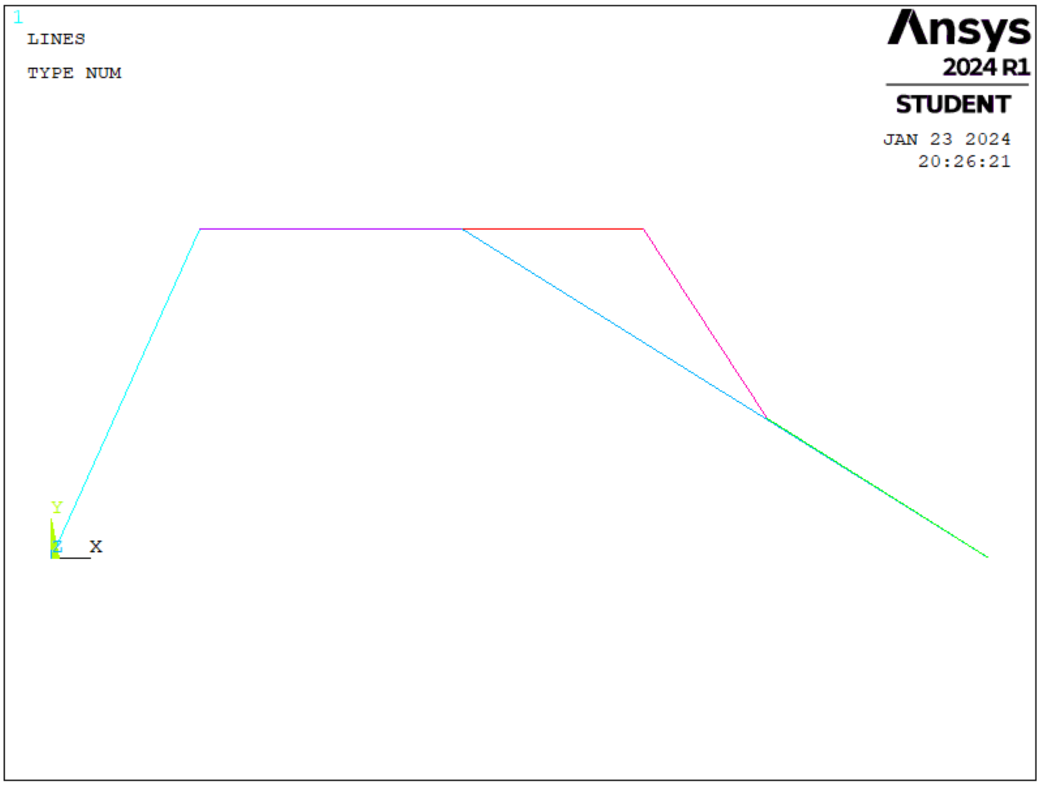
BIWO-03 Energy Methods, FEM

Assignment 5: FEM application using Ansys

The deformation of a fittness bench (truss/beam?) (shown below) due to external loads is to be investigated with the finite element method.

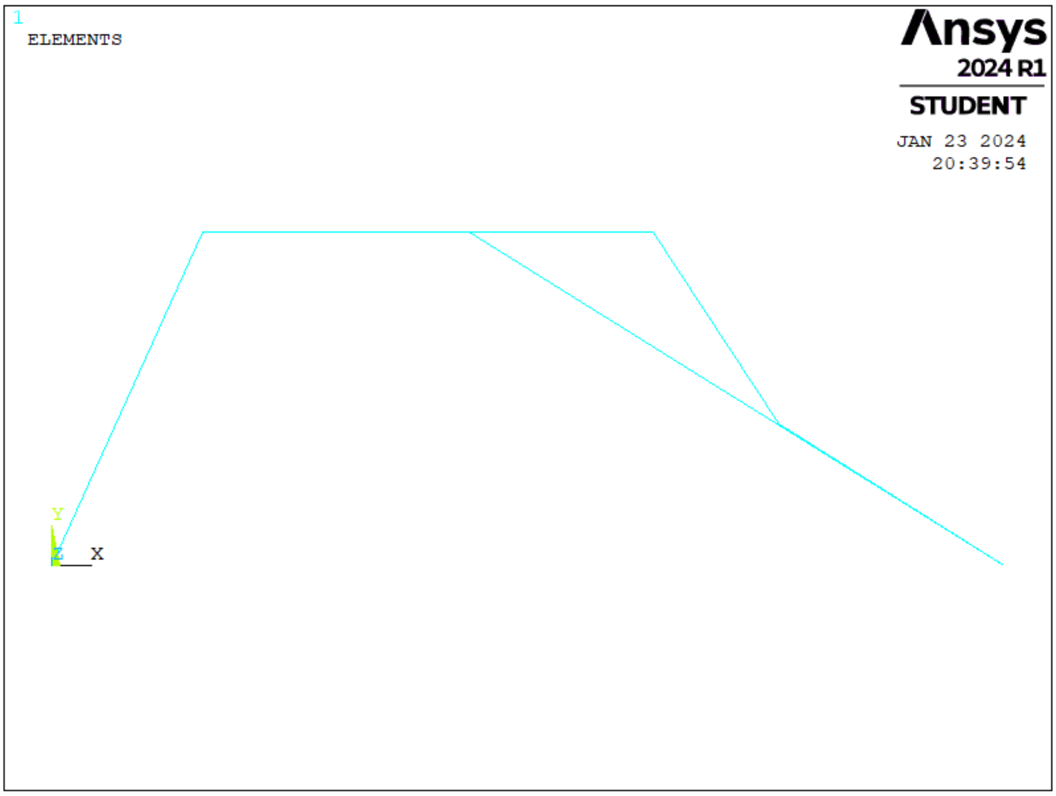
PART I

1. First we consider this structure as a 3D truss bar. The geometry is created by creating keypoints and then creating lines from those keypoints in Ansys:

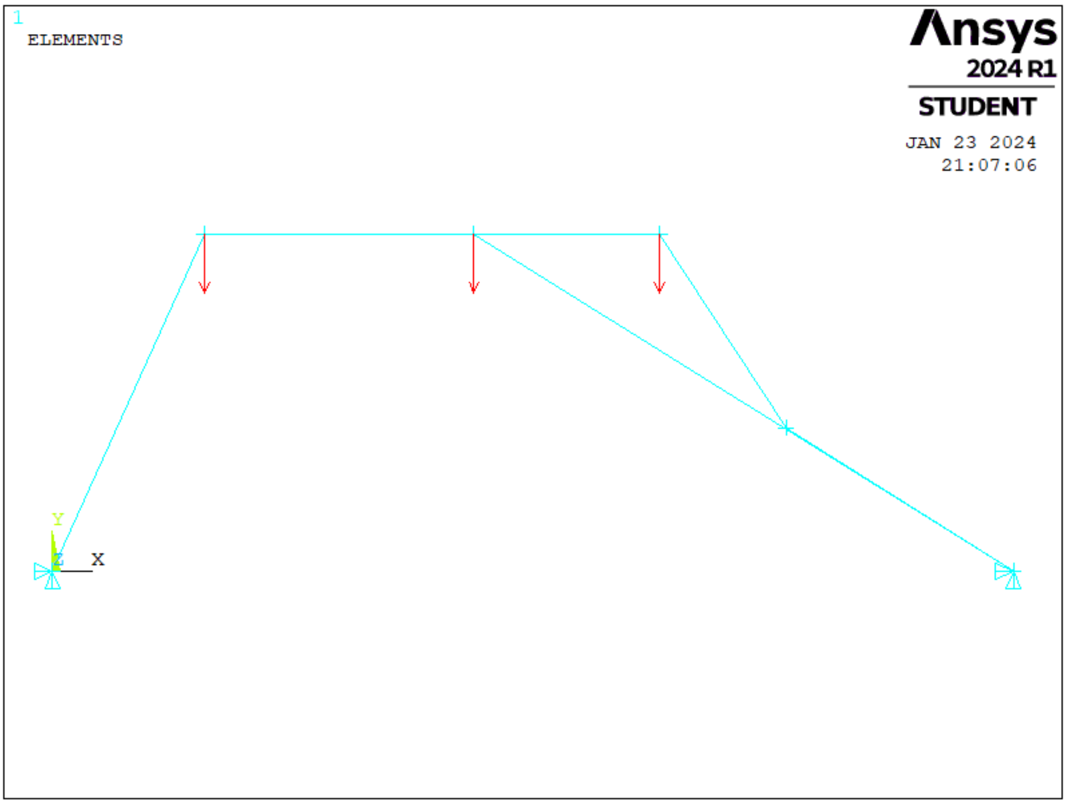


2. The element type chosen for this problem is the 3D truss bar provided through LINK180 in Ansys. The link elements are assumed to have a cross-sectional area of A = 1600 mm^2. In addition, assuming the use of steel, Young's modulus is assumed to be $E = 19000\ MPa$ and the poisson ratio to be $v = 0.3$.

3. The geometry is meshed with one bar per line segment in the geometry (a total of 6 elements):



4. The truss is assumed to be simply supported in plane and to be completely constrained out of plane. In addition on the truss due to external loads is approximated as point loads of 177 N on the joints. Thus, the boundary conditions and loads are showned as:

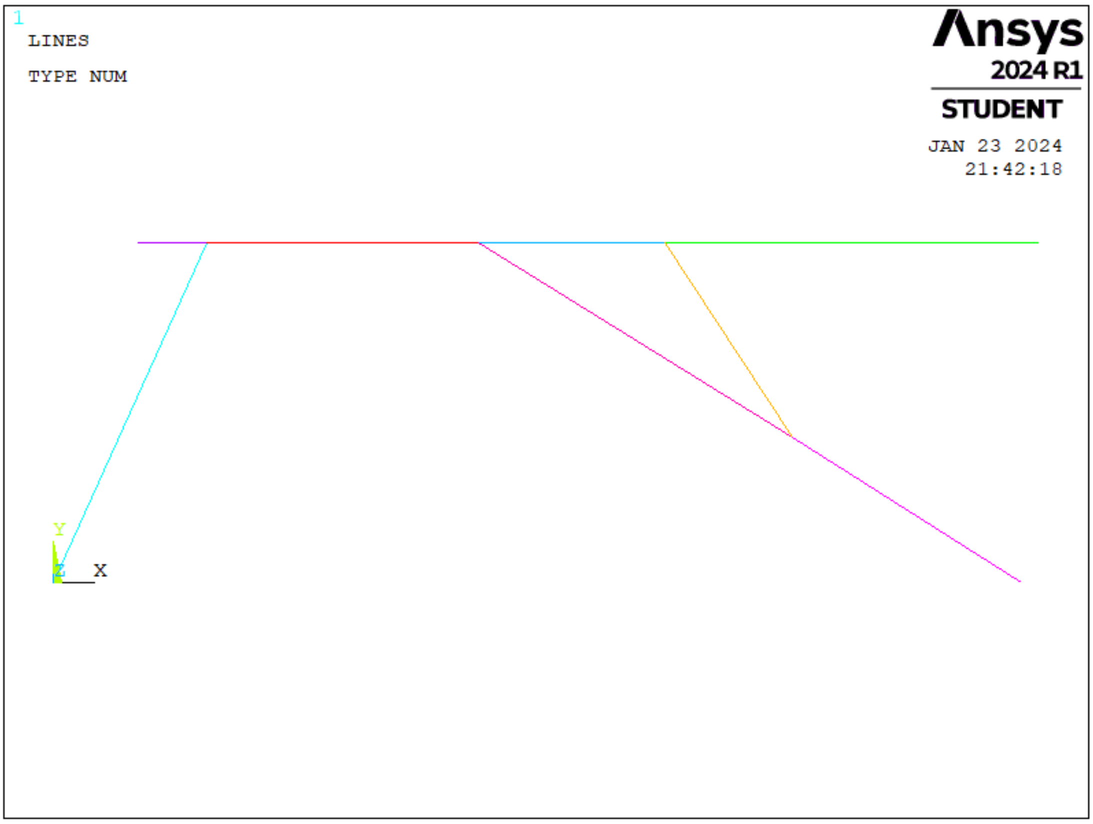


```
 *** ERROR ***                           CP =       0.7
50   TIME= 21:01:15
 There is at least 1 small equation solver pivot term (
e.g., at the UY
 degree of freedom of node 3).  Please check for an ins
ufficiently
 constrained model.
```

This error shows that I choose the false element type. If I change it to the beam element, no error will be reported, see the results below.

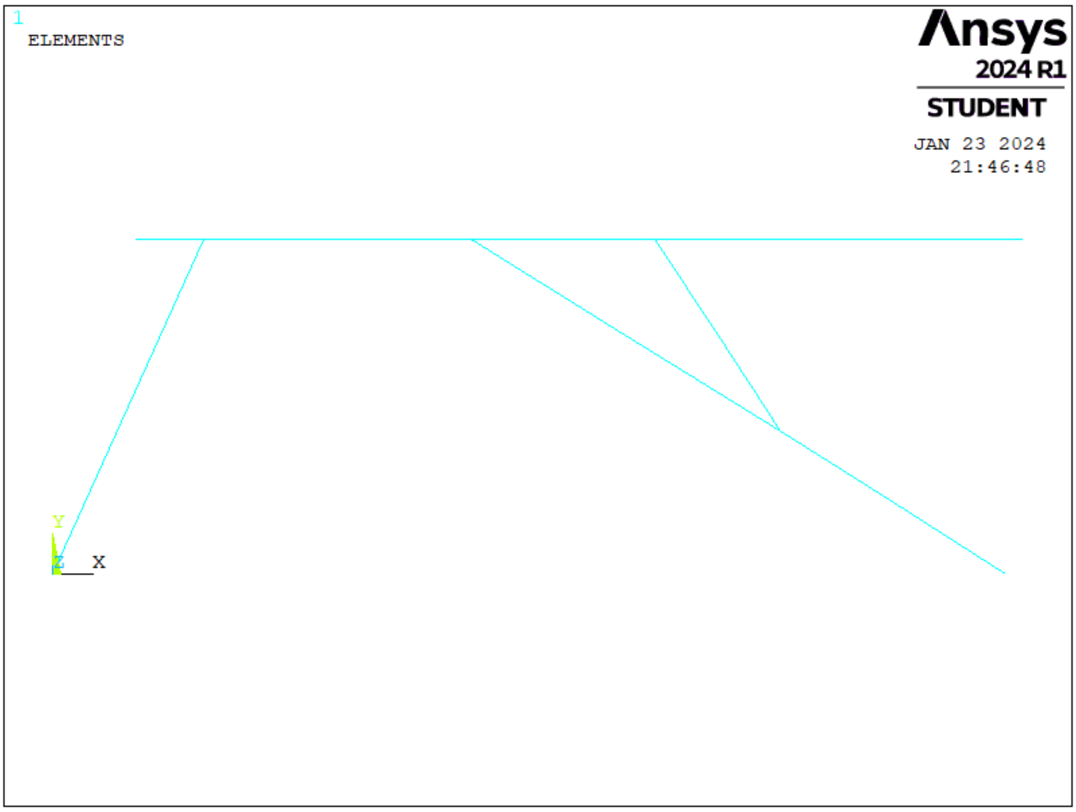
PART 2

1. First we consider this structure as a 3D truss bar. The geometry is created by creating keypoints and then creating lines from those keypoints in Ansys(I add the whole elements of this bench structure, because the beam elements can consider the moment, truss can not):

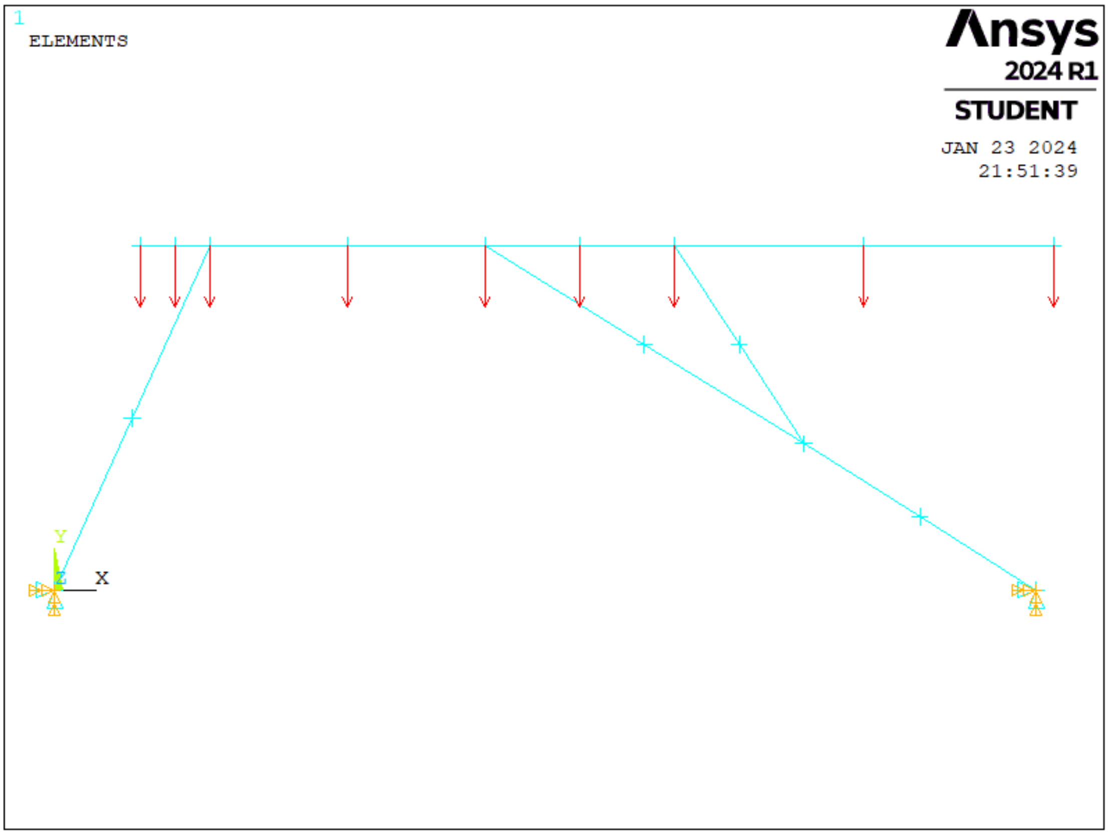


2. The element type chosen for this problem is the 3D truss bar provided through LINK180 in Ansys. The link elements are assumed to have a cross-sectional area of A = 1600 mm^2. In addition, assuming the use of steel, Young's modulus is assumed to be $E = 19000\ MPa$ and the poisson ratio to be $v = 0.3$.

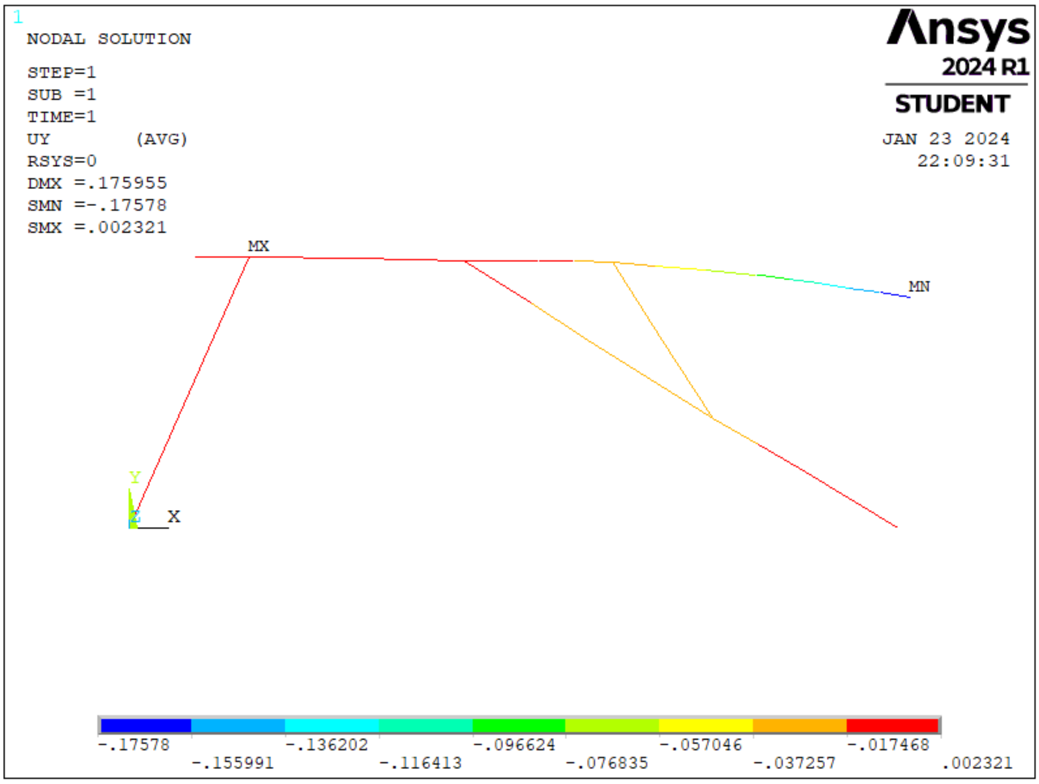
3. The geometry is meshed with one bar per line segment in the geometry (a total of 8 elements):

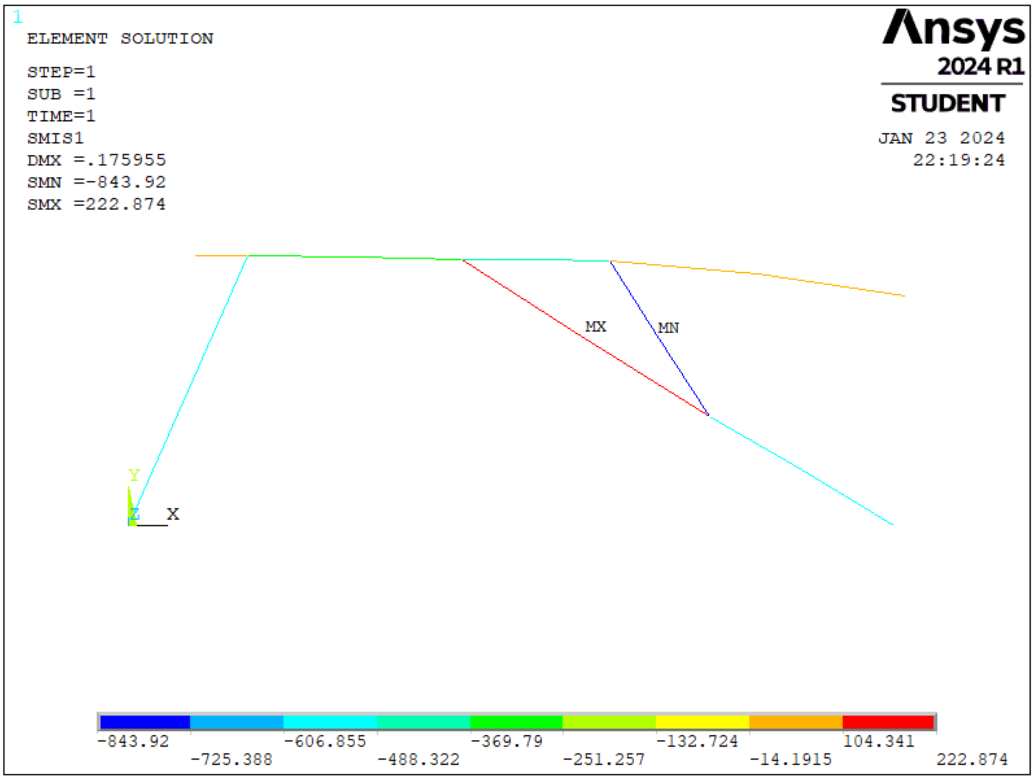


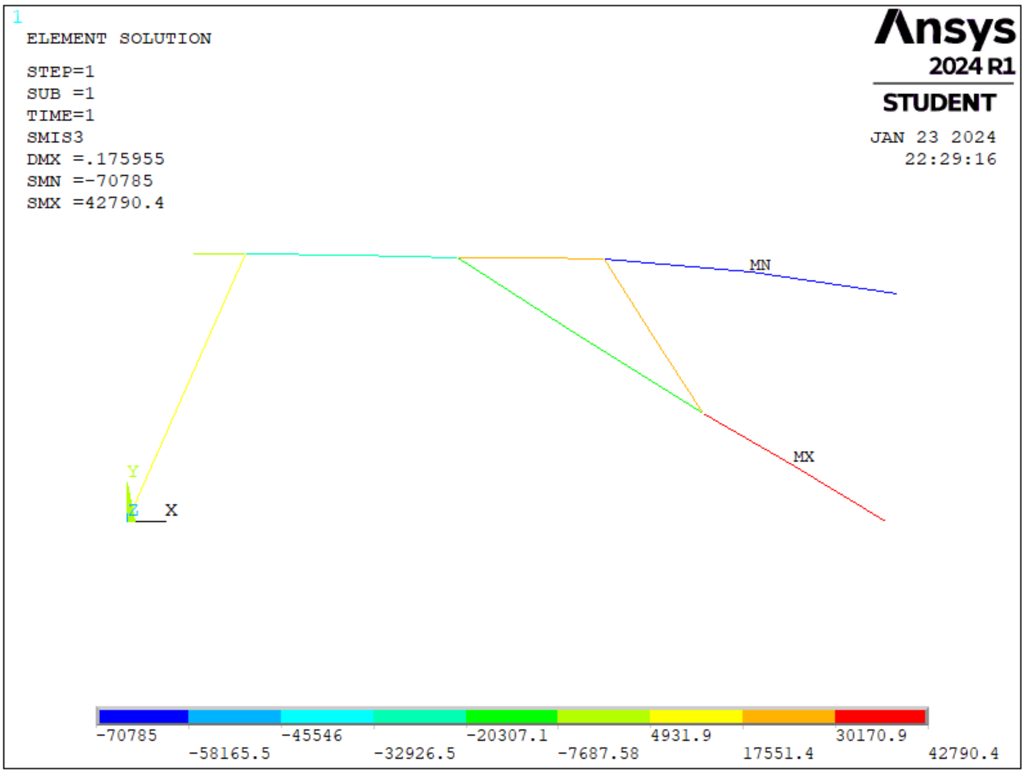
4. The beam structure is assumed to be simply supported in plane and to be completely constrained out of plane. In addition on the beam due to external loads is approximated as point loads of 117 N on the joints. Thus, the boundary conditions and loads are showned as:



5. After solving the problem, a contour plot of the vertical displacement and the forces in the members are shown. Note that the displaced structure is exaggerated to better see how the structure displaces. A maximum downward displacement of -0.17 mm is obtained in the end region of the upper beam. The largest axial compressive stress is of 843.92 Mpa and the largest axial tensile stress is 222.874 Mpa. The maximum compressive stress of bending in the beam is 70785 Mpa and the maximum tensile stress of bending in the beam is 42790.4 Mpa.







6. Input file used for this simulation:

PART 1

```
/clear
/title,

!!!Preprocessing
/prep7                                              ! start preprocessor

!reverse video (to make background white instead of black)
/RGB,INDEX,100,100,100,0
/RGB,INDEX,80,80,80,13
/RGB,INDEX,60,60,60,14
/RGB,INDEX,0,0,0,15

!!!Material definition
MP,EX,1,190000      ! N/mm^2    !Young's Modulus
MP,NUXY,1,0.3                        ! Poisson's ratio


!!! Geometry definition
!Units in mm
K,1,    0,0,0
K,2,    180,400,0
K,3,    500,400,0
K,4,    720,400,0
```

```
K,5,    870,170,0
K,6,    1140,0,0

L,1,2 !line created using keypoints 1 & 2
L,2,3 !line created using keypoints 2 & 3
L,3,4 !line created using keypoints 3 & 4
L,3,6 !line created using keypoints 3 & 7
L,4,5 !line created using keypoints 4 & 5
L,5,6 !line created using keypoints 5 & 6
!plot lines
LPLOT


!!! Element definition
ET,1,LINK 180          !element type 180 = 2 nodes, 3 DOF per node
SECTYPE,1,LINK      !define cross section to be rectangular for link element
SECDATA,40              !40 mm x 40 mm cross section


LESIZE,ALL,,,1          ! only line element per line for all lines
Lmesh,all                    ! mesh all lines selected
eplot


!!! Solution
/solu                                  ! start solution phase
antype,static             ! static analysis
outres,all,all              ! save results of all iterations


!Boundary conditions
d,all,UZ,0                          ! constrain all nodes in out of plane direction(i.e.,z-direction)
nsel,s,loc,X,0.0               ! select all nodes in plane x=0
d,all,all,0                           ! constrain node at this location in all directions
nsel,s,loc,X,1140.0    ! select all nodes in plane x=1140
d,all,all,0                           ! constrain at this location in all directions
!csys,0                   ! fix bottom of the support structure



!Load
nsel,s,loc,Y,400            ! select all nodes in plane y=400
F,all,FY,-177                    !N !apply a force of -177 in y direction for all selected nodes
```

```
allsel,all                              ! select everything
! show boundary conditions
\PBC,all,,1

allsel,all                              ! select all nodes, lines, areas, volumes and elements

solve

finish


!!! Postprocessing
/post1
set,last
PLNSOL,U,Y                         ! plot vertical displacement
PLESOL,SMISC,1            !plot forces in truss members

!/dscale,,1    !this command can be used to change the exaggeration of the displaced
structure
                    !for a multiplier of 1, the displaced structure is at true scale
```

PART 2:

```
/clear
/title,

!!!Preprocessing
/prep7                                                                      ! start preprocessor

!reverse video (to make background white instead of black)
/RGB,INDEX,100,100,100,0
/RGB,INDEX,80,80,80,13
/RGB,INDEX,60,60,60,14
/RGB,INDEX,0,0,0,15

!!!Material definition
MP,EX,1,190000      ! N/mm^2    !Young's Modulus
MP,NUXY,1,0.3                              ! Poisson's ratio


!!! Geometry definition
!Units in mm
K,1,    0,0,0
K,2,    100,400,0
K,3,    180,400,0
K,4,    500,400,0
```

```
K,5,    720,400,0
K,6,    1160,400,0
K,7,    870,170,0
K,8,    1140,0,0

L,1,3 !line created using keypoints 1 & 3
L,2,3 !line created using keypoints 2 & 3
L,3,4 !line created using keypoints 3 & 4
L,4,5 !line created using keypoints 4 & 5
L,4,7 !line created using keypoints 4 & 5
L,5,6 !line created using keypoints 5 & 6
L,5,7 !line created using keypoints 5 & 7
L,7,8 !line created using keypoints 5 & 6
!plot lines
LPLOT


!!! Element definition

ET,1,BEAM189            !element type 180 = 2 nodes, 3 DOF per node
SECTYPE,1,BEAM,RECT     !define cross section to be rectangular for link element
SECDATA,40,40           !40 mm x 40 mm cross section



LESIZE,ALL,,,1          ! only line element per line for all lines
Lmesh,all               ! mesh all lines selected
eplot


!!! Solution
/solu                   ! start solution phase
antype,static           ! static analysis
outres,all,all          ! save results of all iterations


!Boundary conditions
d,all,UZ,0              ! constrain all nodes in out of plane direction(i.e.,z-direction)
nsel,s,loc,X,0.0        ! select all nodes in plane x=0
d,all,all,0             ! constrain node at this location in all directions
nsel,s,loc,X,1140.0     ! select all nodes in plane x=1140
d,all,all,0             ! constrain at this location in all directions
!csys,0                 ! fix bottom of the support structure
```

```
!Load
nsel,s,loc,Y,400             ! select all nodes in plane y=400
F,all,FY,-117                      !N !apply a force of -117 in y direction for all selected nodes

allsel,all                               ! select everything
! show boundary conditions
\PBC,all,,1

allsel,all                               ! select all nodes, lines, areas, volumes and elements

solve

finish


!!! Postprocessing
/post1
set,last
PLNSOL,U,Y                        ! plot vertical displacement
PLESOL,SMISC,1           ! plot axial stress in beam members
PLESOL,SMISC,3           ! plot bending stress in beam members

!/dscale,,1    !this command can be used to change the exaggeration of the displaced
structure
                 !for a multiplier of 1, the displaced structure is at true scale
```