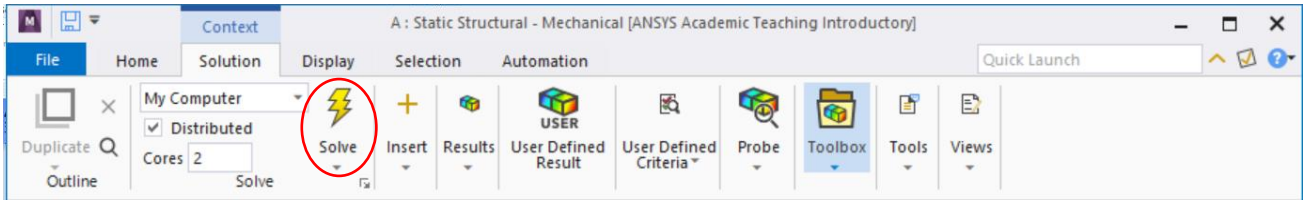




EIT Engineering Institute of Technology.

Step 6: Requesting the solution output

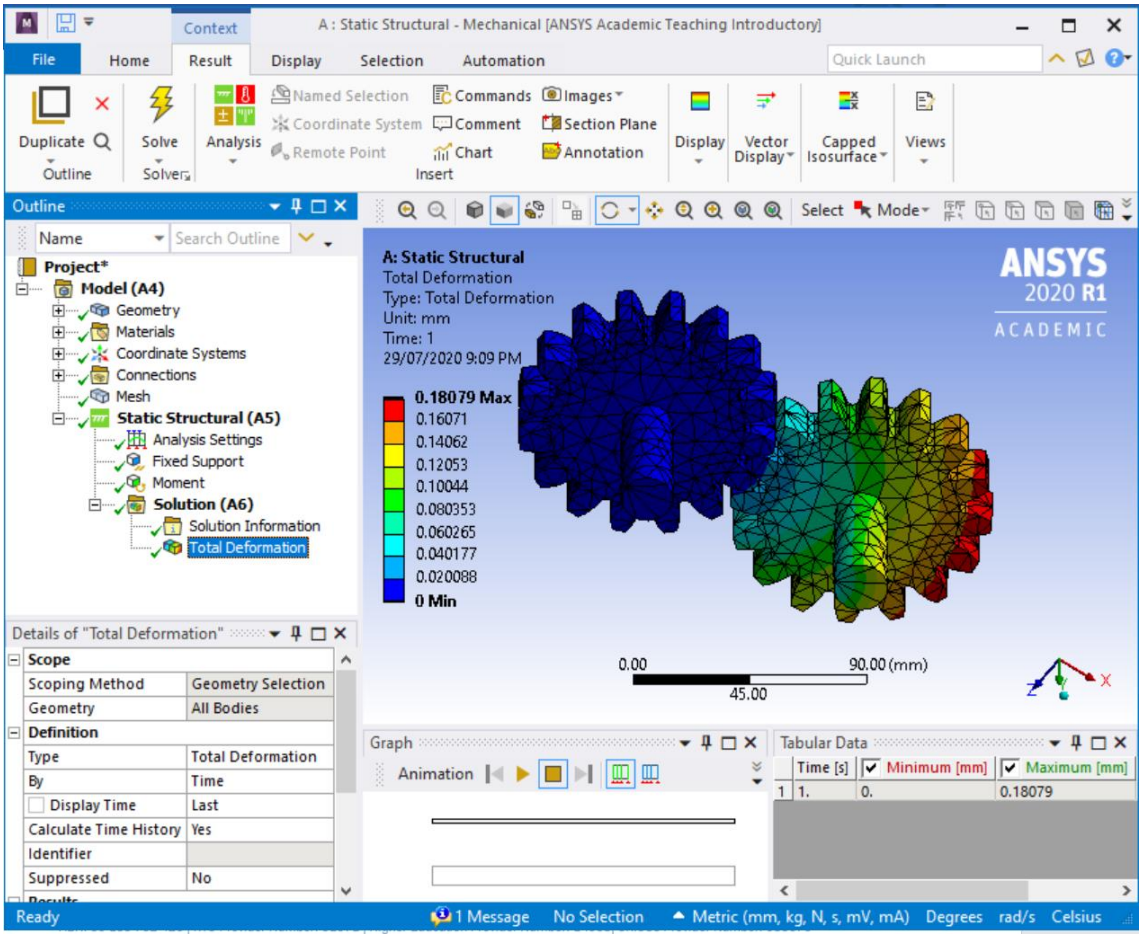
By selecting the **Solution (A6)** branch in the Project tree, you will see the **Solution** toolbox appearing. Take a moment again and have a browse through the different sub-toolboxes in the Solution toolbox.



In out Analysis we want to assess the **total deformation**, the **Equivalent (von Mises) strain** and the **Equivalent (von Mises) stress**. To run the simulation hit the Solve button to submit the Analysis to the solver.

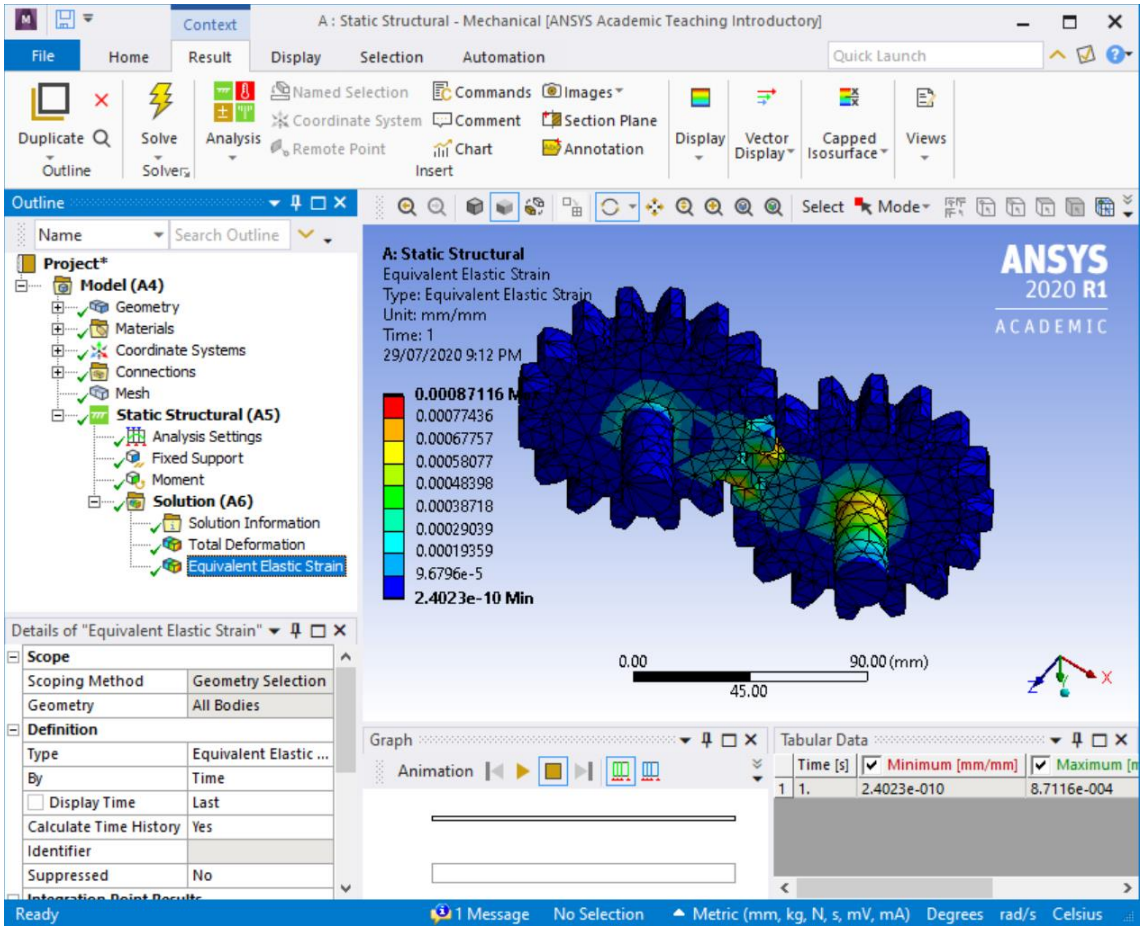
Step 7: Post Processing

To see the result, select Solution (A6)>Results>Deformation>Total. Then right click on the Total Deformation in the Project Tree (in the Solution (A6) branch) and then click “Evaluate All Results”.

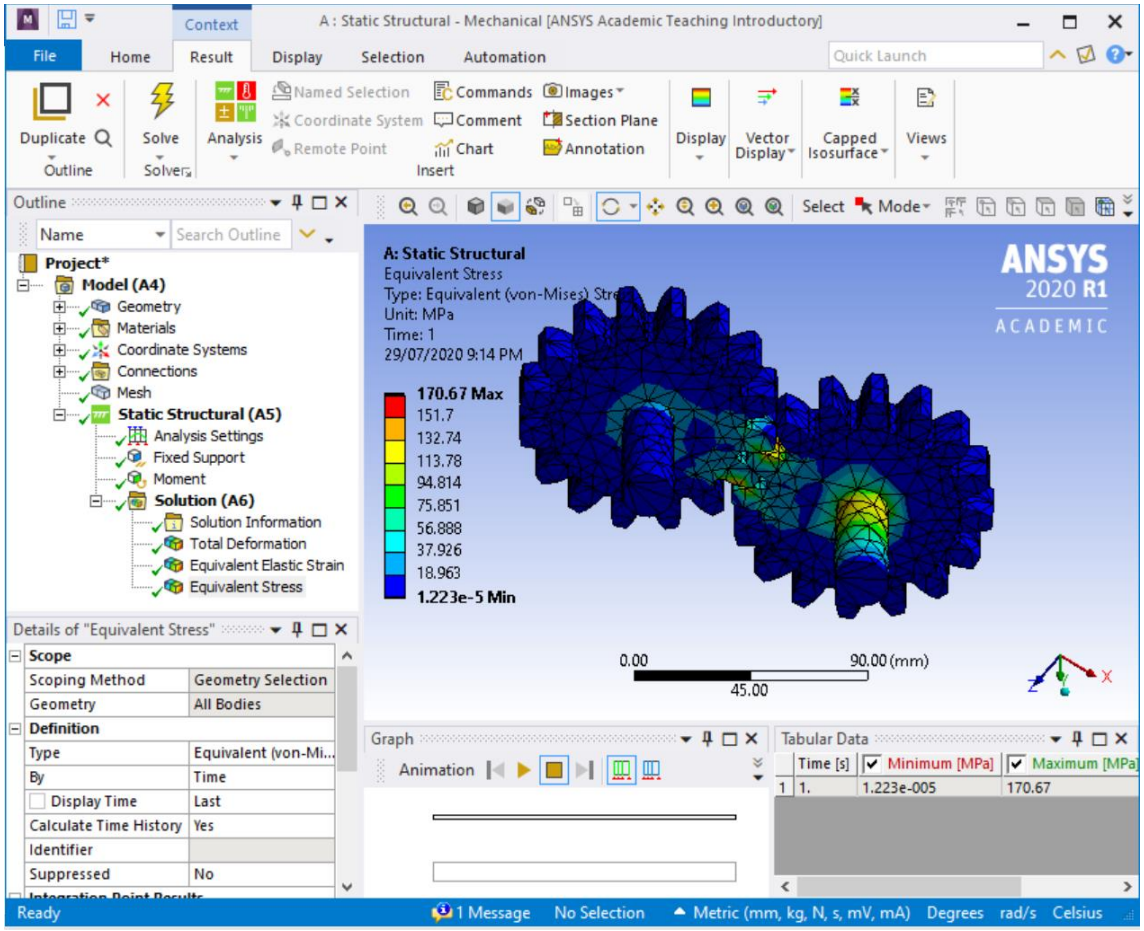


Congrats! You have your first finite element results.
Ctrl+s to save the project.

Similarly, to see the results of Strain, select Solution>Results>Strain>Equivalent (von Mises). Then right click on the Equivalent Elastic Strain in the Project Tree (in the Solution (A6) branch) and then click “Evaluate All Results”.



For Stress calculation, select Solution>Results>Stress>Equivalent (von Mises). Then right click on the Equivalent Stress in the Project Tree (in the Solution (A6) branch) and then click “Evaluate All Results”.


A : Static Structural - Mechanical [ANSYS Academic Teaching Introductory]
Context
File
Home
Result
Display
Selection
Automation
Quick Launch
Duplicate
Outline
Solve
Solvers
Analysis
Named Selection
Coordinate System
Remote Point
Commands
Comment
Chart
Images
Section Plane
Annotation
Insert
Display
Vector Display
Capped Isosurface
Views
Outline
Name
Search Outline
Project*
Model (A4)
Geometry
Materials
Coordinate Systems
Connections
Mesh
Static Structural (A5)
Analysis Settings
Fixed Support
Moment
Solution (A6)
Solution Information
Total Deformation
Equivalent Elastic Strain
Equivalent Stress
Select
Mode
A: Static Structural
Equivalent Stress
Type: Equivalent (von-Mises) Stre
Unit: MPa
Time: 1
29/07/2020 9:14 PM
170.67 Max
151.7
132.74
113.78
94.814
75.851
56.888
37.926
18.963
1.223e-5 Min
ANSYS
2020 R1
ACADEMIC
0.00
45.00
90.00 (mm)
Details of "Equivalent Stress"
Scope
Scoping Method
Geometry Selection
Geometry
All Bodies
Definition
Type
Equivalent (von-Mi...
By
Time
Display Time
Last
Calculate Time History
Yes
Identifier
Suppressed
No
Graph
Animation
Tabular Data
Time [s]
Minimum [MPa]
Maximum [MPa]
1
1.
1.223e-005
170.67
Ready
1 Message
No Selection
Metric (mm, kg, N, s, mV, mA)
Degrees
rad/s
Celsius