University of study of Trento

Department of Industrial Engineering

Master of Science in Mechatronics Engineering

Mechanical design and machine elements

Report homework

Professor

Prof. Matteo Benedetti

Candidate

Francesco Argentieri

ID 183892

academic year 2015/2016

Contents

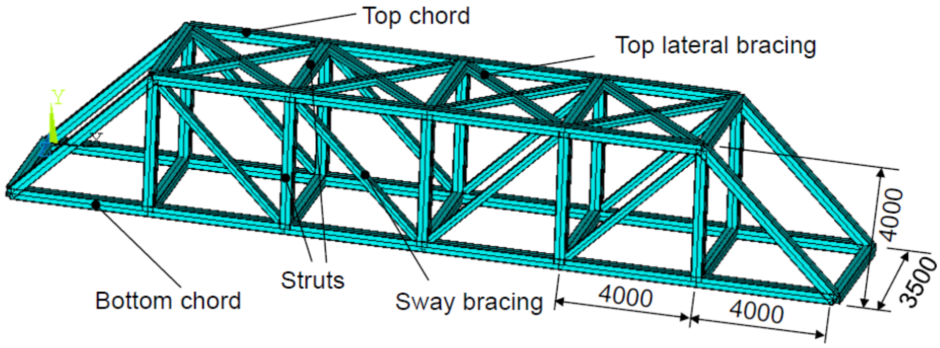
Chapter 1

Homework 1

1.1 Introduction

1.1.1 Problem 1

Elaborate a Finite Element model of the Pratt truss bridge shown in the figure in order to determine nodal deflection, reaction forces and axial stresses

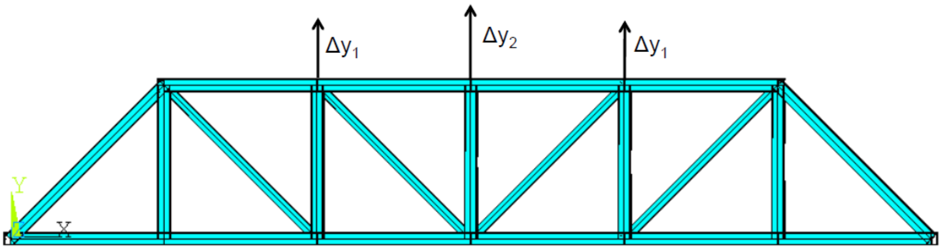


The two bottom chords are subjected to a vertical distributed load with intensity of 20000 N/m. The dimensions are given in mm. One side of the bridge is pinned-supported, the other is roller-supported. The trusses have the following cross-sectional areas:

Bottom chord,								
top chord, struts:	A1	=	1000	mm^2	material:	steel		
Sway bracing:	A2	=	600	mm^2		E	=	210 GPa
Top lateral bracing:	A3	=	400	mm^2		υ	=	0.3

1.1.2 Problem 2

The shape of the bridge is then modified by moving the y coordinate of the nodes of the top chord by $\Delta y1$ and $\Delta y2$ as shown in the figure. Determine the values of $\Delta y1$ and $\Delta y2$ that minimize the maximum deflection.



1.2 Approach the problem

It has defined a simplified diagram of the structure under consideration, making the following assumptions:

- bottom chord's node = 7;
- total lenght = 24 m;
- distritbute load $F = \frac{20000\,\frac{N}{m} * 24\,m}{7}$
- element type: Link180

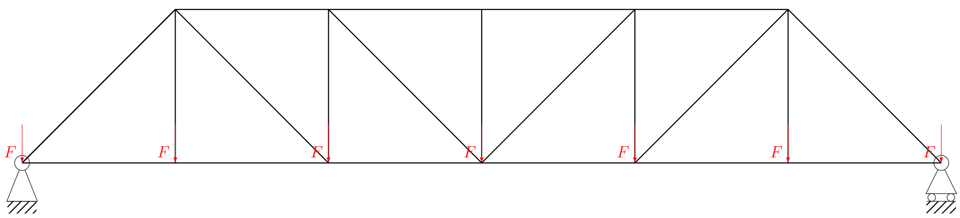


Figure 1.1: Bridge scheme

To fix the problem, we started the construction of the model building nodes and subsequently connected with *"truss"* elements, come the results shows in the figure 1.2. Constraints to the bridge ends were added as requested by the problem. A distributed force was applied along the length of the two bays. Later it was used the same model to analyze the second question.

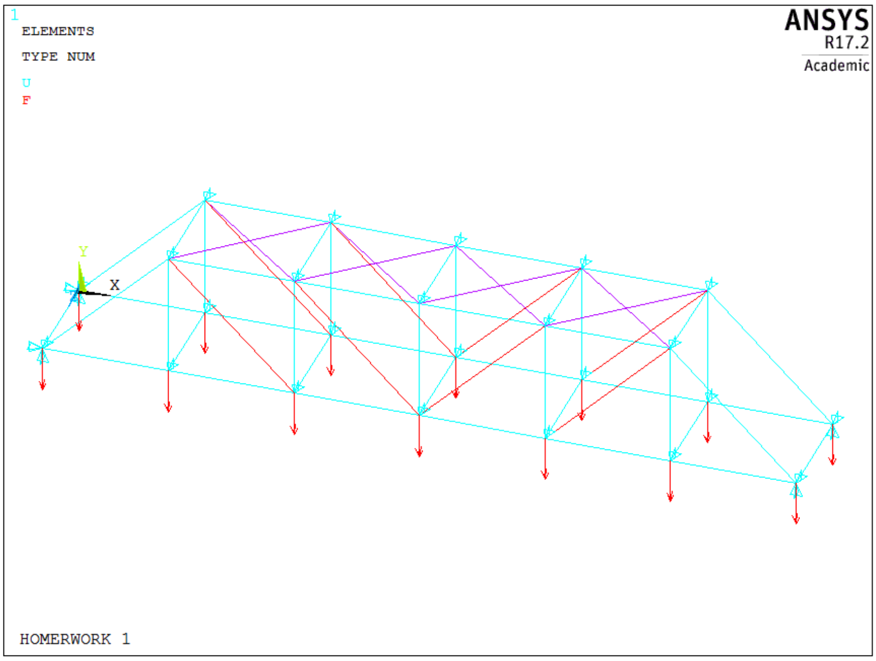


Figure 1.2: Model loaded and bound structure

1.3 Result

1.3.1 Problem 1

In the post processing simulation results are observed in the figures 1.3 and 1.4, where you can observe the distribution of axial forces and the distribution of axial stress respectively.

The displacement of the nodes is shown in the figure 1.5, where it is possible to observe that the maximum displacement is obtained in the vicinity of the nodes and is equal to $62,7759\,mm$.

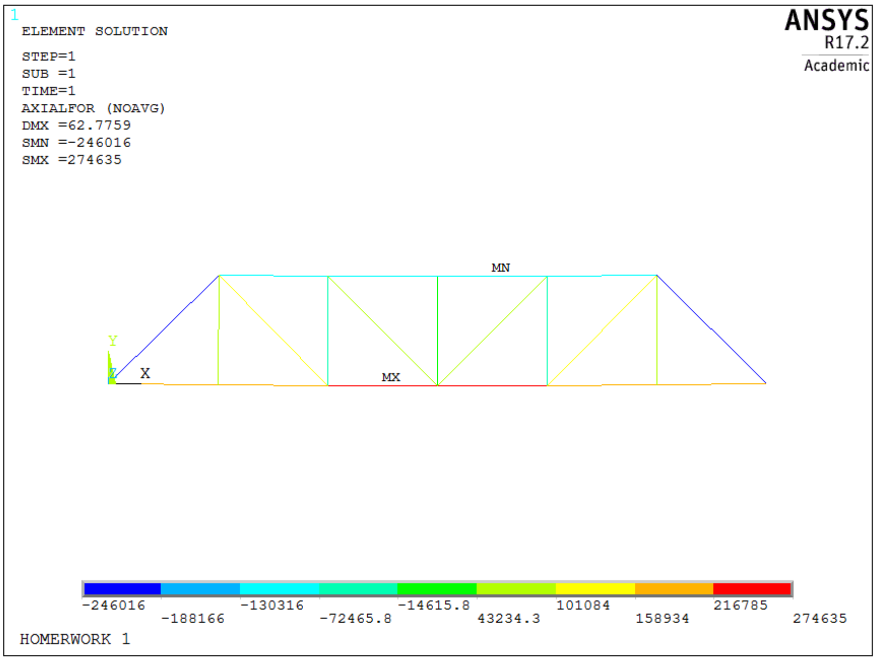


Figure 1.3: Distibution of axial force

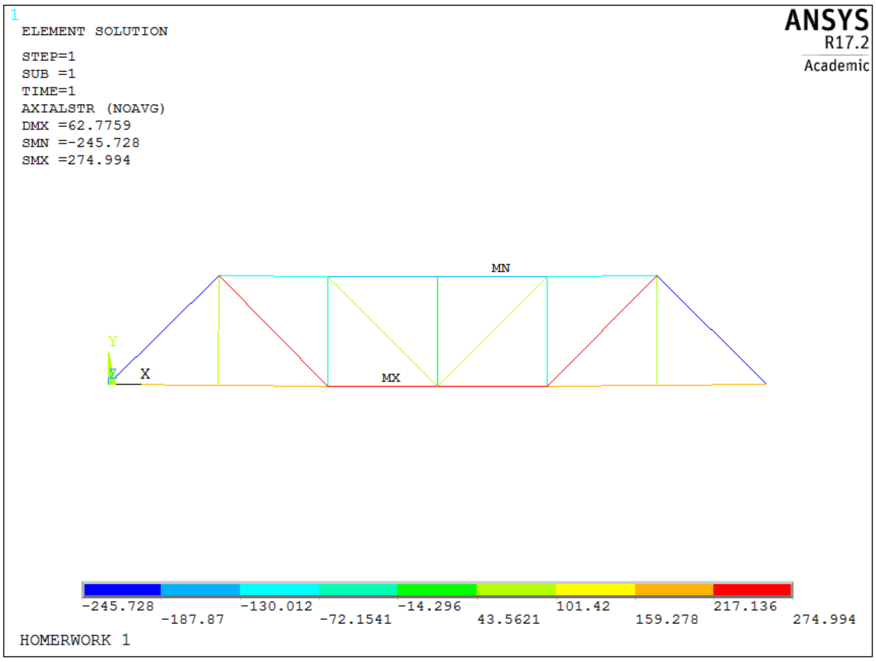


Figure 1.4: Distibution of axial stress

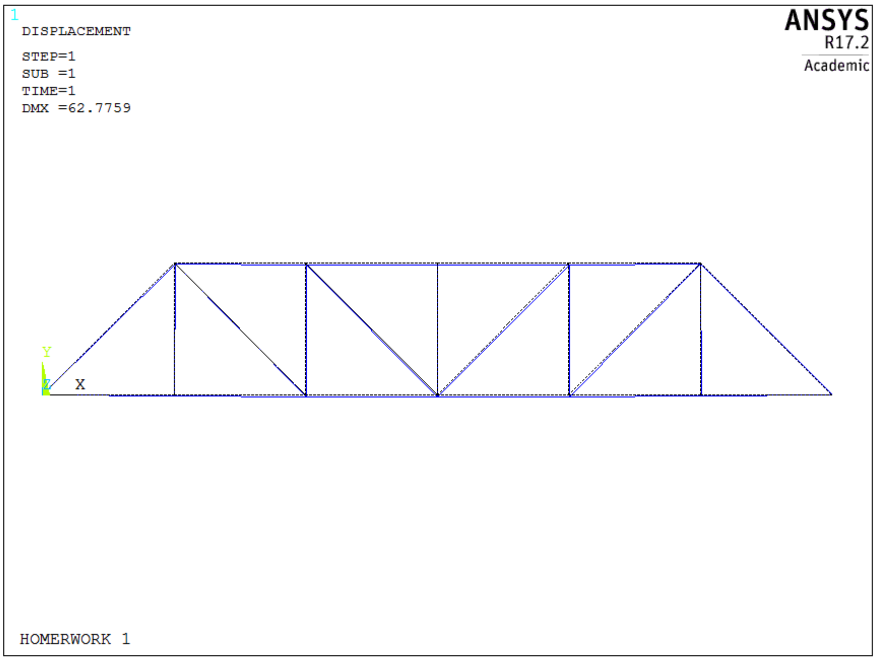


Figure 1.5: Displacement of the structure

1.3.2 Problem 2

For the second question we have used the command *NMODIF* as required to change the position of the nodes by varying the height in order to obtain the minimum deflection of the structure. It has been avoided during the execution of the loop, all those configurations like ”M” shape.

Number interaction	$\Delta y1$	$\Delta y2$	deflaction node 15 [mm]	deflaction node 16 [mm]	deflaction node 17 [mm]
806	7100,00	7100,00	-29,4321048924	-30,9574832350	-29,4321048924
807	7100,00	7200,00	-29,2877380752	-30,3744253498	-29,2877380752
808	7100,00	7300,00	-29,1891942493	-29,8702541041	-29,1891942493
809	7100,00	7400,00	-29,1335772891	-29,4401993777	-29,1335772891
810	7100,00	7500,00	-29,1182105600	-29,0798440735	-29,1182105600
811	7100,00	7600,00	-29,1406179070	-28,7850939406	-29,1406179070
812	7100,00	7700,00	-29,1985064925	-28,5521503107	-29,1985064925
813	7100,00	7800,00	-29,2897512823	-28,3774854310	-29,2897512823
814	7100,00	7900,00	-29,4123810029	-28,2578201188	-29,4123810029
815	7100,00	8000,00	-29,5645654149	-28,1901034907	-29,5645654149

Table 1.1: Displacement of bridge

In the graph 1.6 is observable the needed results of the iterations, we obtain by moving nodes of a value equal to $\Delta y1 = 3100\,mm$ to the node 15 and 17 and an increase equal to $\Delta y2 = 3500\,mm$ to the node 15.
The configuration of the structure with minimum deflection is observable in Figure 1.7 where the displacement is equal to $31,6387\,mm$.

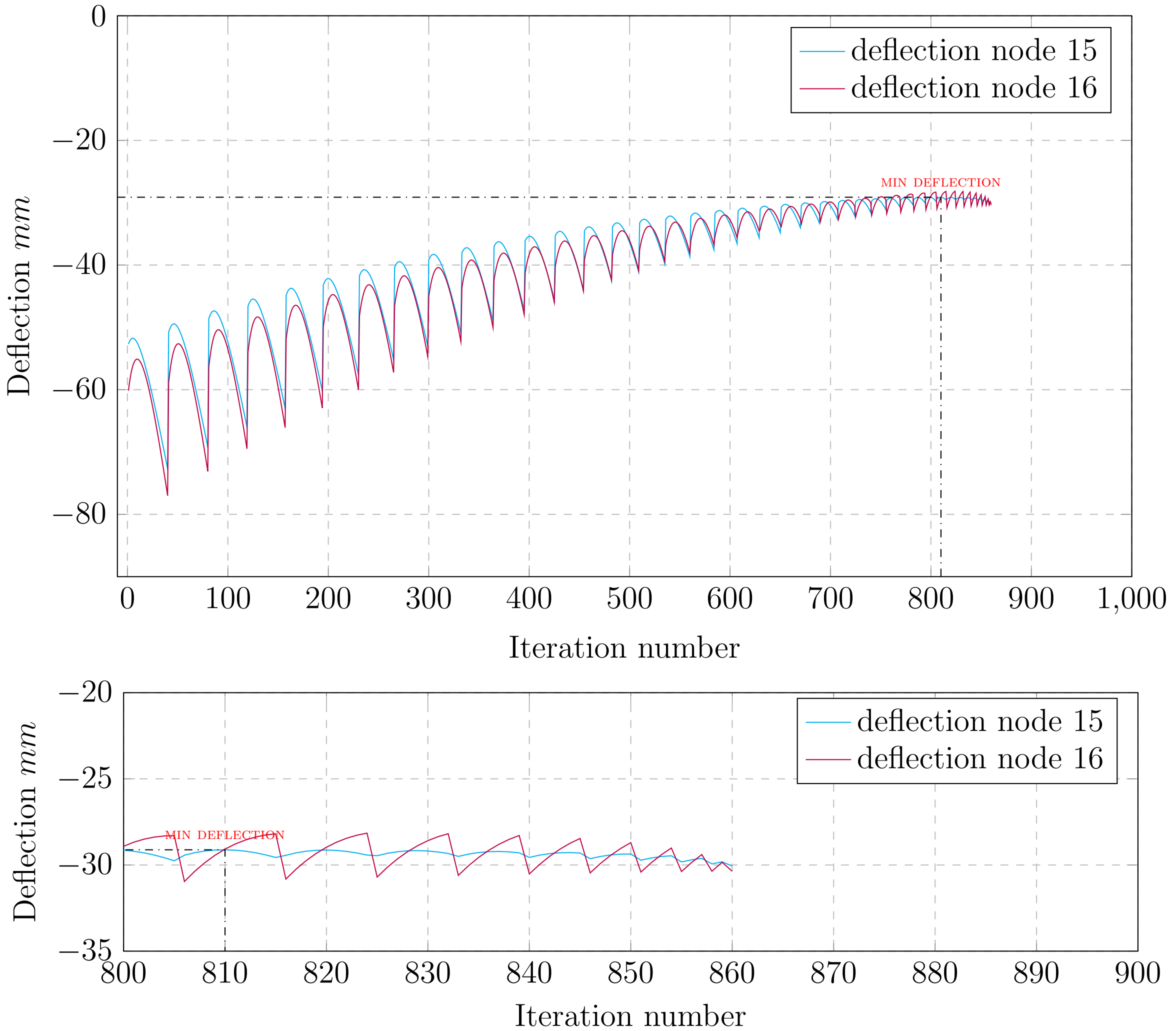


Figure 1.6: Displacement of bridge

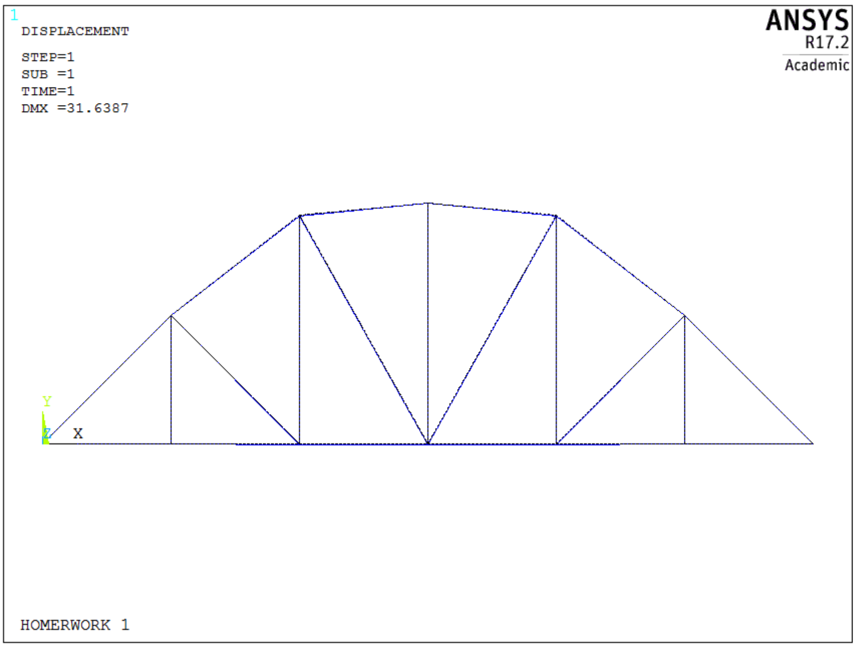


Figure 1.7: minimum deflaction

1.4 Command list

```
!*********************
!PROBLEM: HOMEWORK 1 *
!*********************

FINISH
/CLEAR,START,NEW
/TITLE,HOMERWORK 1
! >>> PARAMETERS MODEL <<<
*SET,bottom_chord,4000
*SET,top_chord,4000
*SET,h_struts,4000
*SET,o_struts,3500
*SET,sez_one,1000
*SET,sez_two,600
*SET,sez_three,400

! >>> PROPERTIES MATERIAL <<<
*SET,E_Young,210000
*SET,ni,0.3

! >>> LOAD CONDITIONS <<<
*SET,F,(20000*(bottom_chord/1000)*6)/7

/PREP7
ET,1,180
ET,2,180
ET,3,180
R,1,sez_one
R,2,sez_two
R,3,sez_three
MP,EX,1,E_Young
MP,PRXY,1,ni
N,1,0,0,0
N,2,bottom_chord,0,0
N,3,bottom_chord*2,0,0
N,4,bottom_chord*3,0,0
N,5,bottom_chord*4,0,0
N,6,bottom_chord*5,0,0
N,7,bottom_chord*6,0,0
N,8,bottom_chord,h_struts,0
N,9,bottom_chord*2,h_struts,0
N,10,bottom_chord*3,h_struts,0
N,11,bottom_chord*4,h_struts,0
N,12,bottom_chord*5,h_struts,0
N,13,0,0,o_struts
N,14,bottom_chord,0,o_struts
N,15,bottom_chord*2,0,o_struts
N,16,bottom_chord*3,0,o_struts
N,17,bottom_chord*4,0,o_struts
N,18,bottom_chord*5,0,o_struts
N,19,bottom_chord*6,0,o_struts
N,20,bottom_chord,h_struts,o_struts
N,21,bottom_chord*2,h_struts,o_struts
N,22,bottom_chord*3,h_struts,o_struts
N,23,bottom_chord*4,h_struts,o_struts
N,24,bottom_chord*5,h_struts,o_struts
SAVE
```

```
! >>> ELEMENT <<<
TYPE,1
Real,1
Mat,1

E,1,2
E,2,3
E,3,4
E,4,5
E,5,6
E,6,7
E,8,9
E,9,10
E,10,11
E,11,12
E,13,14
E,14,15
E,15,16
E,16,17
E,17,18
E,18,19
E,20,21
E,21,22
E,22,23
E,23,24
E,1,13
E,2,14
E,3,15
E,4,16
E,5,17
E,6,18
E,7,19
E,1,8
E,12,7
E,24,19
E,13,20
E,2,8
E,3,9
E,4,10
E,5,11
E,6,12
E,14,20
E,15,21
E,16,22
E,17,23
E,18,24
E,8,20
E,9,21
E,10,22
E,11,23
E,12,24

TYPE,2
Real,2
Mat,1
E,8,21
E,9,22
E,10,23
E,11,24
E,20,9
E,21,10
E,22,11
E,23,12

TYPE,3
Real,2
Mat,1

E,3,8
E,4,9
E,4,11
E,5,12
E,20,15
E,21,16
E,16,23
E,17,24
SAVE

! >>> CONSTRAINT <<<
D,1,ux,0
D,1,uy,0
D,7,uy,0
D,13,ux,0
D,13,uy,0
D,19,uy,0
D,all,uz,0

! >>> FORCE <<<
F,1,fy,-F
F,2,fy,-F
F,3,fy,-F
F,4,fy,-F
F,5,fy,-F
F,6,fy,-F
F,7,fy,-F
F,13,fy,-F
F,14,fy,-F
F,15,fy,-F
F,16,fy,-F
F,17,fy,-F
F,18,fy,-F
F,19,fy,-F
SAVE

! >>> SOLUTION <<<
/SOLU
NLGEOM,ON
SOLCONTROL,ON
TIME,1
PIVCHECK,OFF
SOLVE
```

```
! >>> POST-PROCESS <<<
/POST1

PLDISP,1
PRNSOL,U,COMP

ETABLE,AXIALFORCE,SMISC,1
ETABLE,AXIALSTRESS,LS,1

PLETAB,AXIALFORCE

PLETAB,AXIALSTRESS

PRETAB,AXIALFORCE,AXIALSTRESS
PRRSOL,F

!************************
!SECOND PART HOMEWORK 1  *
!************************

! >>> MODIFY NODES <<<<
*DO,delta1,h_struts,2*h_struts,100
*DO,delta2,delta1,2*h_struts,100
/PREP7
NMODIF,9,bottom_chord*2,delta1,0
NMODIF,10,bottom_chord*3,delta2,0
NMODIF,11,bottom_chord*4,delta1,0
NMODIF,21,bottom_chord*2,delta1,o_struts
NMODIF,22,bottom_chord*3,delta2,o_struts
NMODIF,23,bottom_chord*4,delta1,o_struts

! >>> SOLUTION <<<
/SOLU
NLGEOM,ON
SOLCONTROL,ON
TIME,1
PIVCHECK,OFF
SOLVE

! >>> POST-PROCESS <<<
/POST1
*GET,UMAX15,NODE,15,U,Y
*GET,UMAX16,NODE,16,U,Y
*GET,UMAX17,NODE,17,U,Y

! >>> WRITE THE FILE <<<
*CFOPEN,umaxNEW,txt,,APPEND
*VWRITE,delta1,delta2,UMAX15,UMAX16,UMAX17
(F20.10,F20.10,F20.10,F20.10,F20.10)
*CFCLOS
FINISH
PARSAV,SCALAR,PARAMETRI,PARM
PARRES,NEW,PARAMETRI,PARM
*ENDDO
*ENDDO
```

Chapter 2

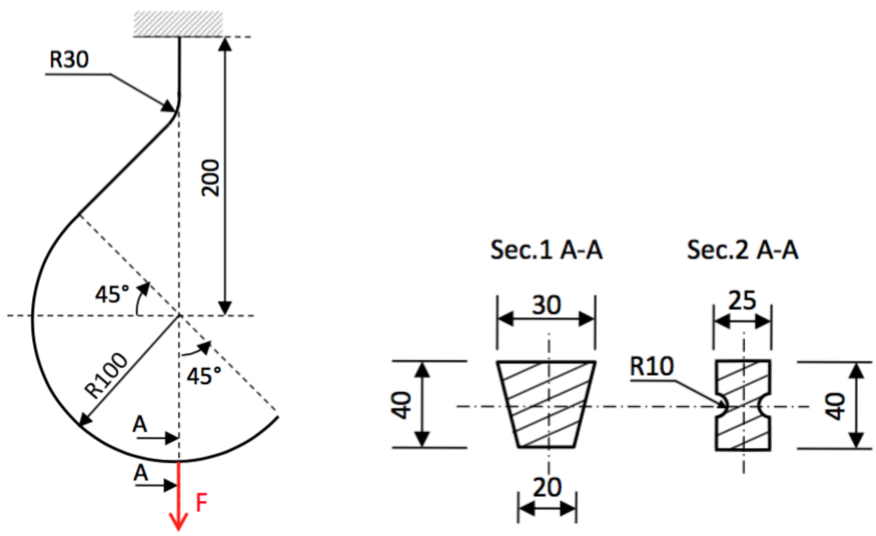
Homework 2

2.1 Introduction

Using beam elements, build up a FE model of the hook shown in the Figure. Determine the maximum displacement, the maximum bending stress, shear and bending moment diagrams. Compare the results obtained considering the cross-sections 1 and 2 shown below. Discuss the need of mesh refinement.

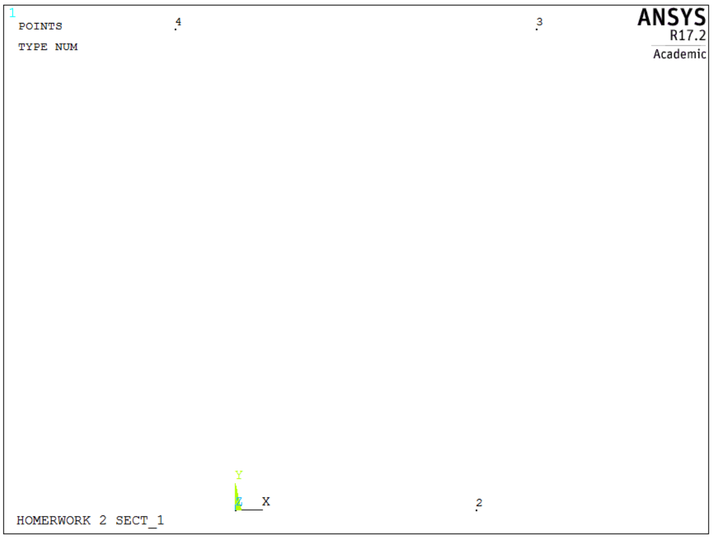
Data:

$F = 20\,kN \quad E = 205\,GPa \quad \upsilon = 0.3$

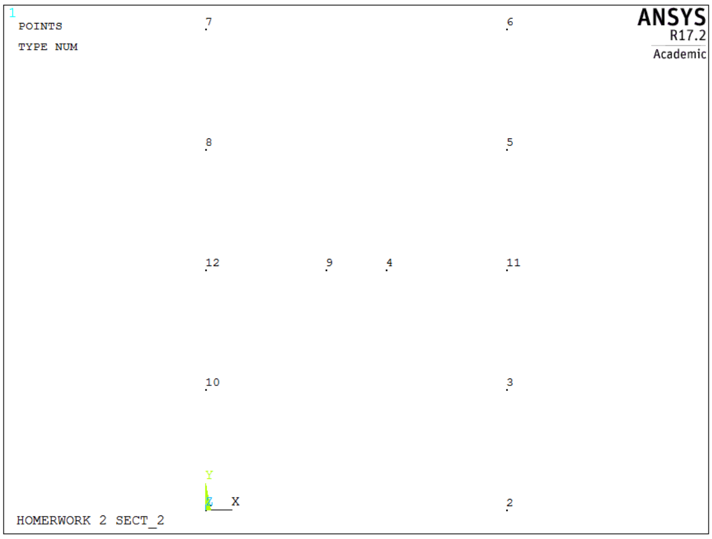


2.2 Approach the problem

For this problem we are first costritute the two custom cross sections, in the figures 2.1, 2.3 and 2.4; can be observed the construction phases then a free mesh is applied to both stat and saved in their respective files.

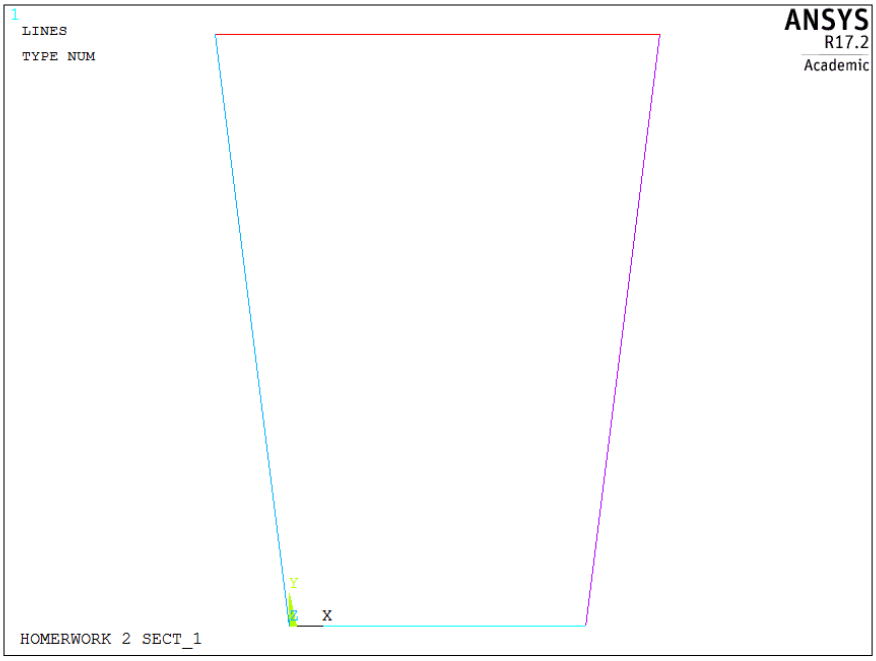


(a) Section 1

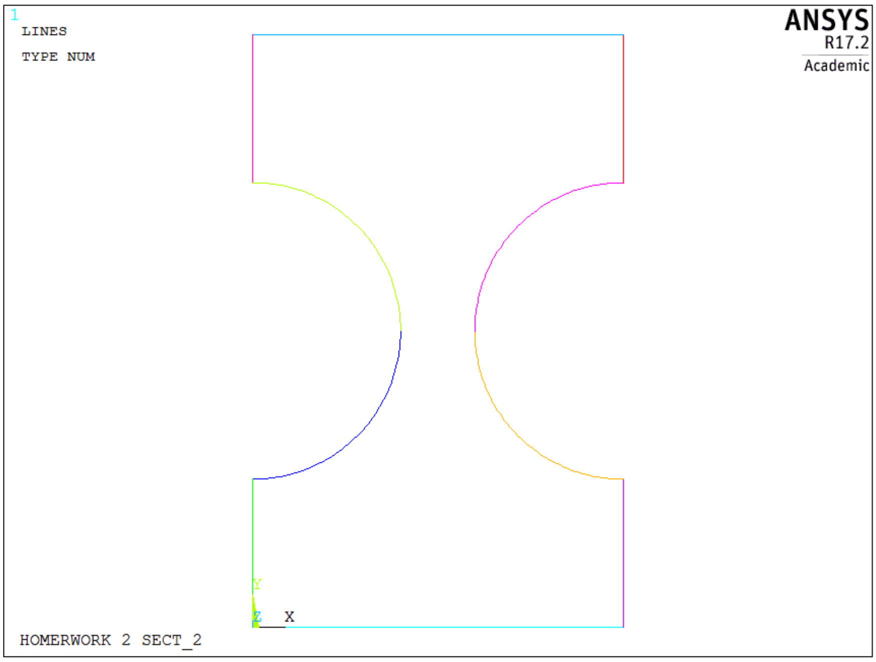


(b) Section 2

Figure 2.1: Keyponit Section



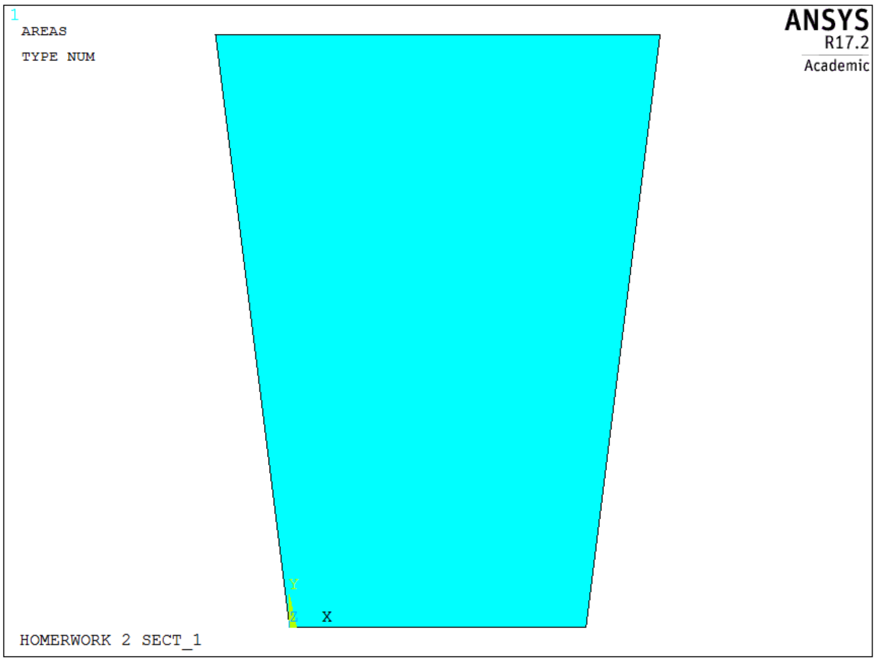
(a) Section 1



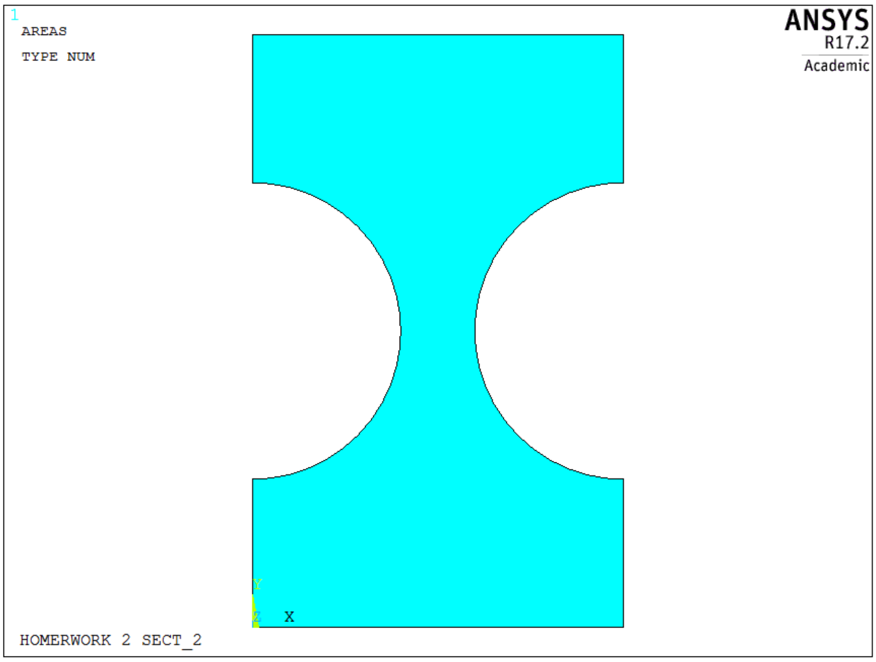
(b) Section 2

Figure 2.2: Geometry of sections

In this case it is defined the geometry of the hook through the use of *keypoints* and subsequently connected by lines, as ahow in fiugure 2.5. Then the mesh using the section $n°1$ was created in the previuos step and then the section $n°2$.
For the section $n°1$ the result in figure 2.4a, while section $n°2$ in picture 2.4b.

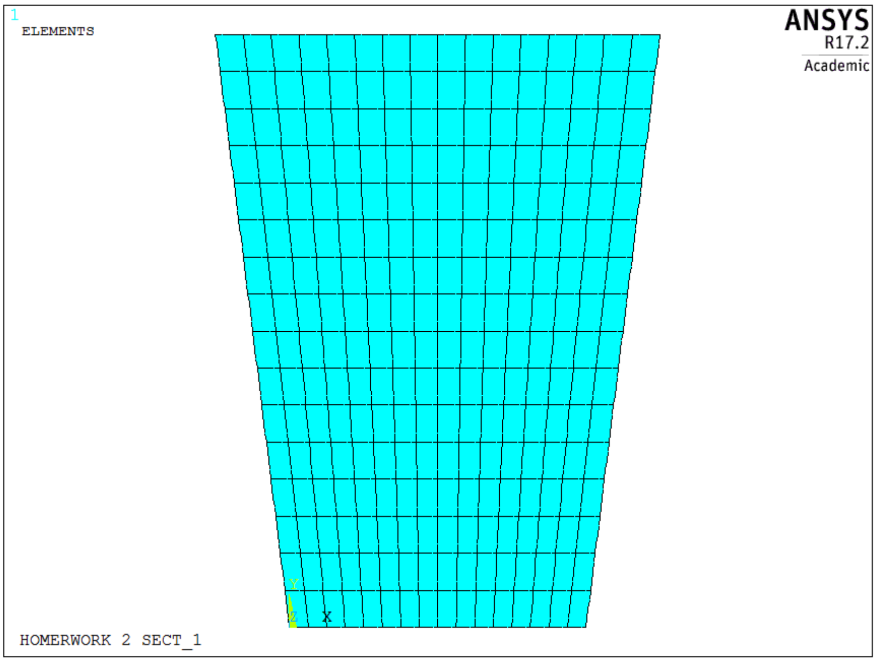


(a) Section 1

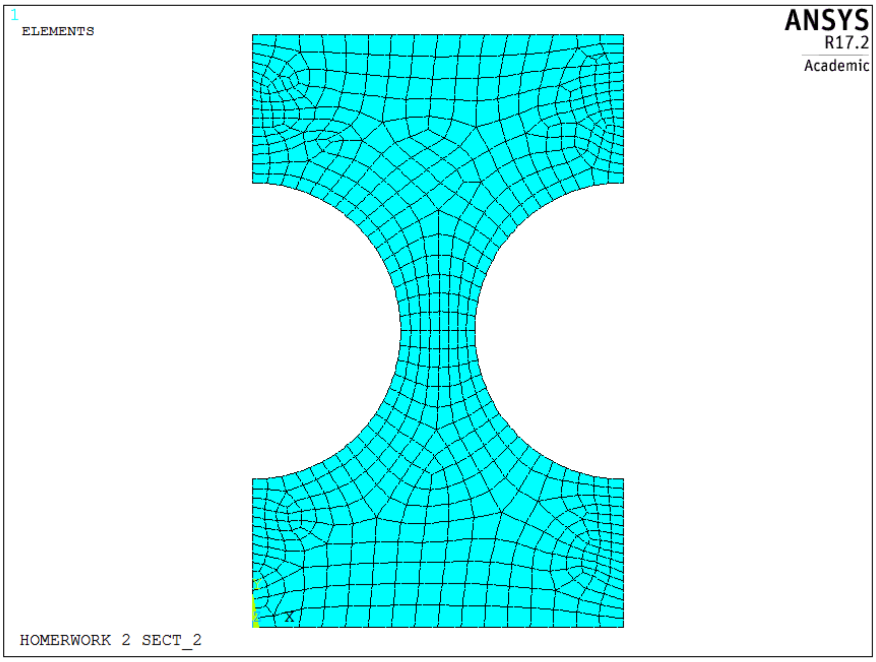


(b) Section 2

Figure 2.3: Area of sections

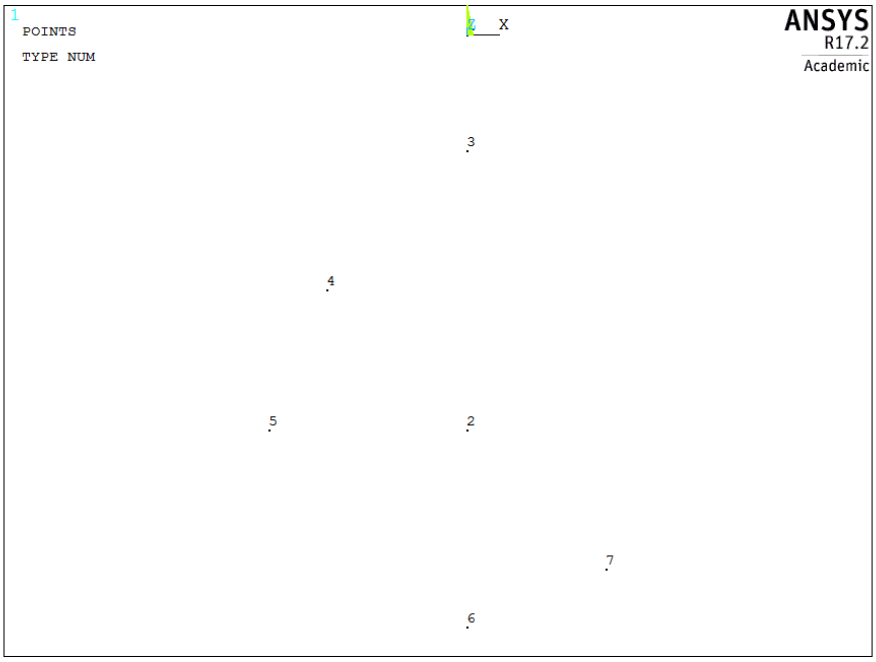


(a) Section 1



(b) Section 2

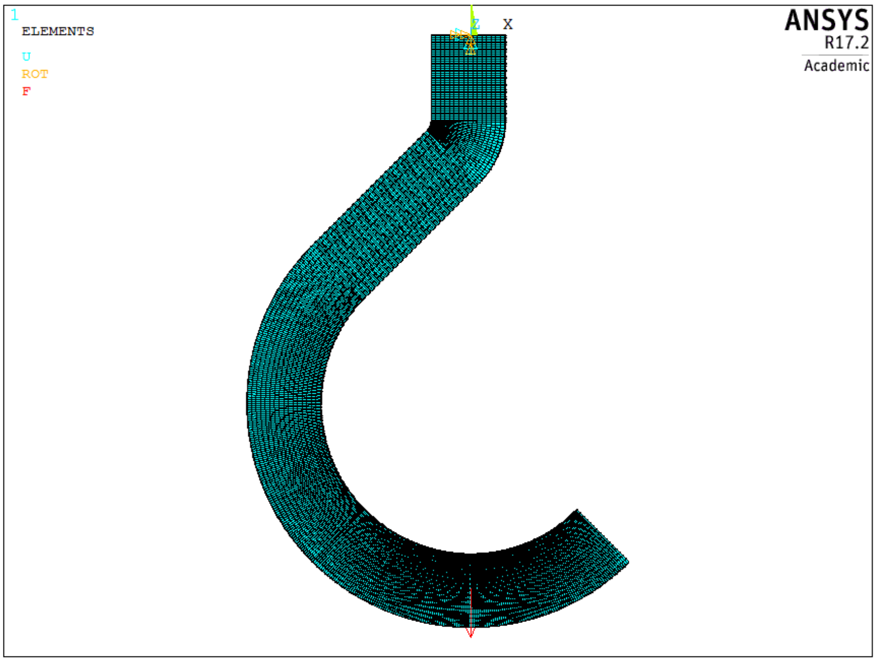
Figure 2.4: Meshed Section



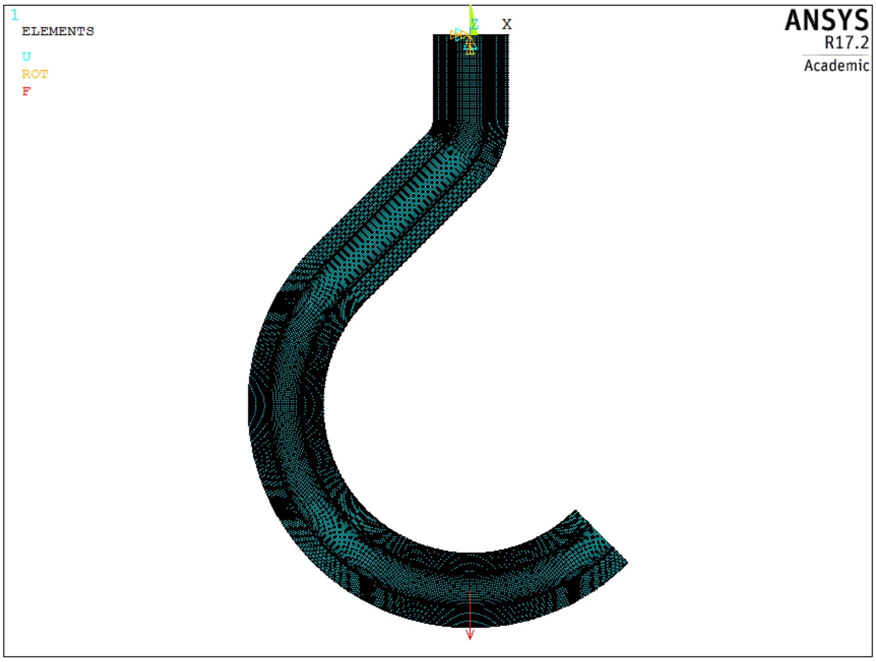
(a) Keypoint

(b) Full geometry

Figure 2.5: Geometry's hook



(a) Section 1



(b) Section 2

Figure 2.6: Hook meshed and costraint

2.3 Result

In conclusion is shown in the table of performance comparison of the two cross section.

Area	Displacement	maximum bending stress	minimum bending stress
mm^2	mm	MPa	MPa
1000	2,59533	283,82	-324,366

Table 2.1: Recap section 1

Area	Displacement	maximum bending stress	minimum bending stress
mm^2	mm	MPa	MPa
658,84	2,8004	318,819	-318,819

Table 2.2: Recap section 2

1. It is observed in the first section a greater deformation despite the area is larger in size when compared with the second.
2. Whereas the difference in the area of the sections, the displacement of the section 2 is similar to the first.
3. Section 2 does not show differences in the stress since Section 1.

In conclusion it is observed a better performance of the section 2 to equal forces applied. Subsequently, the analysis is repeated increased the division into examination number generating, therefore, a more dense mesh from which it is observed that the obtained have results are very close, in the table 2.3, 2.4.

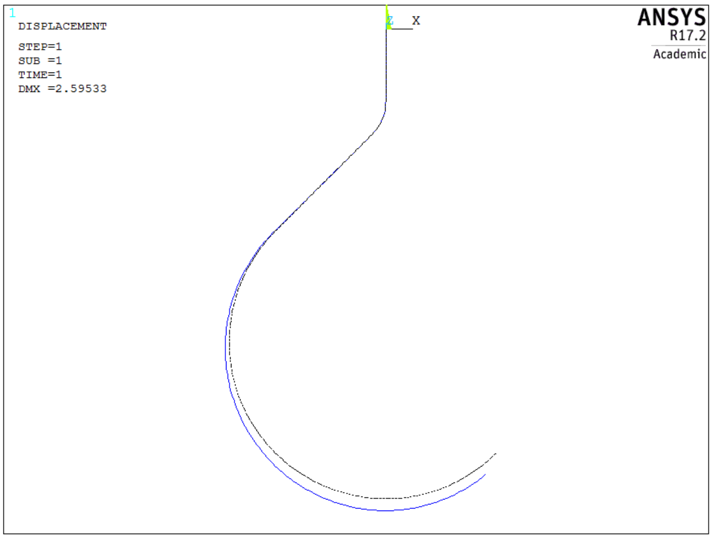
size	section	Bending Stress min	bending stress max	displacement
		MPa	MPa	mm
1	1	-324,327	283,786	2,59533
4	1	-324,366	283,82	2,59533
7	1	-324,45	283,894	2,59533
10	1	-324,584	284,011	2,59533
16	1	-324,989	284,365	2,59533
22	1	-325,36	284,69	2,59533

Table 2.3: Sensibility result to mesh refiniment

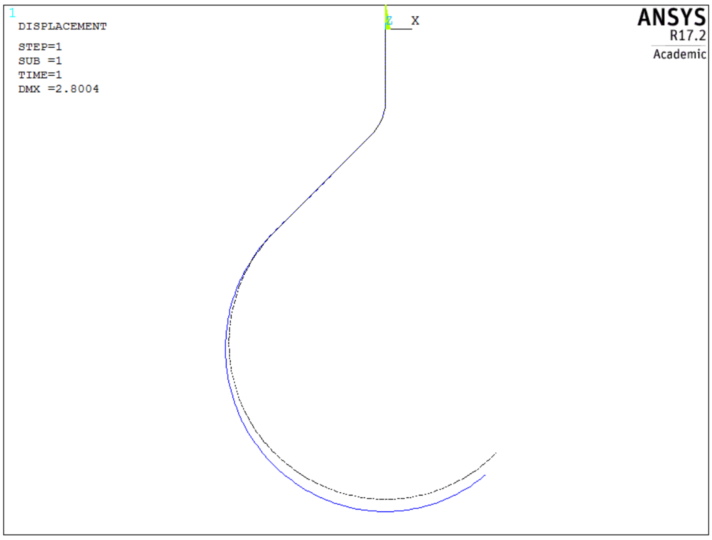
size	section	Bending Stress min MPa	bending stress max MPa	displacement mm
1	2	-318,78	318,78	2,8004
4	2	-318,819	318,819	2,8004
7	2	-318,901	318,901	2,8004
10	2	-319,033	319,033	2,8004
16	2	-319,431	319,431	2,8004
22	2	-319,796	319,796	2,8004

Table 2.4: Sensibility result to mesh refiniment

The following pictures shows the results obtained from the simulation.

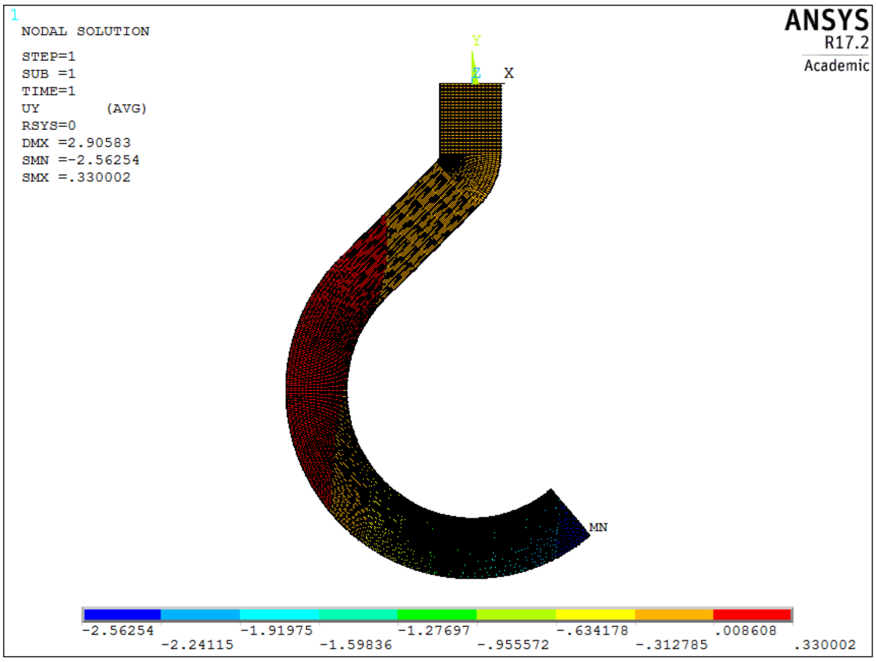


(a) Section 1

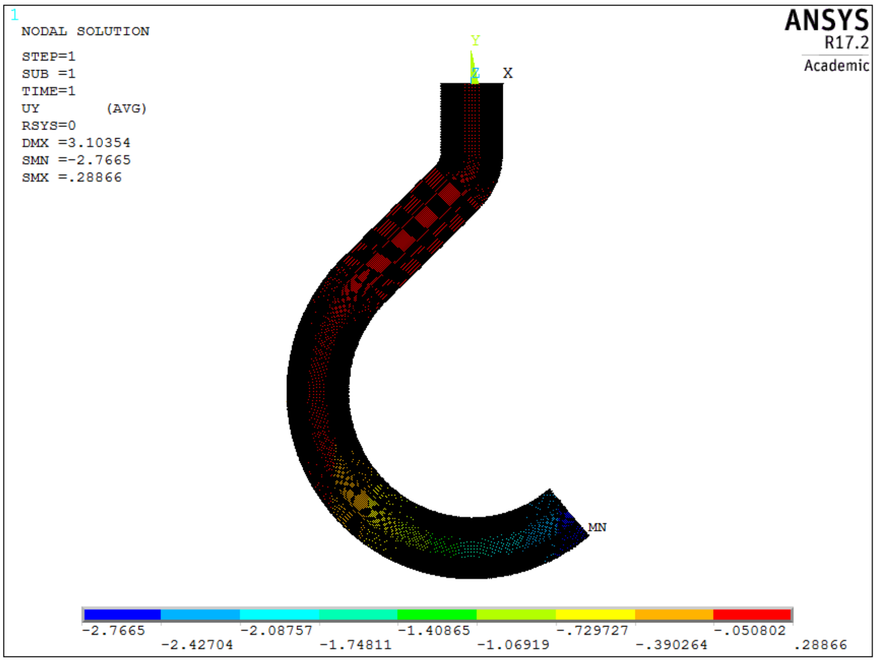


(b) Section 2

Figure 2.7: Result Displacement diagram

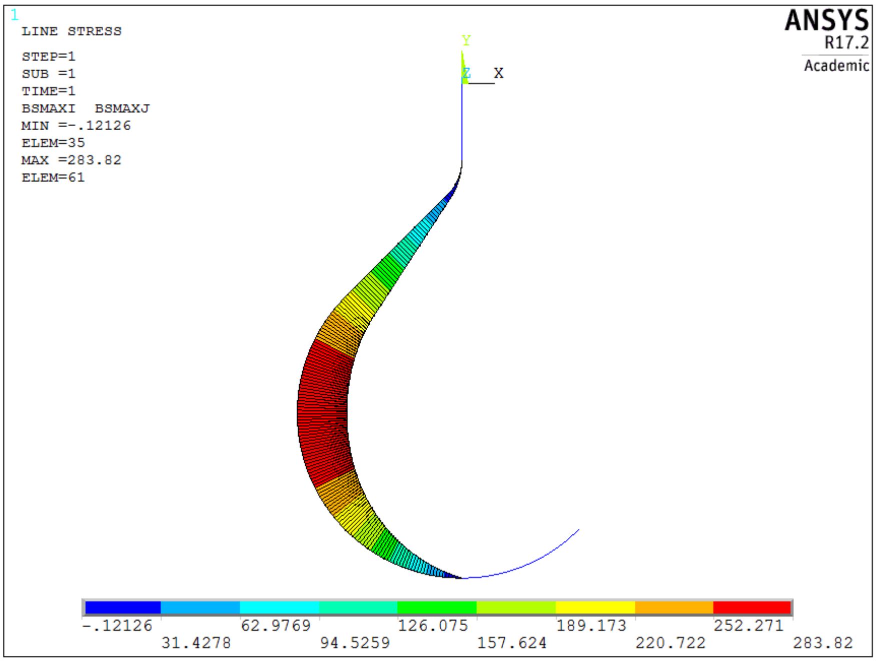


(a) Section 1

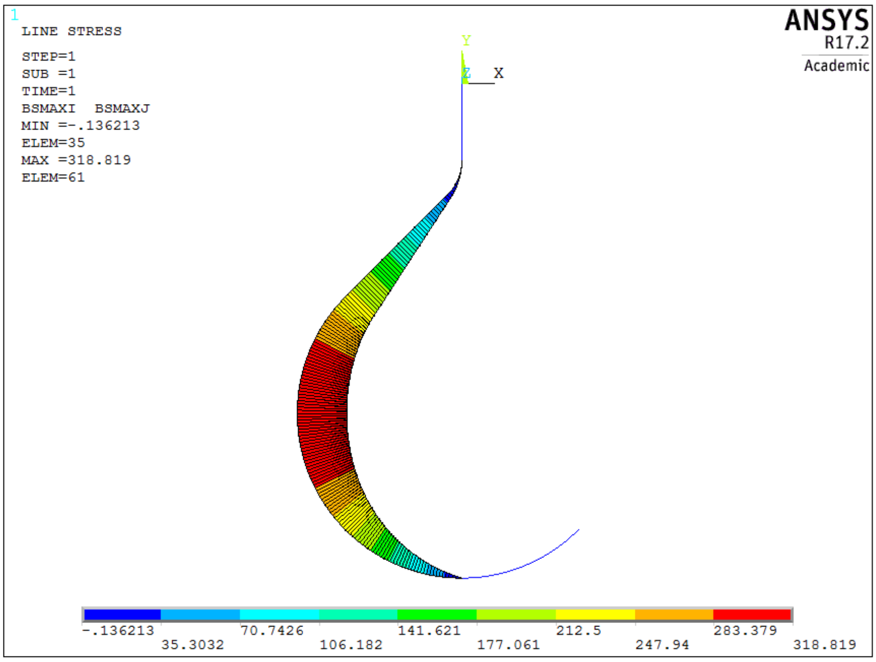


(b) Section 2

Figure 2.8: Result nodal solution diagram

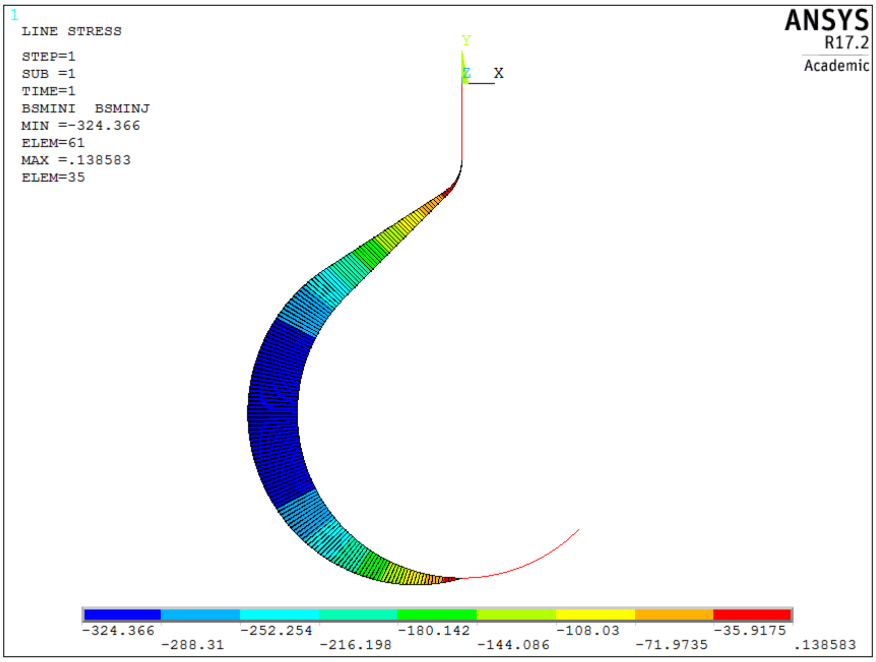


(a) Section 1

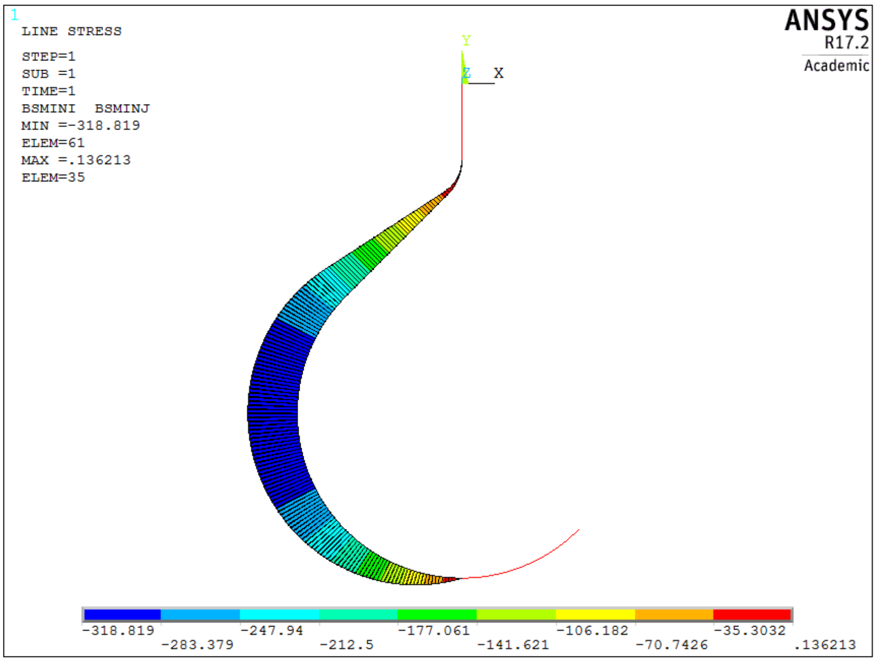


(b) Section 2

Figure 2.9: Result max bending diagram

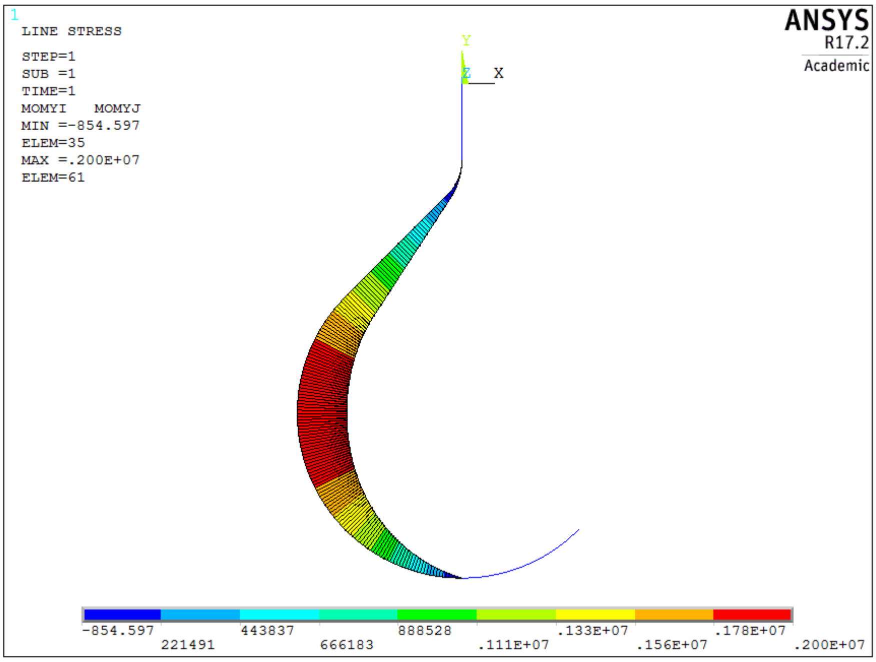


(a) Section 1

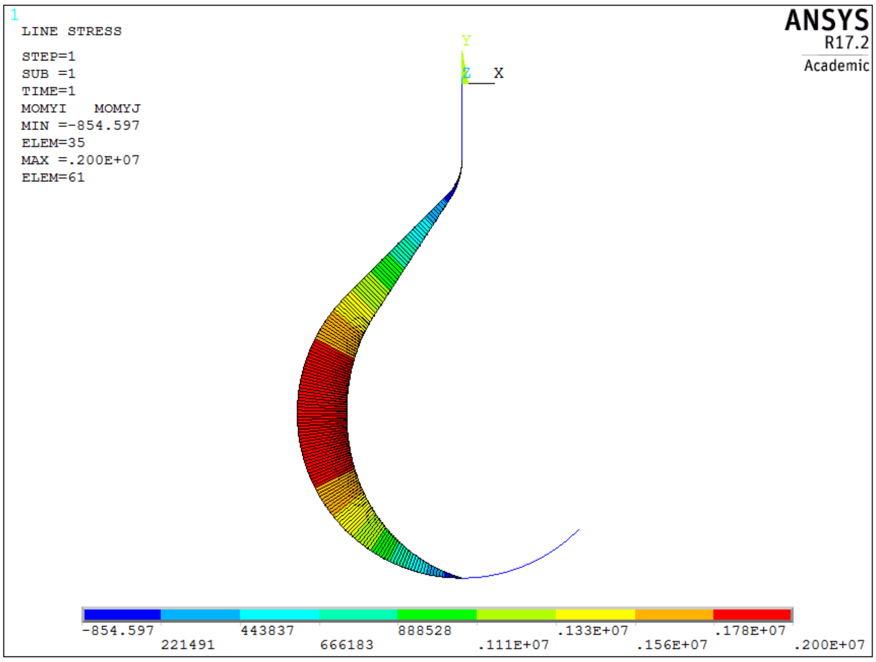


(b) Section 2

Figure 2.10: Result minimum bending stress diagram

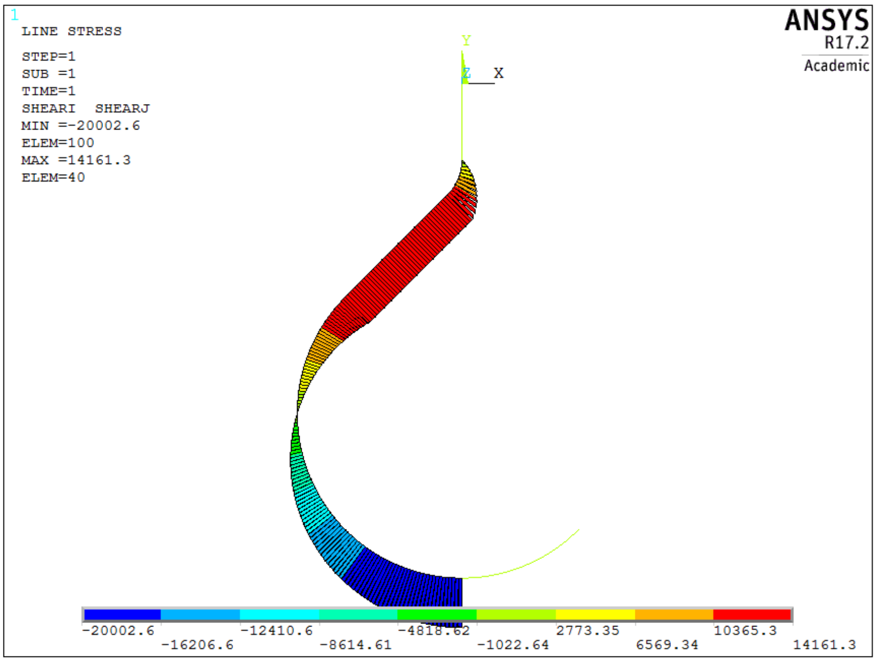


(a) Section 1

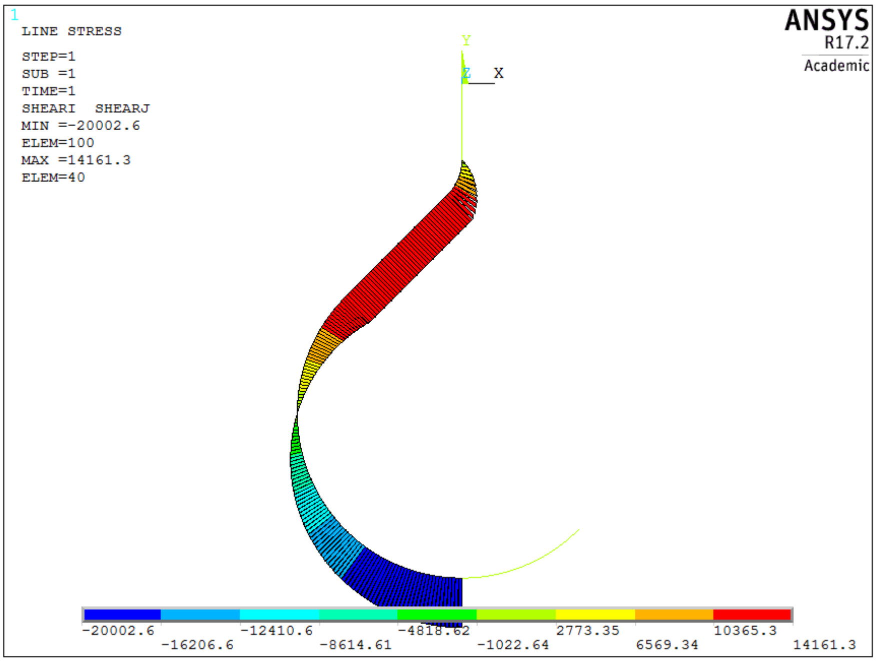


(b) Section 1

Figure 2.11: Result Moment diagram



(a) Section 1



(b) Section 2

Figure 2.12: Result shear diagram

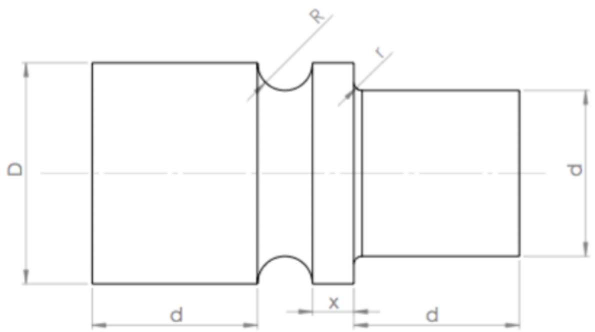
2.4 Command list

```
!**********************
! PROBLEM 2: SECTION 1 *
!**********************

/CLEAR,START,NEW
/TITLE,HOMERWORK 2 SECT_1
! >>> PARAMETER SECTION 1 <<<
*SET,MAX_WIDTH,30
*SET,MIN_WIDTH,20
*SET,HEIGHT,40
*SET,NDIV,16

! >>> PREPROCESS <<<
/PREP7
ET,1,PLANE82

K,1,0,0,0
K,2,MIN_WIDTH,0,0
K,3,MIN_WIDTH+(MAX_WIDTH-MIN_WIDTH)/2,HEIGHT,0
K,4,-(MAX_WIDTH-MIN_WIDTH)/2,HEIGHT,0

FLST,2,4,3
FITEM,2,1
FITEM,2,2
FITEM,2,3
FITEM,2,4
A,P51X

! >>> MESH AND SAVE <<<
LESIZE,ALL,,,NDIV
MSHAPE,0,2D
AMESH,ALL
NUMMRG,ALL
SECWRITE,SECT_1,SECT,,PLANE82
SAVE
FINI

!**********************
! PROBLEM 2: SECTION 2 *
!**********************

/CLEAR,START,NEW
/TITLE,HOMERWORK 2 SECT_2
! >>> PARAMETER SECTION 2 <<<
*SET,WIDTH,25
*SET,HEIGHT,40
*SET,RAD,10
*SET,NDIV,16

! >>> PRE PROCESS <<<
/PREP7
ET,1,PLANE82

K,1,0,0,0
K,2,WIDTH,0,0
K,3,WIDTH,HEIGHT/4,0
K,4,WIDTH-RAD,HEIGHT/2,0
K,5,WIDTH,3*HEIGHT/4,0
K,6,WIDTH,HEIGHT,0
K,7,0,HEIGHT,0
K,8,0,3*HEIGHT/4,0
K,9,RAD,HEIGHT/2,0
K,10,0,HEIGHT/4,0
K,11,WIDTH,HEIGHT/2,0
K,12,0,HEIGHT/2,0

L,1,2
L,2,3
L,5,6
L,6,7
L,7,8
L,10,1
LARC,3,4,11,RAD
LARC,4,5,11,RAD
LARC,8,9,12,RAD
LARC,9,10,12,RAD
SAVE

FLST,2,10,4
FITEM,2,1
FITEM,2,2
FITEM,2,7
FITEM,2,8
FITEM,2,3
FITEM,2,4
FITEM,2,5
FITEM,2,9
FITEM,2,10
FITEM,2,6
AL,P51X

! >>> MESH AND SAVE <<<
LESIZE,ALL,,,NDIV
MSHAPE,0,2D
AMESH,ALL
NUMMRG,ALL
SECWRITE,SECT_2,SECT,,PLANE82
SAVE
FINISH

!*********************
!PROBLEM: HOMEWORK 2 *
!*********************

/CLEAR,START,NEW
/TITLE,HOMERWORK 2 - SECTION 1
! >>> PARAMETERS MODEL <<<
*SET,Pi,ACOS(-1)
```

```
*SET,HEIGTH,200
*SET,FILLET,30
*SET,R,100
*SET,E_LENGTH,i
*SET,EPS,1E-4

! >>> PROPERTIES MATERIAL <<<
*SET,E_YOUNG,205000
*SET,NI,0.3

! >>> LOAD CONDITIONS <<<
*SET,F,20000

! >>> PRE PROCESSING <<<
/PREP7
ET,1,BEAM189

! >>> SECTION <<<
SECTYPE,1,BEAM,MESH,SECT_1
SECREAD,SECT_1,SECT,,MESH
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,205000
MPDATA,PRXY,1,,0.3

K,1,0,0,0
K,2,0,-HEIGTH,0
K,3,0,-HEIGTH+2*R*SIN(Pi/4),0
K,4,-R*COS(Pi/4),-HEIGTH+R*SIN(Pi/4),0
K,5,-R,-HEIGTH,0
K,6,0,-(HEIGTH+R),0
K,7,R*COS(Pi/4),-HEIGTH-R*SIN(Pi/4),0

L,1,3
L,3,4
LFILLT,1,2,FILLET
LARC,4,5,2,R
LARC,5,6,2,R
LARC,6,7,2,R

ESIZE,E_LENGTH
K,100,1000          !ORIENT KEYPOINT
LATT,1,,1,,100,,1
LMESH,ALL
ALLSEL,ALL
SAVE

! >>> SOLUTION <<<
/SOLUTION
ANTYPE,STATIC,NEW
NSEL,S,LOC,Y,-EPS,+EPS
NSEL,R,LOC,X,-EPS,+EPS
D,ALL,ALL
ALLSEL,ALL

KSEL,S,KP,,6
NSLK,S
F,ALL,FY,-F
ALLSEL,ALL
SOLVE
SAVE
FINISH

! >>> POST PROCESS <<<
/POST1
PLDISP,1
PLNSOL,U,Y

!***INTERNAL ACTIONS***
ETABLE,BSMAXI,SMISC,34  !MAX BENDING STRESS I IN Y
ETABLE,BSMAXJ,SMISC,39  !MAX BENDING STRESS J IN Y

ETABLE,BSMINI,SMISC,35  !MIN BENDING STRESS I IN Y
ETABLE,BSMINJ,SMISC,40  !MIN BENDING STRESS I IN Y

PLLS,BSMAXI,BSMAXJ

PLLS,BSMINI,BSMINJ

ETABLE,MOMYI,SMISC,2  !MOMENT Z IN I
ETABLE,MOMYJ,SMISC,15  !MOMENT Z IN J

PLLS,MOMYI,MOMYJ

ETABLE,SHEARI,SMISC,5     !SHEAR I
ETABLE,SHEARJ,SMISC,18    !SHAER J

PLLS,SHEARI,SHEARJ
FINISH
```

Chapter 3

Homework 3



DATA:
$\frac{D}{d} = 1.4$
$\frac{r}{d} = \frac{1}{20}$
$R = \frac{(D-d)}{2}$
Material: steel

The figure illustrates a shouldered shaft carrying a relief groove to reduce the notch stress concentration effect. The shaft is subject to an axial load F. Using axisymmetric plane elements, build up a FE model that allows:

1. determining the stress concentration factor in the absence of the relief groove. Carry out a convergence analysis and compare the obtained result with solutions available in the literature.

2. determining the stress concentration factor as a function of the non-dimensional position x of the relief groove. Try to identify an optimal position. Use a mesh refinement level similar to that obtained in the convergence analysis carried out in point 1).
   The stress concentration factor is defined as: $Kt = \sigma_{1,max}$; $\sigma_{1,max}$: maximum first principal stress in the model; $S_{net} = \frac{F}{\frac{\pi}{4}d^2}$.
   It is required to create a mapped mesh at least in the neighbourhood of the fillets of the shoulder and at the groove. Pay attetion to avoid distorted elements. Apply the axial force as a uniformly distributed pressure.

3.1 Approach the problem

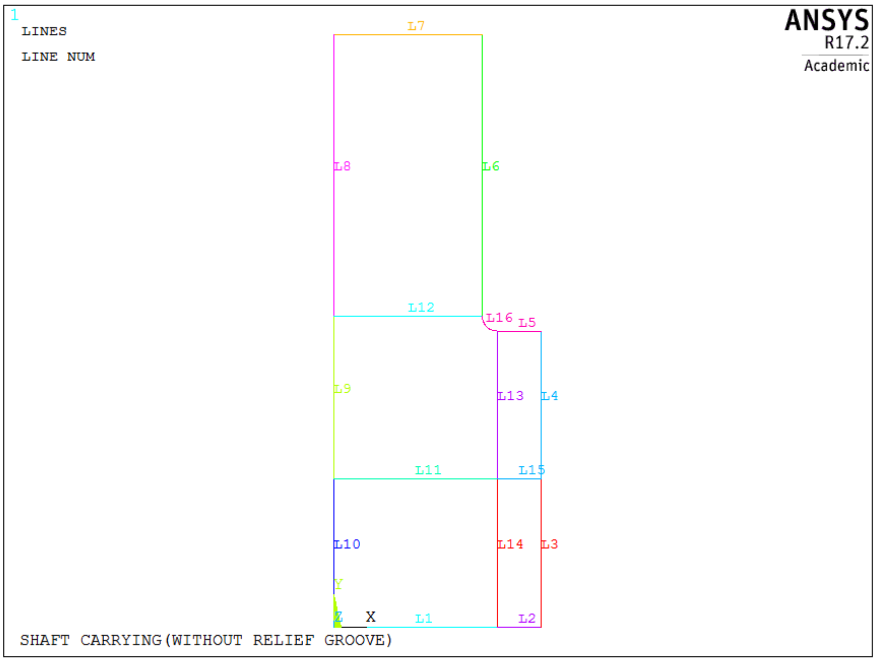
For this problem we have adopted element type PLANE182 for the problem without relief groove, on the other hand we have adoptd element type PLANE183 for the shaft with relief groove because PLANE183 is well suited to model irregular meshes.
The cross section of the shaft is created with axisymmetric elements along the y-direction. A mesh refinement acting along the shoulder surface is made in order to avoid worthless computational costs. The main assumption adopted in the resolution of the problem is the choice of the keyoptions.

- The element technology: Keyoption(1) = 0 due to the only axial traction of the shaft (no bending moment domination);
- Structural behavior: Keyoption(3) = 1 due to the axisymmetry of the problem.

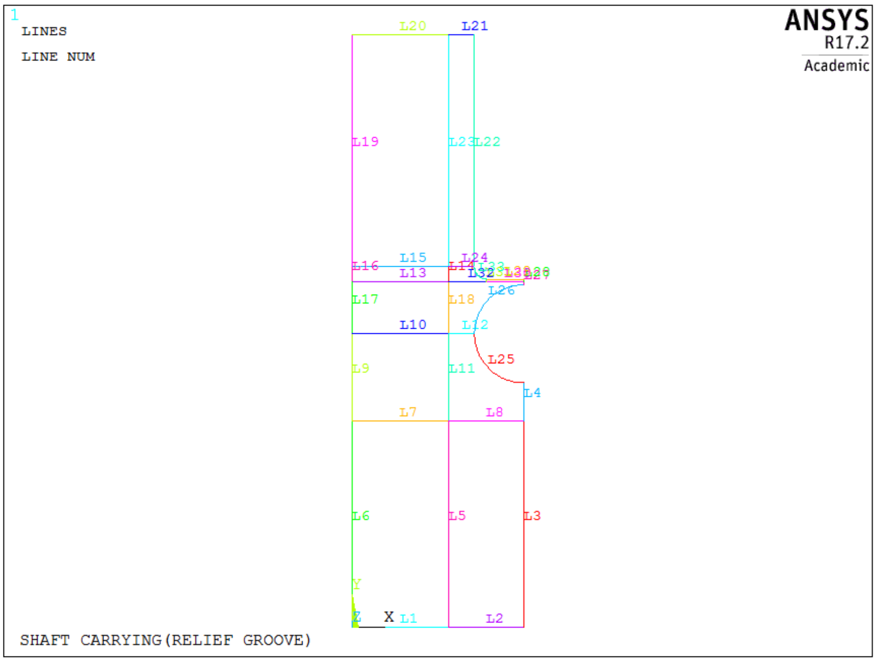
The material present the follow proprities:

- Modulus of elasticity: $210\,GPa$;
- Poissons ratio 0.3.

The shaft is loaded along the axis y with a distributed force.
Finally we set *keyopt* for axisymmetric element behaviour. The two tree models were constructed by placing the keypoint and then connected by lines, both have sub-divisions in areas to generate the mapped mesh. The groove is a function of the parameter x, the geometric model is observable in the figures 3.1, while in subsequent images 3.2, the mesh model.

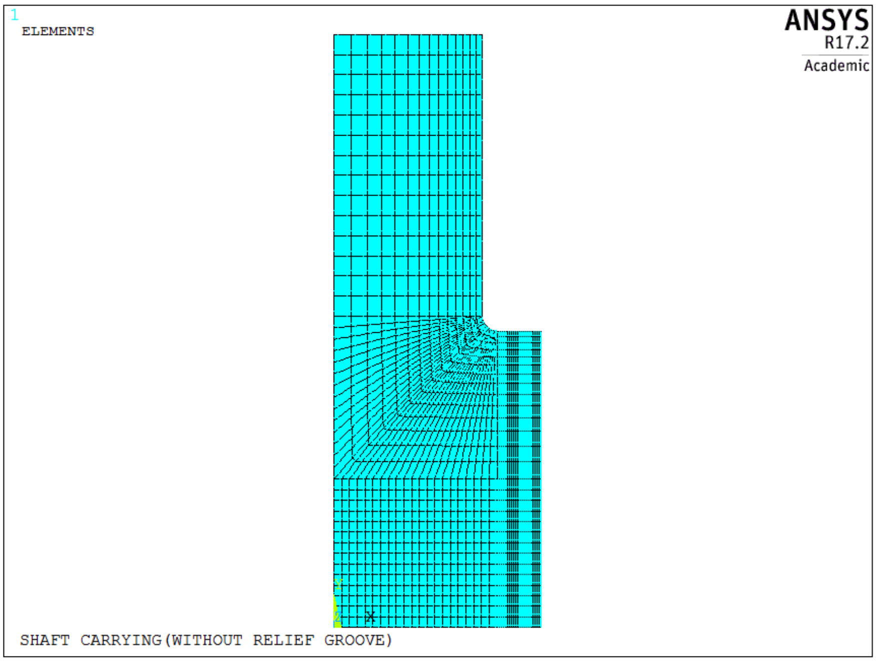


(a) Shaft without relief groove

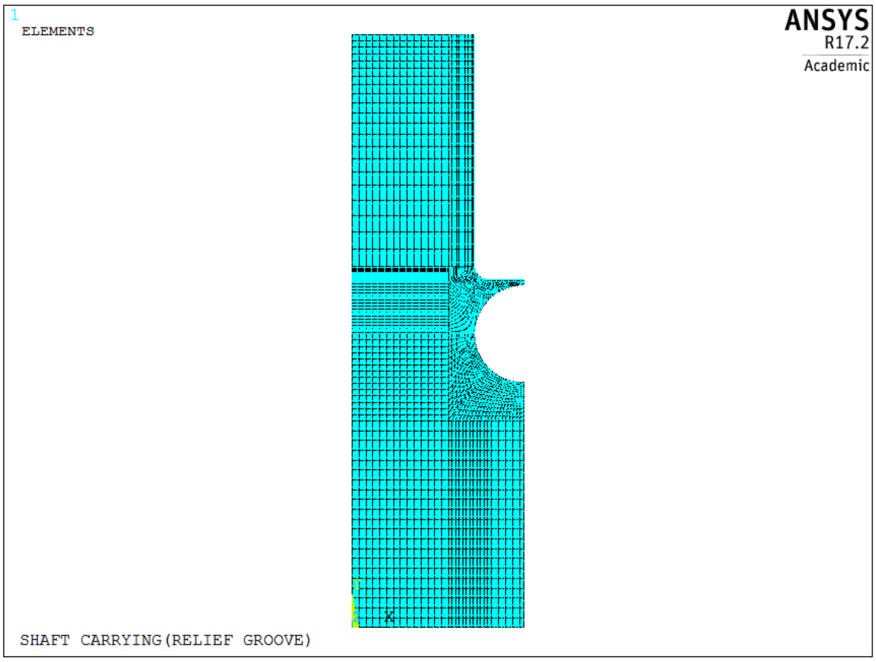


(b) Shaft relief groove

Figure 3.1: Geometry Model



(a) Shaft without relief groove



(b) Shaft relief groove

Figure 3.2: Mapped mesh's model

3.2 Result

The stress concentration factor is defined as:

$$K_t = \frac{\sigma_{1,Max}}{S_{net}}$$

$\sigma_{1,Max}$ is equal to maximum first principal stress in the model:

$$S_{net} = \frac{F}{\frac{\pi}{4}d^2} = 15000\,MPa$$

After analysis of convergence can be observed, in the table 3.1, that as the number of mesh divisions of the K_t value tends to stabilize toward the value $K_t \approx 2,4$. On the other hand it is observed that for a bonus multiplier of the divisions[1] of 10 you get a value close to that available in the literature in fact, the theoretical value is approximately equal to $K_t = 2,3$; show in figure 3.3.

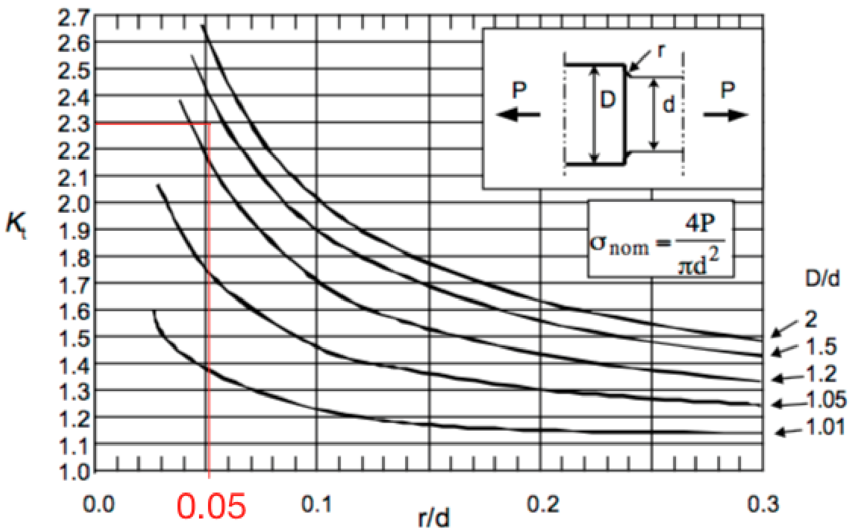


Figure 3.3: stress intensity factor graph

Instead performing a refinement of the mesh in the areas of the fillet it leads to a smaller number of divisions and a comparable number of elements thereby reducing the computational cost.

In fact, just a multiplier 4 to obtain 12 divisions are needed with a level 1 refinement to obtain a number of elements equal to 788 and a $K_t = 2,334$ close to the theoretical value first found, like show in table 3.2. In the second part of the problem, where the

[1] we have adopte: n_div_a, $3 * i$
n_div_b, $2 * i$
n_div_c, $3 * i$

multiplier for n. division	n. elements	$S1_{max}$ MPa	K_t
24	18432	37033,121	2,469
23	16928	36953,023	2,464
22	15488	36866,906	2,458
21	14112	36773,375	2,452
20	12800	36671,621	2,445
19	11552	36560,215	2,437
18	10368	36437,992	2,429
17	9248	36303,395	2,420
16	8192	36154,652	2,410
15	7200	35988,516	2,399
14	6272	35802,207	2,387
13	5408	35591,184	2,373
12	4608	35350,777	2,357
11	3872	35073,625	2,338
10	3200	34751,336	2,317
9	2592	34371,125	2,291
8	2048	33916,500	2,261
7	1568	33363,152	2,224
6	1152	32674,590	2,178
5	800	31797,691	2,120
4	512	30636,627	2,042
3	288	29044,623	1,936
2	128	26681,012	1,779
1	32	22991,930	1,533

Table 3.1: Number of division without refiniment

model with relief grooves, it is set to the same size of the mesh and refinement level of the previous model calculated in the first part of the problem of obtaining a $K_t = 1,762$ as a function of the dimensionless value x where $\text{x} = 1,5$. It can be seen in figure 3.4 trends the fly K_t as x changes.

multiplier for n. division	n. elements	$S1_{max}$ MPa	K_t
24	19913	38378,484	2,559
23	18346	38358,875	2,557
22	16844	38281,844	2,552
20	14041	38218,875	2,548
19	12729	38162,270	2,544
18	11488	38089,398	2,539
17	10306	38068,324	2,538
16	9189	37995,539	2,533
15	8139	37941,855	2,529
14	7155	37834,023	2,522
13	6226	37724,781	2,515
12	5366	37641,707	2,509
11	4570	37510,848	2,501
10	3836	37358,570	2,491
9	3171	37141,441	2,476
8	2564	36991,867	2,466
7	2024	36729,227	2,449
6	1548	36311,852	2,421
5	1136	35868,047	2,391
4	788	35008,906	2,334
3	504	33962,035	2,264
2	284	32086,471	2,139
1	116	29012,170	1,934

Table 3.2: Number of division with refiniment

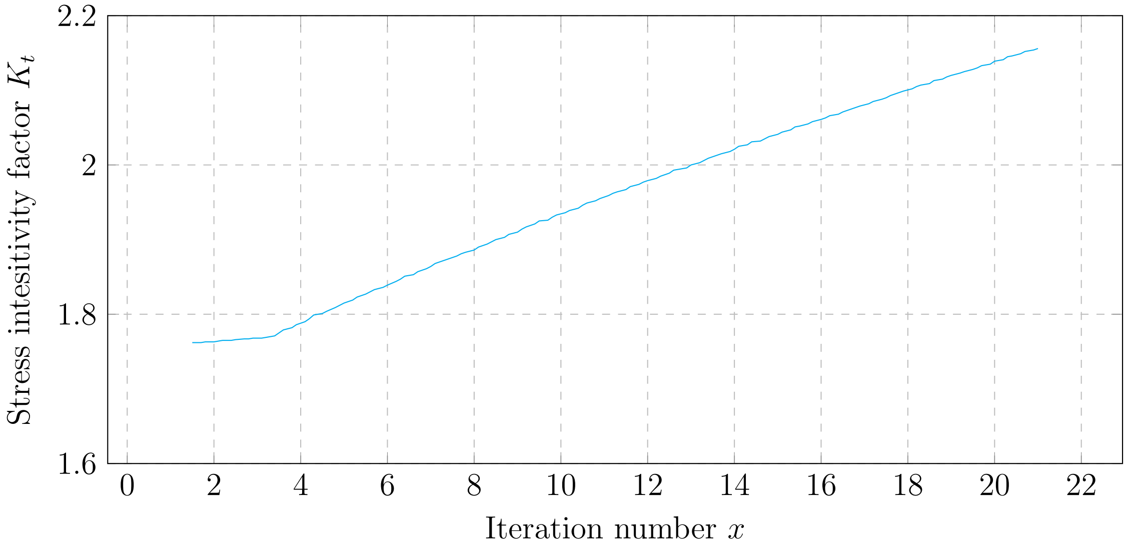
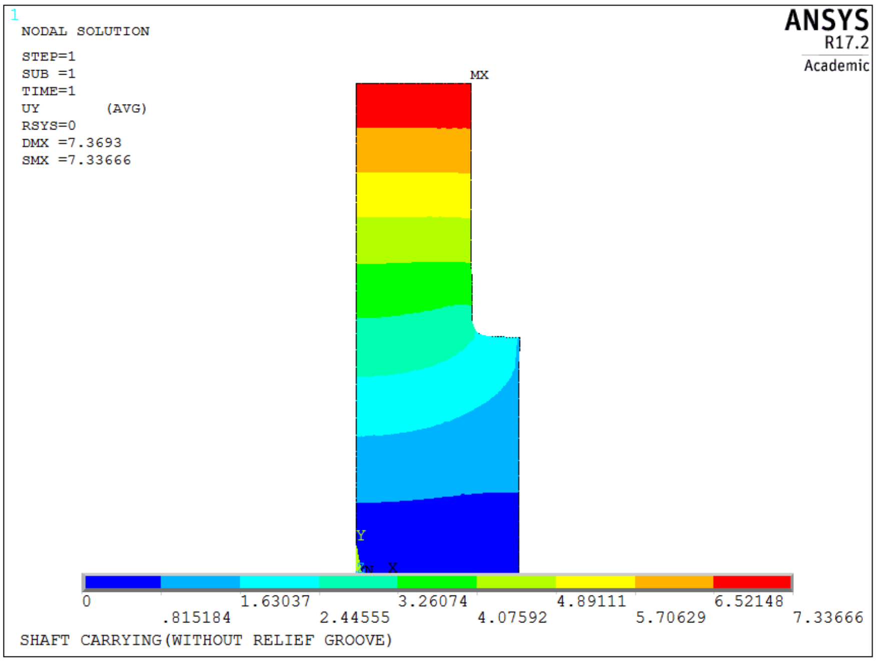
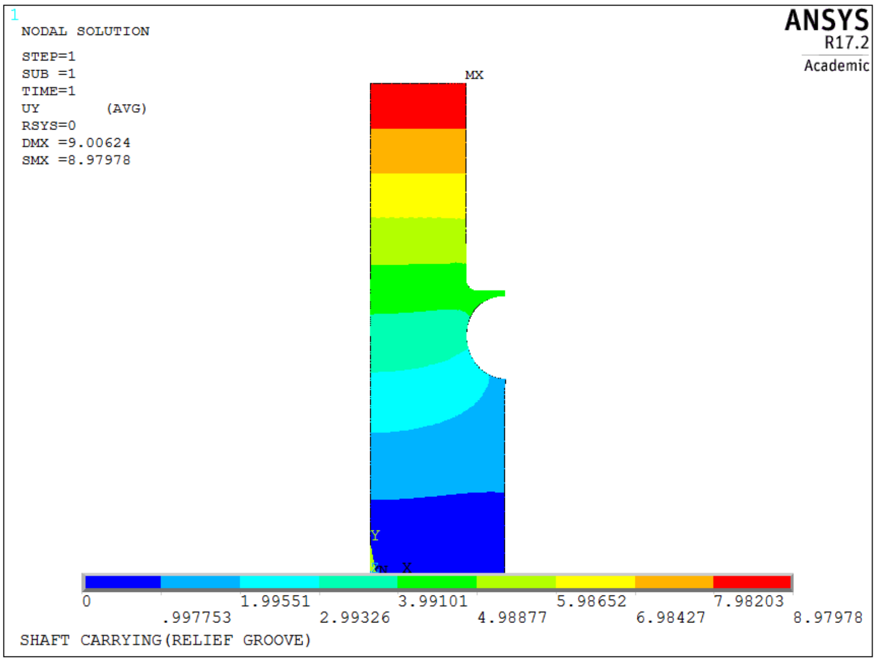


Figure 3.4: K_t function of x

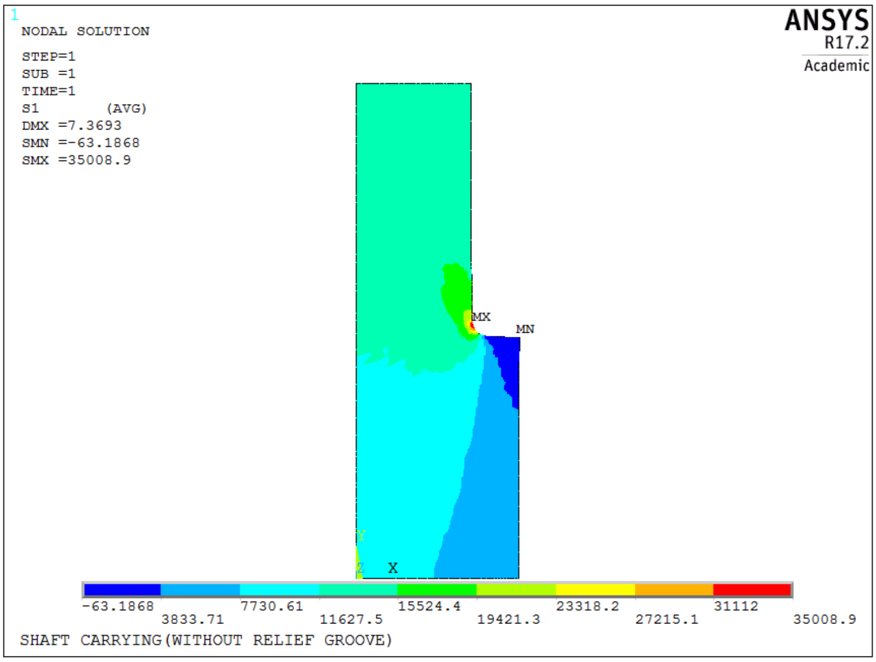


(a) Shaft without relief groove

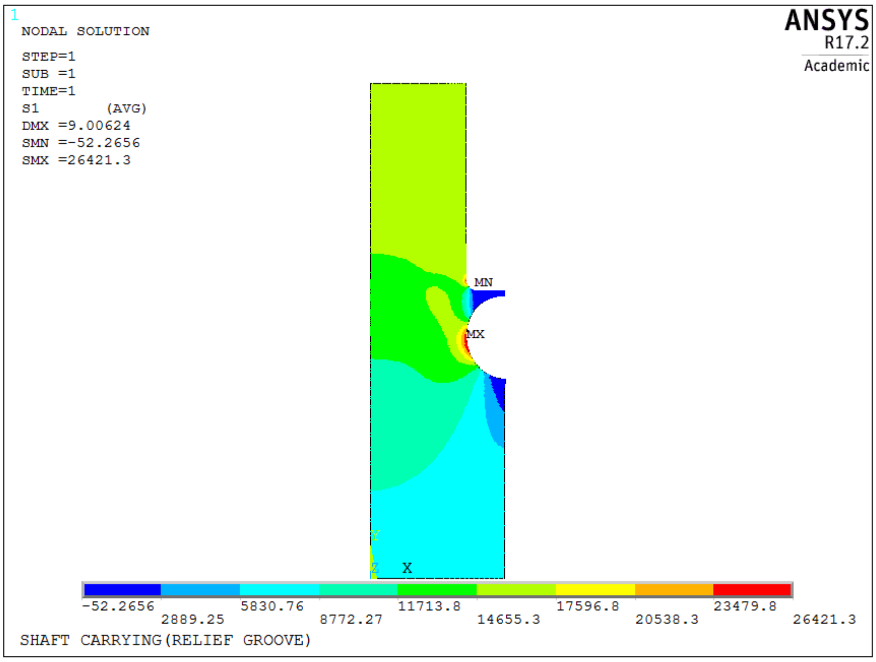


(b) Shaft relief groove

Figure 3.5: Displacement



(a) Shaft without relief groove



(b) Shaft relief groove

Figure 3.6: First Principal Stress

3.3 Conclusion

For the model absence of the relief groove: the analysis without refinement is acceptable, comparing literature value found on graph $K_t \approx 2,3$ and value before found $K_t = 2,334$, that is acceptable.
On the other hand, model with relief groove: the stress concentration decreases with decrease of x coefficient, this behavior is normal for shadow effect, where the stress concentration is minor respect at the case without relief groove.

3.4 Command list

```
!********************
! PROBLEM HOMEWORK 3 *
!********************

FINISH
/CLEAR,START,NEW
/TITLE,SHAFT CARRYING(WITHOUT RELIEF GROOVE)
/FILNAME,HOMEWORK3part1,1
! >>> PARAMETERS MODEL <<<
*SET,Pi,ACOS(-1)
*SET,D,90
*SET,d1,D/(1.4)
*SET,r1,d1/20
*SET,R,(D-d1)/2
*SET,n_div_a,3*7
*SET,n_div_b,2*7
*SET,n_div_c,3*7
*SET,EPS,1E-3

! >>> PROPERTIES MATERIAL <<<
*SET,E_Young,210000
*SET,ni,0.3

! >>>  LOAD<<<
*SET,F,15E3

! >>> PRE PROCESSING <<<
/PREP7

ET,1,PLANE182,0,,1
!KEYOPT,1,1,3
!KEYOPT,1,3,1
MP,EX,1,E_Young
MP,PRXY,1,ni

K,1,0,0,0
K,2,D/2-((D-d1))/2+r1,0,0
K,3,D/2,0,0
K,4,D/2,d1,0
K,5,D/2-((D-d1))/2+r1,d1,0
K,6,D/2-((D-d1))/2,d1,0
K,7,D/2-((D-d1)/2),d1+r1
K,8,d1/2,2*d1
K,9,0,2*d1,0
K,10,0,d1+r1
K,11,0,d1/2,0
K,12,D/2-((D-d1))/2+r1,d1/2,0
K,13,D/2,d1/2,0
SAVE

LSTR,     1,     2
LSTR,     2,     3
LSTR,     3,    13
LSTR,    13,     4
LSTR,     4,     5
LSTR,     7,     8
LSTR,     8,     9
LSTR,     9,    10
LSTR,    10,    11
LSTR,    11,     1
LSTR,    11,    12
LSTR,    10,     7
LSTR,     5,    12
LSTR,    12,     2
LSTR,    12,    13
LARC,7,5,6,-r1
SAVE

AL,1,14,11,10
AL,2,3,15,14
AL,15,4,5,13
AL,9,11,12,13,16
AL,12,8,7,6

LESIZE,11,,,n_div_a
LESIZE,12,,,n_div_b,1/3
LESIZE,13,,,n_div_b,3
LESIZE,9,,,n_div_b
LESIZE,16,,,n_div_b+n_div_a
LCCAT,9,11
```

```
LESIZE,10,,,n_div_b
LESIZE,1,,,n_div_a
LESIZE,14,,,n_div_b
LESIZE,2,,,n_div_a
LESIZE,3,,,n_div_b
LESIZE,15,,,n_div_a
LESIZE,4,,,n_div_b,1/3
LESIZE,5,,,n_div_a
LESIZE,6,,,n_div_b
LESIZE,7,,,n_div_b,3
LESIZE,8,,,n_div_b
SAVE
MSHKEY,1
MSHAPE,0,2D
AMESH,ALL

! >>> CONSTRAINT <<<
DL,1,,uy,0
DL,2,,uy,0
DL,8,,ux,0
DL,9,,ux,0
DL,10,,ux,0
SAVE

! >>> FORCE <<<
SFL,7,PRES,-F
SAVE

! >>> SOLUTION <<<
/SOLU
ANTYPE,STATIC,NEW
NSEL,S,LOC,Y,-EPS,+EPS
NSEL,S,LOC,X,-EPS,+EPS
DSYM,SYMM,X
ALLSEL,ALL
SOLVE
FINISH

!>>>> POST-PROCESS <<<<
/POST1
PLDISP,1
PLNSOL,U,Y
PLNSOL,S,1
PRNSOL,S,PRIN
FINISH

!********************
! PROBLEM HOMEWORK 3 *
!********************

FINISH
/CLEAR,START,NEW
/TITLE,SHAFT CARRYING(RELIEF GROOVE)
/FILNAME,HOMEWORK3Part2,1
! >>> PARAMETERS MODEL <<<
*SET,i,1
*DO,i,1,15,0.1
```

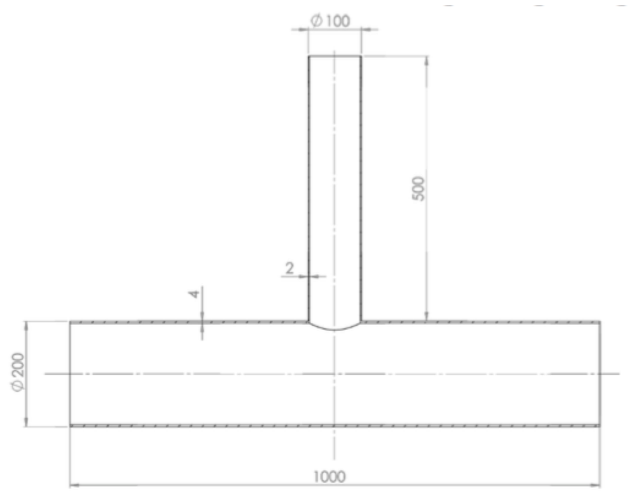
```
*SET,D,90
*SET,d1,D/(1.4)
*SET,r1,d1/20
*SET,R,(D-d1)/2
*SET,n_div_a,3*7
*SET,n_div_b,2*7
*SET,n_div_c,3*7
*SET,EPS,1E-3
*SET,x,(D/d1)*i

! >>> PROPERTIES Material <<<
*SET,E_Young,210000
*SET,ni,0.3

! >>> LOAD <<<
*SET,F,15E3

! >>> PRE PROCESSING <<<
/PREP7

ET,1,PLANE183,0,,1
MP,EX,1,E_Young
MP,PRXY,1,ni

K,1,0,0,0
K,2,D/2-((D-d1))/2+r1-10,0,0
K,3,D/2-((D-d1))/2+r1-10,d1-10,0
K,4,D/2,0,0
K,5,D/2,d1,0
K,6,D/2-R,d1+R,0
K,7,D/2-((D-d1))/2+r1-10,d1+R,0
K,8,D/2,d1+R,0
K,9,D/2,d1+2*R,0
k,10,D/2,d1+2*R+x/2,0
K,11,D/2,(d1+2*R)+x,0
K,12,D/2-((D-d1))/2+r1,(d1+2*R)+x,0
k,13,D/2-((D-d1))/2+r1,(d1+2*R)+x/2,0
K,14,d1/2,(d1+2*R)+x,0
K,15,d1/2,((d1+2*R)+r1)+x,0
k,16,D/2-((D-d1))/2+r1-10,((d1+2*R)+r1)+x,0
K,17,D/2-((D-d1))/2+r1-10,2*(d1+R)+x,0
k,18,D/2-((D-d1))/2+r1-10,d1+2*R+x/2,0
K,19,d1/2,2*(d1+R)+x,0
K,20,0,2*(d1+R)+x,0
K,21,0,((d1+2*R)+r1)+x,0
K,22,0,d1+2*R+x/2,0
K,23,0,d1+R,0
K,24,0,d1-10,0
K,25,D/2,d1-10,0
SAVE

LSTR,       1,       2
LSTR,       2,       4
LSTR,       4,      25
LSTR,      25,       5
LSTR,       2,       3
```

```
LSTR,       1,      24
LSTR,      24,       3
LSTR,       3,      25
LSTR,      24,      23
LSTR,      23,       7
LSTR,       7,       3
LSTR,       7,       6
LSTR,      22,      18
LSTR,      18,      16
LSTR,      16,      21
LSTR,      21,      22
LSTR,      22,      23
LSTR,      18,       7
LSTR,      21,      20
LSTR,      20,      17
LSTR,      17,      19
LSTR,      19,      15
LSTR,      16,      17
LSTR,      16,      15
LARC,5,6,8,R,
LARC,6,9,8,R,
LSTR,       9,      10
LSTR,      10,      11
LSTR,      11,      12
LSTR,      13,      10
LSTR,      13,      12
LSTR,      18,      13
LSTR,      18,       7
LARC,12,15,14,-r1,
SAVE

AL,1,5,7,6
AL,2,3,8,5
AL,8,4,25,12,11
AL,12,26,27,30,32,18
AL,30,28,29,31
AL,32,31,33,24,14
AL,7,11,10,9
AL,10,18,13,17
AL,13,14,15,16
AL,24,22,21,23
AL,15,23,20,19
SAVE

LESIZE,8,,,n_div_a
LESIZE,25,,,n_div_b+n_div_a
LESIZE,4,,,n_div_b,1/3
LESIZE,11,,,n_div_b
LESIZE,12,,,n_div_b,1/3
LCCAT,11,8
SAVE

LESIZE,18,,,n_div_b+n_div_c
LESIZE,32,,,n_div_a
LESIZE,26,,,n_div_b+n_div_c+2*n_div_a
LESIZE,27,,,n_div_b,3
LESIZE,30,,,n_div_a
FLST,2,3,4,ORDE,3
FITEM,2,18
FITEM,2,30
FITEM,2,32
LCCAT,P51X
SAVE

LESIZE,33,,,n_div_b+n_div_a
LESIZE,31,,,n_div_b,1/3
LESIZE,14,,,n_div_b
LESIZE,24,,,n_div_b,1/3
LCCAT,14,32
SAVE

LESIZE,29,,,n_div_a
LESIZE,28,,,n_div_b,1/3
SAVE

LESIZE,22,,,n_div_a,3
LESIZE,21,,,n_div_b,1/3
LESIZE,23,,,n_div_a,1/3
SAVE

LESIZE,20,,,n_div_b
LESIZE,19,,,n_div_a,1/3
LESIZE,15,,,n_div_b
SAVE

LESIZE,10,,,n_div_b
LESIZE,17,,,n_div_b+n_div_c
LESIZE,9,,,n_div_b
SAVE

LESIZE,1,,,n_div_b
LESIZE,6,,,n_div_a
LESIZE,7,,,n_div_b
LESIZE,5,,,n_div_a
SAVE

LESIZE,2,,,n_div_a
LESIZE,3,,,n_div_a
SAVE
MSHKEY,1
MSHAPE,0,2D
AMESH,ALL
SAVE

! >>> CONSTRAINT <<<
DL,1,,uy,0
DL,2,,uy,0
DL,6,,ux,0
DL,9,,ux,0
DL,16,,ux,0
DL,19,,ux,0
SAVE
```

```
! >>> FORCE <<<
SFL,20,PRES,-F
SFL,21,PRES,-F
SAVE

! >>> SOLUTION <<<
/SOLUTION
ANTYPE,STATIC,NEW
NSEL,S,LOC,Y,-EPS,+EPS
NSEL,S,LOC,X,-EPS,+EPS
DSYM,SYMM,X
ALLSEL,ALL
SOLVE
FINI

! >>> POST PROCESSING <<<
/POST1
PLDISP,1
PLNSOL,U,Y
PLNSOL,S,1

*GET,S1max,PLNSOL,0,MAX
*CFOPEN,S1max,txt,,APPEND
*VWRITE,x,S1max,S1max/F
(F5.1,F20.3,F20.3)
*CFCLOS
FINISH
/CLEAR,START
*ENDDO
FINISH
```

Chapter 4

Homework 4



DATA:
Material: steel;
Pressure: $P_i = 10$ BAR;

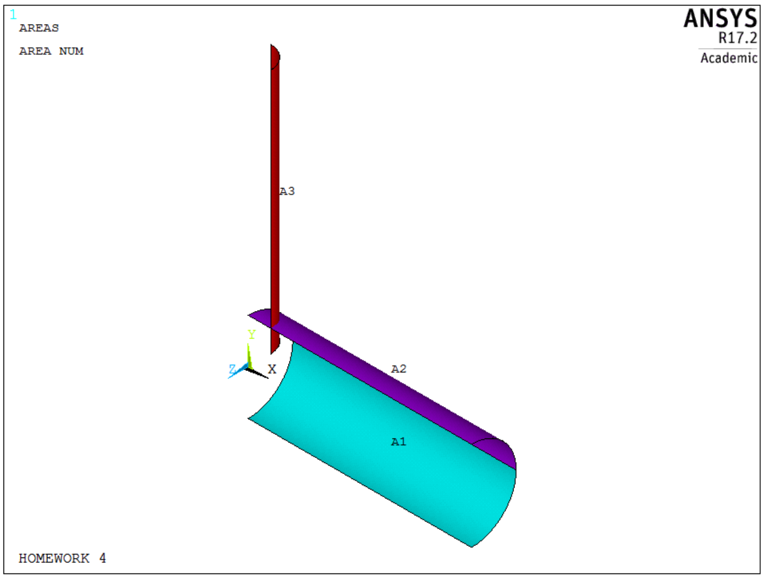
The figure illustrates a T pipe connector to be used in a hydraulic circuit subject to internal pressure p_i. Using shell elements and taking into account the symmetry, build up a FE model composed of a mapped mesh that allows:

1. determining (meridional and circumferential) membrane stresses far from the junction between the two pipes;

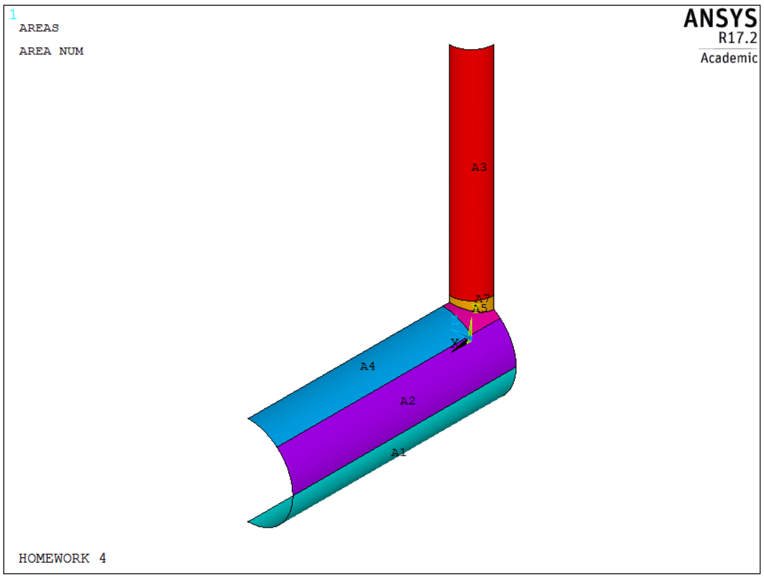
2. determining membrane and bending stress distribution along the periphery of the pipes' junction.

4.1 Approach the problem

the elements that compose the structure: are used in three-dimensional shell elements, SHELL181, having the thickness of the two tubes are first defined. The stress analysis will be conducted on the inner surface and the average plane of the elements. You create keypoint, connect via lines and generating areas for extrusion lines, two local references are defined with cylindrical cooordinate to carry out these operations, as show in figure 4.1a. To realize the junction intersected areas that make up the two tubes and the excess arising from its construction have been eliminated. At this point the areas that compose the model are divided into smaller areas in the vicinity of the junction. This is done later to create a denser mesh in this area. The result that is obtained is the following 4.1b.



(a) First step construction



(b) Second step construction

Figure 4.1: Geometry of T pipe

To realize the mesh are divided all components then mesh mapped areas, show in frame 4.2. Particular attention taken into accoutn at zone of junction, result show in 4.3. Once sure that the unit vectors normal to the shell elements have always the same direction, fig. 4.4, apply the symmetry constraints and internal pressure.
They then analyze two different way:

- It sets to 0 the displacement in the y direction of a node: the constraints on symmetry in fact, they do not eliminate this degree of freedom. It could be a rigid translation of the component;
- they are binding on the long-x node displacements in the flow line (line used to extrude the tube) because it is assumed that the tube can be extended in this direction. Is done the same for the vertical pipe, then set to zero along the y displacement of the nodes present on the line. In this way also avoids the rigid translation of the model.

At this point it solves the structure and proceed with the analysis of the results.

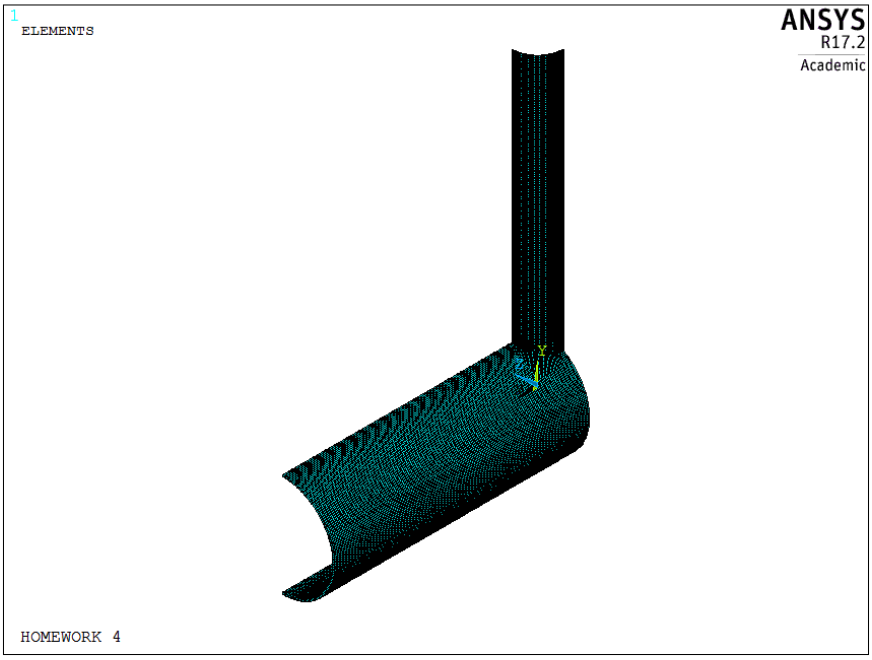


Figure 4.2: Meshed model

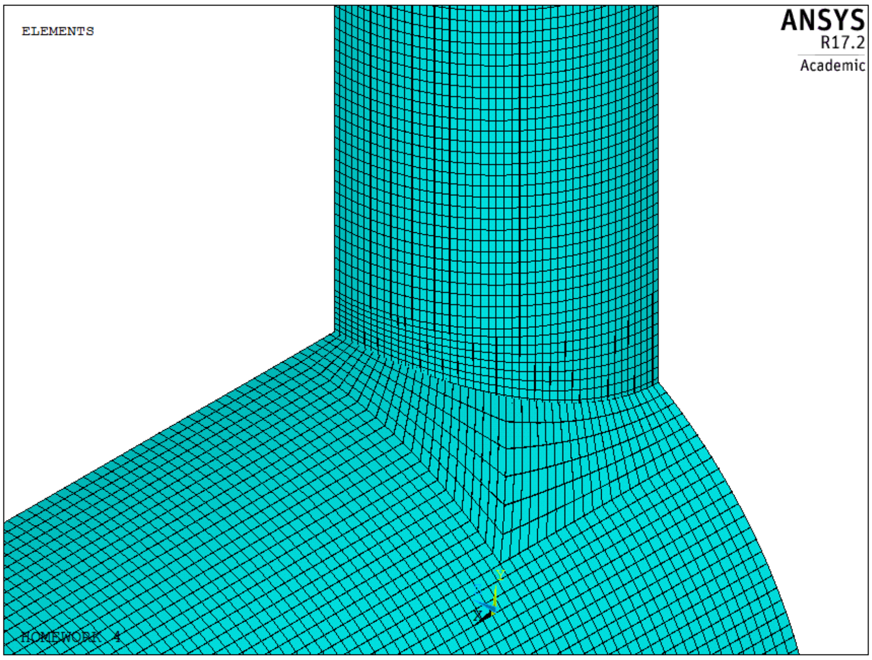


Figure 4.3: Detail of the mesh to the pipe joint

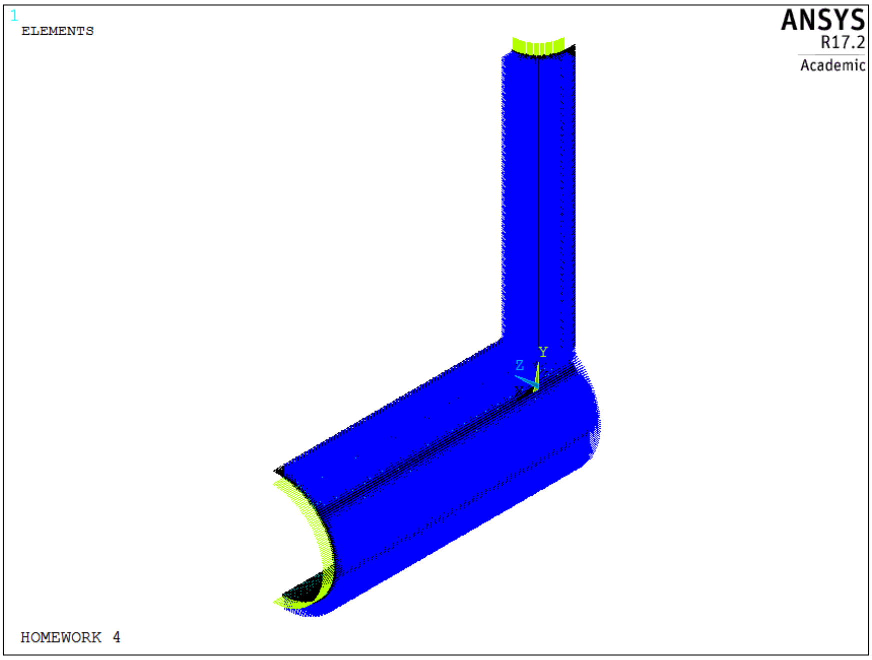
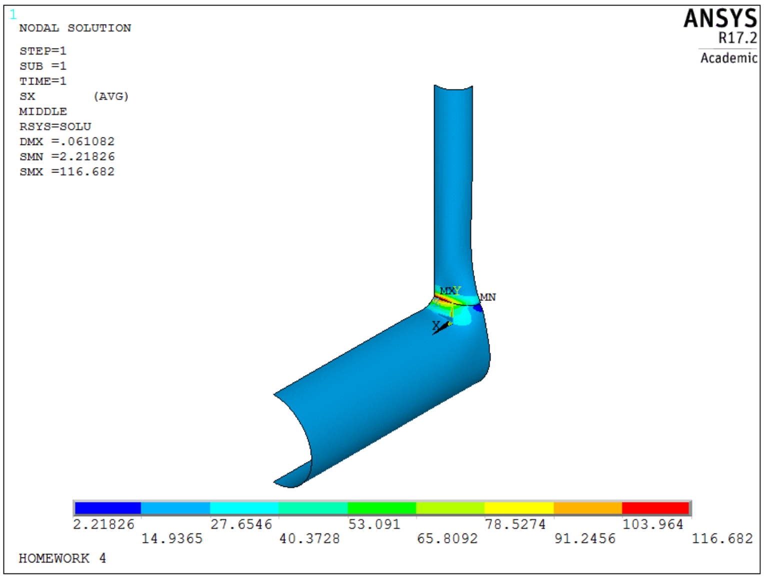


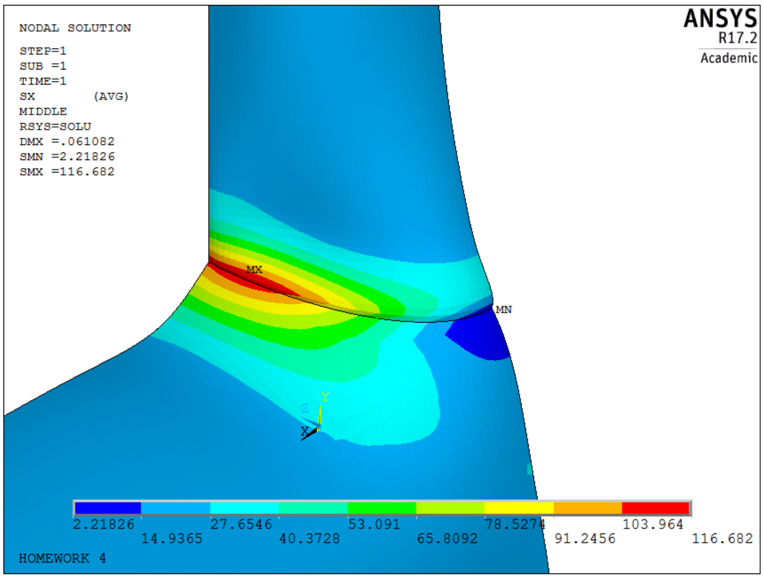
Figure 4.4: Vector normal to surface

4.2 Result

The membrane stress are visible in 4.5a and 4.6a, it is noted that at a certain distance from the junction of the membrane stress assume nearly costant value and the flexural value close to zero. It then sees that in the proximity of junction of the flexural stress become important and are obtained for very high value of stress. Regarding the convergence of the solution, it can be seen that going to densify the mesh the value of the stress on the junction does not converge as this area subject to a strucutral singlularity.

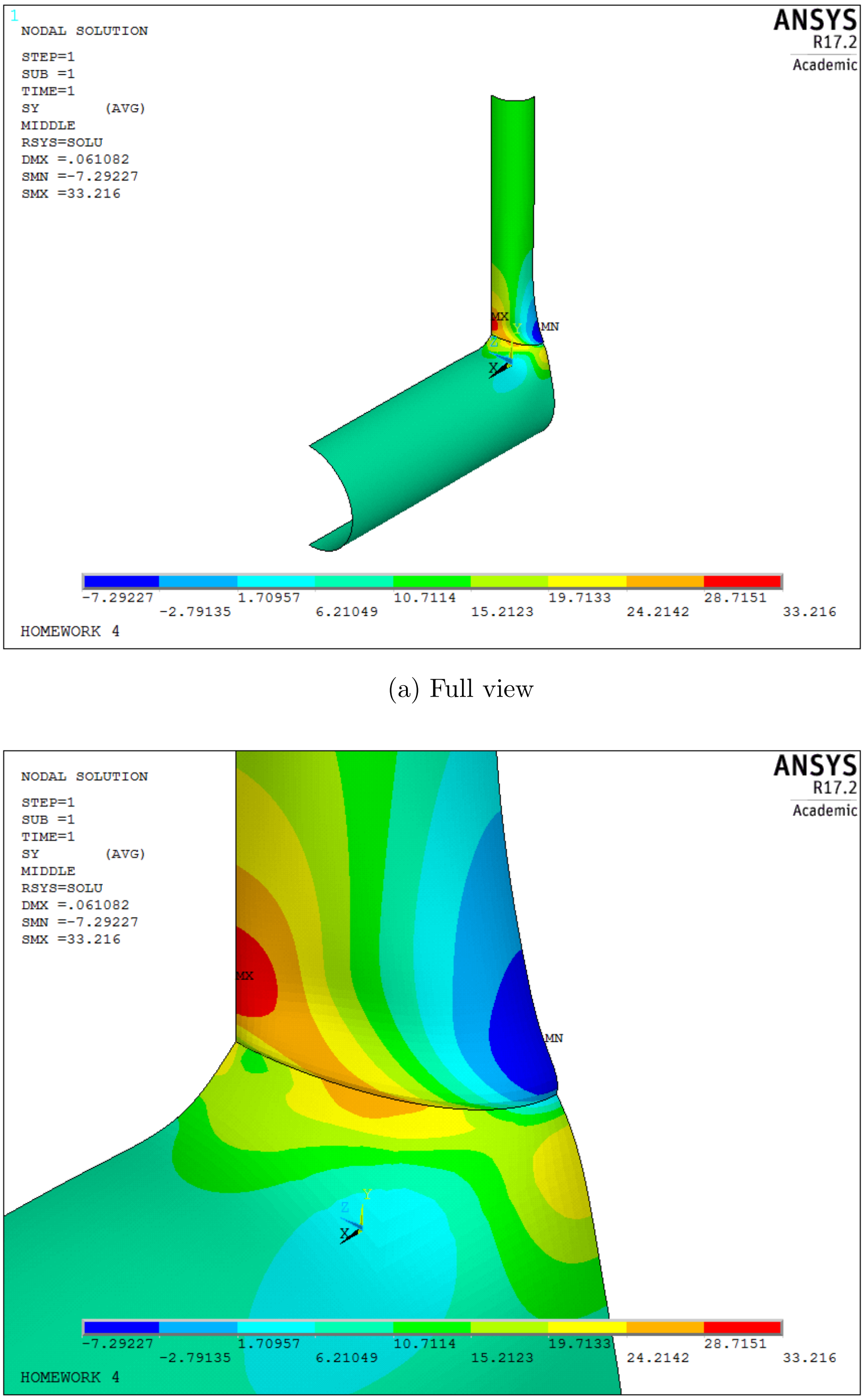


(a) Full view



(b) Detail at junction

Figure 4.5: Membrane meridional stress



(a) Full view

(b) Detail at junction

Figure 4.6: Membrane circumferential stress

It then conducted a more detailed analysis going to represent on a graph the progress of efforts along the axial direction of the two tubes, starting from the most distant areas from the junction until you get near it, the result obtained are show in graphs 4.7 for vertical pipe and 4.8 for horizontal pipe.

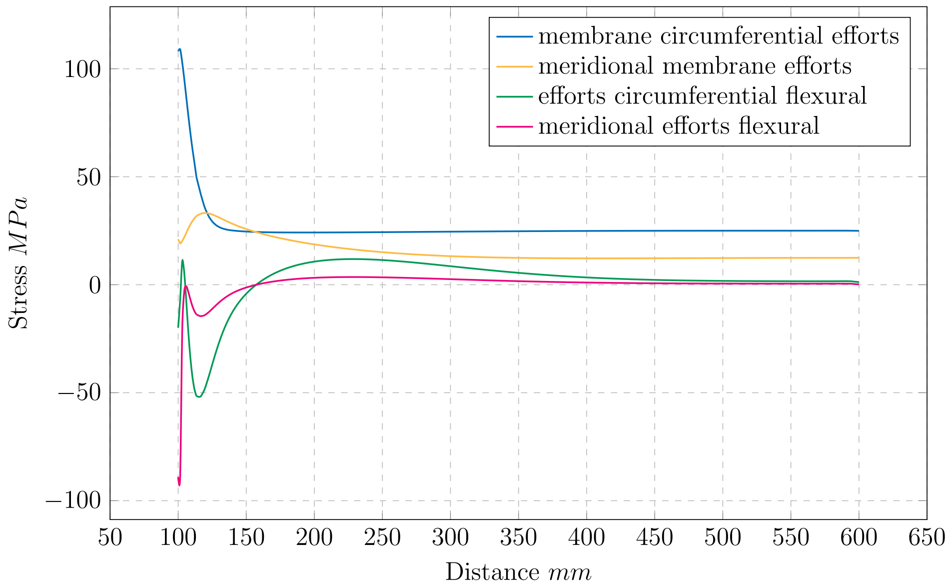


Figure 4.7: Stress vertical pipe

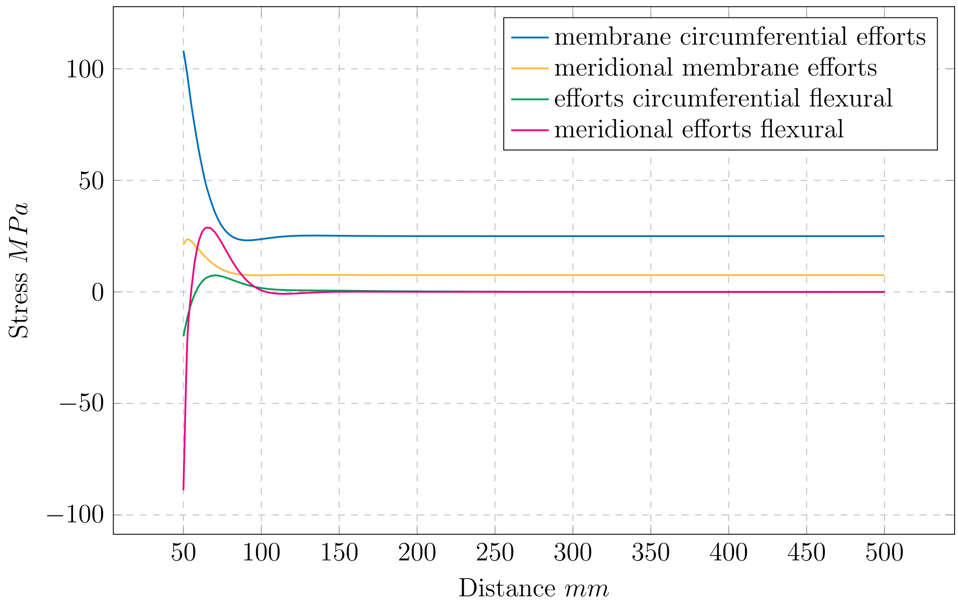


Figure 4.8: Stress horizontal pipe

4.3 Conclusion

The obtained results confirm what was found earlier: the flexural stresses are very low away from the junction and become higher in the vicinity of it; the membrane efforts are constant along the tube and, as regards those circumferential, assume very high values on the junction.

4.4 Command list

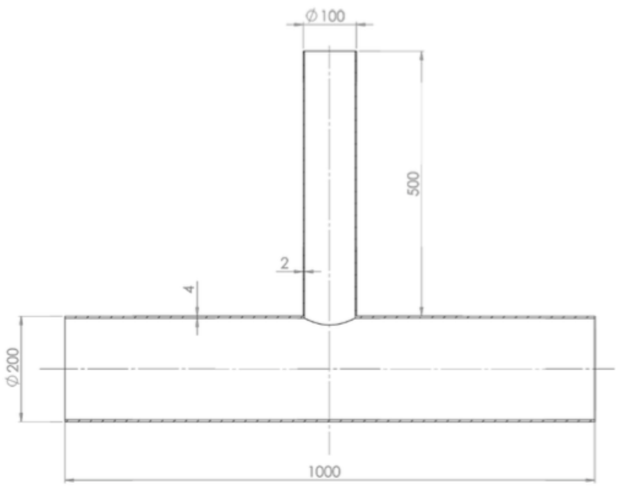
```
!********************
! PROBLEM HOMEWORK 4 *
!********************

FINISH
/CLEAR, START, NEW
/TITLE, HOMEWORK 4
/FILNAME,HOMEWORK4
! >>> PARAMETERS MODEL <<<
*SET,leng_v_pipe,500
*SET,diameter_v_pipe,100
*SET,thick_v_pipe, 2
*SET,leng_h_pipe,1000
*SET,diameter_h_pipe,200
*SET,thick_h_pipe,4
*SET,alpha,90
*SET,EPS,1E-3
*SET,n_div_area,3

! >>> PROPERTIES MATERIAL <<<
*SET,E_Young, 210000
*SET,ni,0.3

! >>> LOAD <<<
*SET,PRESSURE,1

! >>> PRE PROCESSING <<<
/PREP7
ET,1,SHELL181
KEYOPT,1,8,2
SECTYPE,1,shell
SECDATA,thick_h_pipe
MP,EX,1,E_Young
MP,PRXY,1,ni

ET,2,SHELL181
KEYOPT,2,8,2
SECTYPE,2,shell
SECDATA,thick_v_pipe
MP,EX,2,E_Young
MP,PRXY,2,ni

TYPE,1
SECNUM,1
MAT,1
SAVE

! >>> DEFINE RF CYLINDRICAL <<<
CLOCAL,100,CYLIN,0,diameter_h_pipe/2,0,,-alpha
CSYS,0
CLOCAL,200,CYLIN,0,0,0,,,alpha

!***VERTICAL PIPE
CSYS,100
K,1,diameter_v_pipe/2,0,-leng_v_pipe/10
K,2,diameter_v_pipe/2,0,leng_v_pipe
K,3,0,0,0
K,4,0,0,leng_v_pipe
L,1,2
!***HORIZZONTAL PIPE
CSYS,200
K,5,diameter_h_pipe/2,-alpha,0
K,6,diameter_h_pipe/2,-alpha,leng_h_pipe/2
K,7,0,0
K,8,0,0,leng_h_pipe/2
L,5,6
!***GEN AREA H PIPE
AROTAT,2,,,,,,8,7,-2*alpha
!***GEN AREA V PIPE
CSYS,100
AROTAT,1,,,,,,3,4,alpha
SAVE

APTN,2,3
ADELE,4,5
LDELE,10
LDELE,15
LDELE,16
SAVE

CSYS,0
WPOFFS,-50,100
WPROTA,,250
ASBW,7
WPOFFS,50,-10
WPROTA,,-250
WPROTA,,,alpha
WPOFFS,,,64.3
ASBW,3
WPROTA,,alpha
WPOFFS,,,-10
ASBW,6
SAVE

! >>> MESHING <<<
LCCAT,16,21
LCCAT,15,21
AESIZE,ALL,n_div_area

!***HORIZZONTAL PIPE
ESYS,200
TYPE,1
SECNUM,1
MSHKEY,1
MSHAPE,0,2D
AMESH,4
AMESH,5
AMESH,2
AMESH,1
SAVE

!***VERTICAL PIPE
ESYS,100
SAVE
TYPE,2
SECNUM,2
```

```
SAVE
MSHKEY,1
MSHAPE,0,2D
AMESH,7
AMESH,3
SAVE

! >>> VERIFY THE NORMAL VERSOR <<<
EPLOT
/PSYMB,ESYS,1

! >>> SOLUTION <<<
SAVE
SFE,ALL,,PRES,,PRESSURE
/PBC,ALL,,1
LSEL,S,LINE,,20
LSEL,A,LINE,,24
LSEL,A,LINE,,10
LSEL,A,LINE,,1
LSEL,A,LINE,,5
NSLL,S,1
DSYM,SYMM,x

LSEL,S,LINE,,8
LSEL,A,LINE,,2
LSEL,A,LINE,,19
LSEL,A,LINE,,7
LSEL,A,LINE,,23
NSLL,S,1
DSYM,SYMM,z

LSEL,A,LINE,,9
LSEL,A,LINE,,4
LSEL,A,LINE,,6
NSLL,S,1
D,ALL,UX,0

LSEL,S,LINE,,11
NSLL,S,1
D,ALL,UY,0
ALLSEL,ALL
/SOLU
SOLVE
FINISH

! >>> POSTPROCESSING <<<
/POST1
/ESHAPE,1
PLDISP,1

SHELL,MID
RSYS,SOLU
PLNSOL,S,x
PLNSOL,S,y
PRNSOL,S,comp
SAVE

! >>> JUNCTION MEMBRANE STRESS <<<
SHELL,MID
RSYS,SOLU
LSEL,S,LINE,,14
NSLL,S,1
*GET,nnodi,node,,count
*DIM,sx,ARRAY,nnodi
*DIM,sy,ARRAY,nnodi
*DIM,zn,ARRAY,nnodi
*DIM,xn,ARRAY,nnodi
cm,nodi,node
*DO,i,1,nnodi
  *GET,xmin,node,,mnloc,x
  NSEL,r,loc,x,xmin-EPS,xmin+EPS
  *GET,nmin,node,,num,max
  zn(i)=nz(nmin)
  xn(i)=xmin
  *GET,sx(i),node,nmin,s,x
  *GET,sy(i),node,nmin,s,y
  CMSEL,S,nodi
  nsel,u,node,,nmin
  cm,nodi,node
*ENDDO
*CFOPEN,ResultMembraneJunction,txt
*VWRITE
 (7x,'zn',18x,'xn',18x,'sx',18x,'sy')
*vwrite,zn(1),xn(1),sx(1),sy(1)
(F20.10,F20.10,F20.10,F20.10,F20.10,F20.10)
*CFCLOS

SHELL,BOT
RSYS,SOLU
LSEL,S,LINE,,14
NSLL,S,1
*GET,nnodi,node,,count
*DIM,sxt,ARRAY,nnodi
*DIM,syt,ARRAY,nnodi
*DIM,sxb,ARRAY,nnodi
*DIM,syb,ARRAY,nnodi
cm,nodi,node
 *DO,i,1,nnodi
 *GET,xmin,node,,mnloc,x
 NSEL,R,LOC,x,xmin-EPS,xmin+EPS
 *GET,nmin,node,,num,max
 *GET,sxt(i),node,nmin,s,x
 *GET,syt(i),node,nmin,s,y
 sxb(i)=sxt(i)-sx(i)
 syb(i)=syt(i)-sy(i)
 zn(i)=nz(nmin)
 xn(i)=xmin
 CMSEL,S,nodi
 NSEL,U,node,,nmin
 cm,nodi,node
*ENDDO
*CFOPEN,JunctionBendingStress,txt
*VWRITE
 (7x,'zn',18x,'xn',18x,'sxb',18x,'syb')
*vwrite,zn(1),xn(1),sxb(1),syb(1)
(F20.10,F20.10,F20.10,F20.10,F20.10,F20.10)
*CFCLOS
*DEL,ALL

! >>> HORIZZONTAL PIPE <<<
LSEL,S,LINE,,8
NSLL,S,1
*GET,nnodi,node,,count
*DIM,sx,array,nnodi
*DIM,sy,array,nnodi
*DIM,sxt,array,nnodi
*DIM,syt,array,nnodi
*DIM,sxb,array,nnodi
*DIM,syb,array,nnodi
*DIM,pos,array,nnodi
cm,nodi,node
*DO,i,1,nnodi
 *GET,xmin,node,,mnloc,x
 NSEL,R,LOC,x,xmin-EPS,xmin+EPS
 *GET,nmin,node,,num,max
 s1=nx(nmin)
 pos(i)=s1
 shell,mid
 rsys,solu
 *GET,sx(i),node,nmin,s,x
 *GET,sy(i),node,nmin,s,y
 SHELL,BOT
 RSYS,SOLU
 *GET,sxt(i),node,nmin,s,x
 *GET,syt(i),node,nmin,s,y
 sxb(i)=sxt(i)-sx(i)
 syb(i)=syt(i)-sy(i)
 CMSEL,S,nodi
 NSEL,U,node,,nmin
 CM,NODI,node
*ENDDO
*CREATE,ANSUITMP
```

```
*CFOPEN,'HorizzontalStress1','txt',''
*VWRITE
 (7x,'pos',18x,'sx',18x,'sy',18x,'sxb',18x,'syb')
*VWRITE,pos(1),sx(1),sy(1),sxb(1),syb(1)
(F20.10,F20.10,F20.10,F20.10,F20.10)
*CFCLOS
*END
/INPUT,ANSUITMP
*DEL,ALL

LSEL,S,LINE,,19
NSLL,S,1
*GET,nnodi,node,,count
*DIM,sx,array,nnodi
*DIM,sy,array,nnodi
*DIM,sxt,array,nnodi
*DIM,syt,array,nnodi
*DIM,sxb,array,nnodi
*DIM,syb,array,nnodi
*DIM,pos,array,nnodi
cm,nodi,node
*DO,i,1,nnodi
 *GET,xmin,node,,mxloc,x
 NSEL,R,LOC,X,xmin-EPS,xmin+EPS
 *GET,nmin,node,,num,max
 s1=nx(nmin)
 pos(i)=s1
 SHELL,MID
 RSYS,SOLU
 *GET,sx(i),node,nmin,s,x
 *GET,sy(i),node,nmin,s,y
 SHELL,BOT
 RSYS,SOLU
 *GET,sxt(i),node,nmin,s,x
 *GET,syt(i),node,nmin,s,y
 sxb(i)=sxt(i)-sx(i)
 syb(i)=syt(i)-sy(i)
 CMSEL,S,nodi
 NSEL,U,node,,nmin
 cm,nodi,node
*ENDDO
*CREATE,ANSUITMP
*CFOPEN,'HorizzontalStress2','txt',''
*VWRITE
 (7x,'pos',18x,'sx',18x,'sy',18x,'sxb',18x,'syb')
*VWRITE,pos(1),sx(1),sy(1),sxb(1),syb(1)
(F20.10,F20.10,F20.10,F20.10,F20.10)
*CFCLOS
*END
/INPUT,ANSUITMP
*DEL,ALL

! >>> VERTICAL PIPE <<<
LSEL,S,LINE,,7
NSLL,S,1
*GET,nnodi,node,,count
*DIM,sx,array,nnodi
*DIM,sy,array,nnodi
*DIM,sxt,array,nnodi
*DIM,syt,array,nnodi
*DIM,sxb,array,nnodi
*DIM,syb,array,nnodi
*DIM,pos,array,nnodi
cm,nodi,node
*DO,i,1,nnodi
 *GET,xmin,node,,mxloc,x
 NSEL,R,LOC,X,xmin-EPS,xmin+EPS
 *GET,nmin,node,,num,max
 s1=ny(nmin)
 pos(i)=s1
 SHELL,MID
 RSYS,SOLU
 *GET,sx(i),node,nmin,s,x
 *GET,sy(i),node,nmin,s,y
 SHELL,BOT
 RSYS,SOLU
 *GET,sxt(i),node,nmin,s,x
 *GET,syt(i),node,nmin,s,y
 sxb(i)=sxt(i)-sx(i)
 syb(i)=syt(i)-sy(i)
 CMSEL,S,nodi
 NSEL,U,node,,nmin
 cm,nodi,node
*ENDDO
*CREATE,ANSUITMP
*CFOPEN,'VerticalStress1','txt',''
*VWRITE
 (7x,'pos',18x,'sx',18x,'sy',18x,'sxb',18x,'syb')
*VWRITE,pos(1),sx(1),sy(1),sxb(1),syb(1)
(F20.10,F20.10,F20.10,F20.10,F20.10)
*CFCLOS
*END
/INPUT,ANSUITMP
*DEL,ALL

LSEL,S,LINE,,23
NSLL,S,1
*GET,nnodi,node,,count
*DIM,sx,array,nnodi
*DIM,sy,array,nnodi
*DIM,sxt,array,nnodi
*DIM,syt,array,nnodi
*DIM,sxb,array,nnodi
*DIM,syb,array,nnodi
*DIM,pos,array,nnodi
cm,nodi,node
*DO,i,1,nnodi
 *GET,xmin,node,,mxloc,x
 NSEL,R,LOC,X,xmin-EPS,xmin+EPS
 *GET,nmin,node,,num,max
 s1=ny(nmin)
 pos(i)=s1
 SHELL,MID
 RSYS,SOLU
 *GET,sx(i),node,nmin,s,x
 *GET,sy(i),node,nmin,s,y
 SHELL,BOT
 RSYS,SOLU
 *GET,sxt(i),node,nmin,s,x
 *GET,syt(i),node,nmin,s,y
 sxb(i)=sxt(i)-sx(i)
 syb(i)=syt(i)-sy(i)
 CMSEL,S,nodi
 NSEL,U,node,,nmin
 cm,nodi,node
*ENDDO
*CREATE,ANSUITMP
*CFOPEN,'VerticalStress2','txt',''
*VWRITE
 (7x,'pos',18x,'sx',18x,'sy',18x,'sxb',18x,'syb')
*VWRITE,pos(1),sx(1),sy(1),sxb(1),syb(1)
(F20.10,F20.10,F20.10,F20.10,F20.10)
*CFCLOS
*END
/INPUT,ANSUITMP
FINISH
/EXIT,ALL
```

Chapter 5

Homework 5



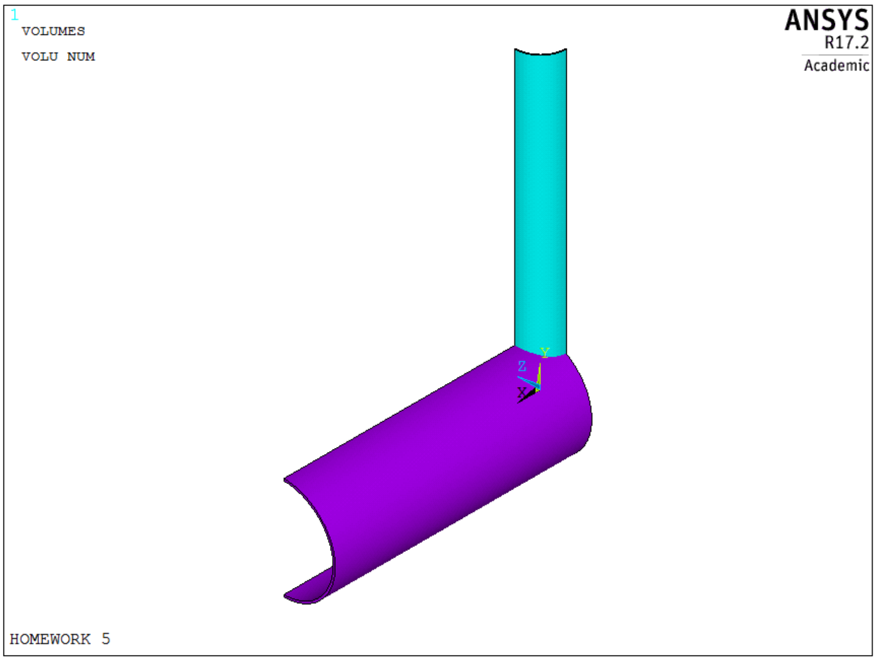
DATA:
Material: steel;
Pressure: $P_i = 10$ BAR;

The T pipe connector analyzed in HW 4 is now filletted at the junction between the two pipes to reduce the stress concentration factor. Using brick elements, build a submodel able to estimate the stress distribution along the periphery of the pipes' junction on the base of the displacement field computed with the shell model developed in HW 4. It is required to:

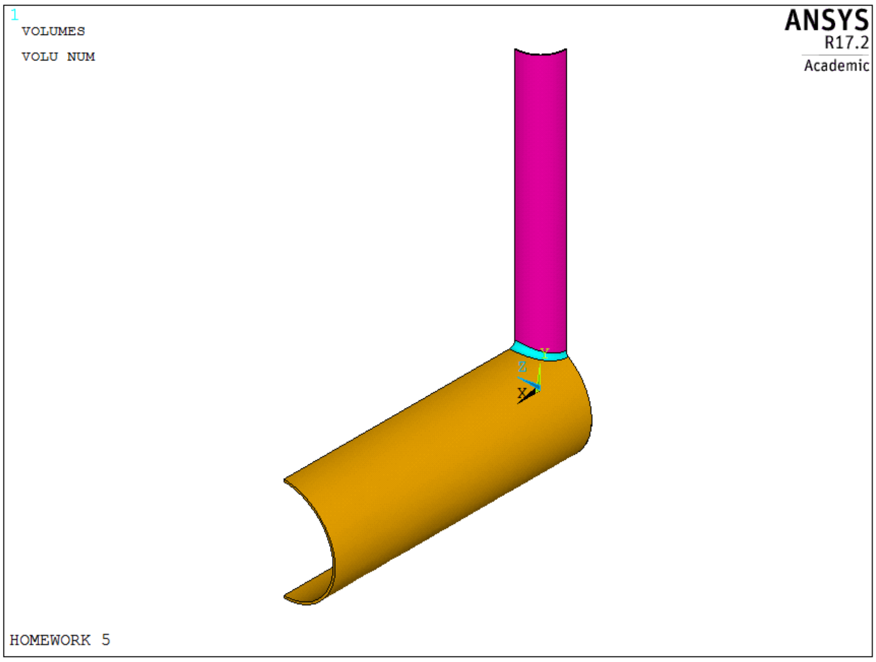
1. check for the sensitivity of the results upon the location of the cutting boundaries of the submodel.

5.1 Approach the problem

For this problem, first designing the pattern seen in homework 4 saving the results and after that we realize the submodel. Using the same work surfaces extrude the two vertical and horizontal cylinders, then they eliminate unneeded volumes. The result as show in figure 5.1a. It builds the fillet eliminating the junction that was present in the previous step, figure 5.1b.



(a) First step construnction



(b) Second step construction

Figure 5.1: 3D Geometry of T pipe

The model is now trimmed to the sub pattern using two planes: one through the vertical plane; the second is horizontal. Finally the separate volumes are united in a single body. It realizes a free mesh with type elements SOLID186, assigned size of the elements is equal one millimeter; such as to ensure that there are at least two elements in the smallest thickness that makes up the vertical cylinder, the result is observable in fig. 5.2.

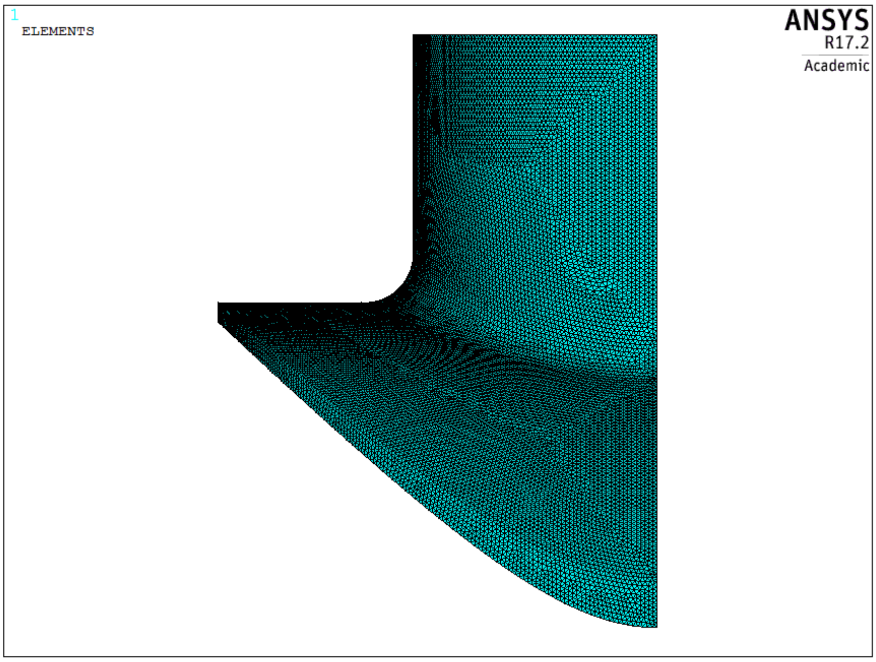


Figure 5.2: Mesh model

At this point, saving the coordinates of the nodes that are located on the surfaces generated from pattern cutting, for later use to define the conditions of the problem outline. The results solution's Homework 4 are recalled and through interpolation is assigned a shift on the nodes that you have saved the coordinates above. Finally applies the same pressure of 10 BAR and constraints.

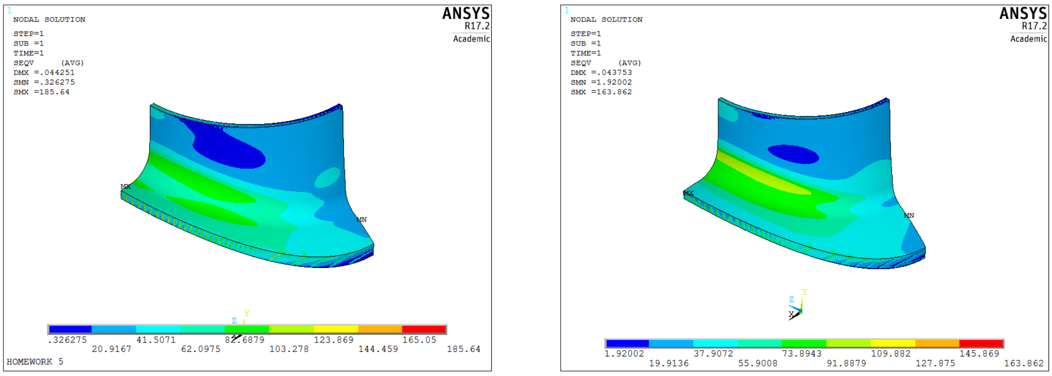
5.2 Result

The study is carried out by varying the distance of the cut boundaries in such a way as not to modify the structure of the problem. defining a sub small model and then increase its size. The analysis is summarized, in the table 5.1, where the cutting distances are quoted with respect to the junction.

iter	Vertical cut boundaries mm	Radius cut boundaries mm
1	30,0000	65,0000
2	35,0000	70,0000
3	40,0000	75,0000
4	45,0000	80,0000
5	50,0000	85,0000
6	55,0000	90,0000

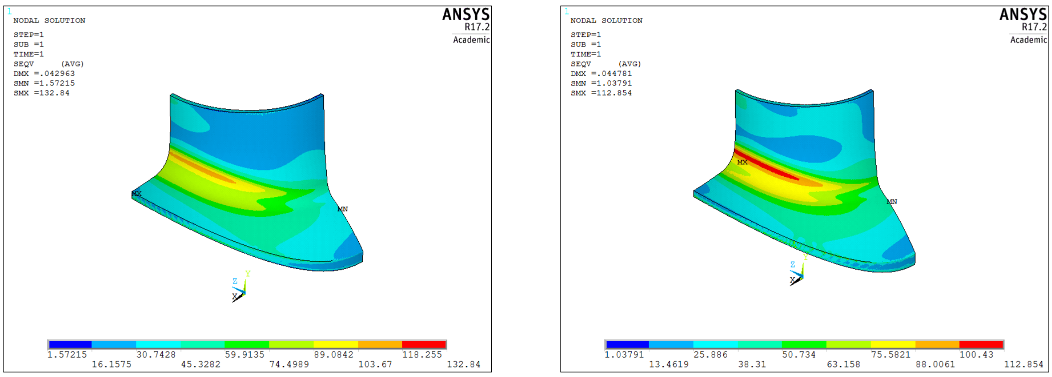
Table 5.1: Cut boundaries distance

The results satisfied the equivalent stress, according to Von Mises, obtained under varying cut boundaries are shown in figures 5.3, 5.4 and 5.5, it referred to as the stress shifts from the cut boundaries to the fillet.



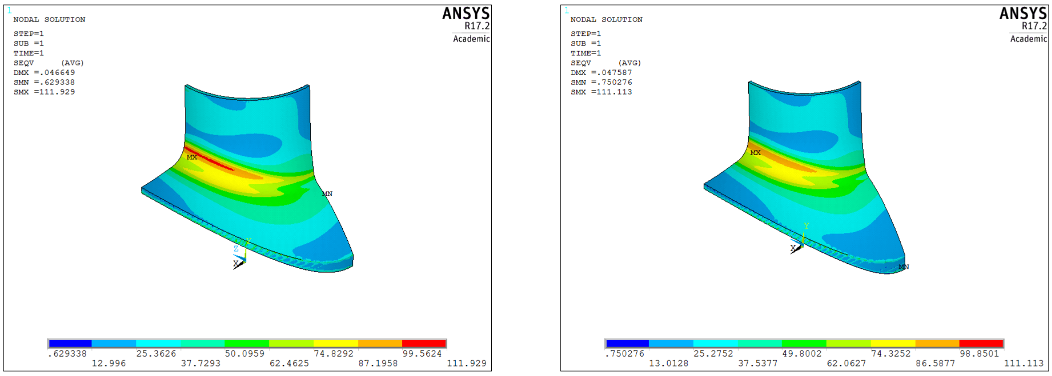
(a) Submodel at cut boundaries 65 mm, 65 mm (b) Submodel at cut boundaries 70 mm, 67, 50 mm

Figure 5.3: Von Mises equivalent stress



(a) Submodel at cut boundaries 75 mm, 70 mm (b) Submodel at cut boundaries 80 mm, 72, 50 mm

Figure 5.4: Von Mises equivalent stress



(a) Submodel at cut boundaries 85 mm, 75 mm (b) Submodel at cut boundaries 90 mm, 77, 50 mm

Figure 5.5: Von Mises equivalent stress

It is observed in the graph, in the figure 5.7, which initially stress is high, then it takes on a downward path until it stabilizes. Increasing the size of the submodel is known that the more efforts are moving from the boundary condition in correspondence of the fitting and inside of the junction.

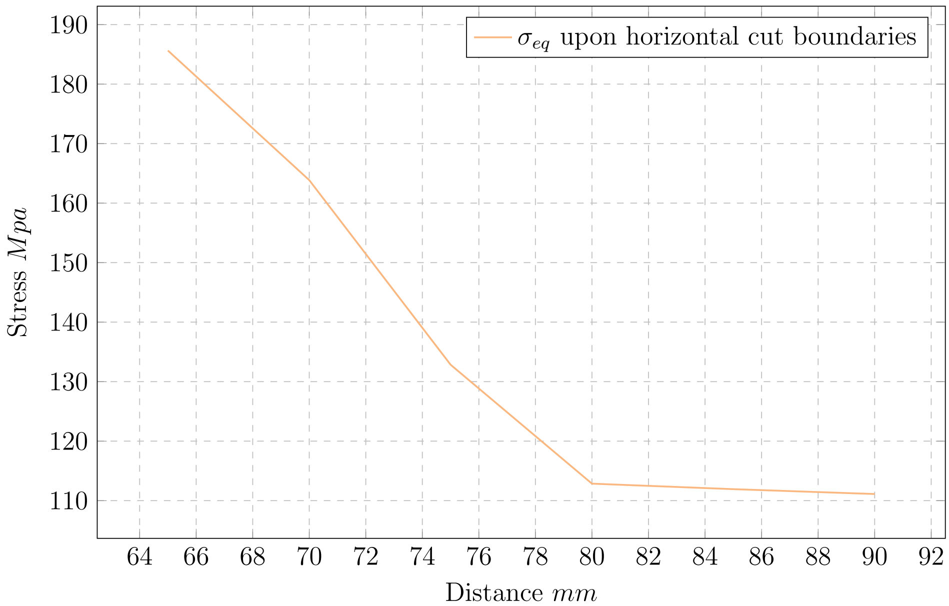


Figure 5.6: Submodel' s sensitivity upon the location of the cutting boundaries

At this point you can graph the stress distribution along the connection by defining a path. Selecting the nodes present on the lines that define the fitting and which pass through the zone more stressed. Interactively define a path by selecting the nodes defining the effort that you want to analyze, equivalent Von Mises, getting the result shown in figure 5.7.

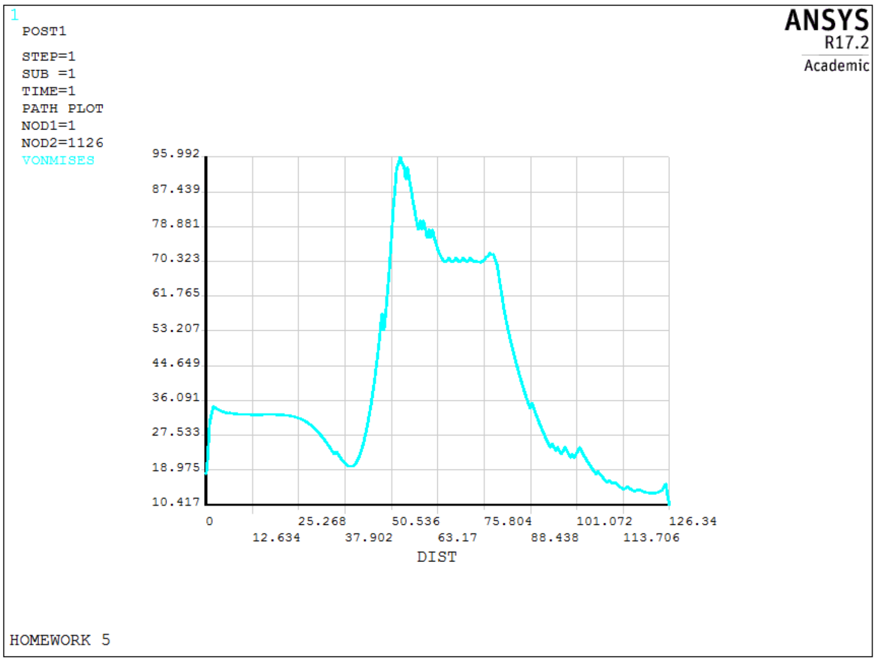


Figure 5.7: Distribution of forces across the junction by defining a path

5.3 Command list

```
!********************
! PROBLEM HOMEWORK 4 *
!********************

FINISH
/CLEAR,START,NEW
/FILNAME,Homework4
! >>> PARAMETERS MODEL <<<
*SET,leng_v_pipe,500
*SET,diameter_v_pipe,100
*SET,thick_v_pipe,2
*SET,leng_h_pipe,1000
*SET,diameter_h_pipe,200
*SET,thick_h_pipe,4
*SET,alpha,90
*SET,EPS,1E-3
*SET,n_div_area,3

! >>> PROPERTIES MATERIAL <<<
*SET,E_Young,210000
*SET,ni,0.3

! >>> LOAD <<<
*SET,pressure,1
```

```
! >>> PRE PROCESSING <<<
/PREP7
ET,1,SHELL181
KEYOPT,1,8,2
SECTYPE,1,shell
SECDATA,thick_h_pipe
MP,EX,1,E_Young
MP,PRXY,1,ni

ET,2,SHELL181
KEYOPT,2,8,2
SECTYPE,2,shell
SECDATA,thick_v_pipe
MP,EX,2,E_Young
MP,PRXY,2,ni

TYPE,1
SECNUM,1
MAT,1
SAVE

! >>> DEFINE RF CYLINDRICAL <<<
CLOCAL,100,CYLIN,0,diameter_h_pipe/2-leng_v_pipe/10,0,,-alpha
CSYS,0
CLOCAL,200,CYLIN,0,0,0,,,alpha
```

```
!***VERTICAL PIPE
CSYS,100
K,1,diameter_v_pipe/2,0,0
K,2,diameter_v_pipe/2,0,leng_v_pipe+leng_v_pipe/10
K,3,0,0,0
K,4,0,0,leng_v_pipe
L,1,2
!***HORIZZONTAL PIPE
CSYS,200
K,5,diameter_h_pipe/2,-alpha,0
K,6,diameter_h_pipe/2,-alpha,leng_h_pipe/2
K,7,0,0
K,8,0,0,leng_h_pipe/2
L,5,6
!***GEN AREA H PIPE
AROTAT,2,,,,,,8,7,-2*alpha
!***GEN AREA V PIPE
CSYS,100
AROTAT,1,,,,,,3,4,alpha
SAVE

APTN,2,3
ADELE,4,5
LDELE,10
LDELE,15
LDELE,16
SAVE

CSYS,0
WPOFFS,-50,100
WPROTA,,250
ASBW,7
WPOFFS,50,-10
WPROTA,,-250
WPROTA,,,alpha
WPOFFS,,,64.3
ASBW,3
WPROTA,,alpha
WPOFFS,,,-10
ASBW,6
SAVE

! >>> MESHING <<<
LCCAT,16,21
LCCAT,15,21
AESIZE,ALL,n_div_area

!***HORIZZONTAL PIPE
ESYS,200
TYPE,1
SECNUM,1
MSHKEY,1
MSHAPE,0,2D
AMESH,4
AMESH,5
AMESH,2
AMESH,1
SAVE

!***VERTICAL PIPE
ESYS,100
SAVE
TYPE,2
SECNUM,2
SAVE
MSHKEY,1
MSHAPE,0,2D
AMESH,7
AMESH,3
SAVE

! >>> VERIFY THE NORMAL VERSOR <<<

! >>> DEFINE RF CYLINDRICAL <<<
CLOCAL,100,CYLIN,0,diameter_h_pipe/2-thick_h_pipe/2,0,,-alpha
EPLOT
/PSYMB,ESYS,1

! >>> SOLUTION <<<
SAVE
SFE,ALL,,PRES,,pressure
/PBC,ALL,,1
NSEL,S,LOC,x,-EPS,+EPS
DSYM,SYMM,x
NSEL,S,LOC,z,-EPS,+EPS
DSYM,SYMM,z
KSEL,S,KP,,6
NSLK,S
DSYM,SYMM,y
ALLSEL,ALL
/SOLU
SOLVE
FINISH

! >>> POSTPROCESSING <<<
/POST1
/ESHAPE,1
PLDISP,1

SHELL,MID
RSYS,SOLU
PLNSOL,S,x
PLNSOL,S,y
PRNSOL,S,comp

!*********************
! PROBLEM HOMEWORK 5 *
!*********************

FINISH
/CLEAR, START, NEW
/TITLE, HOMEWORK 5
/FILNAME,HOMEWORK5,1
! >>> PARAMETERS MODEL <<<
*DO,k,0,150,25
*SET,leng_v_pipe,490
*SET,diameter_v_pipe,100
*SET,thick_v_pipe, 2
*SET,leng_h_pipe,1000
*SET,diameter_h_pipe,200
*SET,thick_h_pipe,4
*SET,fillet,10
*SET,alpha,90
*SET,EPS,1E-3
*SET,e_lenght,1
!***PARAMETERS CUT BOUNDARES
*SET,vCutBun,35+(k/25*5))
*SET,rCutBun,65+(k/25*5)
*SET,hCutBun,(diameter_h_pipe/2)*1.1

! >>> PROPERTIES MATERIAL <<<
*SET,E_Young, 210000
*SET,ni,0.3

! >>> LOAD <<<
*SET,pressure, 1

! >>> PRE PROCESSING <<<
/PREP7
ET,1,SOLID186
MP,EX,1,E_Young
MP,PRXY,1,ni
TYPE,1
SECNUM,1
MAT,1
SAVE
```

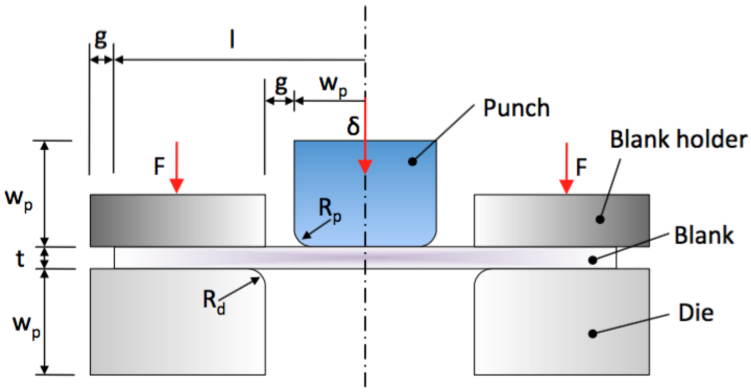
```
CSYS,0
CLOCAL,200,CYLIN,0,0,0,,,alpha

!***VERTICAL PIPE
CSYS,100
WPCSYS,,100
WPOFFS,0,0,-50
CYL4,0,0,(diameter_v_pipe/2)-thick_v_pipe,0,diameter_v_pipe/2,alpha,leng_v_pipe*(1+(109/490))
!***HORIZZONTAL PIPE
CSYS,200
WPCSYS,,200
CYL4,0,0,(diameter_h_pipe/2)-thick_h_pipe,-alpha,diameter_h_pipe/2,alpha,leng_h_pipe/2

VPTN,1,2
VDELE,3,4,,1
AFILLT,33,23,fillet
AL,2,14,33
AL,1,34,16
VA,5,3,6,12,7
SAVE

!***CUT VERTICAL PIPE
WPCSYS,,100
WPOFFS,0,0,vCutBun
VSBW,5,SEPO,DELETE
!***CUT HORIZZONTAL PIPE
CSYS,100
K,100,rCutBun,-(hCutBun-diameter_h_pipe/2),-hCutBun
K,101,rCutBun,hCutBun,-hCutBun
L,100,101
K,102,0,0,-diameter_h_pipe/2
K,103,0,0,diameter_h_pipe/2
L,102,103
ADRAG,5,,,,,,6
VSBA,7,2
!***JOIN SUBMODEL
VADD,3,1,6,4
VSEL,U,VOLU,,7
VDELE,ALL,,,1
ALLSEL,ALL
!***JOINS AREAS
AADD,26,5,31,34
AADD,41,6,32,36
SAVE


! >>> MESHING <<<
CSYS,0
MSHAPE,1,3D
MSHKEY,0
ESIZE,e_lenght
VMESH,ALL
SAVE
/VIEW,1,,,-1
/ANG,1
/REP,FAST

! >>> SHELL TO SOLID SUBMODELS <<<
ASEL,S,AREA,,23,25,2
NSLA,S
NWRITE,subHW5,node,,0
ALLSEL,ALL
SAVE
RESUME,'homework4','db'
/POST1
FILE,HOMEWORK4,rst
SET,first
cbdof,subHW5,node,,DHW5,cbdo,,,,1

RESUME,HOMEWORK5,db
/PREP7
/INPUT,DHW5,cbdo
/INPUT,DHW5,cbdo,,:cb1

! >>> SOLUTION <<<
SAVE
NSEL,S,LOC,x,-eps,+eps
DSYM,SYMM,x
NSEL,S,LOC,z,-eps,+eps
DSYM,SYMM,z
ASEL,S,AREA,,20,24,2
ASEL,A,AREA,,1
NSLA,S
SF,ALL,PRES,pressure
ALLSEL,ALL
/SOLU
SOLVE
FINISH

/VIEW,1,,,-1
/ANG,1
/REP,FAST
! >>> POSTPROCESSING <<<
/POST1
PLDISP,1
PLNSOL,S,Y
PLNSOL,S,X
PRNSOL,S,COMP

PLNSOL,S,X
*GET,stressmax,PLNSOL,,max
*CFOPEN,'Stress','txt',,append
 *VWRITE,k/25,vCutBun,rCutBun,stressmax
   (F20.10,F20.10,F20.10,F20.10)
*cfclos
```

```
PLNSOL,S,eqv
*GET,stressEQV,PLNSOL,,max
*CFOPEN,'stressEQV','txt',,append
 *VWRITE,k/25,vCutBun,rCutBun,stressEQV
  (F20.10,F20.10,F20.10,F20.10)
*CFCLOS

LSEL,S,LINE,,8
LSEL,A,LINE,,2
LSEL,A,LINE,,28
NSLL,S,1
FLST,2,263,1
FITEM,2,1
FITEM,2,56760
FITEM,2,56761
FITEM,2,56762
FITEM,2,56763
FITEM,2,56764
FITEM,2,56765
FITEM,2,56766
FITEM,2,56767
FITEM,2,56768
FITEM,2,56769
FITEM,2,56771
FITEM,2,56771
FITEM,2,56770
FITEM,2,56771
FITEM,2,56771
FITEM,2,56771
FITEM,2,56772
FITEM,2,56773
FITEM,2,56774
FITEM,2,56775
FITEM,2,56776
FITEM,2,56777
FITEM,2,56779
FITEM,2,56779
FITEM,2,56778
FITEM,2,56779
FITEM,2,56780
FITEM,2,56781
FITEM,2,56782
FITEM,2,56783
FITEM,2,56784
FITEM,2,56785
FITEM,2,56787
FITEM,2,56786
FITEM,2,56787
FITEM,2,56786
FITEM,2,56786
FITEM,2,56787
FITEM,2,56788
FITEM,2,56789
FITEM,2,56790
FITEM,2,56791
FITEM,2,56792
FITEM,2,56793
FITEM,2,56794
FITEM,2,56795
FITEM,2,56796
FITEM,2,56797
FITEM,2,56798
FITEM,2,56799
FITEM,2,56800
FITEM,2,56801
FITEM,2,56802
FITEM,2,56803
FITEM,2,56804
FITEM,2,56805
FITEM,2,56806
FITEM,2,56807
FITEM,2,56808
FITEM,2,56809
FITEM,2,56810
FITEM,2,56811
FITEM,2,56812
FITEM,2,56813
FITEM,2,56814
FITEM,2,56815
FITEM,2,56816
FITEM,2,56817
FITEM,2,56818
FITEM,2,56819
FITEM,2,56821
FITEM,2,56821
FITEM,2,56820
FITEM,2,56821
FITEM,2,56822
FITEM,2,56823
FITEM,2,56824
FITEM,2,56825
FITEM,2,56826
FITEM,2,56827
FITEM,2,56828
FITEM,2,56829
FITEM,2,56830
FITEM,2,56831
FITEM,2,56832
FITEM,2,56833
FITEM,2,56834
FITEM,2,56835
FITEM,2,56836
FITEM,2,56837
FITEM,2,56838
FITEM,2,56839
FITEM,2,56840
FITEM,2,56841
FITEM,2,56842
FITEM,2,56843
FITEM,2,56845
FITEM,2,56845
FITEM,2,56844
FITEM,2,56845
FITEM,2,56846
FITEM,2,56847
FITEM,2,56848
FITEM,2,51894
FITEM,2,51925
FITEM,2,51924
FITEM,2,51923
FITEM,2,51922
FITEM,2,51921
FITEM,2,51919
FITEM,2,51919
FITEM,2,51920
FITEM,2,51919
FITEM,2,51918
FITEM,2,51917
FITEM,2,51916
FITEM,2,51914
FITEM,2,51915
FITEM,2,51914
FITEM,2,51914
FITEM,2,51914
FITEM,2,51915
FITEM,2,51915
FITEM,2,51913
FITEM,2,51914
FITEM,2,51913
FITEM,2,51914
FITEM,2,51914
FITEM,2,51913
FITEM,2,51912
FITEM,2,51911
FITEM,2,51910
FITEM,2,51909
FITEM,2,51909
FITEM,2,51909
FITEM,2,51907
FITEM,2,51907
FITEM,2,51909
FITEM,2,51909
```

```
FITEM,2,51907
FITEM,2,51907
FITEM,2,51909
FITEM,2,51907
FITEM,2,51909
FITEM,2,51907
FITEM,2,51909
FITEM,2,51909
FITEM,2,51909
FITEM,2,51909
FITEM,2,51909
FITEM,2,51909
FITEM,2,51907
FITEM,2,51907
FITEM,2,51907
FITEM,2,51907
FITEM,2,51908
FITEM,2,51907
FITEM,2,51906
FITEM,2,51905
FITEM,2,51904
FITEM,2,51903
FITEM,2,51902
FITEM,2,51902
FITEM,2,51902
FITEM,2,51901
FITEM,2,51900
FITEM,2,51899
FITEM,2,51898
FITEM,2,51897
FITEM,2,51896
FITEM,2,51895
FITEM,2,30418
FITEM,2,30660
FITEM,2,30659
FITEM,2,30658
FITEM,2,30657
FITEM,2,30656
FITEM,2,30655
FITEM,2,30654
FITEM,2,30653
FITEM,2,30652
FITEM,2,30651
FITEM,2,30650
FITEM,2,30649
FITEM,2,30648
FITEM,2,30647
FITEM,2,30645
FITEM,2,30645
FITEM,2,30646
FITEM,2,30645
FITEM,2,30645
FITEM,2,30645
FITEM,2,30644
FITEM,2,30643
FITEM,2,30642
FITEM,2,30641
FITEM,2,30640
FITEM,2,30639
FITEM,2,30638
FITEM,2,30636
FITEM,2,30636
FITEM,2,30637
FITEM,2,30635
FITEM,2,30635
FITEM,2,30636
FITEM,2,30634
FITEM,2,30636
FITEM,2,30633
FITEM,2,30634
FITEM,2,30633
FITEM,2,30636
FITEM,2,30635
FITEM,2,30634
FITEM,2,30633
FITEM,2,30632
FITEM,2,30631
FITEM,2,30630
FITEM,2,30629
FITEM,2,30627
FITEM,2,30627
FITEM,2,30628
FITEM,2,30627
FITEM,2,30626
FITEM,2,30625
FITEM,2,30623
FITEM,2,30623
FITEM,2,30624
FITEM,2,30622
FITEM,2,30622
FITEM,2,30623
FITEM,2,30622
FITEM,2,30621
FITEM,2,30620
FITEM,2,30618
FITEM,2,30618
FITEM,2,30620
FITEM,2,30620
FITEM,2,30619
FITEM,2,30618
FITEM,2,30616
FITEM,2,30616
FITEM,2,30618
FITEM,2,30618
FITEM,2,30618
FITEM,2,30618
FITEM,2,30617
FITEM,2,30616
FITEM,2,30615
FITEM,2,30614
FITEM,2,30613
FITEM,2,30612
FITEM,2,30611
FITEM,2,30610
FITEM,2,30609
FITEM,2,30608
FITEM,2,30607
FITEM,2,30606
FITEM,2,30605
FITEM,2,30604
FITEM,2,30603
FITEM,2,30602
FITEM,2,1126
PATH,STREQV,263,30,20,
PPATH,P51X,1
PATH,STAT
AVPRIN,0, ,
PDEF,VONMISES,S,EQV,AVG
/PBC,PATH, ,1
PLPATH,VONMISES
FINISH
/CLEAR,START
*ENDDO
```

Chapter 6

Homework 6



DATA:

l	=	50	mm
g	=	2	mm
t	=	$1,5$	mm
wp	=	25	mm
Rp	=	4	mm
Rd	=	3	mm
δ	=	$4t$	
F	=	100	kN
Blank:		S355JR	steel
E	=	205	GPa
σ_y	=	355	MPa
Ep	=	4	GPa

The figure schematically illustrates the deep drawing of a metal sheet (blank). During the forming process, the blank is pressed against the die applying a preload F, then the punch is gradually displaced by δ in order to push the blank inside the die cavity. The stiffness of punch, die and blank holder is assumed to be much higher than that of the blank. The friction cofficient is $0,1$. Using axisymmetric plane elements, build a FE model able to simulate the forming process. In particular it is required to:

1. determine the distribution of the Von Mises equivalent stress at the end of the travel of the punch and the maximum axial force applied to the punch.

2. determine the punch stroke that maintains the maximum absolute hoop strain below 5%.

3. determine the residual stress distribution on the top and the bottom surface of the blank after the punch removal.

4. determine the elastic springback, i.e. the difference in axial displacement prior to and after the punch removal of the points lying on the blank midplane.

6.1 Approach the problem

This problem has created a model for the respective components: from the punch, through the use of keypoint and tracing lines to data provided by subsequently issue are made blankholder and die using the same procedure.
Then the blank is achieved the model as before, in the figure 6.1.
Summary characteristics of the blank holder, punch and die:

- The stiffness is assumed much higher than the blank, in fact it is not any material properties to the elements was attributed.
- Displacement of punch: $\delta = 4 * 1,5\,mm$;
- Preload between blank holder - blank: $F = 100\,kN$;

Instead, the blank has chosen to use:

- element type: PLANE183;
- using axisymmetric plane elements: KEYOPT(3),1

To realize the contact between the different components was used:

- the friction cofficient is 0.1.
- element type for rigid target body: TARGE169;
- element type for deformable contact body: CONTA172;
- optimization for deformable contact:
  - Contact algorithm: *Augmented Lagrangian*, KEYOPT(2), 0;
  - element level time incrementation control, KEYOPT(7), 1.
    Automatic bisection of increment;
  - To take into account the initial penetration or model initial interference, set KEYOPT(9), 2.
    To ramp the initial penetration with the first load step (to model initial interference problems, for example);
  - contact stiffness update for each iteration: KEYOPT(10), 2.

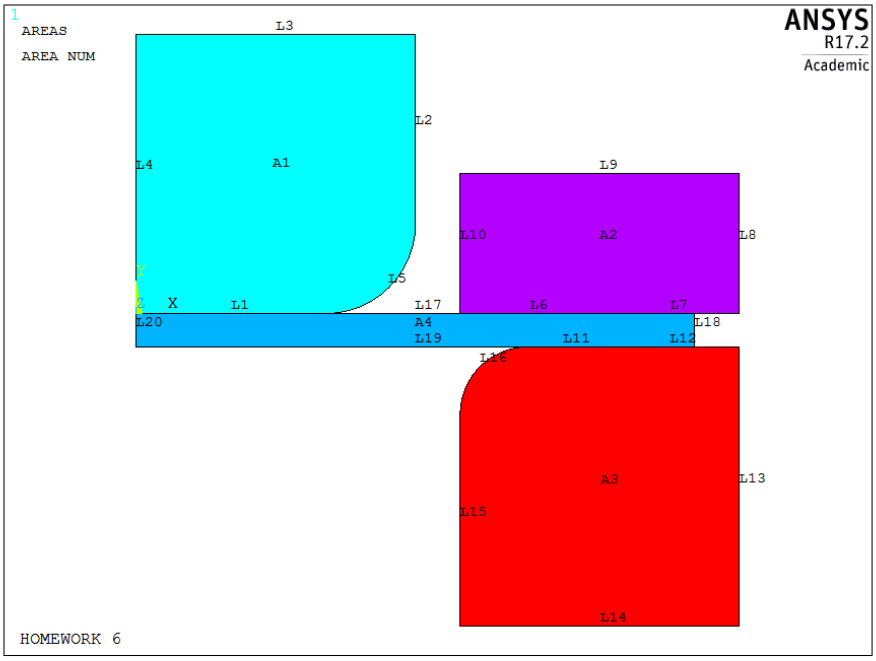


Figure 6.1: Complete geometry model

After these preliminary operations of the geometry creation is attributed to the stat blank mesh composed of elements such PLANE183 with optimization for axisymmetric. The elements have a length of 1 mm, at the same time for the entire height of 12 divisions have been made. Finally you get the mapped mesh as show in figure 6.2.

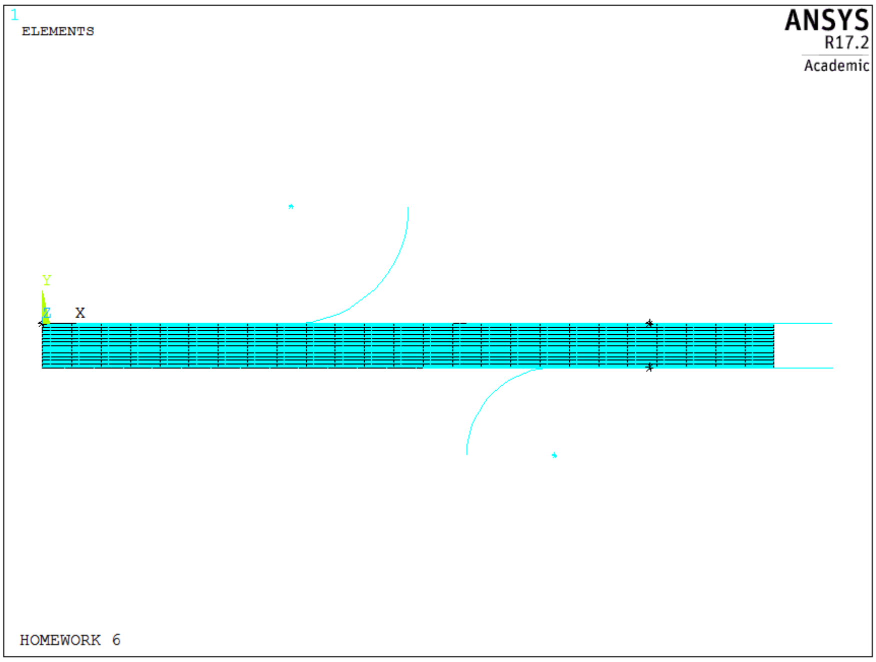


Figure 6.2: Complete meshed model

For molding simulation has chosen to use tree step, a first phase where the force is applied between the blank holder and the blank, as shown in the figure 6.3. The second phase

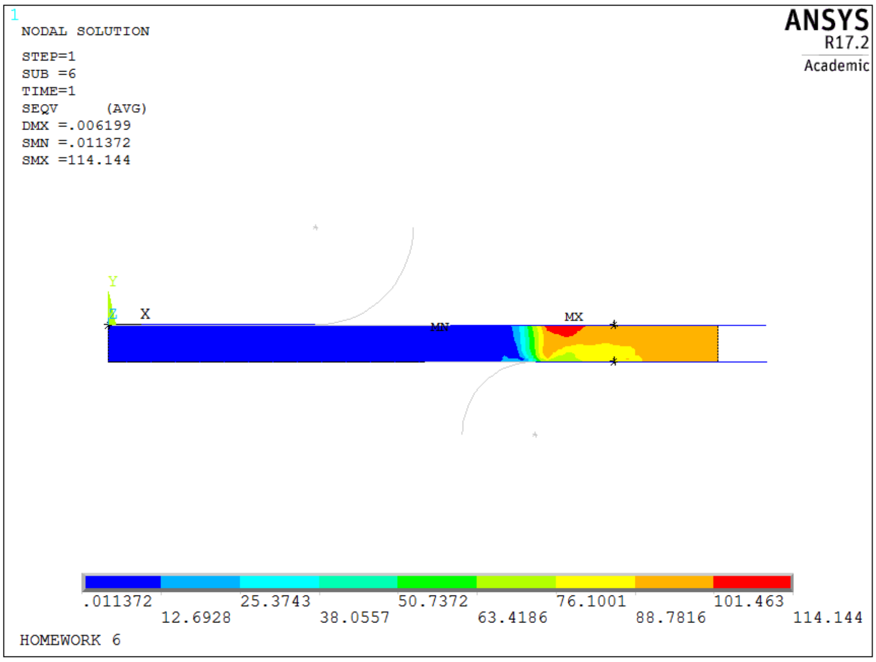


Figure 6.3: Pre gripping

is the descent of the die for a displacement of 6 mm, the last stage consists in stopping the die and in its ascent. The application of the load function is shown in the following chart 6.4. It has opted for a high number of steps to avoid convergence problems during the solution phase, as in the first trials advised the program to increase the number of steps in the phase of application of the force as it was not able to find the solution after several iterations.

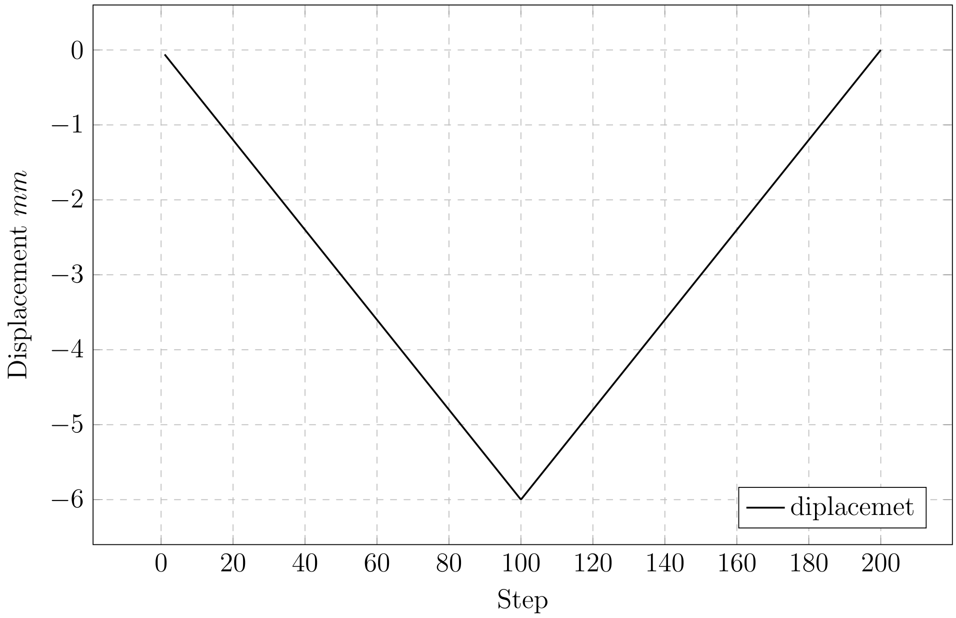


Figure 6.4: Load step die

6.2 Result and Conclusion

The Von Mises equivalent stress at the end of the travel of the punch and the maximum axial force applied to the punch, as show in figure 6.5.
Max axial force applied to the punch is eqaul to $F = -58729,04097\,N$.

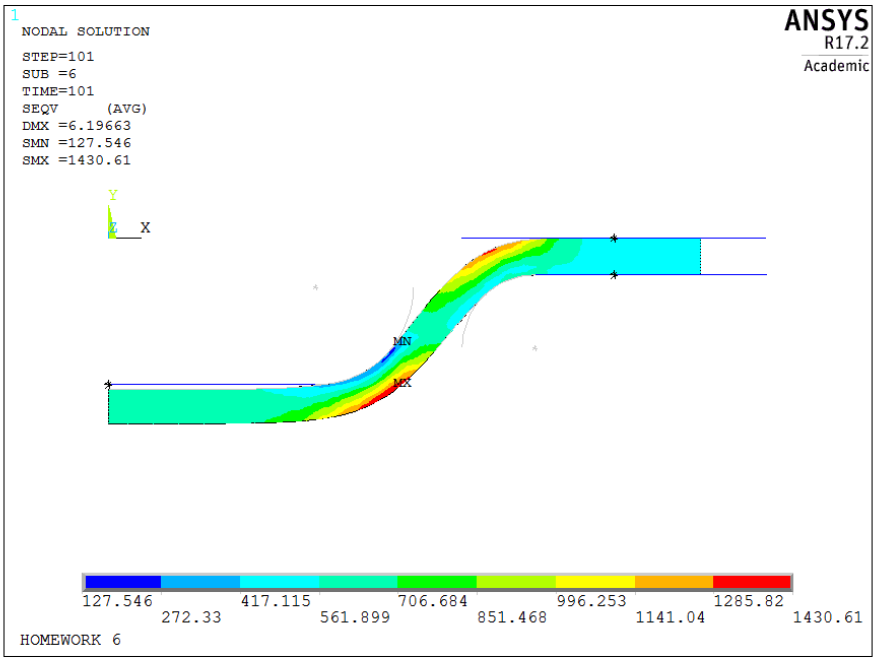


Figure 6.5: Equivalent stress Von Mises

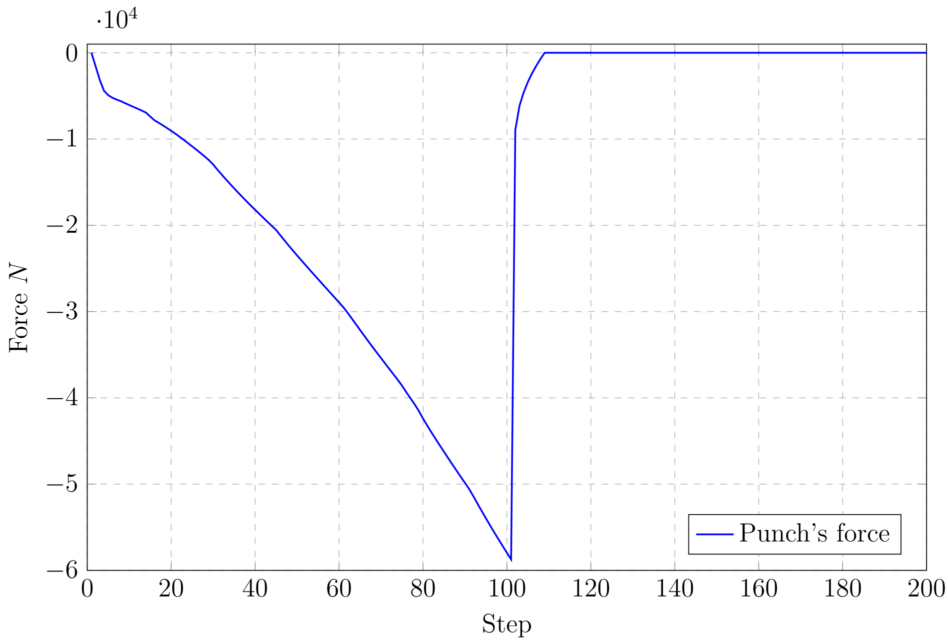


Figure 6.6: Force

The punch stroke that maintains the maximum absolute, where is equal to $\sigma_h = 0,09445\,MPa$, hence hoop stress close to the $5\,\%$ is equal $\sigma_{h5\%} = 0,0047\,MPa$, hence the value of hoop strain is equal to $0,57169\,mm$ as show in figure 6.7.

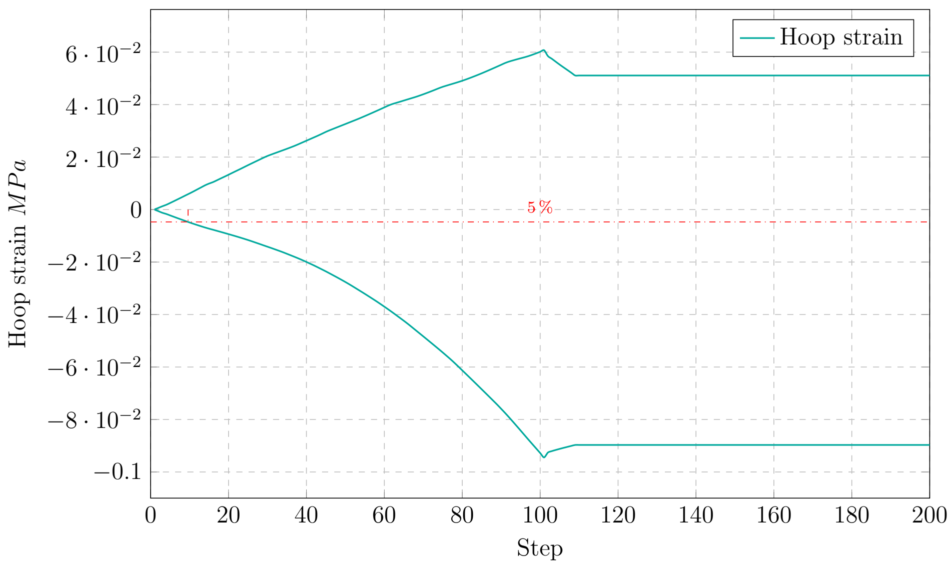


Figure 6.7: Hoop strain below $5\,\%$

When a metal forming tool is planned and designed to deform a work piece, the shape imparted by the tool will be a combination of elastic and plastic deformation, the release of the elastic deformation is the spring back often observed at the end of a metal forming process. The spring back has to be compensated to achieve an accurate result.

The springback effect is visible in the follow graph 6.8.

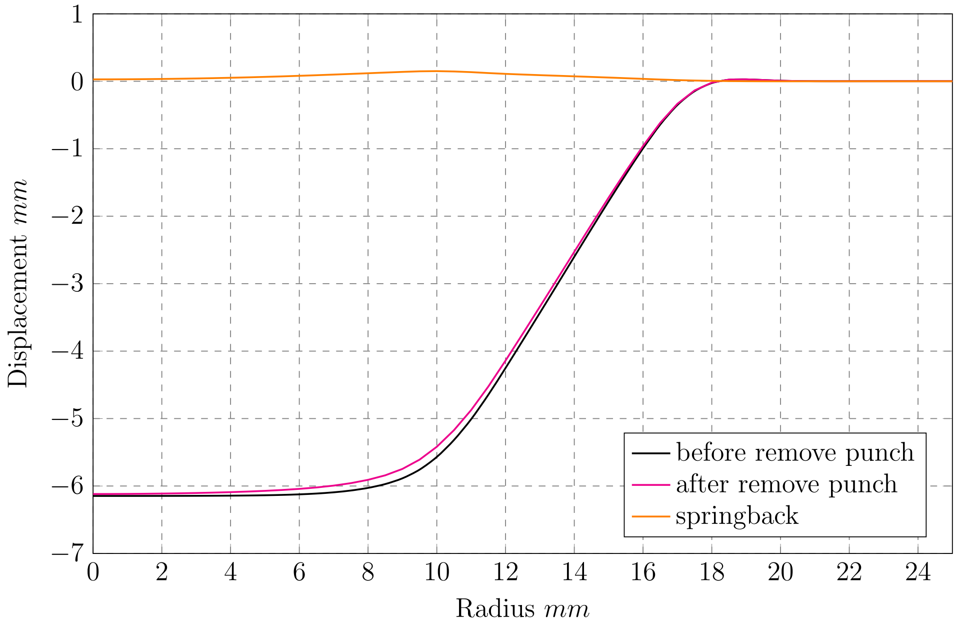


Figure 6.8: Springback effect

Finally shows the plotted data showing the residual stress distribution on top and bottom surface of the blank after the punch removal, as in figures 6.9, 6.10 and 6.11.

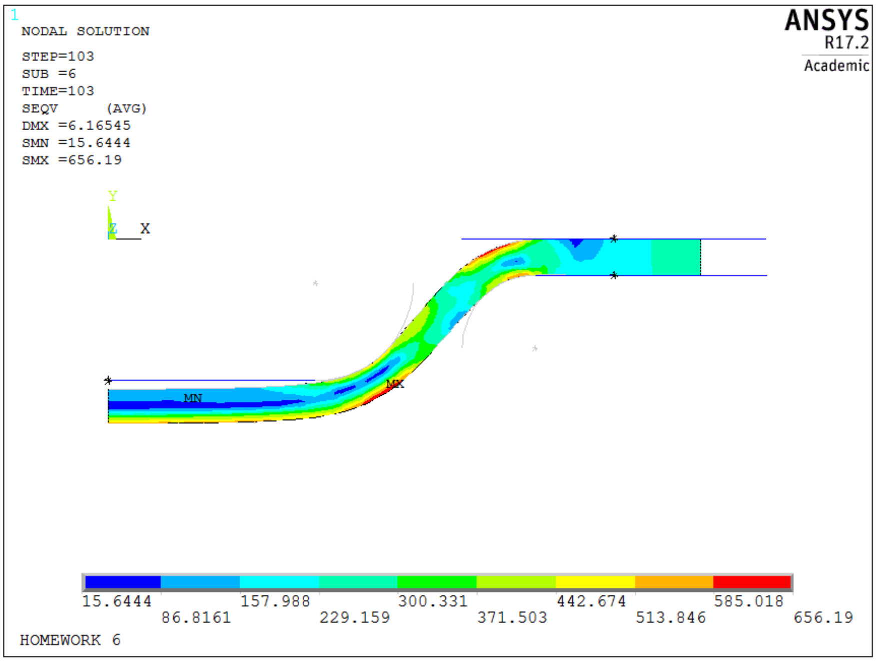


Figure 6.9: Equivalent stress Von Mises

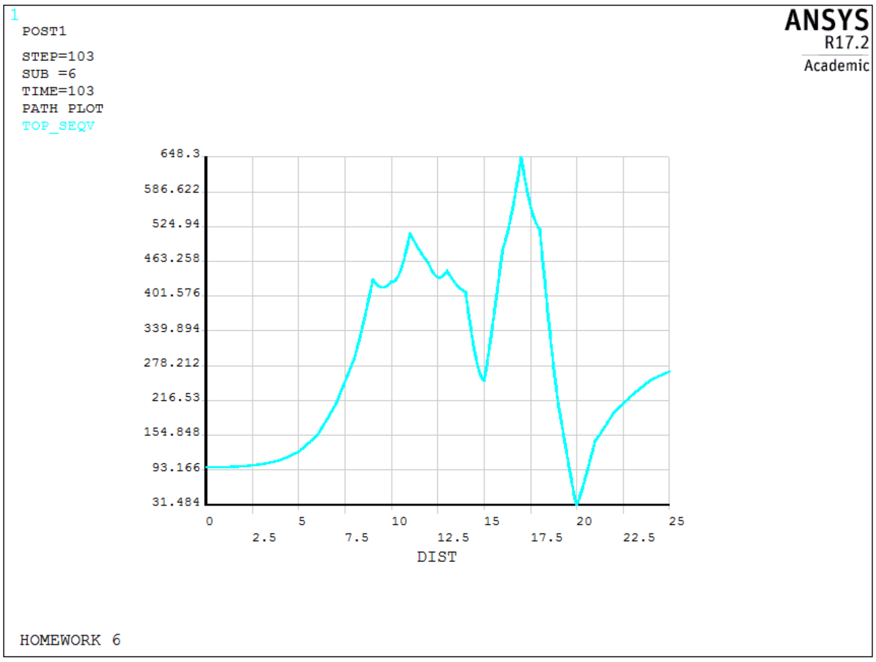


Figure 6.10: Equivalent stress Von Mises top surface

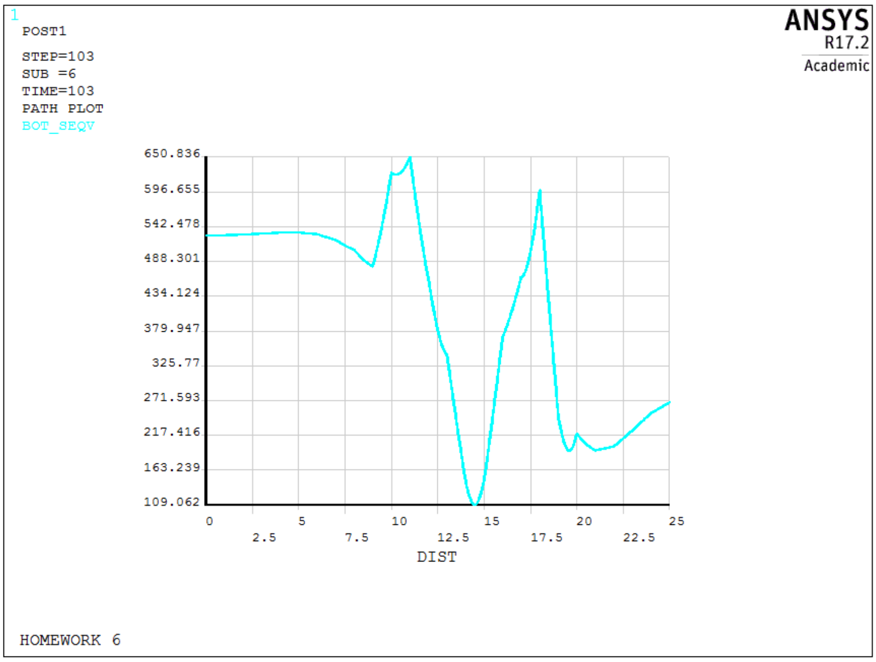


Figure 6.11: Equivalent stress Von Mises bottom surface

6.3 Command list

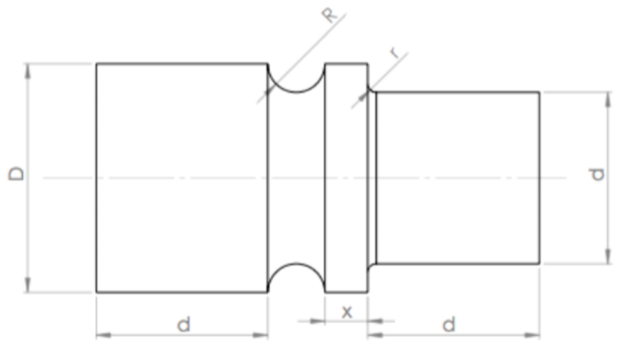
```
!*********************
! PROBLEM HOMEWORK 6 *
!*********************

FINISH
/CLEAR,START,NEW
/TITLE,HOMEWORK 6
/FILNAME,HOMEWORK6
! >>> PARAMETERS MODEL <<<
*SET,l,50/2
*SET,g,2
*SET,t,1.5
*SET,Wp,25/2
*SET,Rp,4
*SET,Rd,3
*SET,delta,4*t
*SET,EPS,1E-3
*SET,n_div,12
*SET,e_lenght,1
*SET,step,100
*SET,x,delta/step

! >>> PROPERTIES MATERIAL <<<
*SET,E_Young,205E3
*SET,ni,0.3
*SET,SigmaYield,355
*SET,E_p,4E3
*SET,friction,0.1

! >>> LOAD <<<
*SET,F,100E3

! >>> PRE PROCESSING <<<
/PREP7
ET,1,PLANE183,,,1

MP,EX,1,E_Young
MP,PRXY,1,ni
MP,MU,1,friction
TB,BKIN,1,1
TBTEMP,0.0
TBDATA,1,SigmaYield,E_p

!***PUNCH MODEL
K,1,0,0
K,2,Wp,0
K,3,Wp,Wp
K,4,0,Wp
L,1,2
L,2,3
L,3,4
L,4,1
LFILLT,1,2,Rp
AL,1,2,3,4,5

!***BLANCK HOLDER MODEL
K,7,Wp+g,0
K,8,Wp+g+(l/4),0
K,9,l+g,0
K,10,l+g,Wp/2
K,11,Wp+g,Wp/2
L,7,8
L,8,9
L,9,10
L,10,11
L,11,7
AL,6,7,8,9,10

!***DIE MODEL
K,12,Wp+g,-t
K,13,Wp+g+(l/4),-t
K,14,l+g,-t
K,15,l+g,-t-Wp
K,16,Wp+g,-t-Wp
L,12,13
L,13,14
L,14,15
L,15,16
L,16,12
LFILLT,15,11,Rd
AL,11,12,13,14,15,16
K,19,0,0
K,20,l,0
K,21,l,-t
K,22,0,-t
L,19,20
L,20,21
L,21,22
L,22,19
AL,17,18,19,20
SAVE

! >>> MESHING <<<
LESIZE,18,,,n_div
LESIZE,20,,,n_div

ESIZE,e_lenght
MSHAPE,0
MSHKEY,1
AMESH,4
SAVE

! >>> DEFINITION CONTACT <<<
ET,2,TARGE169
ET,3,CONTA172
KEYOPT,3,2,0
KEYOPT,3,7,1
KEYOPT,3,9,2
```

```
KEYOPT,3,10,2

!****PUNCH - BLANK
!***DEFORMABLE BLANK
TYPE,3
REAL,1
LSEL,s,LINE,,17
NSLL,s,1
NSEL,R,LOC,x,0,Wp+g
ESURF
ALLSEL,all
SAVE

!***RIGIDBODY PUNCH
TYPE,2
REAL,1
TSHAP,LINE
LMESH,1
TSHAP,ARC
LMESH,5
SAVE
TSHAP,PILO
KMESH,1

!***CONTACT BLANKHOLDER-BLANK
TYPE,3
REAL,2
LSEL,s,LINE,,17
NSLL,s,1
NSEL,R,LOC,x,Wp+g,l+g
ESURF
ALLSEL,all
SAVE

TYPE,2
REAL,2
TSHAP,LINE
LMESH,6,7
TSHAP,PILO
KMESH,8
SAVE

!***CONTACT DIE-BLANK
TYPE,3
REAL,3
LSEL,S,LINE,,19
NSLL,S,1
NSEL,R,LOC,x,Wp,l
ESURF
ALLSEL,all
SAVE

TYPE,2
REAL,3
TSHAP,LINE
LMESH,11,12
LSEL,S,LINE,,11,12
ESLL,S
ESURF,,REVERSE
ALLSEL,ALL
SAVE

TSHAP,ARC
LMESH,16
ESEL,S,,,345
ESURF,,REVERSE
ALLSEL,ALL
TSHAP,PILO
KMESH,13
SAVE

! >>>CONSTRAIN <<<
LSEL,S,,,20
NSLL,S
DSYM,SYMM,X
ALLSEL,ALL,ALL
SAVE

! >>> LOAD CONDITION <<<
!***LOAD STEP 1: PRELOAD
TIME,1
F,981,fy,-F
OUTRES,ALL,ALL
LSWRITE,1
!***LOAD STEP 2: DESCENT OF PUNCH
*DO,i,1,step
 TIME,1+i
 D,976,uy,-x*i
 OUTRES,ALL,ALL
 LSWRITE,1+i
*ENDDO
!***LOAD STEP 3: ASCENT OF PUNCH
*DO,i,step+2,step*2
 TIME,i
 D,976,uy,(-x*step)+(i-step)*x
 OUTRES,ALL,ALL
 LSWRITE,i
*ENDDO
SAVE

! >>> SOLUTION <<<
/SOLU
SOLCONTROL,ON
NSUBST,10,100,2
NROPT,FULL,,OFF
AUTOTS,ON
EQSLV,PCG
NLGEOM,ON
LSSOLVE,1,2*step
FINISH

! >>> POST PROCESSING <<<
```

```
/POST1
*DO,N,1,2*step
 SET,N
 PLNSOL,S,EQV
*ENDDO

!***VON MISES EQUIVALENT STRESS
!END OF THE TRAVEL OF THE PUNCH
SET,step+1
PLNSOL,S,EQV
PRRFOR,FY
*DO,N,1,2*step
 SET,N
 *GET,F,NODE,976,RF,FY
 *CFOPEN,ForcePunch,txt,,APPEND
 *VWRITE,N,F
 (F20.10,F20.10)
 *CFCLOS
*ENDDO

!***PUNCH STROKE
!THE MAXIMUM ABSOLUTE HOOP STRAIN BELOW 5%.
*DO,N,1,2*step
 SET,N
 PLNSOL,EPTO,Z
 *GET,MAXSTR,PLNSOL,0,MAX
 *GET,MINSTR,PLNSOL,0,MIN
 *CFOPEN,HoopStrain,txt,,APPEND
 *VWRITE,N,MAXSTR,MINSTR
 (F20.10,F20.10,F20.10)
 *CFCLOS
*ENDDO
SAVE

!***RESIDUAL STRESS DISTRIBUTION SURFACE
SET,103
PATH,TOP,2,,1000
PPATH,1,,0,0
PPATH,2,,L,0
PDEF,TOP_SEQV,S,EQV
PLPATH,TOP_SEQV

!***RESIDUAL STRESS
!DISTRIBUTION BOTTOM SURFACE
PATH,BOTTOM,2,,1000
PPATH,1,,0,-T
PPATH,2,,L,-T
PDEF,BOT_SEQV,S,EQV
PLPATH,BOT_SEQV

!***AXIAL DISPLACEMENT AFTER THE PUNCH REMOVE
SET,step+1
NSEL,S,LOC,y,-(1.5/2)-0.05,-(1.5/2)+0.05
*GET,nnodi,NODE,,count
*DIM,dy,ARRAY,nnodi
*DIM,xn,ARRAY,nnodi
CM,nodi,NODE
*do,i,1,nnodi
 *GET,xmin,NODE,,mnloc,x
 nsel,r,loc,x,xmin-EPS,xmin+EPS
 *GET,nmin,NODE,,NUM,MAX
 xn(i)=xmin
 *GET,dy(i),NODE,nmin,u,y
 CMSEL,S,nodi
 NSEL,U,NODE,,nmin
 CM,nodi,NODE
 *CFOPEN,set%step+1%,txt
 *VWRITE,xn(1),dy(1)
 (F20.10,F20.10)
 *CFCLOS
*ENDDO
*DEL,ALL

SET,103
NSEL,S,LOC,y,-(1.5/2)-0.05,-(1.5/2)+0.05
*GET,nnodi,NODE,,COUNT
*DIM,dy,ARRAY,nnodi
*DIM,xn,ARRAY,nnodi
CM,NODI,NODE
*DO,i,1,nnodi
 *GET,XMIN,NODE,,MNLOC,X
 NSEL,R,LOC,X,xmin-EPS,xmin+EPS
 *GET,NMIN,NODE,,NUM,MAX
 xn(i)=XMIN
 *GET,DY(i),NODE,NMIN,U,Y
 CMSEL,S,NODI
 NSEL,U,NODE,,NMIN
 CM,NODI,NODE
 *CFOPEN,set103,txt
 *VWRITE,xn(1),dy(1)
 (F20.10,F20.10)
 *CFCLOS
*ENDDO
FINISH
```

Chapter 7

Homework 7



DATA:
$\frac{D}{d} = 1.4$
$\frac{r}{d} = \frac{1}{20}$
$R = \frac{(D-d)}{2}$
Material: steel

Determine the stress concentration factor (defined as ratio between the maximum first principal stress in the model and the remote stress acting on the cross-secton with diameter d), in the presence and in the absence of the optimized relief groove analyzed in Homework 3, when the shaft is subject to uniform bending and torsional moment.

7.1 Approach the problem

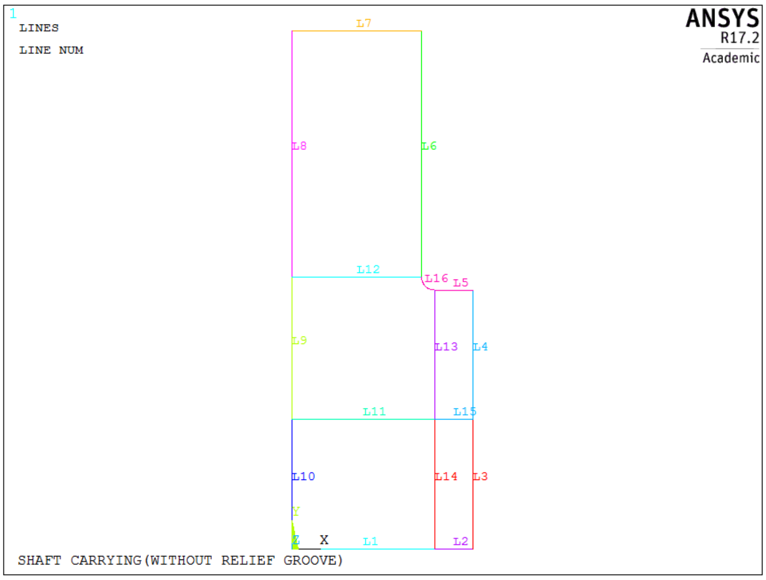
In this issue we used the geometric model produced in homework 3, setting the x previously found at a value equal to $x = 1,5$ for the shaft with relief groove. Then going to replace the elements of the mesh with the type Plane25. The mesh is mapped with the same number of divisions seen in Homework 3, obtaining the result shown in figures 7.1. The material present the follow proprities:

- Modulus of elasticity: $210\,GPa$;
- Poissons ratio 0.3.

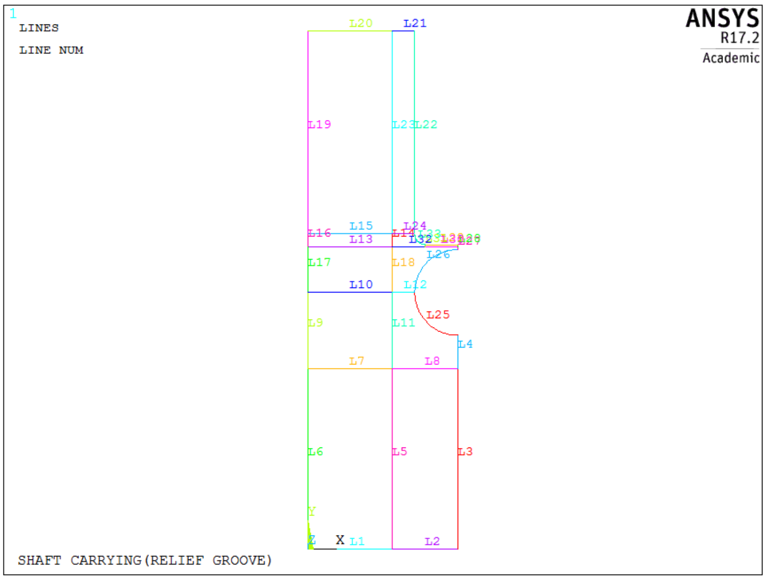
Assumptions adopted:

- Element type for both model Plane25;

- $x = 1.5$ optimal position;
- Bending moment : $1 * 10^5 Nm$;
- Torque moment: $1 * 10^6 Nm$;

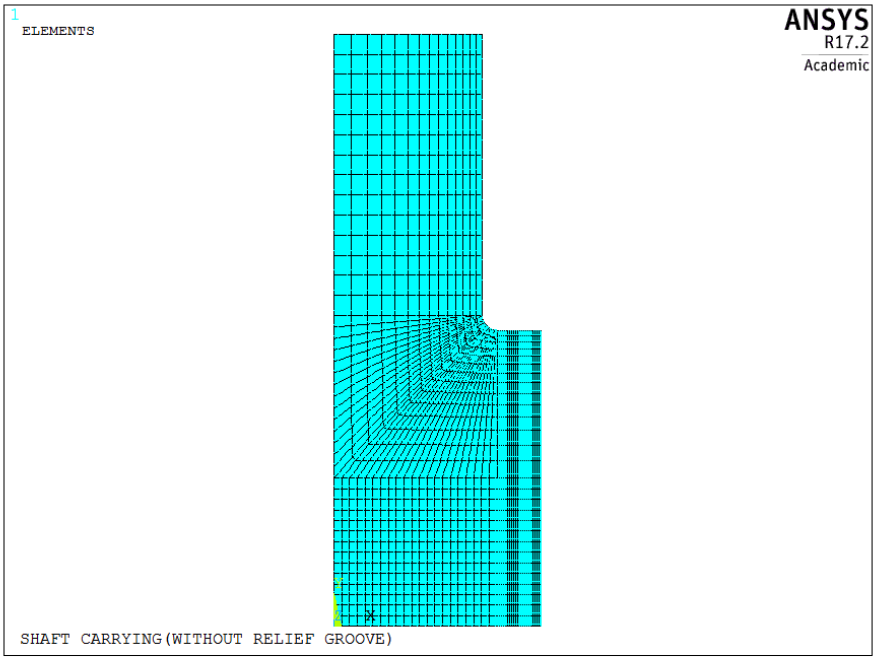


(a) Gemotry - Shaft without relief groove

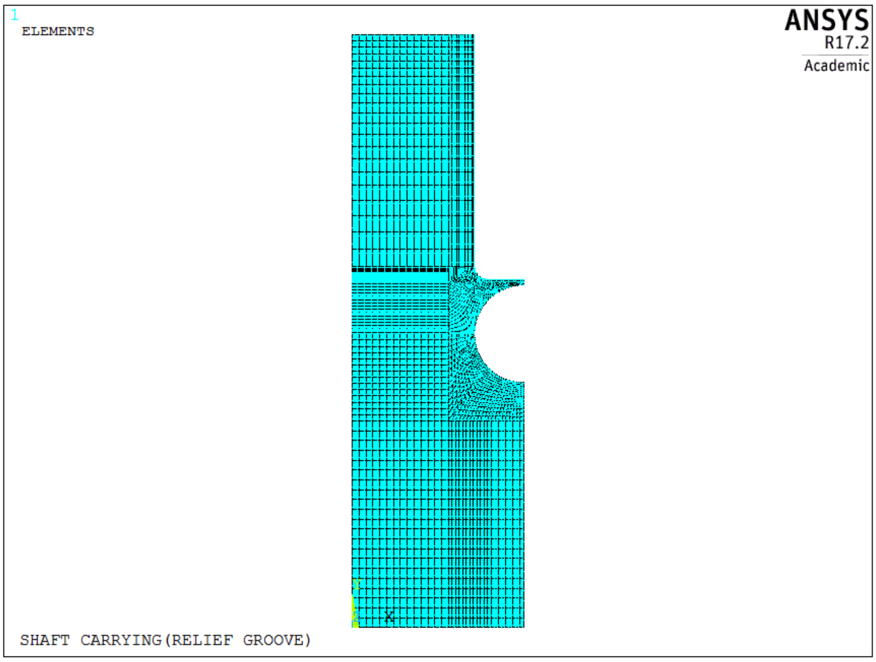


(b) Gemotry - Shaft relief groove

Figure 7.1: Model of the shaft used in previous homework 3



(a) Shaft without relief groove



(b) Shaft relief groove

Figure 7.2: Mapped mesh's model

7.2 Result

An analysis is performed only by first applying a bending moment at the end of the two shafts and compared the values obtained in the presence or not of the relief groove.

It is repeated the same analysis this time, by applying a torque. Finally, it analyzed the combination of the moments of both shafts. The stress concentration factor subject to:

7.2.1 Bending moment

Impose a bending moment eqault to Mf $= 1 * 10^5 Nm$.
The stress concentration factor is defined as: $K_t = \frac{\sigma_{1,max}}{S_{nom}}$, where $\sigma_{1,max}$ = maximum first principal stress in the model

$$S_{nom} = \frac{32Mf}{\pi d^3} = 3,8340\, MPa$$

Ratio of reduction $= \frac{K_{t1} - K_{t2}}{K_{t1}} = 47,53\%$

	Maximim first principal stress in the model $\frac{N}{mm^2}$	stress concentraton factor of analysis
normal shaft	$\sigma_{1,max} = 8,55803$	$K_{t1} = 2,23$
shaft with relief groove	$\sigma_{1,max} = 6,55013$	$K_{t1} = 1,71$

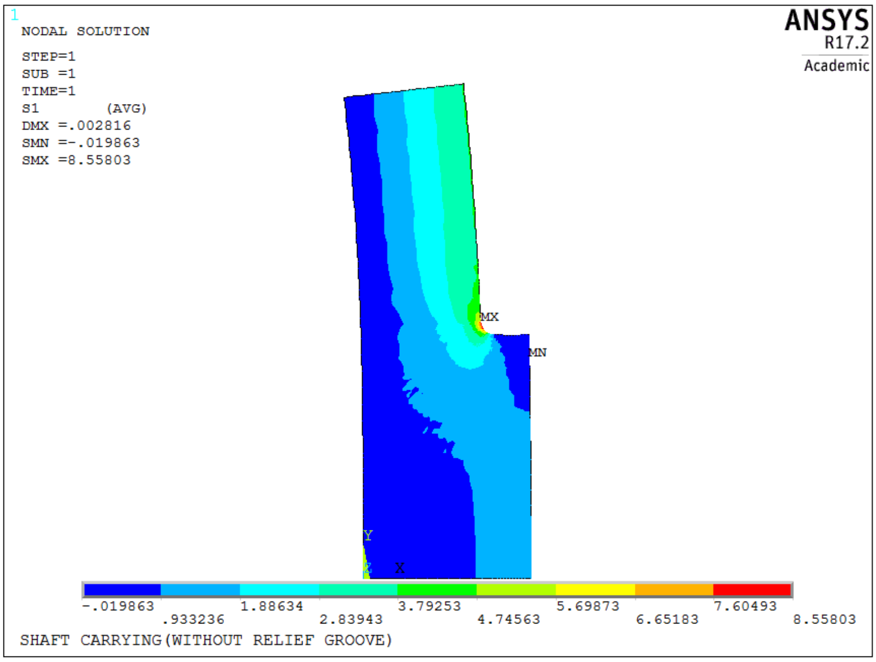


Figure 7.3: Max First Stress Bending Moment - Shaft without relief groove

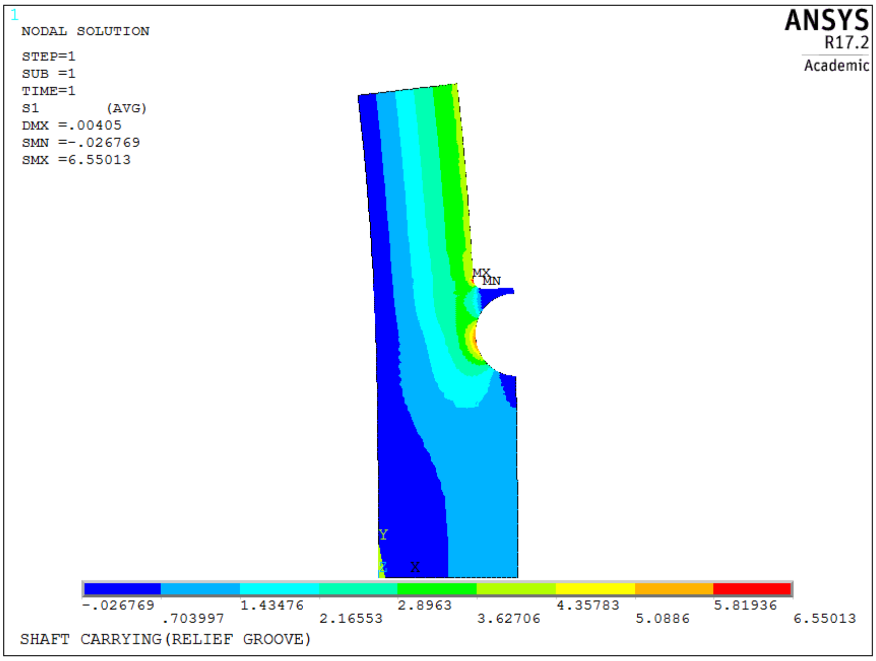


Figure 7.4: Max First Stress Bending Moment - Shaft relief groove

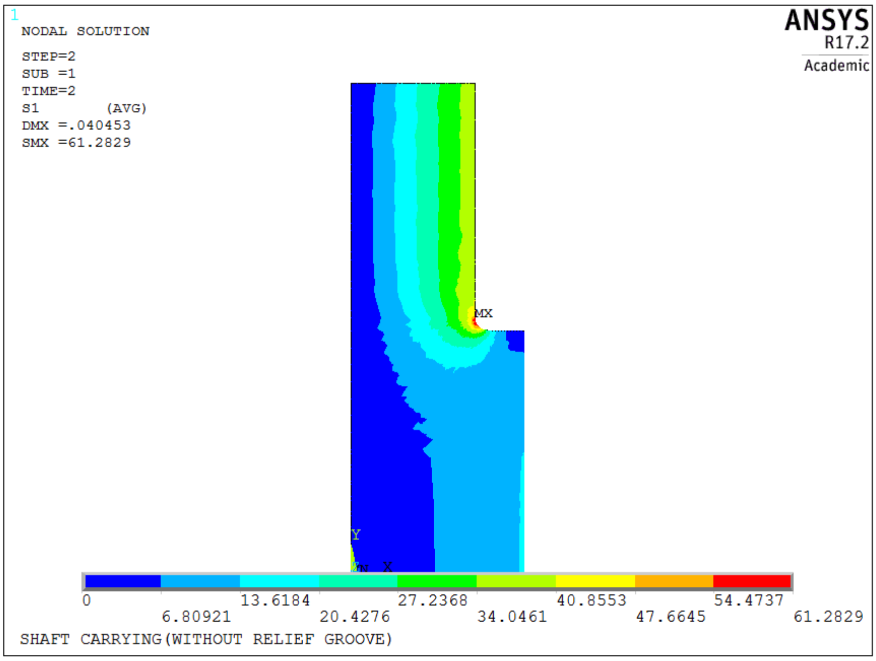
7.2.2 Torque moment

Impose a torque equal to Mt $= 1 * 10^6 Nm$.
The stress concentration factor is defined as: $K_t = \frac{\sigma_{1,max}}{S_{nom}}$, where $\sigma_{1,max}$ = maximum first principal stress in the model

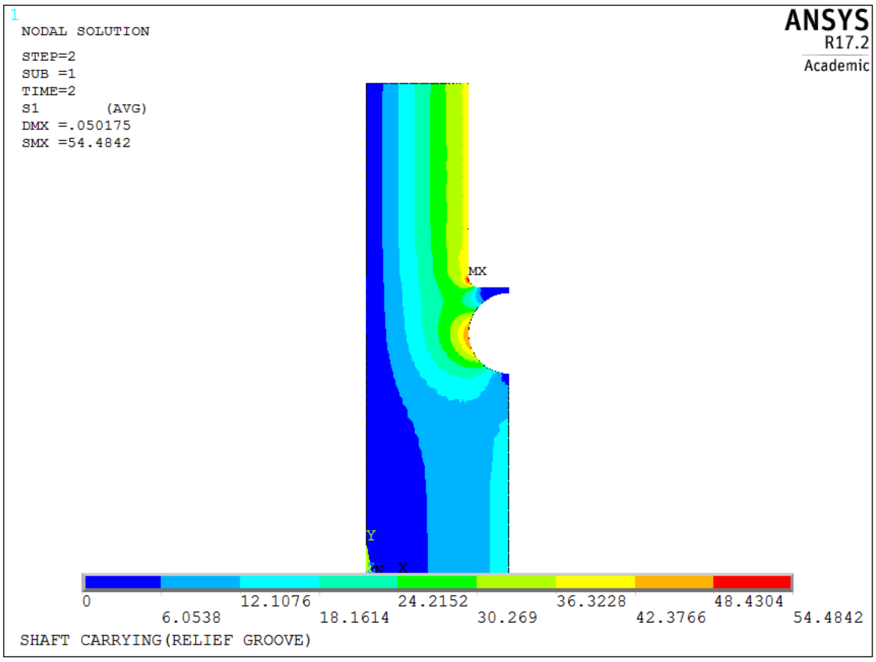
$$S_{nom} = \frac{16Mt}{\pi d^3} = 19,1702\,MPa$$

Ratio of reduction $= \frac{K_{t1} - K_{t2}}{K_{t1}} = 10,97\%$

	Maximim first principal stress in the model $\frac{N}{mm^2}$	stress concentraton factor of analysis
normal shaft	$\sigma_{1,max} = 61,2829$	$K_{t1} = 3,19$
shaft with relief groove	$\sigma_{1,max} = 54,4842$	$K_{t1} = 2,84$



(a) Shaft without relief groove



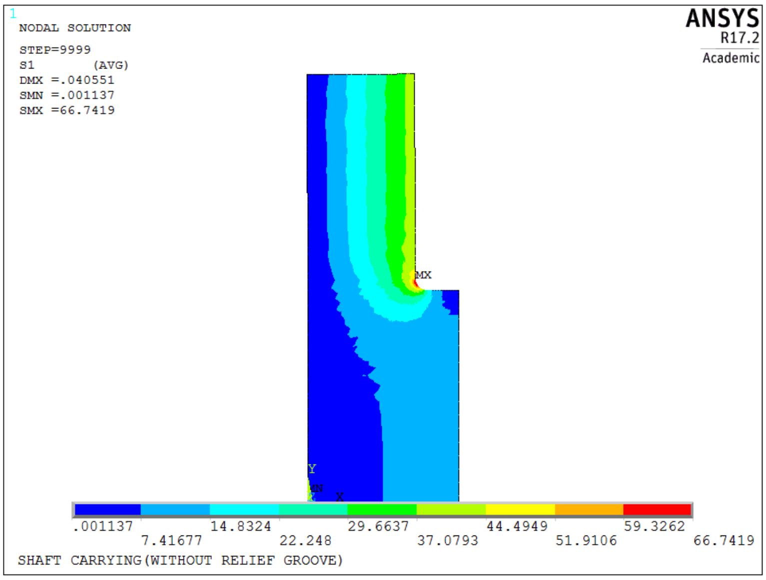
(b) Shaft relief groove

Figure 7.5: Max First Stress Torsion Moment

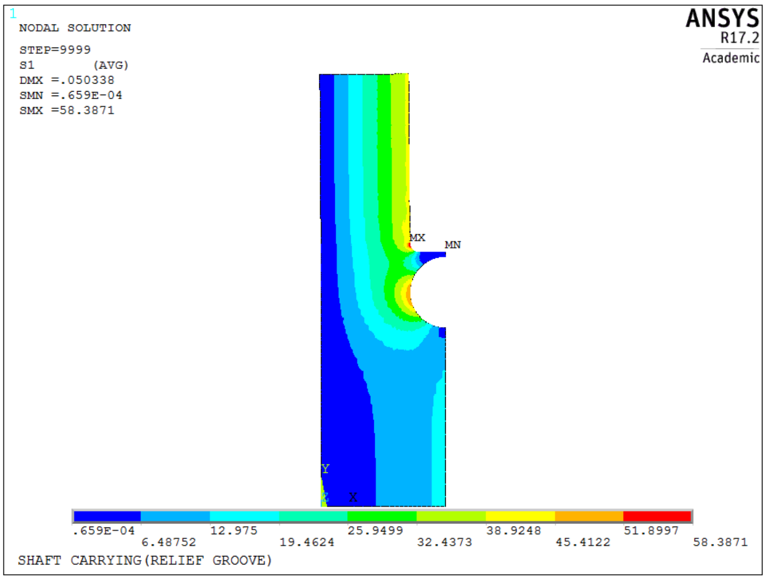
7.2.3 Combined moments

For last analysis impose a combination of previuos bending moment and torque, Mf $= 1 * 10^5 Nm$ and Mt $= 1 * 10^6 Nm$.

	Maximim first principal stress in the model $\frac{N}{mm^2}$
normal shaft	$\sigma_{1,max} = 66,7419$
shaft with relief groove	$\sigma_{1,max} = 58,3871$



(a) Shaft without relief groove



(b) Shaft relief groove

Figure 7.6: Max First Stress Combined Moment

7.3 Conclusion

As we can see from results obtained, the stress concentration factor of the two cases, underlined that: in case of bending moment the presence of relief groove obtained a ratio of reduction of 47,53%. Otherwise in case of torque moment the ratio of reduction obtained 10,71%. Hence the presence of relief groove, decrease especially the bending stress concentration. In conclusion also in case of combined stress in presence of relief groove the stress state is more less than normal shaft.

7.4 Command list

```
!********************
! PROBLEM HOMEWORK 7 *
!********************

FINISH
/CLEAR,START,NEW
/CWD,'E:\Homework\HW7\Part1'
/TITLE,SHAFT CARRYING(WITHOUT RELIEF GROOVE)
/FILNAME,HOMEWORK7part1,1
/PLOPTS,DATE,0
! >>> PARAMETERS MODEL <<<
*SET,Pi,ACOS(-1)
*SET,D,90
*SET,d1,D/(1.4)
*SET,r1,d1/20
*SET,R,(D-d1)/2
*SET,n_div_a,3*4
*SET,n_div_b,2*4
*SET,n_div_c,3*4
*SET,EPS,1E-3

! >>> PROPERTIES MATERIAL <<<
*SET,E_Young,210000
*SET,ni,0.3

! >>>  LOAD <<<
*SET,Mf,  100000
*SET,Mt,1000000

! >>> MOMENT INERTIA <<<
*SET,J,(Pi*d1**4)/64
*SET,sigma_max,Mf/J*(d1/2)

! >>> PRE PROCESSING <<<
/PREP7

ET,1,PLANE25,,,,,,2
MP,EX,1,E_Young
MP,PRXY,1,ni
K,1,0,0,0
K,2,D/2-((D-d1))/2+r1,0,0
K,3,D/2,0,0
K,4,D/2,d1,0
K,5,D/2-((D-d1))/2+r1,d1,0
K,6,D/2-((D-d1))/2,d1,0
K,7,D/2-((D-d1)/2),d1+r1,0
K,8,d1/2,2*d1,0
K,9,0,2*d1,0
K,10,0,d1+r1,0
K,11,0,d1/2,0
K,12,D/2-((D-d1))/2+r1,d1/2,0
K,13,D/2,d1/2,0
SAVE

LSTR,     1,     2
LSTR,     2,     3
LSTR,     3,    13
LSTR,    13,     4
LSTR,     4,     5
LSTR,     7,     8
LSTR,     8,     9
LSTR,     9,    10
LSTR,    10,    11
LSTR,    11,     1
LSTR,    11,    12
LSTR,    10,     7
LSTR,     5,    12
LSTR,    12,     2
LSTR,    12,    13
LARC,7,5,6,-r1
SAVE

AL,1,14,11,10
AL,2,3,15,14
AL,15,4,5,13
AL,9,11,12,13,16
AL,12,8,7,6
SAVE
LESIZE,11,,,n_div_a
```

```
LESIZE,12,,,n_div_b,1/3
LESIZE,13,,,n_div_b,3
LESIZE,9,,,n_div_b
LESIZE,16,,,n_div_b+n_div_a
LCCAT,9,11
LESIZE,10,,,n_div_b
LESIZE,1,,,n_div_a
LESIZE,14,,,n_div_b
LESIZE,2,,,n_div_a
LESIZE,3,,,n_div_b
LESIZE,15,,,n_div_a
LESIZE,4,,,n_div_b,1/3
LESIZE,5,,,n_div_a
LESIZE,6,,,n_div_b
LESIZE,7,,,n_div_b,3
LESIZE,8,,,n_div_b
MSHKEY,1
MSHAPE,0,2D
AMESH,ALL
SAVE

! >>>  WRITE FILE <<<
*GET,Xnode,NODE,506,LOC,X

*CREATE,ansuitmp
*CFOPEN,temp,txt,,
*VWRITE,%506%,Xnode
(F5.0,F20.10)
*DO,i,513,507,-1
 *GET,Xnode_i,NODE,i,LOC,X
 *VWRITE,i,Xnode_i
 (F5.0,F20.10)
*ENDDO
*GET,Xnodelast,NODE,498,LOC,X
*VWRITE,%498%,Xnodelast
(F5.0,F20.10)
*CFCLOS
*END
/INPUT,ansuitmp

! >>> SOLUTION <<<
/SOLU
ANTYPE,STATIC
! >>> CONSTRAINT <<<
NSEL,S,LOC,Y,-EPS,EPS
D,ALL,uy,0
NSEL,R,LOC,X,(D/2)-EPS,(D/2)+EPS
D,ALL,ux,0
ALLSEL,ALL

! >>> BENDING <<<
MODE,1,1
NSEL,S,LOC,y,(2*d1)-EPS,(2*d1)+EPS
SFGRAD,PRES,0,X,0,sigma_max/(d1/2)
SF,ALL,PRES,0
ALLSEL,ALL
LSWRITE,1

! >>> TORQUE <<<
LSCLEAR,ALL
MODE,0
NSEL,S,LOC,Y,-EPS,EPS
D,ALL,uy,0
NSEL,R,LOC,X,(D/2)-EPS,(D/2)+EPS
D,ALL,ux,0
ALLSEL
NSEL,S,LOC,Y,-EPS,EPS
D,ALL,uz,0
ALLSEL

*DIM,force,TABLE,n_div_b+1,,,X
*TREAD,force,torsion,txt
NSEL,S,LOC,y,(2*d1)-EPS,(2*d1)+EPS
F,ALL,FZ,%force%
ALLSEL,ALL
LSWRITE,2
LSSOLVE,1,2
FINISH

! >>> POST-PROCESS <<<
/POST1
SET,1,1,,,,180
PLNSOL,S,1
SET,2,1
PLNSOL,S,1

!superposition of the two loading cases
SET,1,1,,,,180
LCWRITE,1
SET,2,1,,,,180
LCOPER,ADD,1
LCWRITE,1
PLNSOL,S,1
FINISH

!********************
! PROBLEM HOMEWORK 7 *
!********************

FINISH
/CLEAR,START,NEW
/CWD,'E:\Homework\HW7\Part2'
/TITLE,SHAFT CARRYING(RELIEF GROOVE)
/FILNAME,HOMEWORK7Part2,1
! >>> PARAMETERS MODEL <<<
*SET,Pi,ACOS(-1)
*SET,i,4
*SET,D,90
*SET,d1,D/(1.4)
*SET,r1,d1/20
*SET,R,(D-d1)/2
*SET,n_div_a,3*4
*SET,n_div_b,2*4
```

```
*SET, n_div_c ,3*4
*SET, EPS, 1E-3
*SET, x , 1.5

! >>> PROPERTIES Material <<<
*SET, E_Young ,210000
*SET, ni ,0.3

! >>>  LOAD <<<
*SET, Mf,  100000
*SET, Mt, 1000000

! >>> MOMENT INERTIA <<<
*SET, J , ( Pi*d1**4)/64
*SET, sigma_max , Mf/J*(d1/2)

! >>> PRE PROCESSING <<<
/PREP7

ET, 1 , PLANE25, , , , , , 2
MP, EX, 1 , E_Young
MP, PRXY, 1 , ni

K, 1 , 0 , 0 , 0
K, 2 ,D/2-((D-d1))/2+r1-10,0,0
K, 3 ,D/2-((D-d1))/2+r1-10,d1-10,0
K, 4 ,D/2 ,0 ,0
K, 5 ,D/2 ,d1 ,0
K, 6 ,D/2-R, d1+R, 0
K, 7 ,D/2-((D-d1))/2+r1-10,d1+R,0
K, 8 ,D/2 ,d1+R, 0
K, 9 ,D/2 ,d1+2*R, 0
k, 10 ,D/2 ,d1+2*R+x/2,0
K, 11 ,D/2 ,( d1+2*R)+x ,0
K, 12 ,D/2-((D-d1))/2+r1 ,( d1+2*R)+x ,0
k, 13 ,D/2-((D-d1))/2+r1 ,( d1+2*R)+x/2,0
K, 14 , d1/2 ,( d1+2*R)+x ,0
K, 15 , d1/2 ,(( d1+2*R)+r1)+x ,0
k, 16 ,D/2-((D-d1))/2+r1-10 ,((d1+2*R)+r1)+x ,0
K, 17 ,D/2-((D-d1))/2+r1-10 ,2*(d1+R)+x ,0
k, 18 ,D/2-((D-d1))/2+r1-10,d1+2*R+x/2,0
K, 19 , d1/2 ,2*( d1+R)+x ,0
K, 20 ,0 ,2*( d1+R)+x ,0
K, 21 ,0 ,(( d1+2*R)+r1)+x ,0
K, 22 ,0 , d1+2*R+x/2,0
K, 23 ,0 , d1+R, 0
K, 24 ,0 , d1-10,0
K, 25 ,D/2 , d1-10,0
SAVE

LSTR,      1,      2
LSTR,      2,      4
LSTR,      4,     25
LSTR,     25,      5
LSTR,      2,      3
LSTR,      1,     24
LSTR,     24,      3
LSTR,      3,     25
LSTR,     24,     23
LSTR,     23,      7
LSTR,      7,      3
LSTR,      7,      6
LSTR,     22,     18
LSTR,     18,     16
LSTR,     16,     21
LSTR,     21,     22
LSTR,     22,     23
LSTR,     18,      7
LSTR,     21,     20
LSTR,     20,     17
LSTR,     17,     19
LSTR,     19,     15
LSTR,     16,     17
LSTR,     16,     15
LARC, 5 , 6 , 8 ,R,
LARC, 6 , 9 , 8 ,R,
LSTR,      9,     10
LSTR,     10,     11
LSTR,     11,     12
LSTR,     13,     10
LSTR,     13,     12
LSTR,     18,     13
LSTR,     18,      7
LARC, 12 , 15 , 14 , -r1 ,
SAVE

AL, 1 , 5 , 7 , 6
AL, 2 , 3 , 8 , 5
AL, 8 , 4 , 25 , 12 , 11
AL, 12 , 26 , 27 , 30 , 32 , 18
AL, 30 , 28 , 29 , 31
AL, 32 , 31 , 33 , 24 , 14
AL, 7 , 11 , 10 , 9
AL, 10 , 18 , 13 , 17
AL, 13 , 14 , 15 , 16
AL, 24 , 22 , 21 , 23
AL, 15 , 23 , 20 , 19
SAVE

LESIZE, 8 , , , n_div_a
LESIZE, 25 , , , n_div_b+n_div_a
LESIZE, 4 , , , n_div_b ,1/3
LESIZE, 11 , , , n_div_b
LESIZE, 12 , , , n_div_b ,1/3
LCCAT, 11 ,8
SAVE

LESIZE, 18 , , , n_div_b+n_div_c
LESIZE, 32 , , , n_div_a
LESIZE, 26 , , , n_div_b+n_div_c+2*n_div_a
LESIZE, 27 , , , n_div_b ,3
LESIZE, 30 , , , n_div_a
```

```
FLST,2,3,4,ORDE,3
FITEM,2,18
FITEM,2,30
FITEM,2,32
LCCAT,P51X
SAVE

LESIZE,33,,,n_div_b+n_div_a
LESIZE,31,,,n_div_b,1/3
LESIZE,14,,,n_div_b
LESIZE,24,,,n_div_b,1/3
LCCAT,14,32
SAVE

LESIZE,29,,,n_div_a
LESIZE,28,,,n_div_b,1/3
SAVE

LESIZE,22,,,n_div_a,3
LESIZE,21,,,n_div_b,1/3
LESIZE,23,,,n_div_a,1/3
SAVE

LESIZE,20,,,n_div_b
LESIZE,19,,,n_div_a,1/3
LESIZE,15,,,n_div_b
SAVE

LESIZE,10,,,n_div_b
LESIZE,17,,,n_div_b+n_div_c
LESIZE,9,,,n_div_b
SAVE

LESIZE,1,,,n_div_b
LESIZE,6,,,n_div_a
LESIZE,7,,,n_div_b
LESIZE,5,,,n_div_a
SAVE

LESIZE,2,,,n_div_a
LESIZE,3,,,n_div_a
SAVE
MSHKEY,1
MSHAPE,0,2D
AMESH,ALL
SAVE

! >>>  WRITE FILE <<<
*GET,Xnode,NODE,4475,LOC,X
*CREATE,ansuitmp
*CFOPEN,temp,txt,,
*VWRITE,%4475%,Xnode
(F5.0,F20.10)
*DO,i,4496,4508,1
  *GET,Xnode_i,NODE,i,LOC,X
  *VWRITE,i,Xnode_i
  (F5.0,F20.10)
*ENDDO
*DO,i,4181,4194,1
  *GET,Xnode_i,NODE,i,LOC,X
  *VWRITE,i,Xnode_i
  (F5.0,F20.10)
*ENDDO
*GET,Xnodelast,NODE,4160,LOC,X
*VWRITE,%4160%,Xnodelast
(F5.0,F20.10)
*CFCLOS
*END
/INPUT,ansuitmp

! >>> SOLUTION <<<
/SOLU
! >>> bending <<<
ANTYPE,STATIC
NSEL,S,LOC,Y,-EPS,EPS
D,ALL,uy,0
NSEL,R,LOC,X,(D/2)-EPS,(D/2)+EPS
D,ALL,ux,0
ALLSEL,ALL

MODE,1,1
NSEL,S,LOC,y,2*(d1+R)+x-EPS,2*(d1+R)+x+EPS
SFGRAD,PRES,0,x,0,sigma_max/(d1/2)
SF,ALL,PRES,0
ALLSEL,ALL
LSWRITE,1
/REPLOT

! >>> TORQUE <<<
LSCLEAR,ALL
MODE,0
NSEL,S,LOC,Y,-EPS,EPS
D,ALL,uy,0
NSEL,R,LOC,X,(D/2)-EPS,(D/2)+EPS
D,ALL,ux,0
ALLSEL
NSEL,S,LOC,Y,-EPS,EPS
D,ALL,uz,0
ALLSEL

*DIM,force,TABLE,2*n_div_b+1,,,X
*TREAD,force,torsion,txt
NSEL,S,LOC,y,2*(d1+R)+x-EPS,2*(d1+R)+x+EPS
F,ALL,FZ,%FORCE%
ALLSEL,ALL
LSWRITE,2
LSSOLVE,1,2
FINISH

! >>> POST PROCESSING <<<
/POST1
```

```
SET,1,1,,,,180
PLNSOL,S,1
SET,2,1
PLNSOL,S,1

!SUPERPOSITION OF THE TWO LOADING CASES
SET,1,1,,,,180
LCWRITE,1
SET,2,1,,,,180
LCOPER,ADD,1
LCWRITE,1
PLNSOL,S,1
FINISH
```

Bibliography

[1] O. Zienkiewicz, R. Taylor, and J. Zhu, *The Finite Element Method: Its Basis and Fundamentals*. Elsevier Science, 2013. [Online]. Available: https://books.google.it/books?id=7UL5Ls9hOF8C

[2] R. Juvinall and K. Marshek, *Fundamentals of Machine Component Design, 5th Edition*. John Wiley & Sons, 2011. [Online]. Available: https://books.google.it/books?id=0vUbAAAAQBAJ

[3] J. Shigley, R. Budynas, J. Nisbett, D. Amodio, and G. Santucci, *Progetto e costruzione di macchine*, ser. Collana di istruzione scientifica. McGraw-Hill Education, 2013. [Online]. Available: https://books.google.it/books?id=mayongEACAAJ