# BEAM-FRAME SOLVER

Install "**anastruct**" python library before running this code

```
pip install anastruct
```

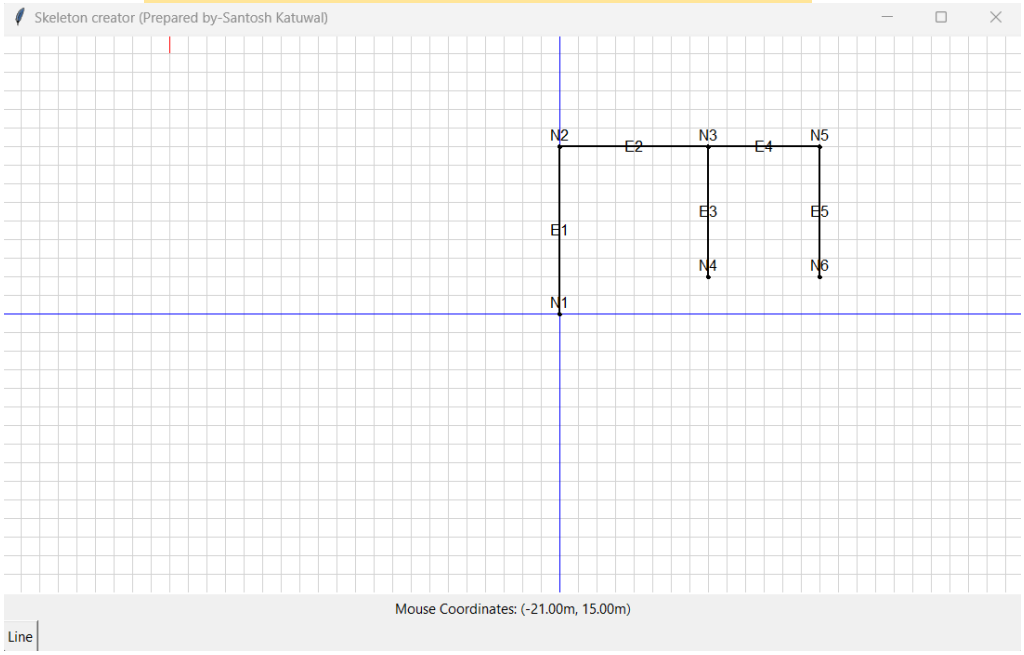
**Step 1.** Run ***run_me.py***

**Step 2.** **Prepare model**

In this stage, you need to sequentially perform the following four steps:

1. **Draw structure:** You can utilize the drawing canvas to create your geometric shapes by utilizing the mouse as a guiding tool.
   *Command:*

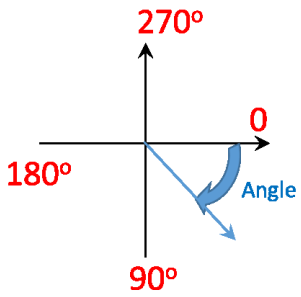
   ```
   frame.draw()
   ```

   Note: You have to click two different points each time to create an element. After closing the drawing canvas, all the input data will be automatically saved as "*nodes.csv*" and "*elements.csv*" in the designated "input" folder

2. **Assign point load and moment at nodes:** It will automatically detects the number of nodes from the drawing canvas.
   →Enter magnitude of point load in kN.
   →Enter angle made by point load: The angle provided is measured in a clockwise direction from the positive x-axis.

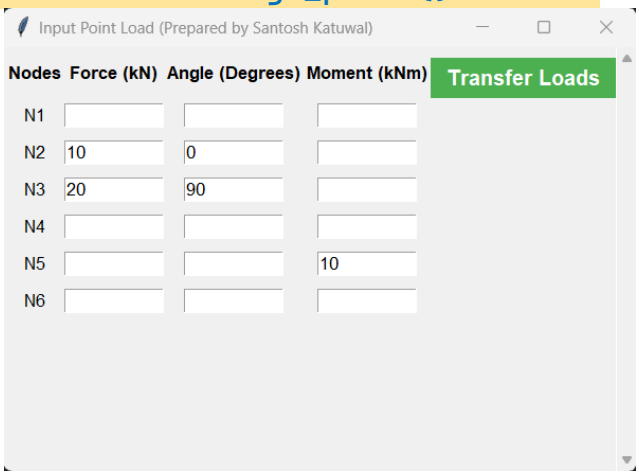
Prepared By: Santosh Katuwal




→Enter moment in kNm: Assign a positive moment for clockwise rotation and a negative moment for counterclockwise rotation.

*Command:*

```
frame.assign_pload()
```

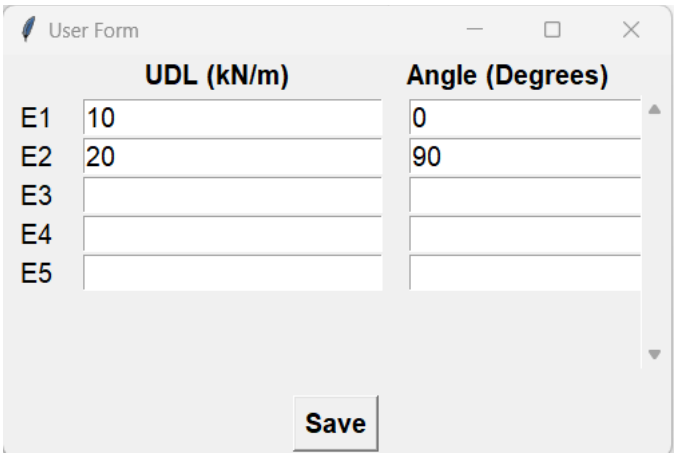



In this figure,
→At node 2, 10kN load is applied towards right
→At node 3, 20 kN load is applied downward
→Clockwise 10kNm moment is applied at node 5
Click on "Transfer Loads" close the window to transfer and save all user input data in the "input" folder.

3. **Assign distributed load**

*Command:*

```
frame.assign_dload()
```

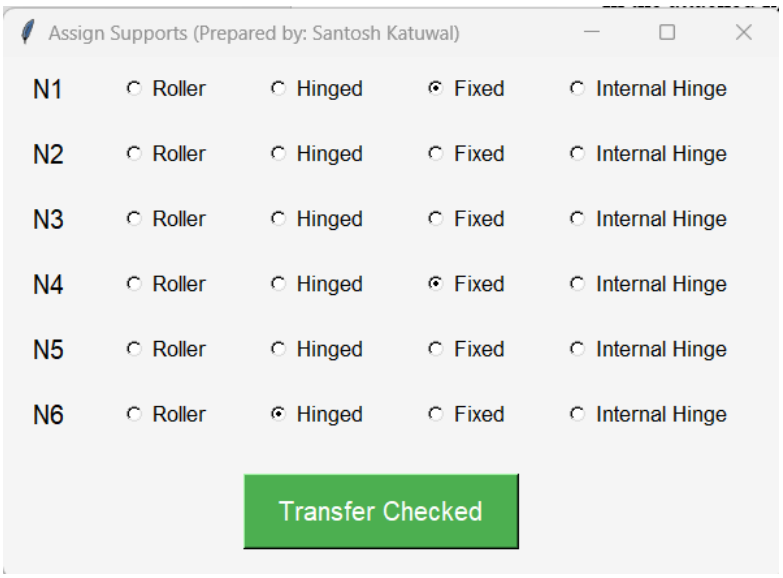



This automatically detect the number of elements. Enter the intensity of UDL in kN/m and angle in degrees. As you click on "Save" button, it creates "udl.csv" file in the input folder.

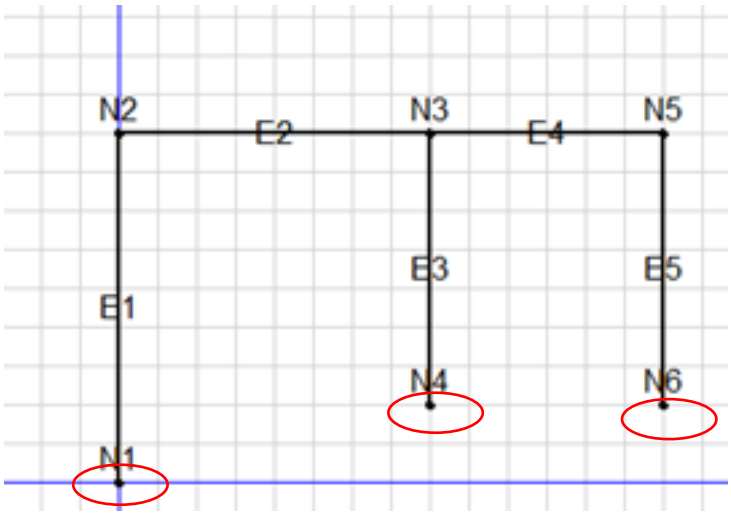
In the attached figure,
→Element 1: 10 kN/m load is applied towards right direction.
→Element 2: 20 kN/m load is applied vertically downward.

4. **Assign supports**
*Command:*

```
frame.assign_support()
```




Ensure that you only select the radio button for supports that need to be assigned.
In the attached figure,
→Fixed support is provided at node 1 and node 5.
→Hinged support is provided at node 6.

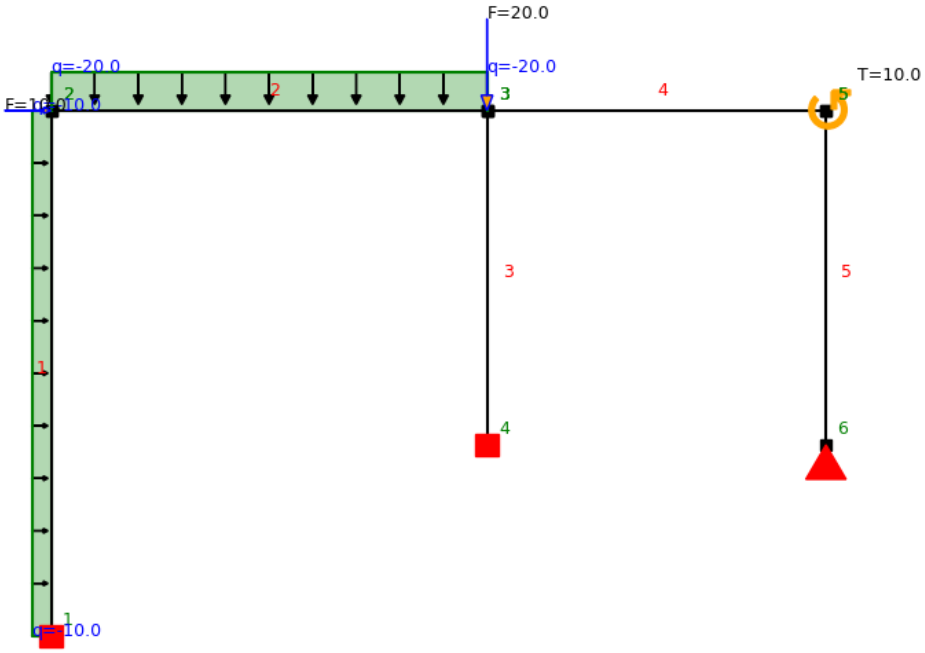



Click on “Transfer Checked” to apply support and save checked supports; it creates “supports.csv” file in the “input” folder.

**Step 3.** **Preview and solve the frame:** You can see and solve your model based on defined parameters in step 2.

*Command:*

```
frame.preview_solve()
```




This will prompt the following message:

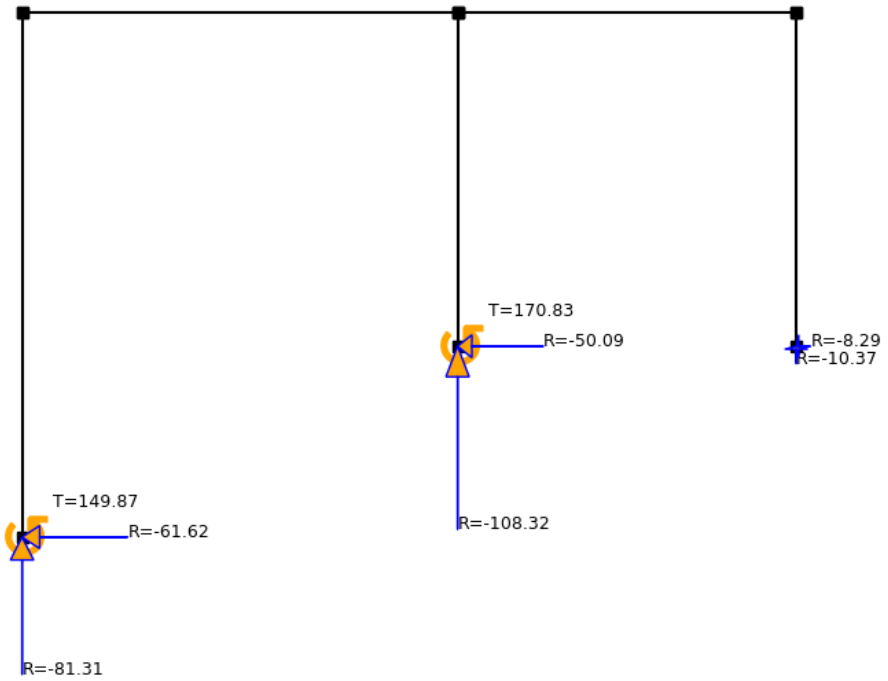
*Solved successfully !!*
*use following commands*
*frame.RD() to see Reaction Diagram*
*frame.AFD() to see Axial Force Diagram*
*frame.SFD() to see Shear Force Diagram*
*frame.BMD() to see Bending Moment Diagram and*

*frame.DSD() to see Deformed Shape Diagram*

**Step 4.** **See result:** Run following commands to display the result.

1. **See reaction diagram**

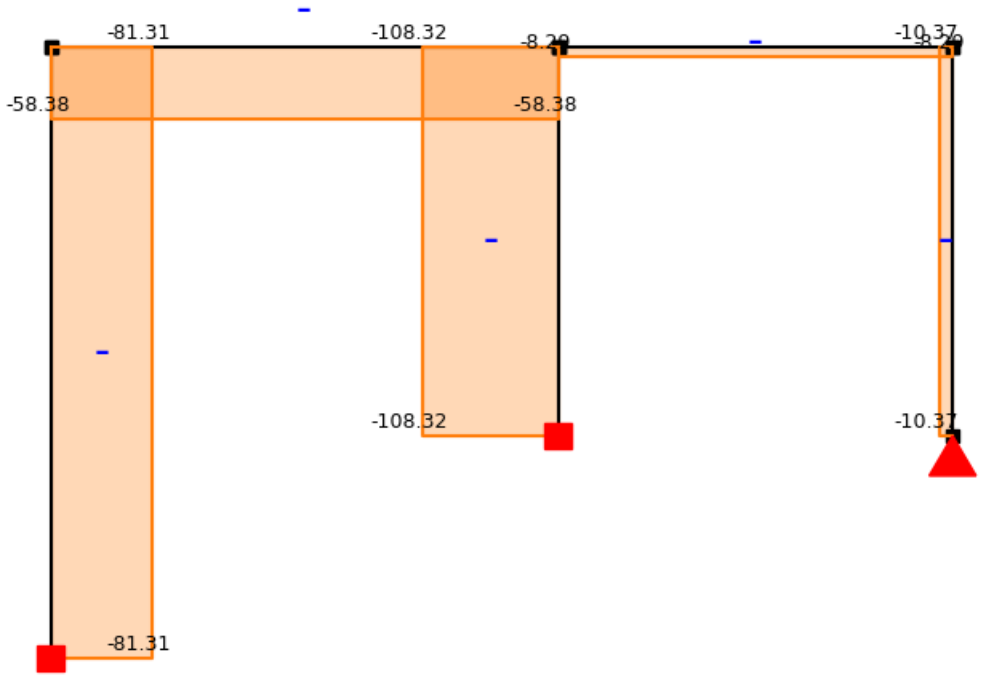
```
frame.RD()
```




2. **See axial force diagram**

*Command:*

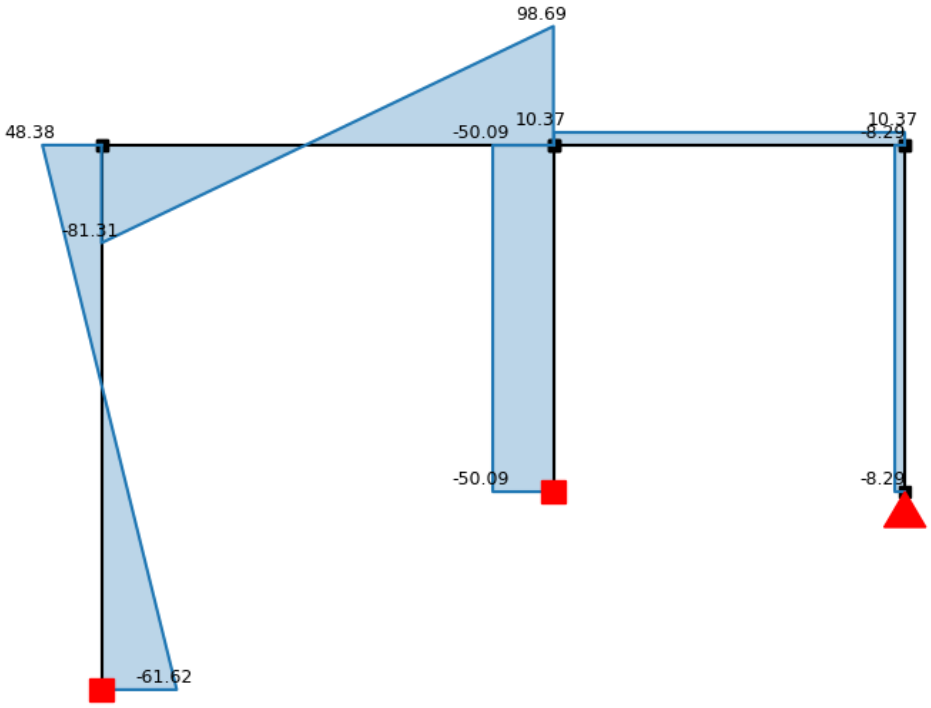
```
frame.AFD()
```




3. **See shear force diagram**

*Command:*

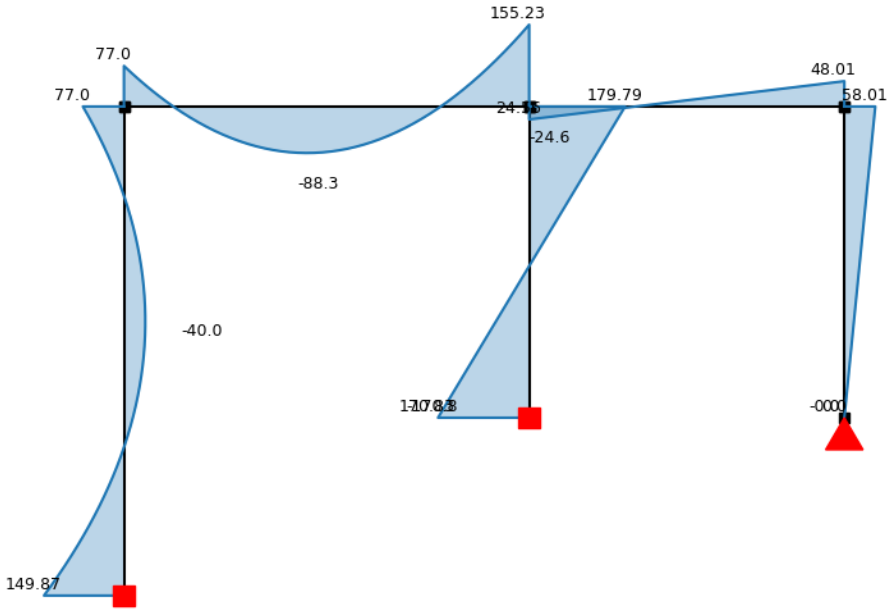
```
frame.SFD()
```




4. **See bending moment diagram**

*Command:*

```
frame.BMD()
```




**5. See deformed shape diagram**

*Command:*

```
frame.DSD()
```