APPENDIX TO TASK 1
MODELING IN ANSYS

General information about working in ANSYS.

To start working with ANSYS it is better to launch ANSYS Product Launcher. In ANSYS Product Launcher select Simulation Environment→ANSYS, choose the working directory and define Job Name for your project. If you already have an ANSYS database for your project, then after launching ANSYS resume your database from File→ Resume Jobname.db.

An ANSYS *command file* (text file with extension .inp or .txt), written in APDL ANSYS, can be executed from File → Read Input from… When creating a command file, it is useful to copy the commands from input file into the command line, execute them step by step and watch the results.

It is recommended to save the results from time to time! The current database should be cleared before performing new analysis (Menu path: File→Clear and Start New).

Solid modeling in ANSYS.

It is convenient to build a solid model of the domain with complex geometry using a such called "bottom-top" approach, i.e. start from building low-level entities such as Keypoints and finish with Areas for 2D problems or Volumes for 3D problems. In order to build a 2D domain, it can be useful to build the keypoints of the domain, then connect them with lines and after that generate the areas by a list of lines defining the areas so that they should form a simply connected closed curve.

Command line mode.

Use **K** command in order to build the keypoints. The first argument stands for the reference number for the keypoint. The remaining arguments define the coordinates of the keypoint. For example, the command **K, 2, R1, -H/2** will create a keypoint with the number 2 and the coordinates x=R1, y=-H/2, z=0 (zero is the default value). In order to build the line between two keypoints, use **L** command. For example, the command **L, 2, 3** will create a line in the active coordinate system between keypoints with numbers 2 and 3. The lines are numbered automatically by ANSYS, starting with the lowest available number. Areas can be built using **A** or **AL** command. The areas are also numbered automatically. **AL** command creates an area bounded by previously defined lines. The list of lines (10 lines maximum) must be input either in clockwise or counterclockwise direction. **A** command defines an area by connecting keypoints. The keypoints defining the area must be input in a clockwise or counterclockwise order around the area (maximum 18 keypoints in the list). The existing lines between adjacent keypoints will be used; and the missing lines will be generated "straight" in the active coordinate system and will be assigned the lowest available numbers.

Generating finite element model in ANSYS.

There are two main meshing methods in ANSYS: free and mapped.

A *free* mesh does not follow any pattern and has no element shape restrictions. Free mesh is suitable for complex shaped areas and volumes. Free mesh is usually defined by an average finite element size.

On the contrary, a mapped mesh restricts the element shape (for example, quadrilaterals for areas, or hexahedrals for volumes) and typically has a regular pattern with obvious rows of elements. Mapped mesh is beneficial for areas and volumes of simple shape, such as rectangles or bricks. The use of a mapped mesh facilitates to reduce the computational time. Usually a mapped mesh is built on the base of the divisions and spacing ratio specified for its constituting lines.

Defining mechanical boundary conditions in ANSYS

Degree-of-freedom (DOF) constraints can be defined either for the entities of finite element model (on nodes and elements) or for the entities of solid model (on keypoints, lines or areas). All solid boundary conditions will be converted to finite element boundary conditions at the stage of solution. For conflicting boundary conditions solid boundary conditions have priority over finite element boundary conditions at the same boundary.

Interactive mode: menu path **Main Menu> Solution >Define Loads>Apply>Structural->…**
Command line mode: commands **D, SF, F**, which set the boundary conditions on nodes or commands **DL, SFL, FL**, which set the boundary conditions on lines.

Commands for entities selection:

- Select nodes with coordinate x=a.

```
NSEL,S,LOC,X,A ! Select all nodes with coordinate X=A
```

- Select nodes with coordinates a<=y<=b and c<=y<=d

```
NSEL,S,LOC,y,A,b    ! (s-new set)
NSEL,A,LOC,Y,C,D ! (a-add set)
```

- Select line L2

```
LSEL,S,LINE,,2
```

- Select lines L3,L5 and L8.

```
LSEL,S,LINE,,3,5,2  ! Select lines 3 and 5 (2 is the step between
the line numbers)
LSEL,A,LINE,,8 ! Add line 8 to the set
```

- Select all nodes on line L2

```
LSEL,S,LINE,,2 ! Select line 2
NSLL,S,1 ! Select all nodes lying on selected line
! 2nd argument is the key to select the nodes: 1 – internal and key
points (line ends); 0 – select only nodes interior to selected line
```

- Select all nodes lying on lines L3,L5 and L8.

```
LSEL,S,LINE,,3,5,2  ! Select lines 3 and 5 (2 is the step between
the line numbers)
LSEL,A,LINE,,8 ! Add line 8 to the set
NSLL,S,1 ! Select all nodes lying on selected line
```

1) **Symmetry boundary conditions**
- In *interactive mode*: …->Symmetry B.C. -> On lines или On nodes.
- In *command line mode*: use **D, DL** or **DA** command. In DL and DA command use symmetry label **SYMM** (for example: **DL,8,,SYMM**). Another way is to select necessary nodes with selecting command and then use D command. For example, the nodes belonging to the line L8, can be selected by the following commands:
  LSEL,S,LINE,,8
  NSLL,S,1 ! select nodes associated with selected lines
  ! 2nd argument is the key specifying whether or not only interior line nodes are to be selected (here 1 stands for selecting both interior and outer nodes)
  The command **D,ALL,UX,0** will define the component of the displacement vector UX=0 for all selected nodes that corresponds to the symmetry boundary conditions with respect to *Oy* axis for the problem with degrees of freedom UX and UY. Analogously, the command **D,ALL,UY,0** will the component of the displacement vector UY=0 for all selected nodes that corresponds to the symmetry boundary conditions with respect to *Ox* axis for the problem with degrees of freedom UX and UY.
2) **Rigidly constrained boundary**: for the problem with degrees of freedom UX and UY both components of the displacement vector are equal to zero, i. e. UX=0 and UY=0.
  - In *interactive mode*: …->.Displacement -> On lines or On nodes.
  - In *command line mode*: use the following block of comments using D or DL commands:
  - **D,ALL,UX,0**
    D,ALL,UY,0
    or: **D,ALL,ALL,0** (for the case when UX and UY are the only degrees of freedom).
  - **DL,ALL,,UX,0**

DL,ALL,,UY,0

3) **Free boundary.** This is default boundary condition in ANSYS, no need to be defined.

4) **Distributed load (pressure) boundary condition**

- In *interactive mode*: …-> Pressure -> On lines or On nodes (or On areas for 3D problems).

- In *command line mode*: use **SF** command that specifies surface loads at nodes or commands like **SFL, SFA** that specify surface loads on lines or areas, respectively.

Example:

- **SF,ALL,PRES,-P** ! defines distributed load PRES = -P (to apply the pressure acting out) for all selected nodes

- **LSEL,S,LINE,,2** ! select line 2
  SFL,ALL,PRES,-P ! apply distributed load PRES = -P (to apply the pressure acting out) for all selected lines

5) **Concetrated load (force) boundary condition**

- In *interactive mode*: …-> Force/Moment -> On lines or On nodes (or On areas for 3D problems).

- In *command line mode*: use **F** command that specifies force loads at nodes or **FK** command that specifies force load at keypoints.

Viewing results in ANSYS (interactive mode)

The results are contained in files with extensions like *.RST, *.RFL, *.RTH, *.RMG (for different types of analyses). Any results file can be loaded for postprocessing.

It is convenient to view results in graphical (interactive) mode, although the commands for viewing results are also available. The results can be accessed via postprocessor.

There are two postprocessors in ANSYS: General Postprocessor and Time History Postprocessor.

General Postprocessor is used for reviewing results for a specifically defined combination of loads at a single time (or frequency). To enter the ANSYS general postprocessor, issue the /POST1 command or use menu path: Main Menu> General Postproc. It is the only postprocessor available for static or steady state analysis.

Time History Postprocessor is used for reviewing results at specific locations in the model as a function of time, frequency, or some other change in the analysis parameters that can be related to time. Time History Postprocessor can be used to construct graphics displays, chart representations or tabular listings, or perform math operations on the data sets. Time History Postprocessor is available for transient, modal or harmonic analyses. A typical time-history task would be to graph result items versus time in a transient analysis, or to graph force versus deflection in a nonlinear structural analysis. To enter the ANSYS Time History Postprocessor, issue the /POST26 command or use menu path: Main Menu-> TimeHist PostPro that will launch the Variable Viewer.

View results in General Postprocessor

For structural analysis you can review the following results. Enter <u>Main Menu</u> and choose General Postprocessor.

To display the deformed shape of the structure, select

General Postproc → Plot Results → Deformed Shape → select Def + undef edge (to display deformed finite element mesh and undeformed edge of the model)

Plot results for the Degree of freedom solution (solution in terms of unknown variables)

General Postproc → Plot Results → Contour Plot → Nodal Solu → DOF Solution → X-Component of displacement (to obtain a picture of displacements *Ux* distribution)

→ Y-Component of displacement (to obtain a picture of displacements *Uy* distribution)

$\rightarrow$ Displacement vector sum (to display the distribution of the displacement vector magnitude)

Plot results for the solution derivatives (such as stresses, strains, etc.)

Important note: in all modern versions of ANSYS the display of the results for the solution derivatives, such as stresses and stresses, may not be available Plot Results immediately. In this case when working in interactive mode one can open Results Viewer first or read the results of the last set General Postproc $\rightarrow$ Read Results $\rightarrow$ Last Set. In a command mode or for an input file, just add a command to read the results of the last set to get access to these results: **SET, LAST**

General Postproc $\rightarrow$ Plot Results $\rightarrow$ Contour Plot $\rightarrow$ Nodal Solu $\rightarrow$
$\rightarrow$ Nodal Solu $\rightarrow$ Stess   X-Component of stress (to display the distribution of the stresses T_{xx}, labeled by SX in ANSYS)
$\rightarrow$ Nodal Solu $\rightarrow$ Elastic strain   X-Component of elastic strain (to display the distribution of the strains S_{xx}, labeled by EPELX in ANSYS)

One can also plot Element Solution results:
$\rightarrow$ Element Solu… etc.
(Nodal Solution differs from Element Solution by the use of different numerical methods for computing field gradients over the available nodal results).

Plot vector variables
To display the distribution of the vector variables, choose Vector Plot in General Postprocessor
General Postproc $\rightarrow$ Plot Results $\rightarrow$ Vector Plot $\rightarrow$ Predefined
DOF solution $\rightarrow$ Translation U (to display the distribution of the displacement vector)

All results can also be accessed from the Upper menu:
Plot $\rightarrow$ Results $\rightarrow$ Deformed Shape…

Saving images from Ansys
There are different ways to save an image into a graphical file.
- PlotCtrls->Capture Image. An image will appear in a new window. Use menu path File->Save as… to save an image as a *.bmp graphical file without reversing black and white colors.
- Plot Ctrls->Hard Copy->To File. This menu path will open a Graphics Hard Copy window. Here you can choose a color scale (Monochrome, Gray Scale, Color) and an extension for the graphical file (.bmp, postscript, .tiff, .jpeg, .png), you can also specify the file name and choose whether or not use a Reverse Video for reversing black and white colors. There is no option to choose the place where to save the image. The graphical file will be saved in the working directory of ANSYS that was selected at the start.
- PlotCtrls->Write Metafile. Here you can also tick Invert White/Black for reversing black and white colors. You can specify the directory where to save the image. The graphical file will be saved as a Windows metafile with an extension .emf or .wmf.

Window options in ANSYS (interactive mode)

For viewing results in an active window you can specify various window options.
In Upper menu choose PlotCtrls $\rightarrow$ Window Controls $\rightarrow$ Window Options
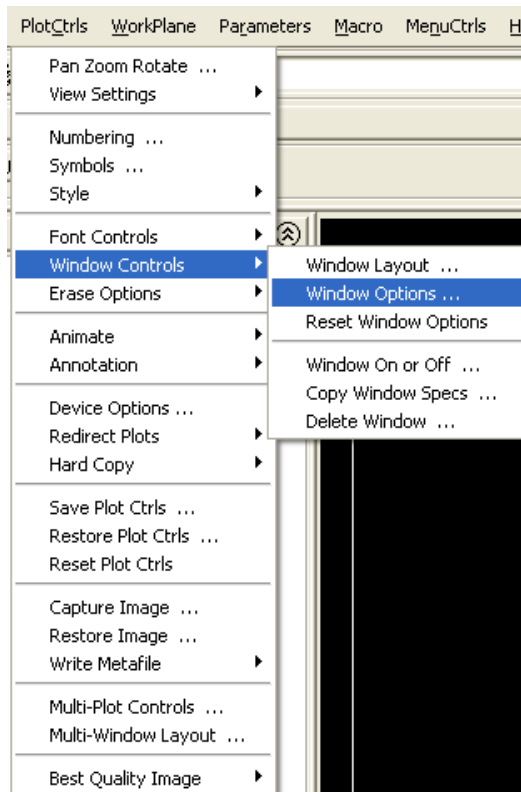
In Window Options you can select whether or not to show: title, window frame, ANSYS logo, legend, coordinate triad and so on: