Problem #2:

Step by Step modeling, analysis and design:
Create a new 2D model starting from scratch:
The following GEOMETRY window will be opened:

Right click on the ‘joint tool’ in the ‘shortcuts’ task bar (left) and the following pop-up window will be opened:
o Now put the coordinates values of the joints as labelled by circles in the problem formulation; the numbering of the joints follows the sequence of adding the joints. To see the numbering of the joints ‘the joint tool’ button in the bottom taskbar should be kept ‘on’.

o Right click on the ‘Add member’ in the left ‘Shortcuts’ toolbar and the following pop-up window will be opened:
Choose the ‘Single member’ and press ‘ok’. Change ‘Continue from last joint’ button (in the fourth taskbar from the top) to ‘yes’ if you want to draw the members sequentially.

Right click on the ‘Add Element’ in the left ‘Shortcuts’ toolbar and the following pop-up window will be opened; Choose the ‘beam’ type element and press ‘ok’.
Make sure that the ‘Member type’ in the fourth taskbar (from the top) is also chosen as ‘beam’ as following:

Right click on the ‘Add section’ in the left ‘Shortcuts’ toolbar and the following pop-up window will be opened; Click on the ‘Dbase’ and choose the necessary section. In this case it should be an ‘I’ section named W360X158. The name and color of the chosen section can be changed. In this problem the color is chosen as red.
ENGI 7704: Design of Steel Structure

Use of S-Frame and S-Steel
Choose the section from the drop down menu at the fourth taskbar from the top and click on the specific members those needed to be defined as the chosen section. In this case all the members are having the same section and colored as red.

Right click on the ‘Add support’ in the left ‘Shortcuts’ toolbar and the following pop-up window will be opened.
Choose the required support and click on the joints those should have the chosen support.

Click on the ‘Loads’ from the left ‘Shortcuts’ toolbar and the following pop-up window will be opened.
- From the 'Edit' > 'New Load Case' define a new load case. In this case it is named as 'PL'.

- Save the model in preferred directory.
Click on the ‘Joint Loads’ from the left ‘Shortcuts’ toolbar and define the joint loads according to the problem. In this case there are two joint loads=20kN downward and 60kN-m clockwise moment. Choose the required direction and values from the fourth taskbar and click on the required joint to apply the load.

Right click on the ‘Global Load Tool’ in the left ‘Shortcuts’ toolbar and the following pop-up window will be opened; Choose the necessary UDL and apply on the required member by single click.
Save your model and from ‘Run’ > ‘Analysis’ > ‘S-Steel’ analyze the saved model.
The ‘Graphical’ window will be opened. Click on the ‘Reaction tool’ from the left ‘Shortcuts’ toolbar and then you will be able to see the reactions as follows:
The software results are similar with the analytical results:

- Click on the ‘Settings’ > ‘Diagram’ and choose ‘Display numerical values on diagrams’ which will show the numerical values on the diagrams.
Click on the axial force, shear force and bending moment diagram from the left toolbar.
To check the results by member, double click on the specific member and you will see the detailed results as follows.
The Shear force and bending moment diagrams also match with the analytical results:
ENGI 7704: Design of Steel Structure
Use of S-Frame and S-Steel

Kshama Roy (ksr037@mun.ca)
To see the deflection, go to the ‘Settings’ > ‘Diagram’ and click on the ‘Automatic Scale’. Click the ‘deflection’ from the left toolbar.
The deflection shape also matches with the analytical results:

![Deflected shape](image)

Go to the ‘Code Input’ from the left toolbar and choose all members by single click on the members.
From ‘Run’ > ‘Code check’, check whether the chosen sections are okay for the members or not:
o Check the utilization ratios; double click on the specific member to get the code details:

o For design, choose the ‘Design Input’ from the left toolbar and click on the members that you need to design by S-Steel.
Click ‘Run’ > ‘Design’ from the top taskbar and you will get the design results from the S-Steel:

If you double click on the members, you will see the proposed sections and now you need to play with it to choose the proper section.
Finally from ‘Run’ > ‘Re-analyze’ reanalyze the model with the proposed section and check whether the utilization ratios are satisfactory or not.

Practice problems:
ENGI 7704: Design of Steel Structure
Use of S-Frame and S-Steel

Problem # 3:

Kshama Roy (ksr037@mun.ca)