

# Ansys Lab — 3D Solids

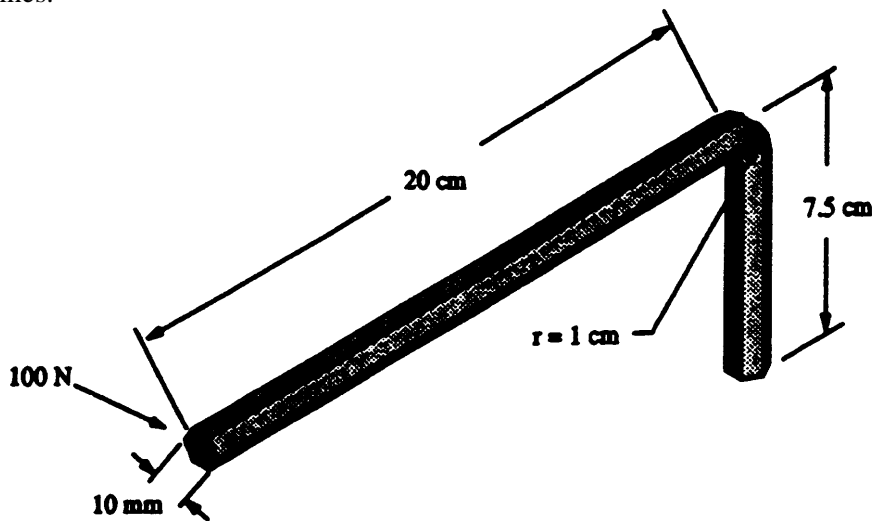
---

## Purpose

- Learn how to perform static analysis of a 3D solids
- Using parameters in ANSYS
- Learn to utilize log and lgw files

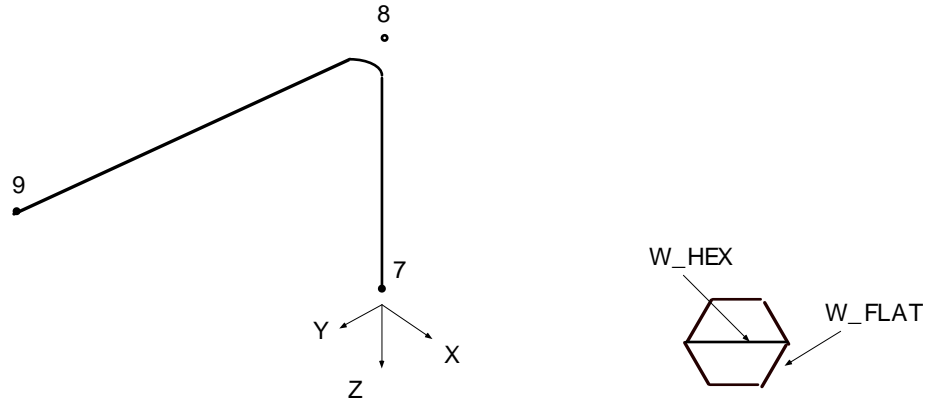
## Lab exercise

A 10-mm allen wrench is subjected to a torque by means of a 100-N force at its end. The objective is to determine the stress intensity in the wrench under this loading condition. The wrench geometry is generated by creating a hexagon primitive and "dragging" it along a set of lines.



Given data:

- |                                  |  |
|----------------------------------|--|
| Width across flats = 10 mm       | Configuration = hexagonal                      |
| Length of shank = 7.5 cm         | Length of handle = 20 cm                       |
| Bend radius = 1 cm               | Modulus of elasticity = $2.07 \times 10^5$ MPa |
| Applied tangential force = 100 N |  |



## 1. Preprocessing

### 1.1. Define parameters for geometry and material

*In ANSYS, the value of physical quantities can be input using parameters, namely variables storing specified values. In the cases when repeated analysis is needed and each analysis involves only the change of, for example, geometric dimensions, use of parameters can reduce the amount of repetitive works.*










- ANSYS Utility Menu > Parameters > Angular Units > [\*AFUN]  [ Degrees DEG ] > OK
- ANSYS Utility Menu > Parameters > Scalar Parameters
- Scalar Parameters
  - Selection [ **E<sub>XX</sub>=2.07E11** ] > <Enter>
  - Selection [ **W\_HEX=0.01** ] > <Enter>
  - Selection [ **W\_FLAT=W\_HEX\*TAN(30)** ] > <Enter>
  - Selection [ **L\_SHANK=0.075** ] > <Enter>
  - Selection [ **L\_HANDLE=0.2** ] > <Enter>
  - Selection [ **BENDRAD=0.01** ] > <Enter>
  - Selection [ **L\_ELEM=0.0075** ] > <Enter>
  - Selection [ **NO\_D\_HEX=2** ] > <Enter>
- Selection [ **TOL = 25E-6** ] > <Enter>
- Parameters > Save Parameters > OK
- ANSYS Toolbar > SAVE\_DB

### 1.2. Define the element type (SOLID45) and material properties.

- ANSYS Main Menu
- Preprocessor > Element Type
  - Structural **Solid** > [ **Brick 8node 45** ] > OK > Close
- Preprocessor > Material Props > Material Model > Structural > Linear > Elastic > Isotropic
  - > EX [ **E<sub>XX</sub>** ]
  - > PRXY [ **0.3** ]
- ANSYS Toolbar > SAVE\_DB

### 1.3. Begin the model creation.

*In this example the solid model (wrench) is created by extruding the cross section area along a set of lines.*

- ANSYS Main Menu
- ! Create hexagonal area.
- Preprocessor > -Modeling- **Create**
  - Areas- **Polygon** > By Side length
  - NSIDES [ 6 ] LSIDE [ W\_FLAT ] > OK
- ! Create keypoints 7, 8 and 9 as in the figure.
- Preprocessor > -Modeling- **Create**
  - Keypoints
  - > In Active CS... > NPT [ 7 ] X, Y, Z [ 0 ] [ 0 ] [ 0 ] > Apply
  - > In Active CS... > NPT [ 8 ] X, Y, Z [ 0 ] [ 0 ] [ -L\_SHANK ] > Apply
  - > In Active CS... > NPT [ 9 ] X, Y, Z [ 0 ] [ L\_HANDLE ] [ -L\_SHANK ] > OK
- ANSYS Utility Menu > PlotCtrls > View Settings > View Direction > [/VIEW] [ 1 ] [ 1 ] [ 1 ] > OK
- ANSYS Utility Menu > PlotCtrls > Numbering > KP [ On ] LINE [ On ] > OK
- ANSYS Utility Menu > Plot > Keypoints > [Keypoints](#)
- Preprocessor > -Modeling- **Create**
  - Lines- **Lines** > In Active Coord +
  - > ANSYS Input > 1,4 < Enter > > OK
- Preprocessor > -Modeling- **Create**
  - Lines- **Lines** > In Active Coord +
  - > ANSYS Input > 7,8 < Enter > > OK
- Preprocessor > -Modeling- **Create**
  - Lines- **Lines** > In Active Coord +
  - > ANSYS Input > 8,9 < Enter > > OK
- Preprocessor > -Modeling- **Create**
  - Lines- **Line Fillet**
  -  [ Line 8 and 9 ] > OK > RAD [ BENDRAD ] > OK
- ANSYS Utility Menu > PlotCtrls > Pan,Zoom,Rotate >  **Front**
-  **Zoom** [ Pick up the hexagon area ]
- ANSYS Utility Menu > Plot > Lines
- ANSYS Main Menu
- ! Hex section is cut into two quadrilaterals to satisfy mapped meshing requirements for brick.
- Preprocessor > -Modeling- **Operate**
  - Booleans- **Divide** > Area by Line
  -  [ Hexagon area ] > OK >  [ Center horizontal line ] > OK
- ! Drag both quadrilaterals along path pattern following L8,L10 and L9.
- Preprocessor > -Modeling- **Operate**
  - Extrude/Sweep > -Areas- **Along Lines**
  -  [ Both polygon areas ] > OK
  - ANSYS Input > 8,10,9 < Enter > > OK
- Verify > Yes
- Verify > Yes
- Warning > Close
- ANSYS Utility Menu > PlotCtrls > Pan,Zoom,Rotate >  **ISO**  **Fit** > Close
- ANSYS Utility Menu > Plot > Volumes
- ANSYS Utility Menu > PlotCtrls > Style > Hidden - **Line Options**
- [/TYPE/ [Type of Point](#)]  [ **Precise hidden** ] > OK

#### 1.4. Generate the mesh.

! Select lines at Z = 0 plane.

- ANSYS Utility Menu > Select > Entities... > [ Lines ] [ By location ]  
[ Z coordinates ] Min, Max [ 0 ] > OK
- ANSYS Utility Menu > Plot > Lines
- ANSYS Main Menu
- ! Number of divisions along selected lines.
- Preprocessor > -Meshing- **Size Control** → **Manualsize-**  
**-Lines- Picked Lines+**  
[ L1, L2, L3, L4, L5 and L6 ] > OK > NDIV [ **NO\_D\_HEX** ] > OK
- ANSYS Utility Menu > Select > Everything
- ANSYS Main Menu
- Preprocessor > -Meshing- **Size Contrls ->**  
**-ManualSize-...> -Global- Size**  
**SIZE [ L\_ELEM ] > OK**
- Preprocessor > -Meshing- **Mesh**  
**-Volumes- Mapped > 4 to 6 sided > Pick All > OK**

#### 1.5 Create a DB log file

- ANSYS Utility Menu > File > Write DB Log File > OK

*The Database log file is an abbreviated log file which contains all the commands pertaining to geometric modeling and meshing. By issuing the above command, a log file is created in the current directory. Once this file is created, it can be used as an input file to a new analysis. The lgw file can be edited using a text editor (outside ANSYS). In the lab problem, you may need to edit the lgw file to make changes to the values assigned to the parameters.*

## 2. Solution

### 2.1. Apply displacement constraints and applied forces.

! Select bottom area of shank.

- ANSYS Utility Menu > Select > Entities... > [ Areas ] [ By location ]  
[ Z coordinates ] Min, Max [ 0 ] > Apply

! Select exterior lines of the selected areas.

- ANSYS Utility Menu > Select > Entities.. > [ Lines ] [ Exterior ] > Apply

! Select nodes which attached to the selected lines.

- ANSYS Utility Menu > Select > Entities.. > [ Nodes ] [ Attached to ]  
[ Lines, all ] > OK
- ANSYS Main Menu
  - **Solution** > **-Define Loads- Apply**  
**-Structural- Displacements**  
**On Nodes > Pick All**
  - Lab2 [ All DOF ] > OK

- ANSYS Utility Menu > PlotCtrls > Symblos > [ /PBC ] **Applied B.C.'s** > OK
- ANSYS Utility Menu > Select > Everything
- ANSYS Utility Menu > Plot > Elements
- ANSYS Main Menu
  - **Solution** > - **Define Loads** - Apply
  - Structural**- Force/Moment > **On Keypoints**
  - ■ ■ [The keypoint where forces applied] > OK
  - Lab [ FX ] VALUE [ **100** ] >OK

## 2.2. Initiate the solution

- ANSYS Main Menu
- Solution > -Solve- **Current LS** > OK

## 3. Postprocessing

### 3.1. Review the resultss using the general postprocessor (POST1).

- ANSYS main Menu
- General Postproc > -Read Results- First Set > Plot Results > Deformed Shape  
KUND > **Def + undeformed** > OK
- General Postproc > Plot Results  
-Contour plot- **Nodal Solu**
- Item, Comp ■ ■ ■ [Stress] ■ ■ ■ [ **Intensity SINT** ] > OK

**Note:** The stress intensity  $\sigma_1$  (output quantity SINT) is the largest of the absolute values of  $\sigma_1 - \sigma_2$ ,  $\sigma_2 - \sigma_3$ , or  $\sigma_3 - \sigma_1$ , which is essentially a measure of the shear stress.:

$$\sigma_1 = \text{MAX}(|\sigma_1 - \sigma_2|, |\sigma_2 - \sigma_3|, |\sigma_3 - \sigma_1|)$$

## Lab Problems

Change the length of the shank to 6cm, the length of handle to 15 cm and the bend radius 1.2 cm and re-analyze the problem. Submit the following plots in the lab report.

1. The finite element mesh
2. The deformed shape
3. Contour plot for SINT.

## Tips

There are two convenient ways to re-run an analysis. The first is to read the solid model and the mesh information from the lgw file created in Step 1.5. The boundary conditions and applied forces need to be input again. The second approach is to use the log file. The log file is a record of all the commands used in the analysis, including possibly mistakes. If the new analysis involves only parametric change, the best way is to editing the log file (outside ANSYS) to change the values assigned to the parameters.