ORDER OF IMPLEMENTING 2D BAR STRUCTURES INTO ROBOT STRUCTURAL SOFTWARE (FOR THE PURPOSE OF THE SUBJECT "SELECTED TOPICS OF STRUCTURAL MECHANICS")

1. LANGUAGE

Toolbars at the top of the screen:

Click: Tools (Narzędzia) / Preferences (Preferencje) / Languages (Języki):
Change "Regional Settings" (Parametry Regionalne) into Great Britain
Click "Yes (Tak)" then "Yes (Tak)" and "OK"

The program will reopen and be available in English

2. UNITS

Toolbars at the top of the screen:

Click: Tools / Job preferences / Units and Formats:

- **Dimensions:**
- change the unit of "Structure Dimensions" into [m]
- change the unit of "Section Dimensions" into [m]
- change the unit of "Section Properties" into [m]
By using the arrows on the right you can change the number of significant figures

- **Forces:**
Change the unit of "Stresses" into kN/m^2 by clicking on the "three dots" symbol and then change the unit of force into kN.
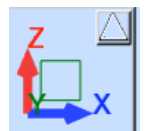
- **Other:**
Change the unit of "Displacement" into [m].

Angle/Rotation (data) – change the unit into [Rad] if you want to implement the rotational elastic support ↻ or if you want to apply the imposed displacement (rotation) of supports (needed when solving tasks from the Displacement Method).

Click „OK" to accept all your changes.

3. AXIS DEFINITION

In Robot Structural Software, the coordinate system for implementing plane 2D structures is as shown in
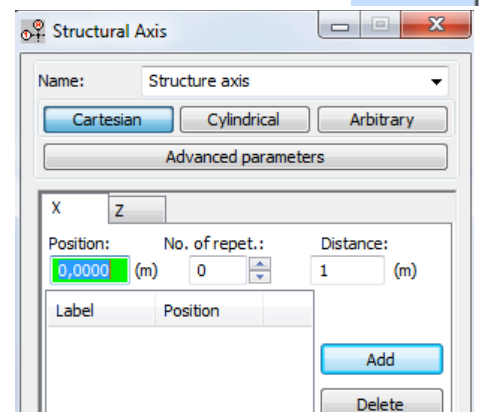
the adjacent figure.

Toolbars on the right side of the screen: Click the first icon.

On the tab „X" next to the word „Position" - enter the coordinates that are located on axis "X". Then click the "Enter" or "Add" button.

On the tab „Z" next to the word „Position" - enter the coordinates that are located on axis "Z". Then click the "Enter" or "Add" button.
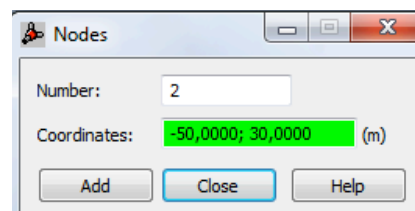
At the end click "Apply" and "Close".

4. NODES

In order to enter nodes you can either enter their coordinates in the open window "Nodes", or click on the screen in places where you want the nodes to be implemented (they will be marked with small red dots).

After finishing, click "Close".

You can display the node numbers by clicking the first icon (a yellow dot with the letter "n") in the bottom left corner (below the coordinate system)
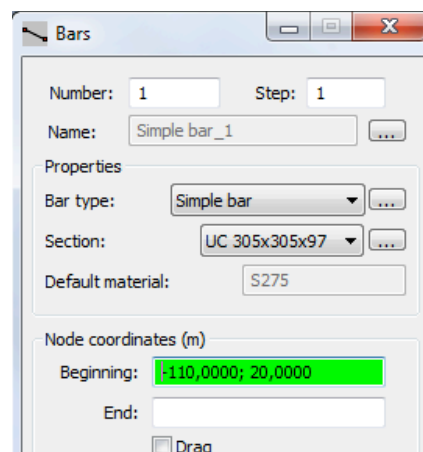
5. BARS

In order to implement bars you can either enter the node coordinates (the beginning and the end) in the open "Bars" window and click "Add",

or click on the screen on the node which is the beginning of your bar and then the node at the end (connecting nodes with bars).
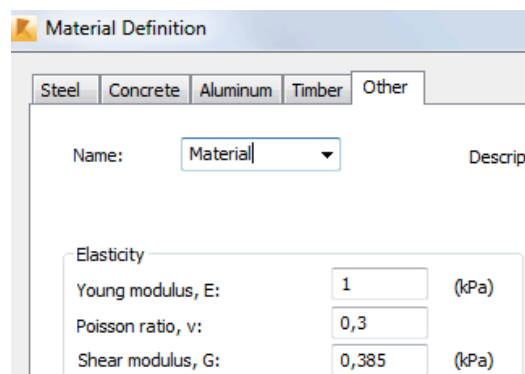
After finishing, click "Close".

6. MATERIAL DEFINITION

Click: Tools / Job preferences / Materials / Modification / Other

Next to the word „Name" enter the name of your material eg. "Material".

- Change the value of the Young Modulus [E] into 1 [kPa]

- Change the value of the Shear Modulus [G] into 0.385 [kPa]

CLICK „ADD", „OK" AND ONE MORE TIME „OK"!!!

Only then will the material be saved in the Robot library of materials.

7. BAR SECTIONS

Toolbars on the right side of the screen: Click the fifth icon.

Click the "white paper" symbol (New Section Definition) in the top left corner of the window.

Click the tab "A_x, I_y, I_z"

Next to the word „Label" write e.g. EI or EA

Next to the symbol of A_x, enter:
- for frame or beam structures - a value of at least 100 000 [m^2]
- for truss structures - a value of 1[m^2] for bars with 1 EA stiffness, and respectively 2[m^2] for bars with 2EA stiffness.

Next to the symbol I_y, enter:
- for frame or beam structures - a value of 1 [m^4] for bars with EI stiffness, and respectively the value of 2 [m^4] for bars with 2EI stiffness.
- for truss structures - a value of 1[m^4].

Click "Add". When all the sections are added click "Close".
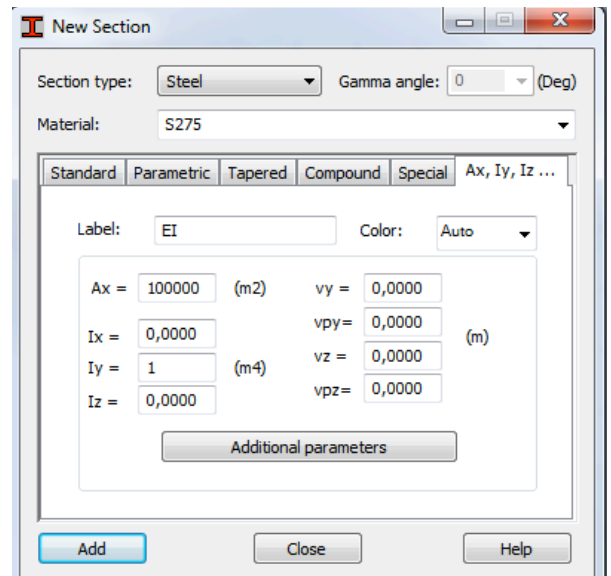
When the "Sections" window is still open click on the appropriate section e.g. "EI" and then click on the bar on the screen to assign it to this bar. After finishing, click "Close".

8. MATERIAL APPLICATION

Toolbars on the right side of the screen: Click the sixth icon.

Select the material that you created in Point 6 (Material Definition). Click on the name of the bar section you created in Point 7 (Bar Sections) and then click "Apply". Repeat this for each section. Click "Close".
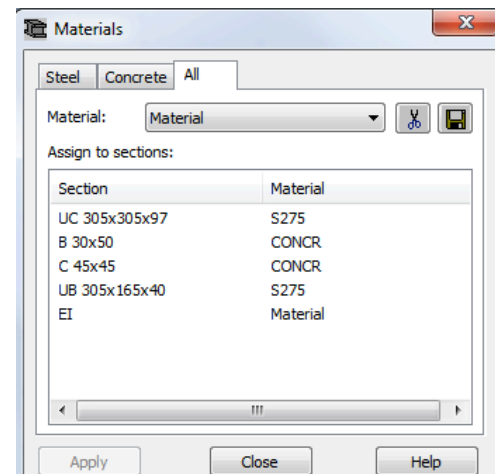
9. SUPPORTS

Toolbars on the right side of the screen: Click the seventh icon.

Click the "white paper" symbol (New Support Definition) in the top left corner of the window.

- RIGID SUPPORTS:

You can create rigid supports on the tab "Rigid".

In order to create a rigid support, enter the name of your support e.g. Roller support or S1 next to the word "Label". Afterwards, fix the directions of its reactions. A support can also be rotated by clicking on the "Direction" button and typing the angle of rotation.

After fixing the appropriate directions click the "Add" button- you can now create another support. When all supports are created, click the "Close" button.

From the list of supports select one of them and click on the screen on a node where you want the support to be.

- ELASTIC SUPPORTS:

You can create elastic supports on the tab "Elastic".

In order to create an elastic support, enter the name of the support next to the word "Label" and then:

- to create a translational elastic support: do not fix any direction, and next to the direction of a reaction (KX or KZ) enter a value of the kΔ coefficient. Then click "Add".

- to create a rotational elastic support: do not fix any direction, change the units to Radians (see point 2.), then enter the value of the kφ coefficient next to HY .

Then click "Add" and "Close".

From the list of created elastic supports select one of them and click on the screen on the node where you want it to be.
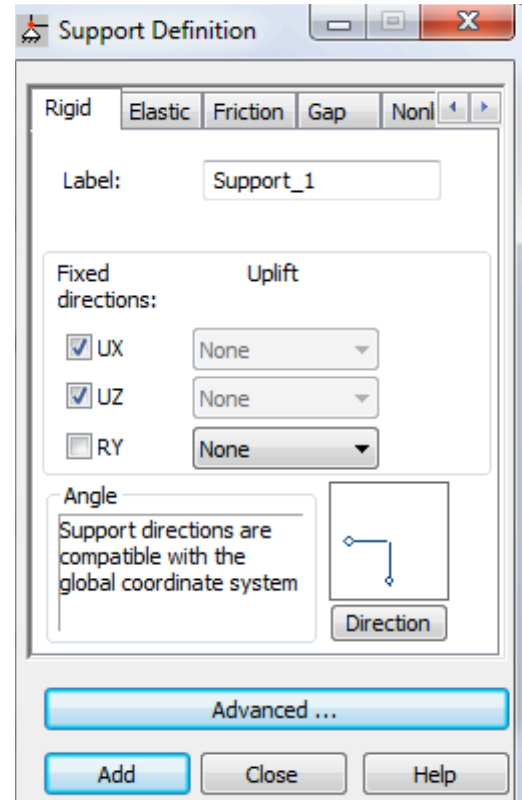
10. HINGES (IT APPLIES ONLY FOR BEAM OR FRAME STRUCTURES)

Toolbars at the top of the screen:
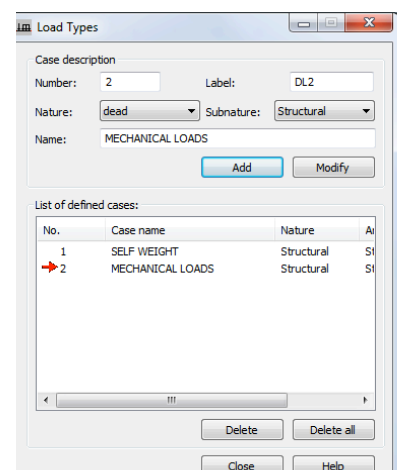
Click: Geometry / Releases

From the open list, select the type of release and then click on the appropriate rod.

11. LOAD CASES

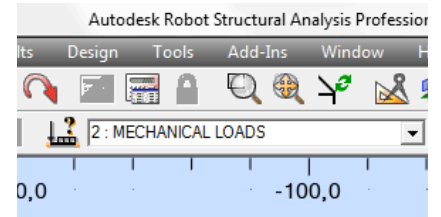Toolbars on the right side of the screen: Click the eighth icon.

Next to the word "Name" enter the name of a load case e.g. self-weight or mechanical loads. Then click "Add".

After entering all load cases, click the "Close" button.

12. LOAD DEFINITION

Toolbars on the right side of the screen: Click the ninth icon.

MECHANICAL LOADING:

In order to enter mechanical loading, first select an appropriate load case e.g."mechanical loads" in the middle of the screen.
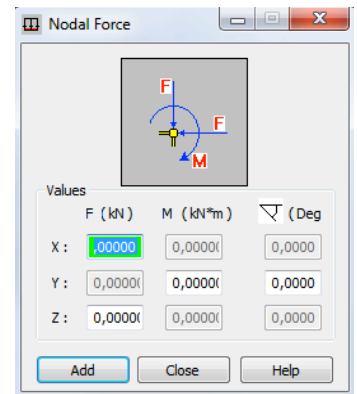
- NODAL:

Click the first icon:

Enter the value of a force or moment in the appropriate column. Each load must be defined separately.

After entering the value of the force click "Add" and then click on the node on the screen.

- ON A BAR:

In order to enter a load on a bar click the ninth icon in the toolbars on the right side of the screen and then choose the tab "bar". The following types of loads can be implemented:

- uniform load - first two icons . After entering the value of the load click "Add" and then choose the bar on the screen,

- bar force - ,

After entering the value of the load and its location click "Add" and then choose the bar on the screen,
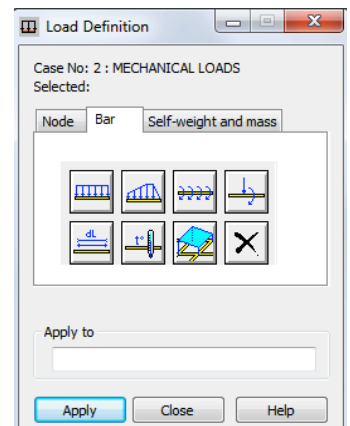
NON-MECHANICAL LOADING:

In order to enter the mechanical loading, first select an appropriate load case e.g. "non-mechanical loading" in the middle of the screen.

- NODAL – IMPOSED SUPPORT DISPLACEMENTS,

On the tab "node" click the second icon

After entering the value click the "add"button and then an appropriate node on the screen.

- BAR – TEMPERATURE AND BAR EXTENSION

- click the first icon in the second row [icon] and implement the bar extension

- click the second icon in the second row [icon] to implement the temperature.

After entering the value of the load and its location click "Add" and then choose the bar on the screen.

WARNING! – To remove unwanted loads go to the top toolbar and click "Load" and then select "Load table". In the table, loads can be selected and removed or changed. Afterwards, close the table.

13. <u>ANALYSIS - CALCULATIONS</u>

<u>Toolbars at the top of the screen:</u>

Click Analysis / Calculations or the red calculator icon at the top of the screen. [icon]

14. <u>RESULTS</u>

<u>Toolbars at the top of the screen:</u>

Click "Results" and then "Diagrams for bars":

- INTERNAL FORCE DIAGRAMS

On the first tab "NTM", select and tick the forces for which you want the diagrams to be checked and then click "Apply". If you want to see values on the diagram go to the last tab "Parameters" and then select "description".

- REACTIONS

Go to the tab "Reactions". Select and tick the support reactions that you want to display and also choose "description" and then "Apply"

- DISPLACEMENTS

To read the displacement of a node, click on it with the right mouse button and select "Object Properties". Now select "Displacement".