ME304 Finite Element Analysis, Project 2: Design a Bracket.
Ken Lulay, December 3, 2019

The following document describes results of the bracket design project for ME304. See Memo from K. Lulay, November 22, 2019.

Table 1 – bracket design criteria given in the project memo

#	Criteria	Description	Priority
1	Bracket size	300mmX200mmX10mm maximum	Essential
2	Attachment	Welded to a steel column along one vertical edge.	Essential
3	Load	1kN, straight downward	Essential
4	Load location	Applied along the top of the bracket uniformly distributed over an area from 250mm to 300mm from the wall	Essential
5	Material	Steel (no specific alloy)	Important
6	Stress	Minimize the largest first principal stress (σ_1)	Important

The following is work done to validate the baseline FEA model and presents a solution to meet the criteria in Table 1.

Purpose: Validate that the "baseline" (200mmX300mm) FEA model of 1000N distributed load applied at the end:

a) Demonstrate that σ_x in the FEA model is equal to the bending stress equation (Mc/I).
b) Show that the distributed load is 1000N total

Part a, Solution: Bending stress at top middle of bracket = Mc/I = 2.25 (MPa) based on:

M=(L/2)*F = 150*1000 = 150,000 (N-mm) [bending moment mid-length]

c = h/2 = 200/2 = 100 (mm)

I = $bh^3/12 = 10*(200)^3/12$ = 6.67E6 (mm^4)

From PATH through the mid-length, bending stress at top mid-length is about 2.3 (MPa). FEA and hand-calculation (Mc/I) are very similar, validating the FEA model. QED

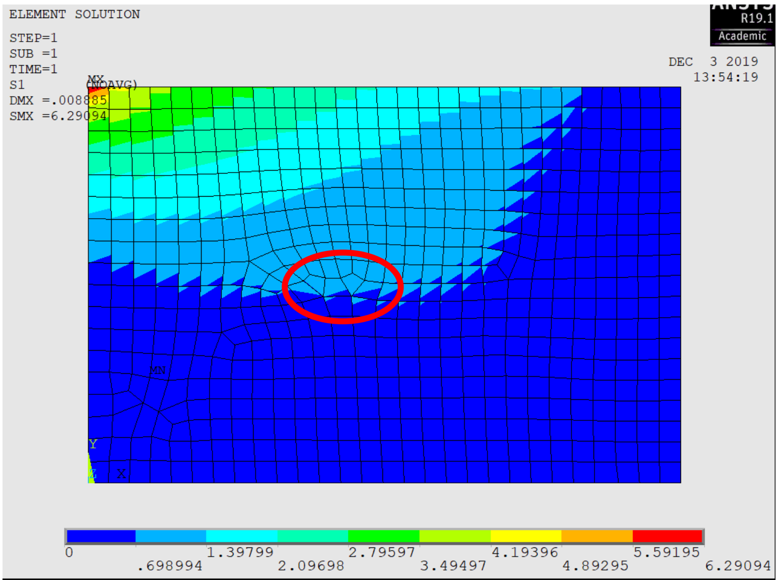


Figure 1 – baseline bracket, SMRTSIZE,1. Circled region indicates distorted elements (very non-90 degree angles). These are well away from the high stress regions and therefore should have little effect on the maximum first principal stress results.

Part b, Solution:

Baseline bracket with point load (plot not shown here), using PATH shows σ_1= 6.77MPa at upper left corner.

Baseline bracket with distributed load. PATH shows σ_1= 6.29MPa at upper left corner (similar to point load) QED

Also, checked GUI: General Postproc>List Results>Reaction Solu> total reaction force in y-direction is 1000. Therefore, the total distributed load is 1000 (N). Double QED.

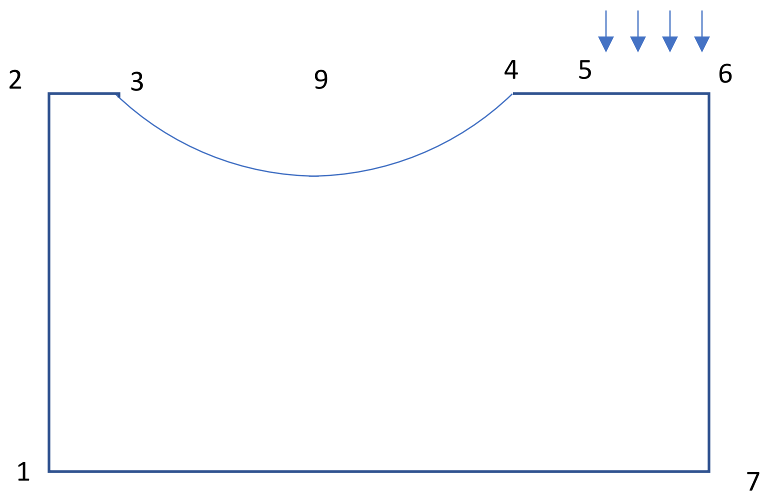
CONCLUSION: the FEA model with a distributed load of 1000N total provides reasonable solution.

Purpose: revise the baseline model (200mmX300mm) to reduce the maximum first principal stress (σ_1) to meet criterion 6 in Table 1.

Approach: The greatest first principal stress in the baseline model exists at the upper corner. This is due to this location having the highest bending stress (greatest bending moment) and the constraint at the left edge acts as a stress concentration (it's a sharp corner). To reduce the stress in this location, a smooth curvature was created at the top of the bracket. This directs some of the stress away from the corner reducing the stress in that location.

Model outline showing key points, shown below.

- The load is distributed between KP5 and KP6.
- KP9 is used to define side of curvature for the LARC command.
- The left edge (all nodes between KP1 and KP2) are constrained in all DOF (UX, UY).



Various diameters of the arc and locations were evaluated – a total of 19 different models are discussed below.

First design matrix (3 variables, 2 levels each). Note that the radius affected the results more significantly than the other two variables.

#	RAD	KP3	KP4*	Sig 1
1	300	15	10	4.522
2	200	15	10	5.056
3	**300**	**5**	**10**	**4.446**
4	200	5	10	5.327
5	**300**	**15**	**3**	**4.358**
6	200	15	3	5.148
7	300	5	3	4.540
8	200	5	3	5.433

*KP4=KP5-10 for designs 1-4, KP4=KP5-3 for designs 5-8

Design #5 produced best results for SMRTSIZE,3 – but the results for SMRTSIZE,1 were significantly worse (5.538). Therefore, design #3 (which produced the next best results) was studied further by refining values. Since the radius (RAD) has the most significant effect on principal stress, three runs were conducted to evaluate how close RAD=300 is to the best result. The other parameters were: KP3=5 and KP4-KP5-10.

RAD	KP3	KP4*	Sig 1
250	5	10	4.75
275	5	10	4.57
325	5	10	4.84

*KP4=KP5-10

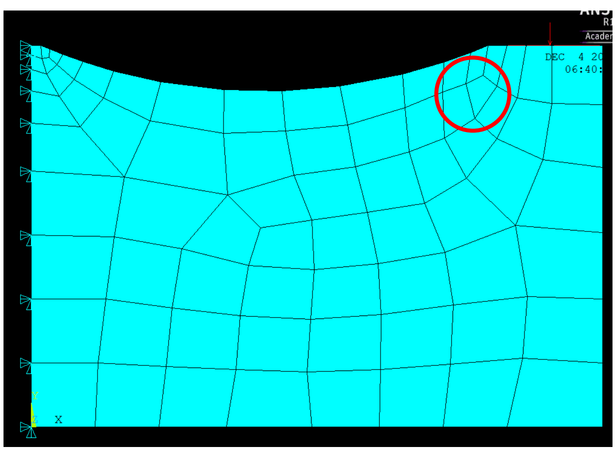
None of the three radii (RAD) produced better results than RAD=300. Therefore, it appears that RAD=300 is near the ideal. An additional set of runs was conducted to refine the conclusion – shown in the following table.

#	RAD	KP3	KP4*	Sig 1
21	310	7	7	4.530
22	290	7	7	4.482
23	310	3	7	4.567
24	290	3	7	4.636
25	310	7	13	4.618
26	290	7	13	4.475
27	310	3	13	4.672
28	290	3	13	4.512

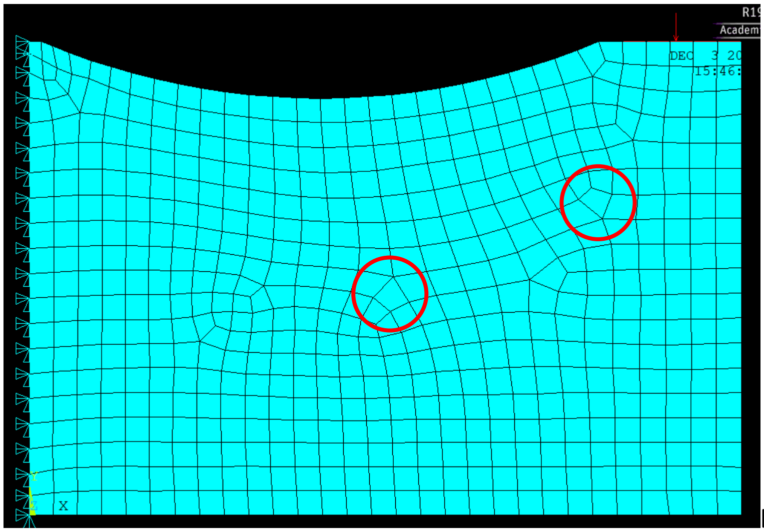
*KP4=KP5-the value in the table (KP4=KP5-7 for designs 21-24, KP4=KP5-13 for designs 25-28)

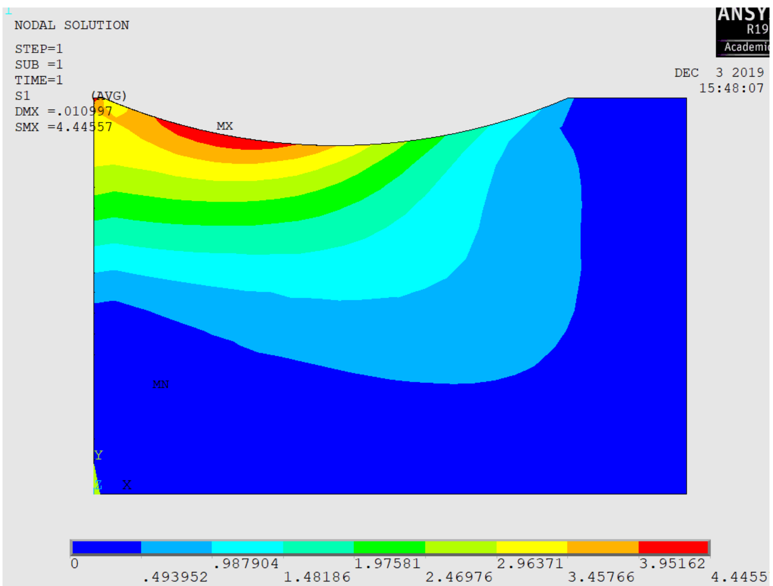
CONCUSION: the best results were roduced with RAD=300, KP3=5, and KP4=KP5-10. See APDL script below for details.

The following are the results using RAD=300, KP3=5, and KP4=KP5-10

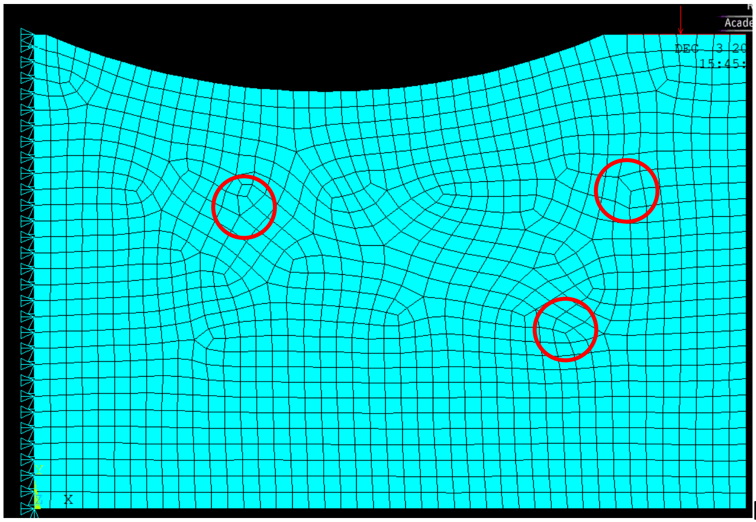


Elements and constraints using SMRTSIZE,6 (not required in the memo). Included as part of the convergence study (Table 1). Circle indicates very non-90 degree element.

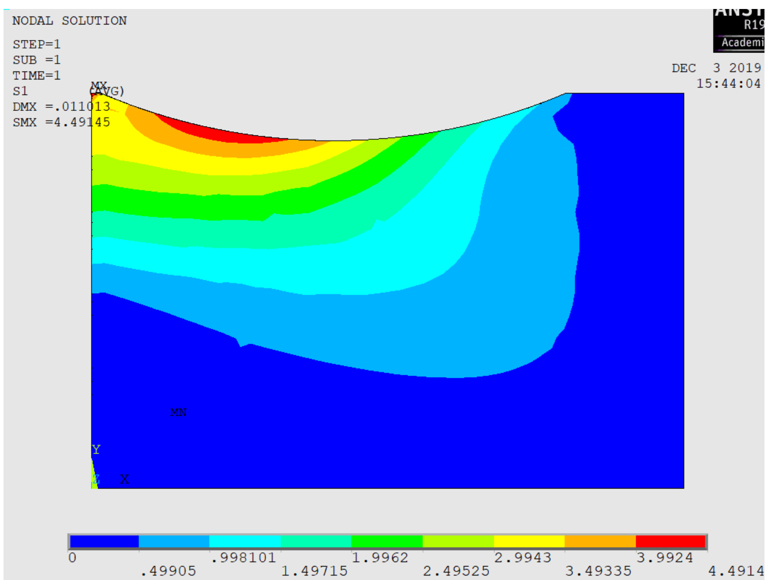

Elements and constraints using SMRTSIZE,3. Circles indicate highly non-90 degree elements.



SMRTSIZE,3; First principal stress plot (4.44557 max)



Elements and constraints using SMRTSIZE,1. Circles indicate highly non-90 degree elements.



SMRTSIZE,1; First principal stress plot (4.49145 max)

Table 1 – results from different meshes (convergence study) and designs. The mesh size had little effect on the final design model. Note, SMRTSIZE,6 was not required in the memo rules, but was included to help validate the model. The final design shows at least 29% improvement over the baseline.

	Final Design, σ_1 (MPa)	Baseline, σ_1 (MPa)	Difference
SMRTSIZE,6	4.53412		
SMRTSIZE,3	4.44557	6.29094	29% improved
SMRTSIZE,1	4.49145	6.88848	35% improved
Difference	2%	8.7%	

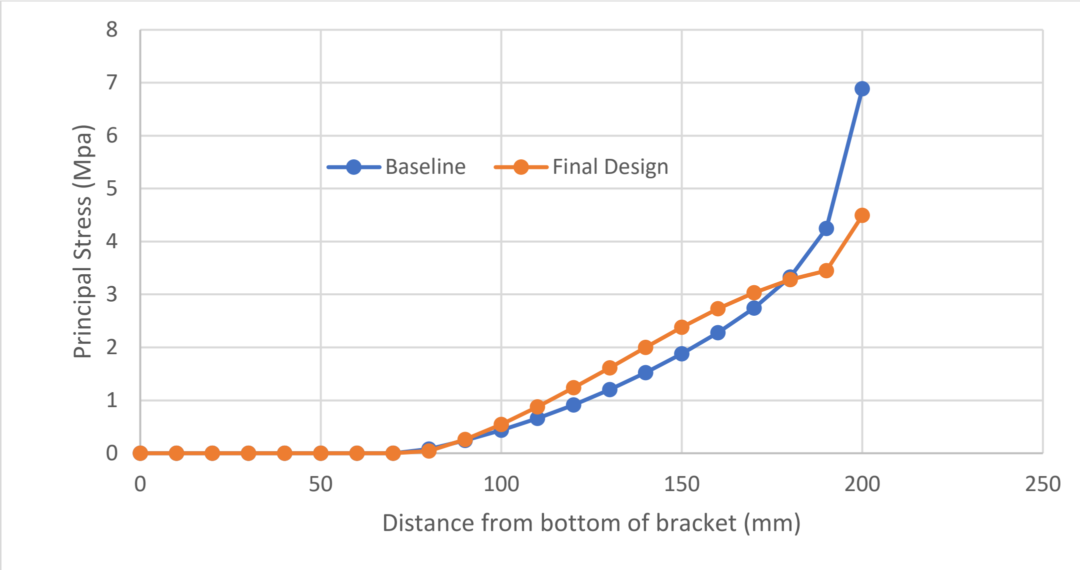


Figure 2 – first principal stress at the wall for the baseline and final design. This plot demonstrates that the curvature helps direct stress away from the top corner (200mm location). The final design has higher stress than the baseline from about 100mm to 175mm from the bottom, but has lower maximum stress than the baseline. Based on "weakest link" design philosophy (failure will occur at the point of highest stress), the final design should be able to carry about a 30% greater load than the baseline design.

ELEMENT WARNINGS: Element shape warnings/errors were checked on all models – no errors were identified with any mesh. On a few models (especially the baseline), "triangular" elements were included in the mesh. However, these elements were well away from high stress regions and should have little effect on the maximum first principal stresses.

CONCLUSION: Based on trying various radii and locations of the arc, the final design (APDL script provided below) produced the lowest maximum first principal stresses of 4.49MPa (based on SMRTSIZE,1 – the poorer result of "required" mesh sizes).

!APDL SCRIPT:

```
!PLANE182: 4-nodes, 2D element, 2DOF/NODE (UX,UY)
!Steel, E=210E3, Poisson's ratio 0.3
!Units are mm and N (therefore, E and stress are N/mm^2 = MPa)
!ME304 Project: Cantilever bracket
!This model is based on rectangular bracket 300mmX200mm

FINISH     !Finishes any previous activity
/CLEAR     !Clears any previous activity
/BATCH     !Works in "batch" mode
/PREP7

!Define geometry and load parameters.

THICKNESS=10
LENGTH=300
HEIGHT=200
LOADLENGTH=50
RAD=300
!Define locations of key points used to create the arc:
KP3=5
KP5=LENGTH-LOADLENGTH
KP4=KP5-10

!BEST VALUES SO FAR (11/22/19) KP3=5,KP4=KP5-10,RAD=300
!TRIED SAME VALUES, BUT PLACED LOAD AT BOTTOM RIGHT. SLIGHTLY WORSE
     RESULT - plus it violates the rules.

PRESSAREA=LOADLENGTH*THICKNESS   !The area that the applied load
     ("pressure" acts upon)
APPLFORCE=1000  !Total force applied
PRSR=APPLFORCE/PRESSAREA

ET,1,PLANE182
MP,EX,1,210E3
MP,PRXY,1,0.3
KEYOPT,1,3,3    !Use plane stress (through the thickness)
R,1,THICKNESS   !Use "THICKNESS" as the through thickness dimension

!Define locations of key points.
K,1,0,0,0
K,2,0,HEIGHT,0
K,3,KP3,HEIGHT,0
K,4,KP4,HEIGHT,0
K,5,KP5,HEIGHT,0
K,6,LENGTH,HEIGHT,0
K,7,LENGTH,0,0
K,9,LENGTH/2,1.2*HEIGHT !locates side of curvature for LARC
```

```
L,1,2 !Create a line connecting Key Points
L,2,3
L,4,5
L,5,6
L,6,7
L,7,1

LARC,3,4,9,RAD  !Create an arc from KP3 to KP4, with center of
     curvature on side of KP9, radius RAD

/PNUM,AREA,1
AL,ALL
SMRTSIZE,1
AMESH,ALL

FINISH
/SOLU

!Constrain in x directions the nodes at left edge:
NSEL,S,LOC,X,0        !select all nodes along X=0
D,ALL,ALL,0           !Prevents ALL displacement of selected nodes

!Apply distributed force along line (SFL)
SFL,4,PRES,PRSR !Line 4, apply a PRES(pressure)of magnitude PRSR

ALLSEL     !Since we have used NSEL to select specific nodes, we now
     need ALLSEL to select all of the nodes for solution
SOLVE
FINISH
/POST1
/ESHAPE,1  !Display element shapes using section data
/RGB,INDEX, 0, 0, 0,15     !set text color to black
/COLOR,WBAK,14  !Set background color to light grey
/DSCALE,ALL,1   !Plot using true scale
!/VIEW,1,1,1,1
FINISH          !Finish and exit the post-processor
SAVE       !Save the data base
```