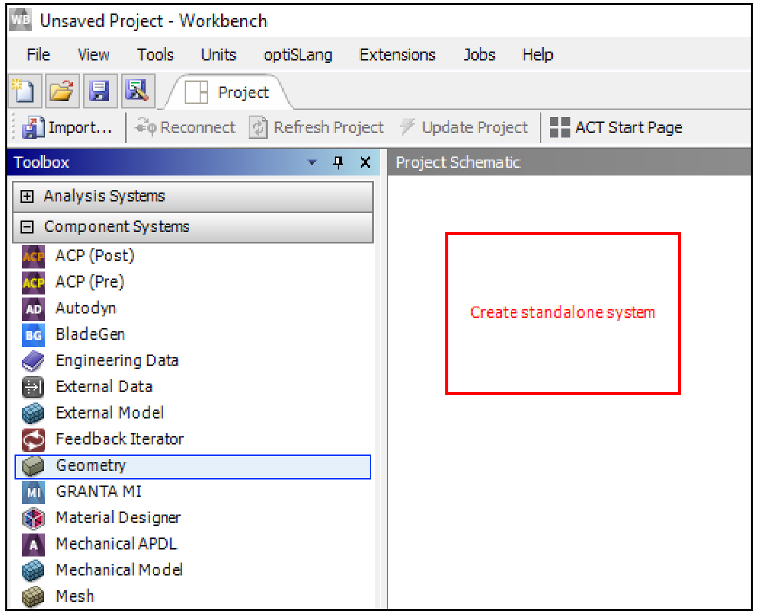
Workshop 05.1: Electromagnetic-Mechanical Coupling

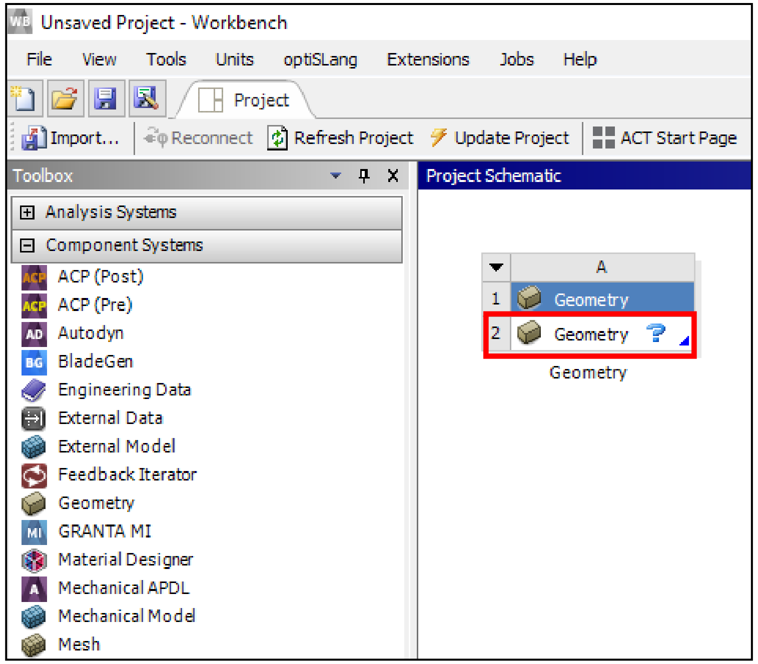
Overview

- **Electromagnetic – Mechanical Coupling**
  - This workshop will discuss how to use the Ansys Maxwell inside Ansys Workbench to study mechanical deformation due to electromagnetic forces
  - The following tasks will be performed:
    - The Geometry of a simple helical Coil will be imported in Design Modeler
    - The Geometry will be shared to a Maxwell 3D Design
    - An Electromagnetic Magnetostatic analysis will be carried out
    - The Maxwell Solutions will be sent to a structural design
    - The structural analysis will be realized
    - The deformation will be shown

Project Startup

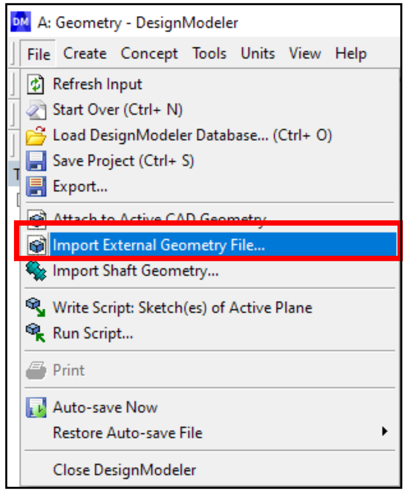
- Create the Project
  - Start Workbench
  - *Start → All Programs → ANSYS 2020 R1 → Workbench 2020 R1*
  - Expand Component Systems and drag and drop a Geometry component system into the Project Schematic
  - Double click on the Geometry cell (A2) to start Design Modeler

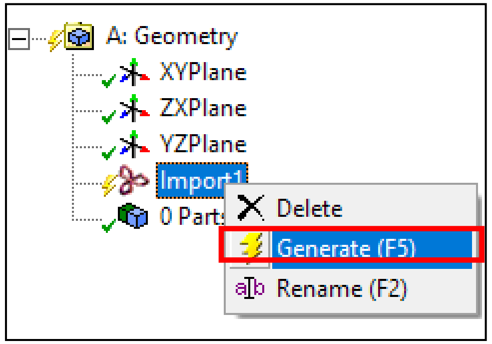


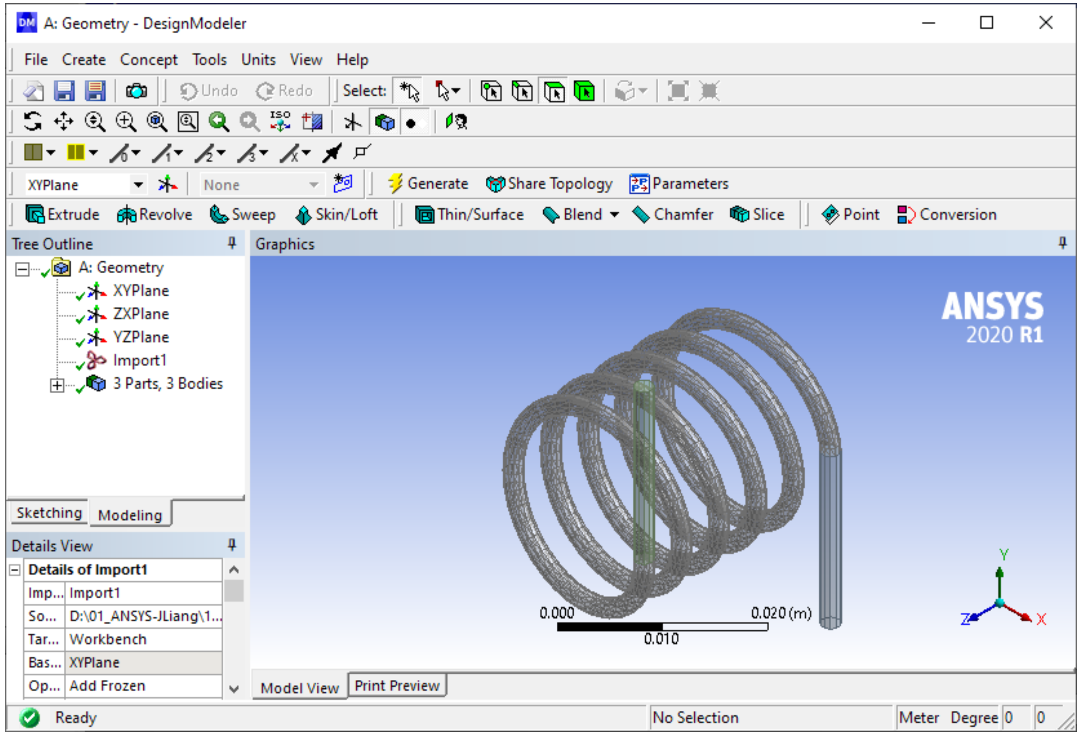


Import

- **Import a Step file of Coil Model**
  - *[Menu] → File → Import External Geometry File...*
  - Select "Coil.stp" and import it with Default settings
  - RMB on "Import1" and press "Generate (F5)"

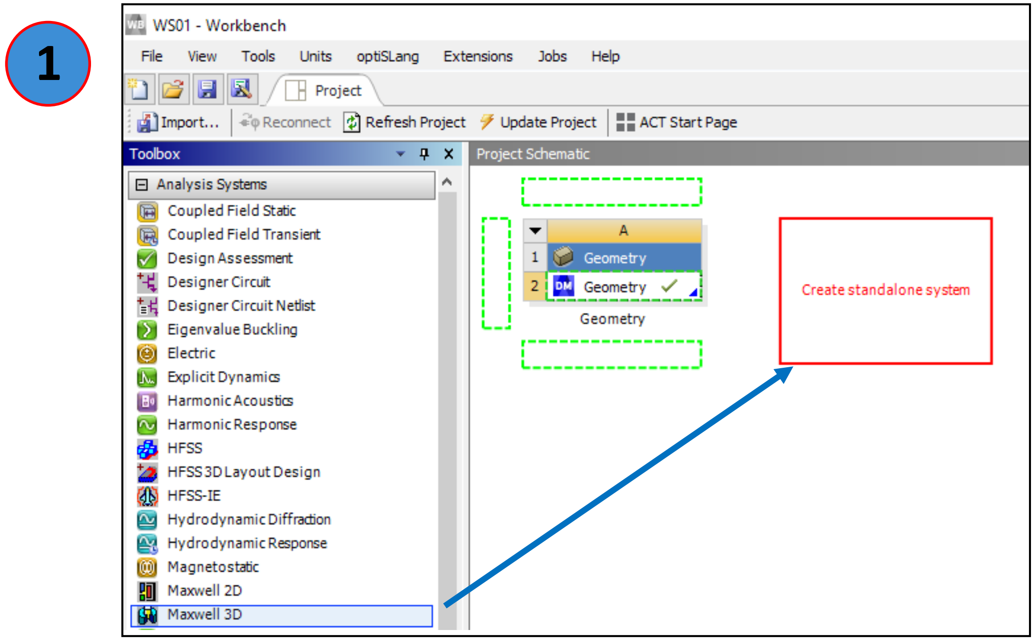


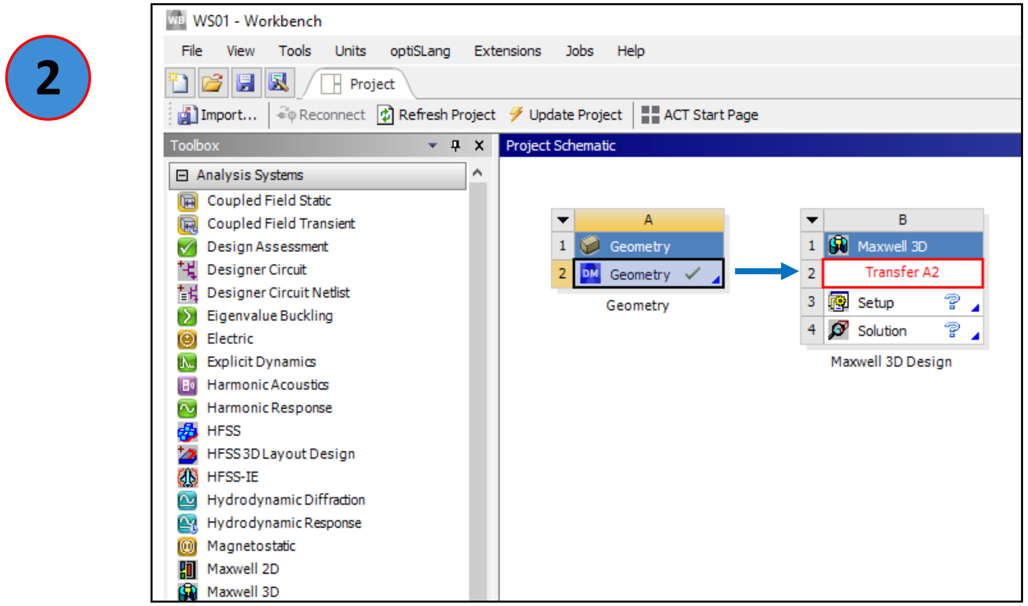


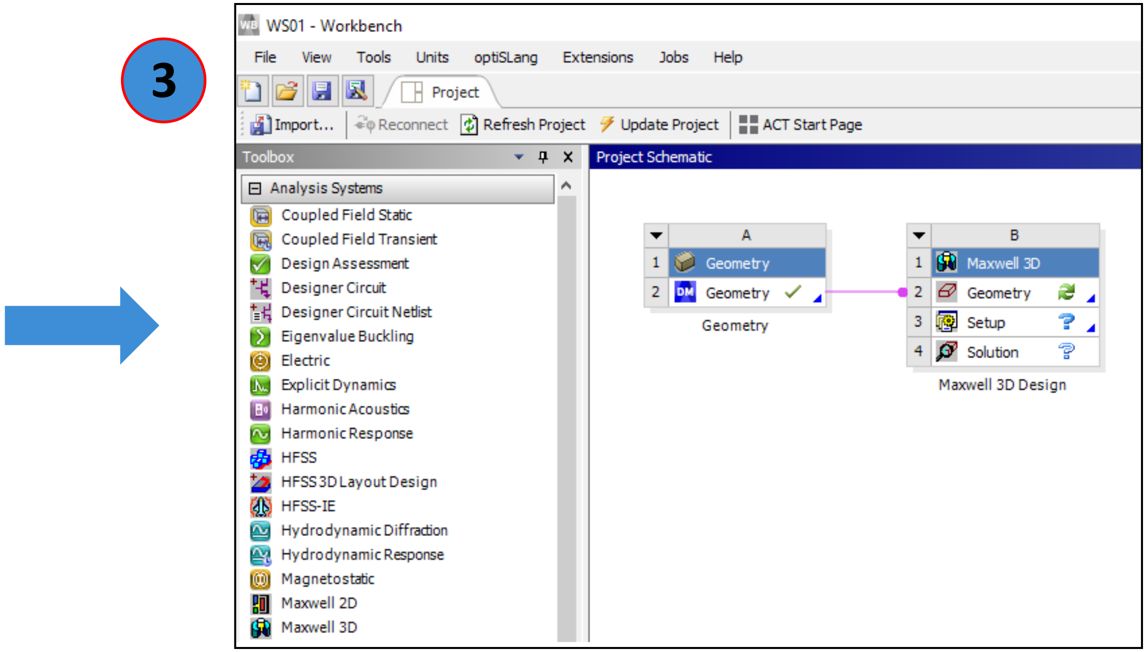


Insert Maxwell Analysis

- **Select Maxwell 3D**
  - **drag and drop a Maxwell 3D analysis system into the Project Schematic**
  - **Select the Geometry cell (A2) and drag it on the Maxwell cell B2**

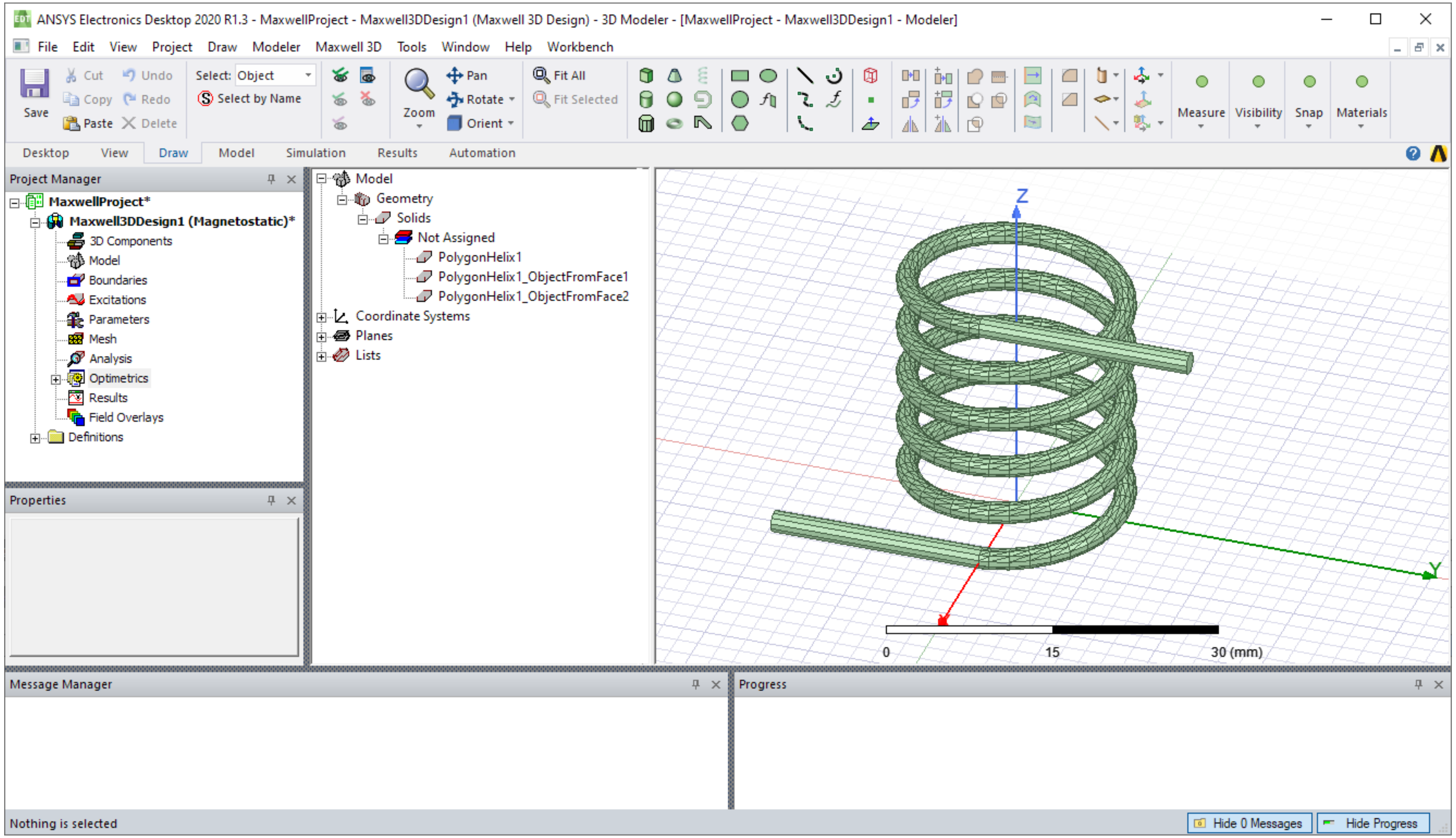






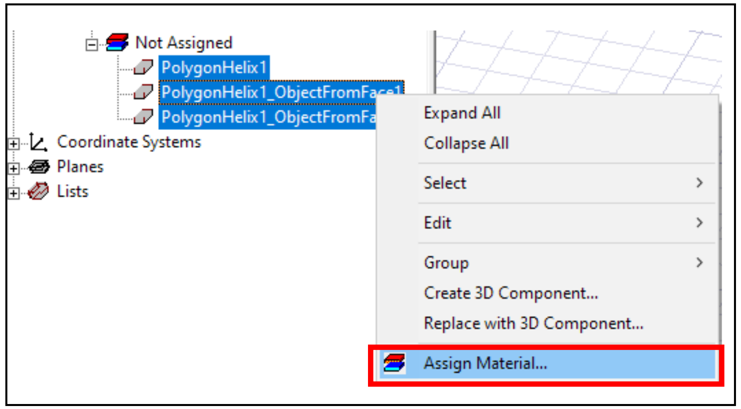
Open Maxwell

- Double click on Maxwell cell B2 to open Maxwell 2020 R1
- The Geometry is automatically imported

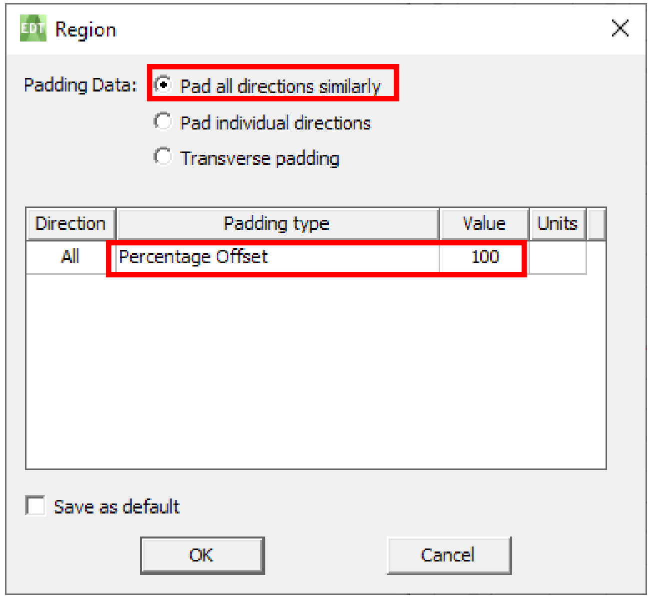


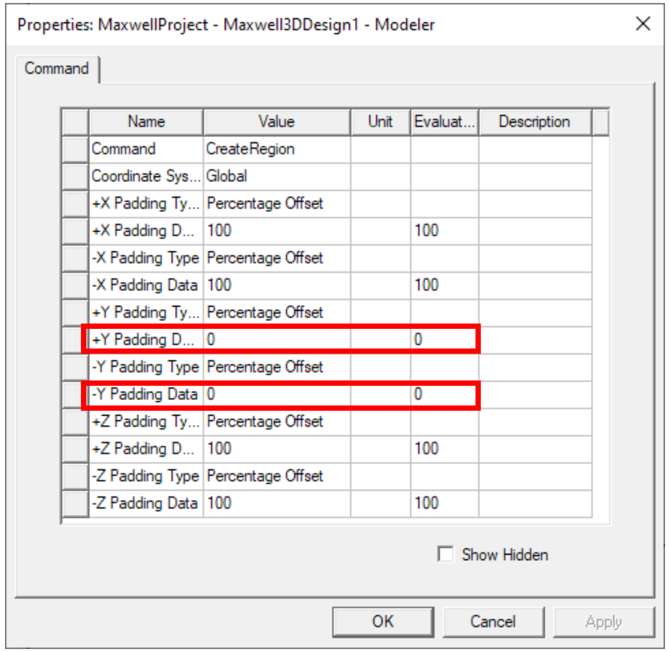
Setup Problem

- **Assign Material**
  - With Ctrl key pressed select the three objects
  - *RMB → Assign Material → Copper*



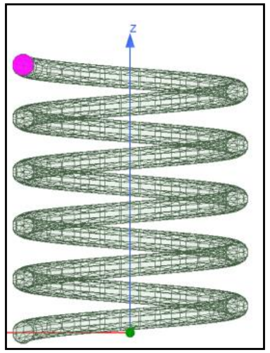
- **Create Simulation Region**
  - Select the menu item *Draw → Region* or click on the icon 
    - Pad all directions similarly: ☑ Checked
    - Padding Type: Percentage Offset
    - Value: 100
    - Press OK
  - Double click on Create Region
    - +Y Padding Data: 0
    - -Y Padding Data: 0

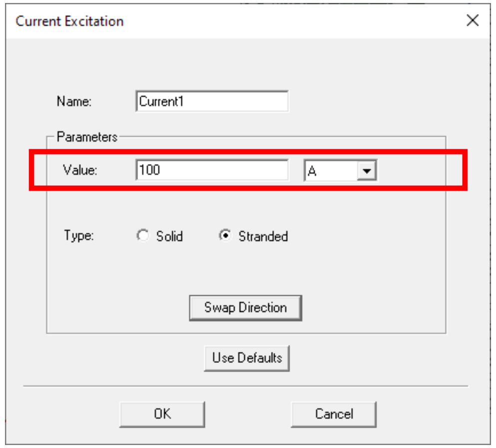


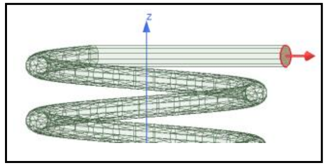


Setup Problem

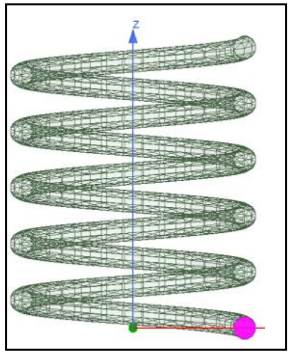
- **Assign Excitations**
  - Press the F button to select Faces
  - Select one terminal section of the coil
  - *RMB → Assign Excitation → Current*
    - Value: 100 A
    - Stranded: ☑ Checked
    - Swap Direction

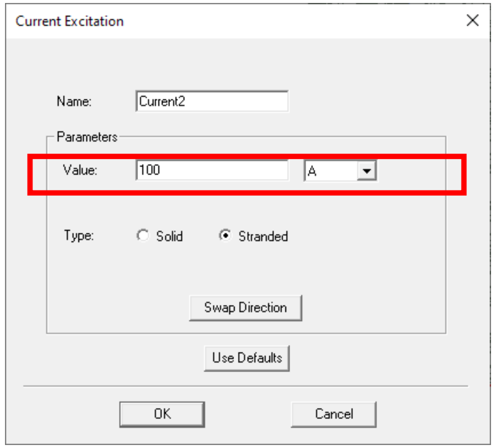


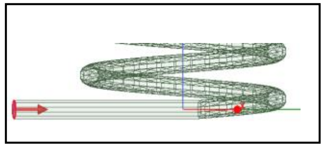




  - Select the second coil terminal section
  - *RMB → Assign Excitation → Current*
    - Value: 100 A
    - Stranded: ☑ Checked

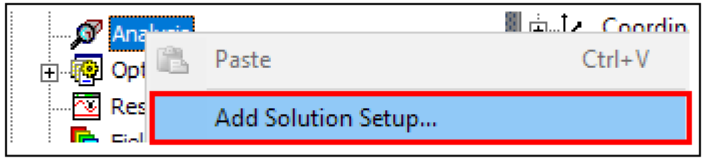


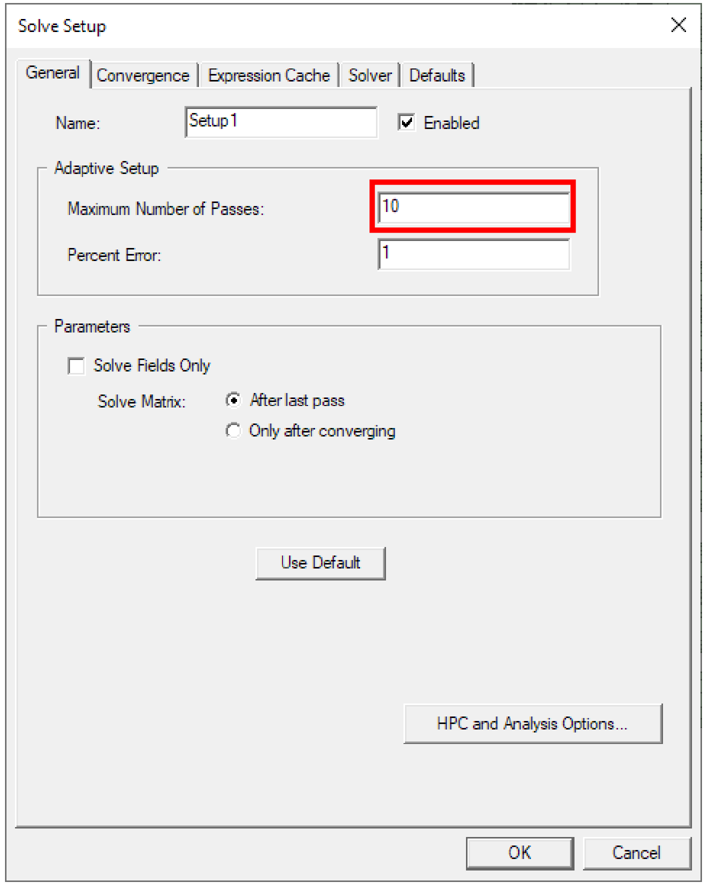


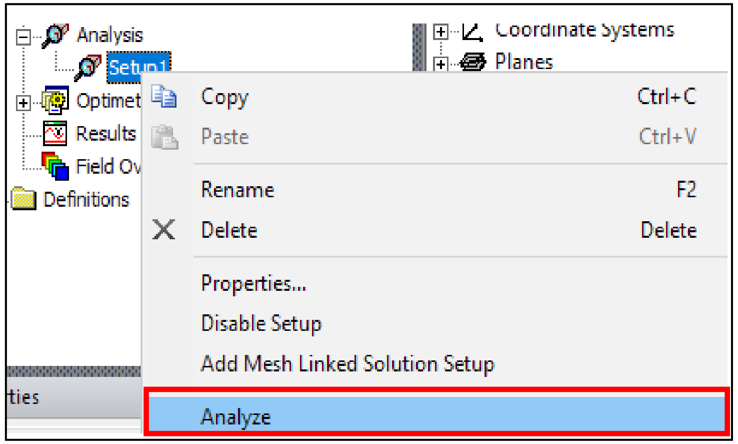


Analyze

- **Create an analysis setup:**
  - *RMB on Analysis → Add Solution Setup*
  - **Solution Setup Window:**
    - **General Tab**
    - **Maximum Number of Passes: 10**
    - **Press OK**
- **Start the solution process:**
  - *RMB on Setup1 → Analyze*
- **Save and close the Electronics Desktop**





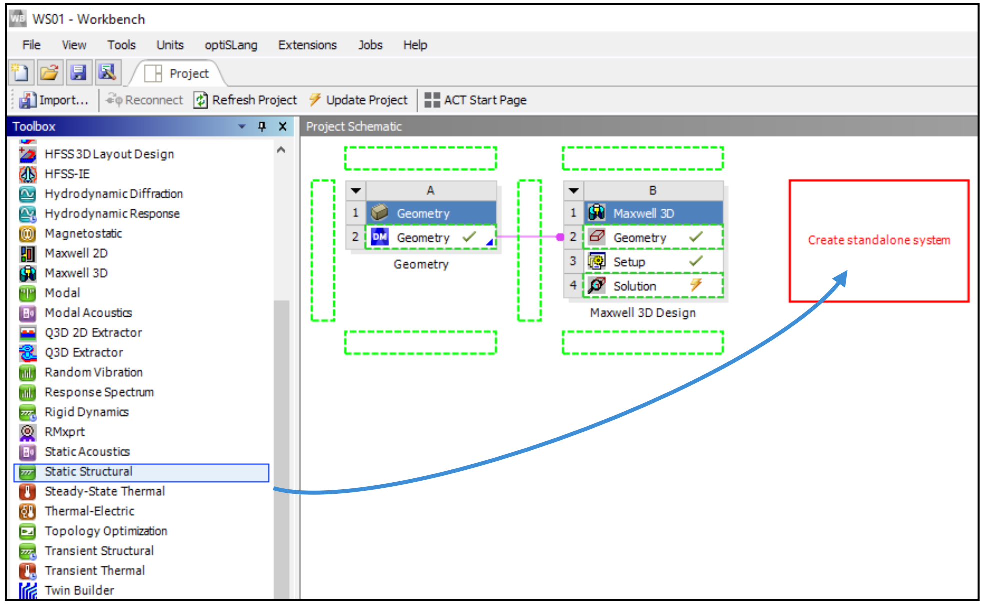


Ansys

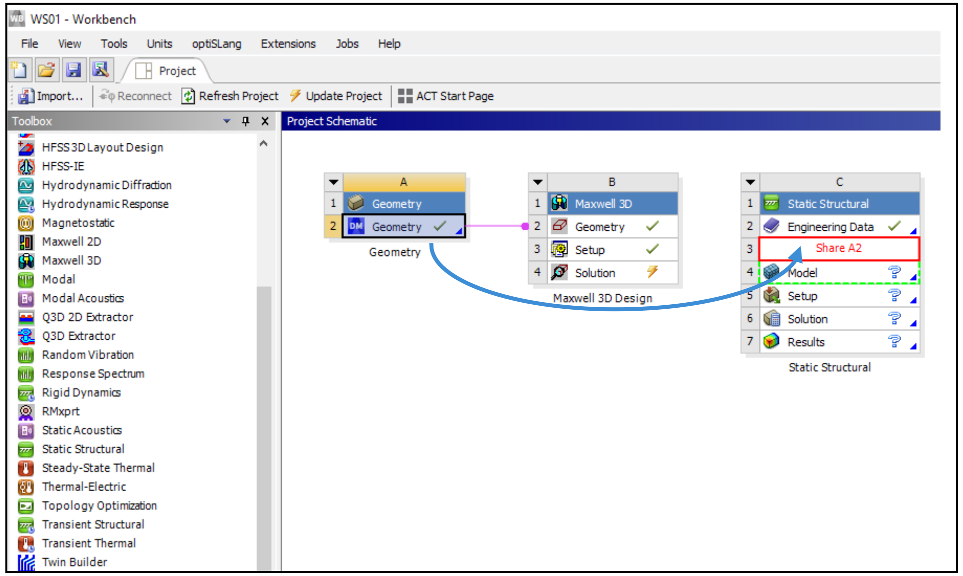
Insert Static Structural Analysis

- **Select Static Structural**
  - drag and drop a Static Structural analysis system into the Project Schematic
  - Select the Geometry cell (A2) and drag it on the Static Structural cell C3
  - Double click the cell C2 to change material assignment

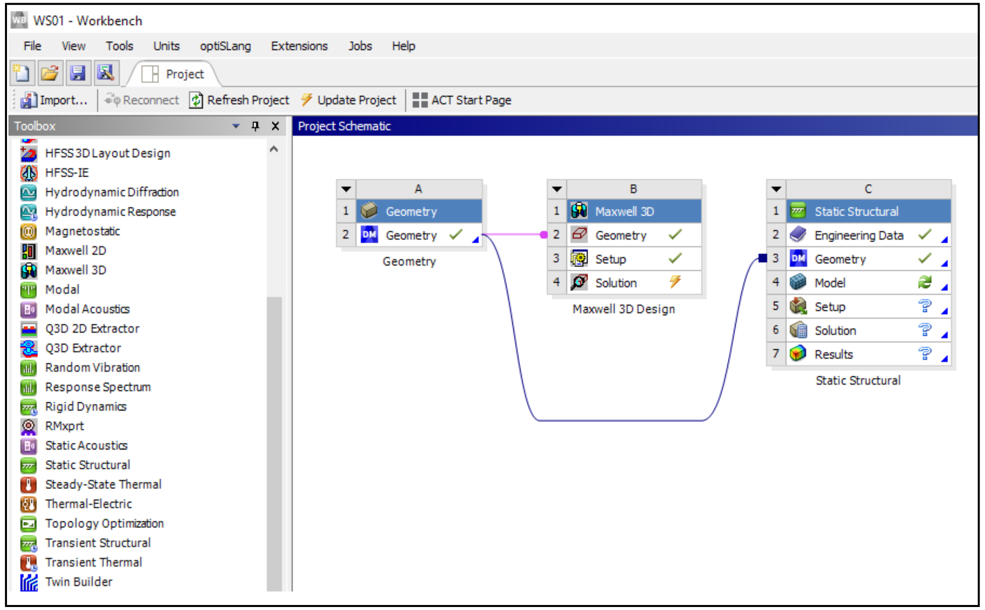
1



2



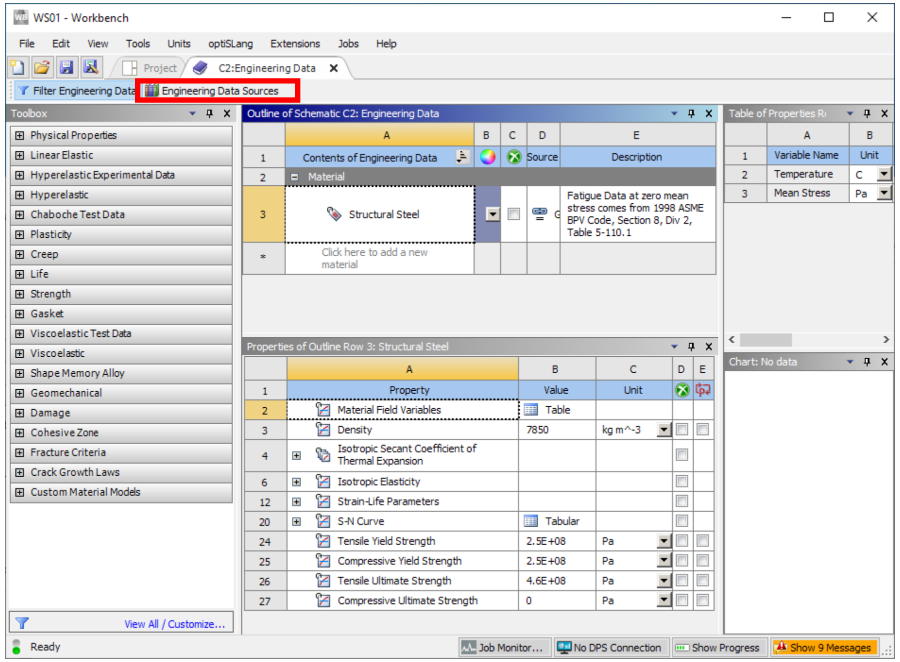
3



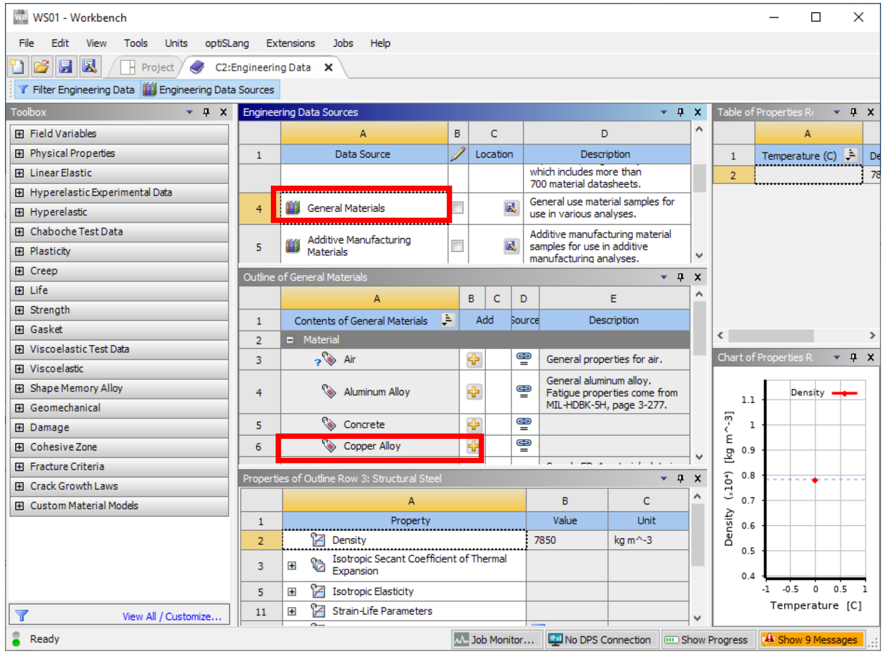
Change material in Ansys Structural

- **Click the Engineering Data Sources button**
  - **Select General Materials library**
  - **Press the "+" symbol near Copper Alloy**
  - **Close the C2:Engineering Data Tab**

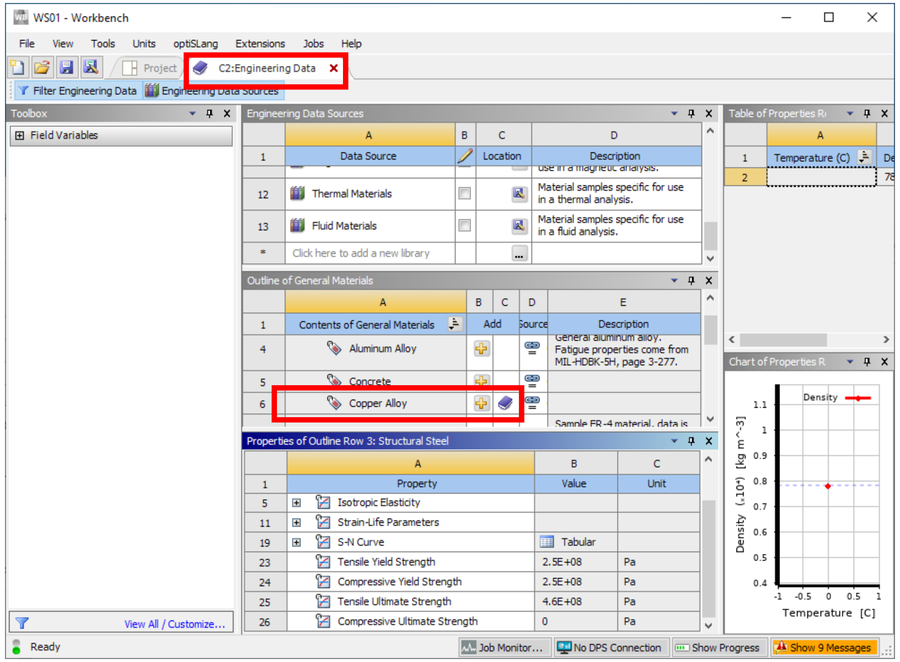
1



2



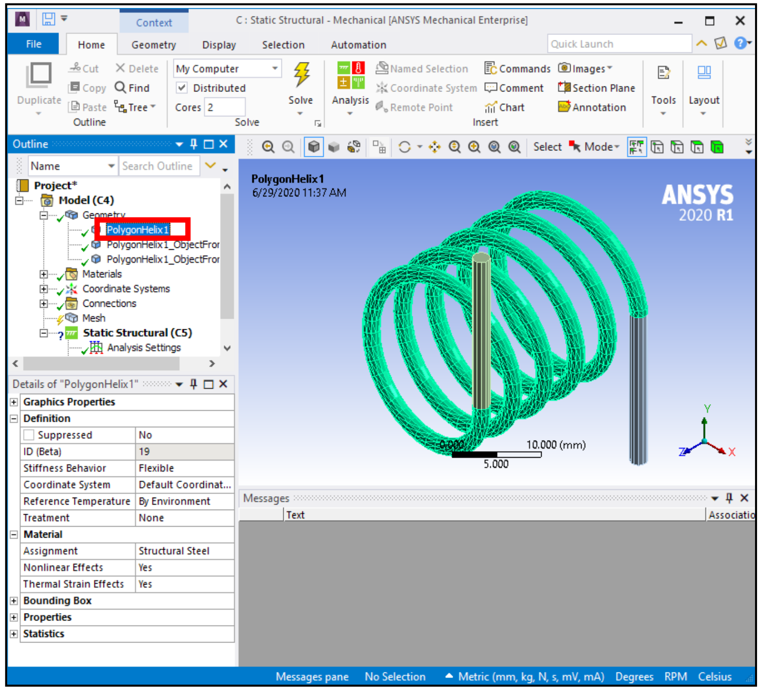
3



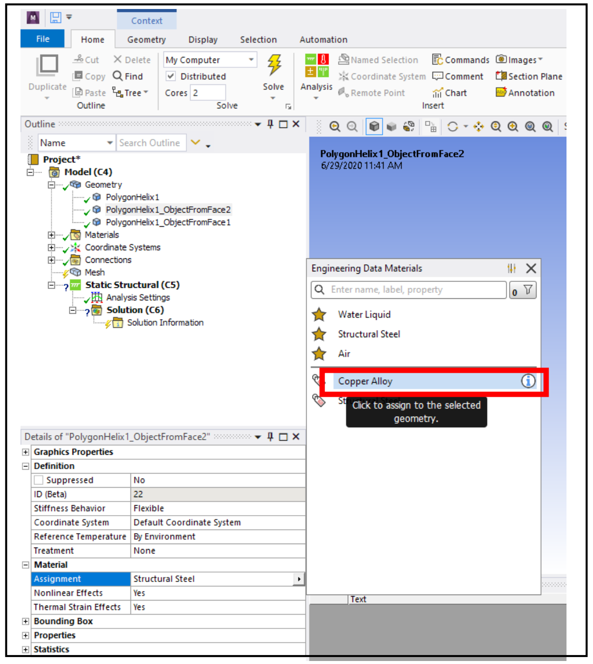
Change material in Ansys Structural

- Double Click on cell C4 to open Ansys Structural
  - Expand Geometry and select the first body “PolygonHelix1”
  - Select *Material → Assignment* and press the small arrow on the right
  - Choose Copper Alloy
  - Repeat the procedure for the other two bodies
  - On the Workbench window, update Cell C4

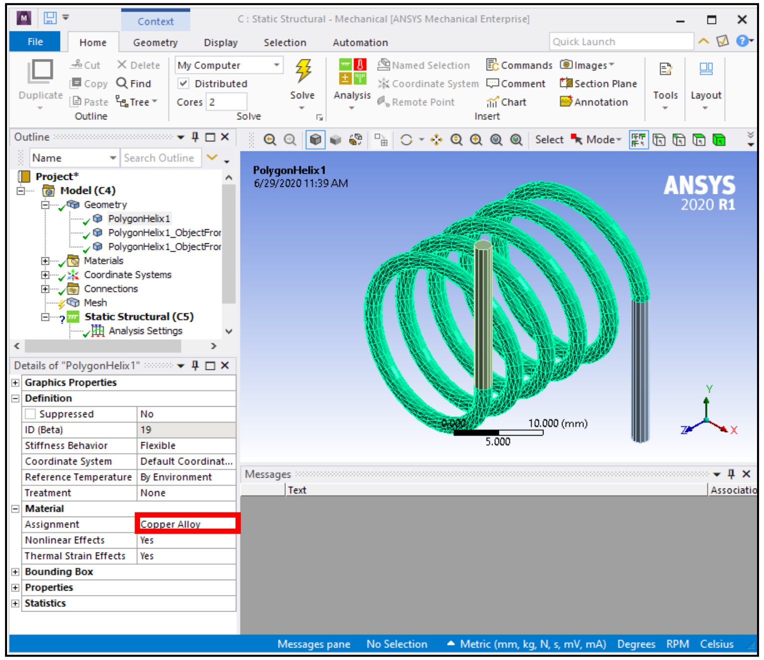
1



2

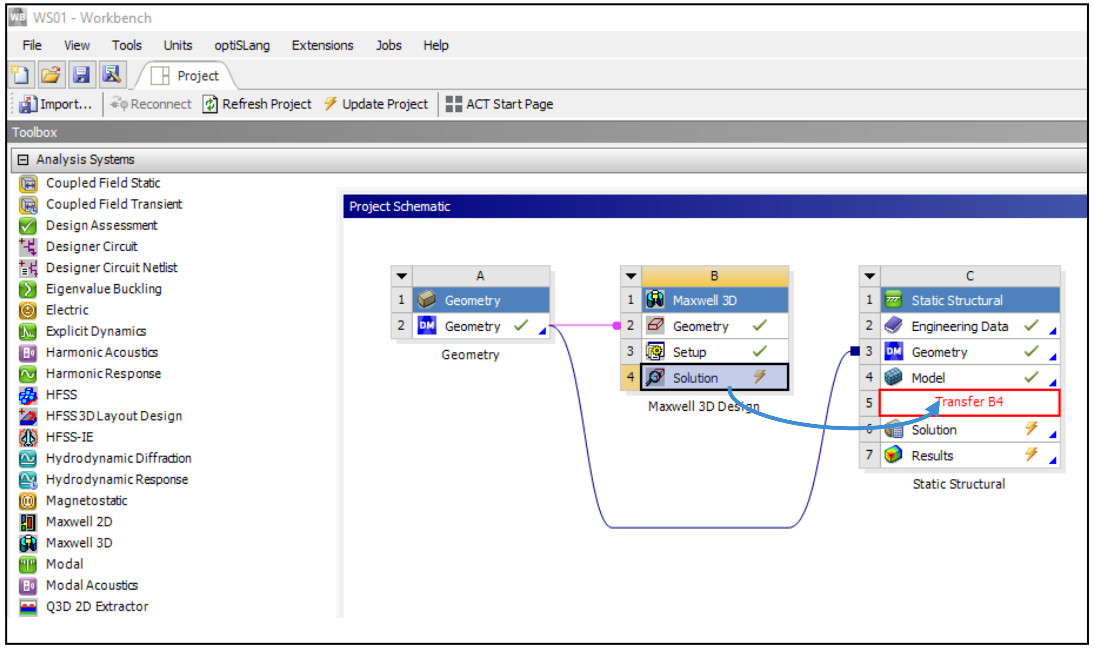


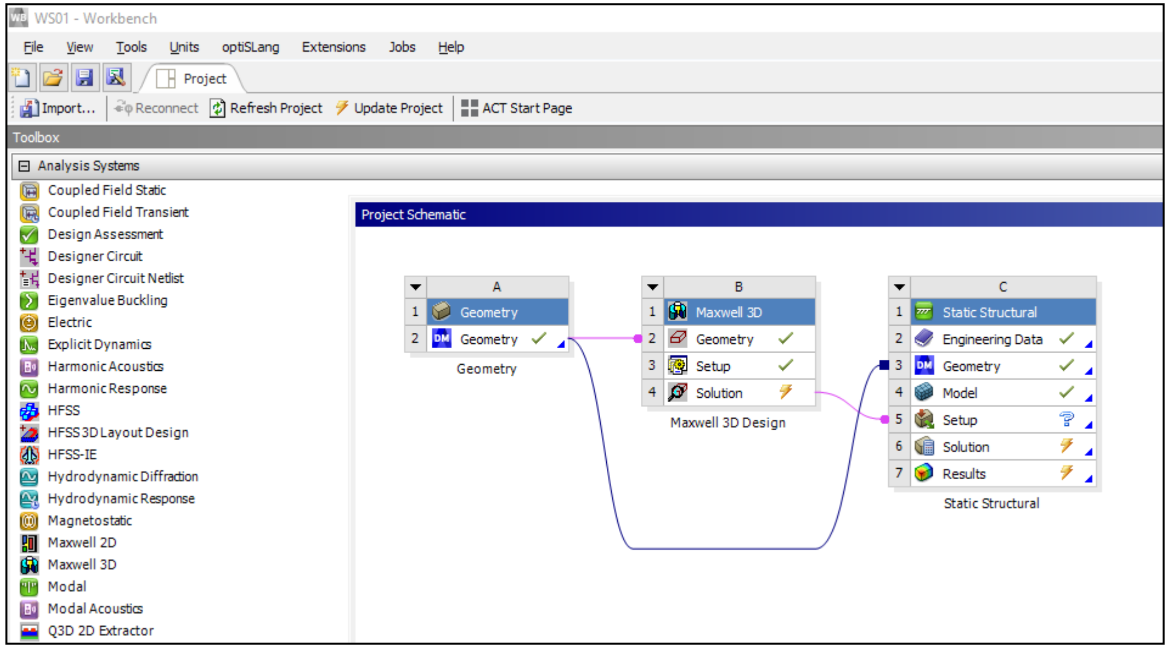
3



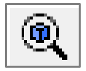
Connect Maxwell and Static Structural Analysis

- **Select Maxwell**
  - Drag and drop cell B4 (Solution) on cell C5 (Setup)
  - Update cell B4 (Maxwell Solution) and refresh cell C5
  - Double click on cell C5 to open Ansys Mechanical

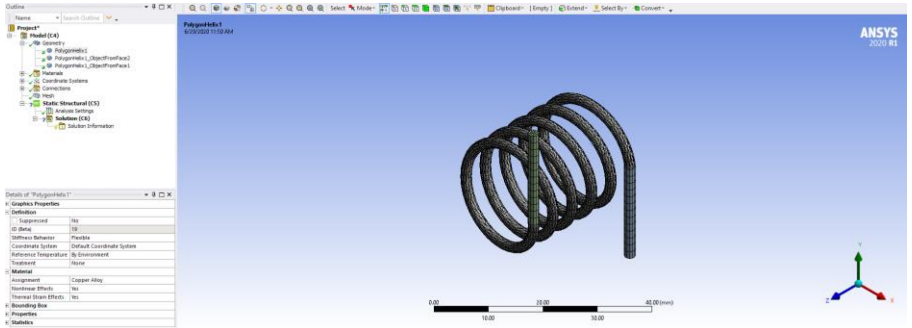




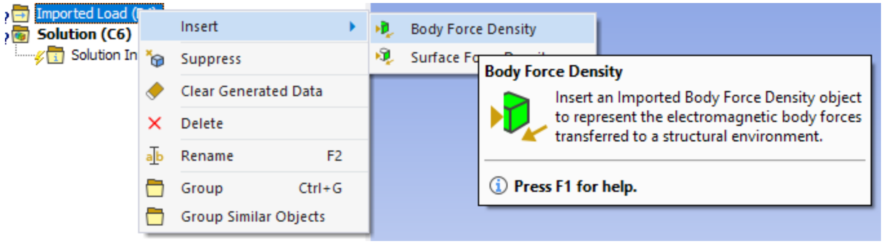
Import the load from Maxwell

- **In Ansys Mechanical**
  - Click F7 or icon  to fit the view
  - *RMB on Imported Load (B4) → Insert → Body Force Density*
  - Select Body Force Density
  - Select the three geometric bodies
  - Under Details select Geometry and Apply
  - *RMB on Body Force Density → Import Load*

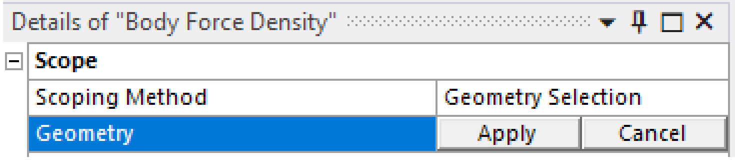
1



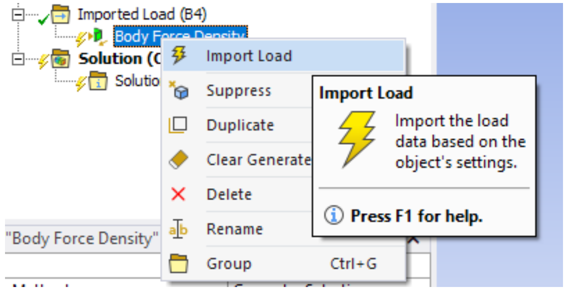
2



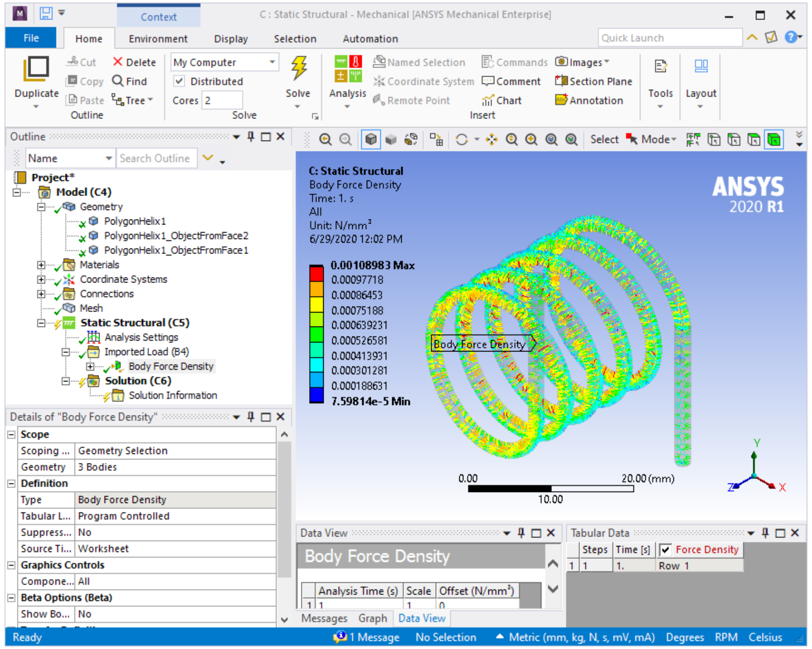
3



4



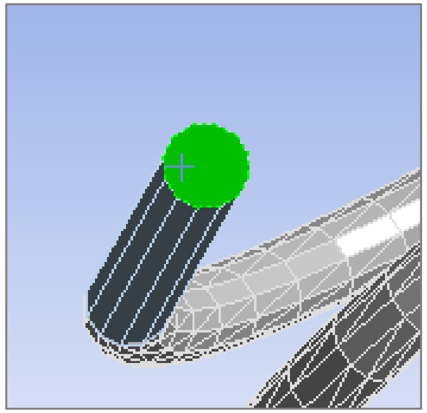
5



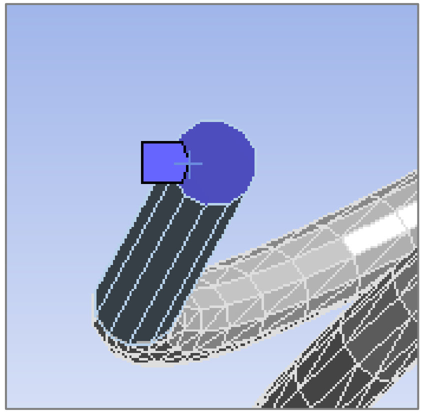
Set Mechanical Constraints

- **In Ansys Mechanical**
  - **Click the icon to select the faces**
  - **Select one coil terminal cross section**
  - ***RMB on Static Structural (C5) → Insert → Fixed Support***
  - **Select one coil terminal cross section**
  - ***RMB on Static Structural (C5) → Insert → Fixed Support***

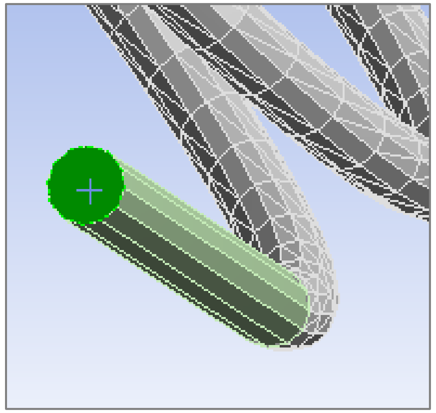
1



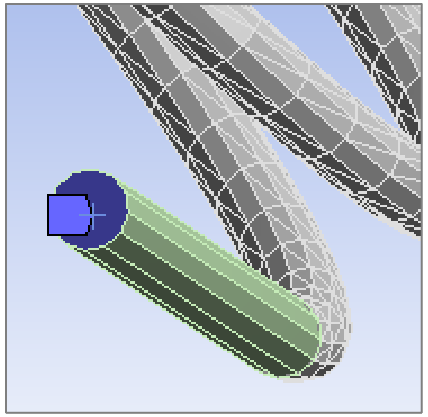
2



3

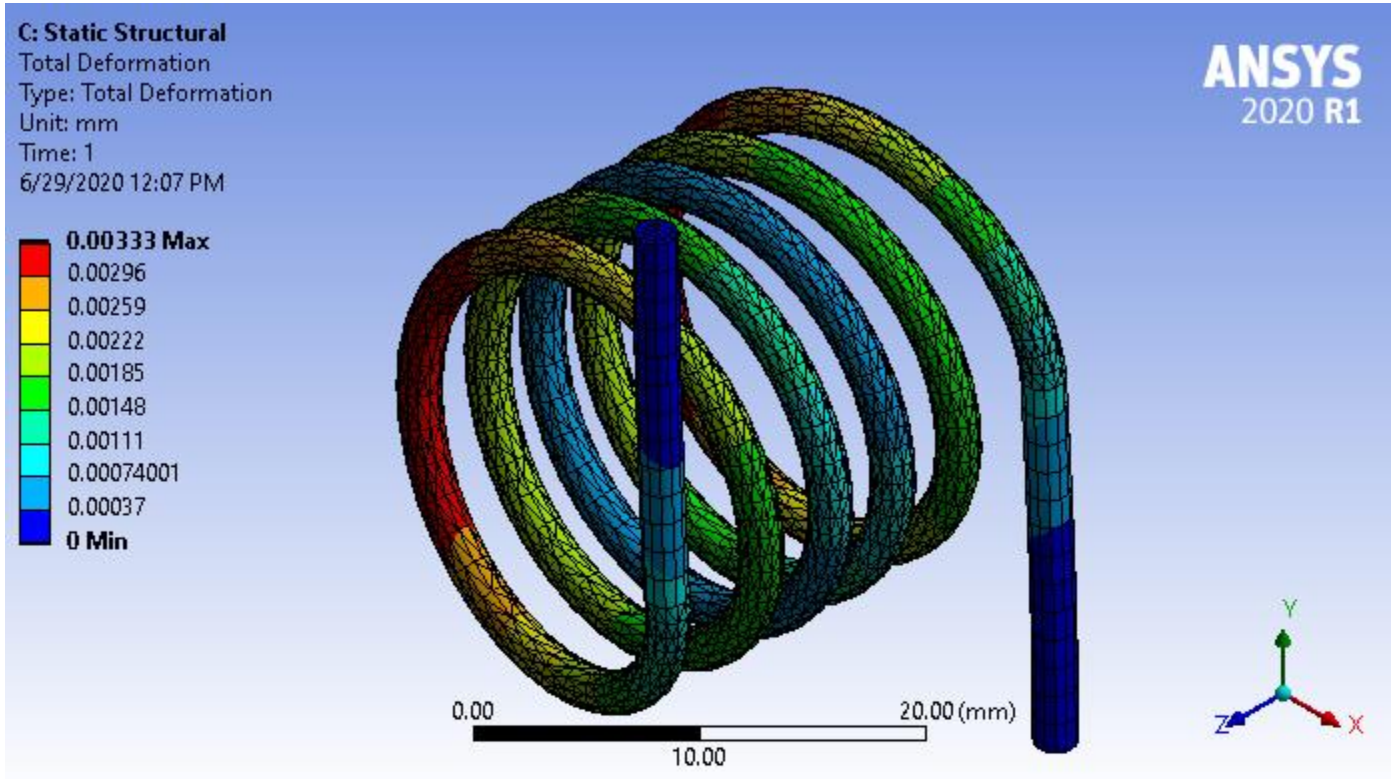


4



Solve the Mechanical Design

- **In Ansys Mechanical**
  - **Click the icon** Solve **to solve the mechanical Analysis**
  - ***RMB on Solutions (C6) → Insert → Deformation → Total***
  - **Click the icon** Solve **to evaluate results**
  - **Deformation is then shown graphically**



Saving the Project

- **This completes the workshop**
- **Save the file with the name Workshop 05.1 in the working folder**