DATE: November 22, 2019
TO: Members of ME 304 Section A and B
FROM: Kenneth E. Lulay, Ph.D., PE *KEL*
REF: Bracket design project description (Project 2)

Congratulations, you did such a good job on the turbine blade project that Rolls-Royce has decided to extend your employment for at least a little longer! I understand that you have been taking advantage of professional development opportunities offered here at RR and you have learned to analyze stresses using ANSYS. That is an essential skill for this next project.

I encourage you to work on teams of 2 to 4 engineers per team, but if you choose to go solo without a team, that is fine. Each engineer will be responsible for design and analysis of one load carrying bracket. The details are provided below.

Bracket details

Criteria for the bracket is as follows:

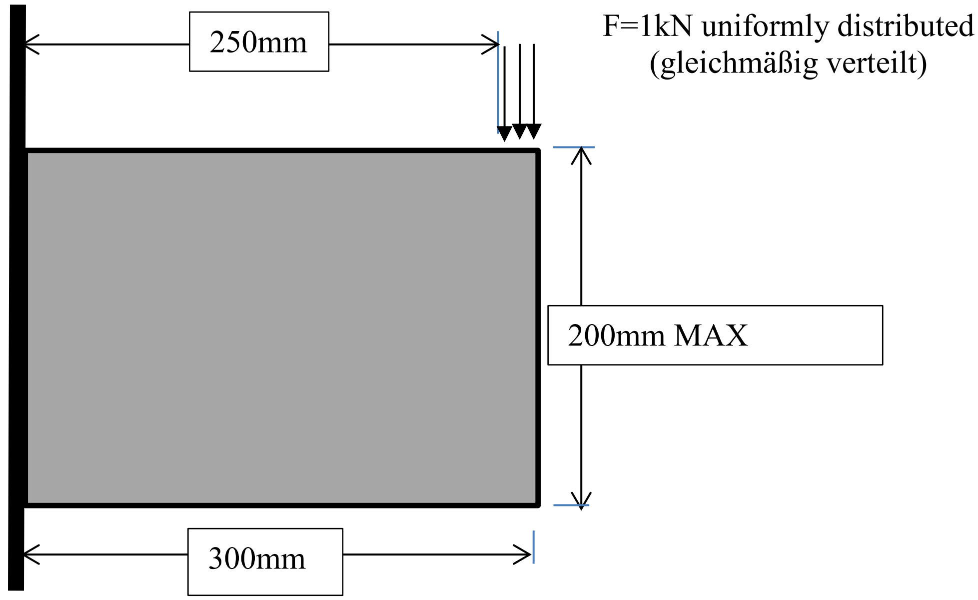
Table 1 – bracket design criteria

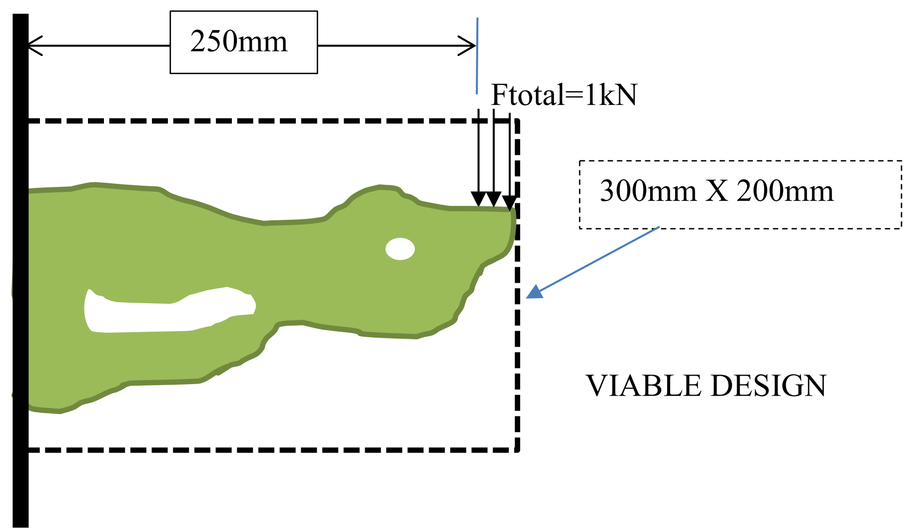
#	Criteria	Description	Priority
1	Bracket size	300mmX200mmX10mm maximum	Essential
2	Attachment	Welded to a steel column along one vertical edge.	Essential
3	Load	1kN, straight downward	Essential
4	Load location	Applied along the top of the bracket uniformly distributed (gleichmäßig verteilt)over an area 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 goal for each team is to have one design that is better than all others in the class.

The steel plate is ductile, so why is minimizing the first principal stress the primary design criterion?

Answer: this bracket is being designed to have a long fatigue life. The first principal stress is the primary stress of concern for fatigue in all isotropic materials so it needs to be minimized. Fatigue will be discussed in Machine Design.





FEA Model requirements to be eligible for extra credit:

Element type: PLANE182

The weld at the wall shall be modeled by constraining all nodes at the wall in all directions

SMRTSIZE of both 1 and 3 (2 different runs) to evaluate element size effects (convergence study).

No serious warnings should occur (such as "triangle" elements) – if warnings only occur for one of the SMRTSIZE but not both, and the one with the warning has worse results (higher stress), that is acceptable and the team will still be eligible for extra credit.

Project Grading:
Team work: 25% (same grade applied to all engineers on the team)
Individual Design: 75% (individual grades assigned to individual design work)
Bonus: The best designs from each team will be compared and extra credit will be given to all team members on the team that adequately demonstrates the minimum first principal stress.

- 15% extra credit for the "winning" team – 20% extra credit if the winning team also beats Prof. Lulay, PhD, PE
- 10% extra credit for the second-place team
- 5% extra credit for the third-place team.

Winning models will be subject to Dr. Lulay' scrutiny to make sure there were no rules violations.

Memo requirements: Clear, concise, effective communication including:
Header/title information (everyone should sign or initial, the title should be meaningful, etc.)
First paragraph: Brief intro/purpose/overview of the project and memo
Design requirements/criteria
Brief discussion as to why the FEA results seem reasonable. Discuss any "problematic" FEA elements, etc.
Brief description and small sketch of each design. Each design should have a brief but informative title
Summary of the results (largest first principal stress in each of the designs) – think about how to best present this data (clear, concise, easy to find, easy to understand)
Conclusion: which is the best design and what is the maximum first principal stress in it?
List the attachments (the design work) at the end of the memo, and include a brief but descriptive title for each. Each set of design work should be its own attachment.

Design Work
One per team: validate the basic FEA model by the following (**this needs to be discussed briefly in the memo)**

a. APDL script for a rectangular bracket is available on the course web page. It has a distributed load – but you will need to figure out certain details. One engineer should modify this script replacing the distributed load with a point load. Use this to confirm the distributed load has the correct magnitude (similar stresses). For teams, this can be the design for one engineer.
b. Hand calculations of the rectangular beam in part (a) to estimate the expected stresses for FEA. This must follow the standard problem-solving format.
c. For parts (a) and (b) above, do not compare stresses near boundary conditions. The "path" command would be useful for this. How to use "path" commands is give at the end of this document. Include "path" data in the report.

Each student is responsible for one unique design (substantially different than each other's):

a. Dimensioned drawing (CAD or neatly hand done with straight edge) of your bracket design.
b. Sketch of the bracket with the following identified on the sketch:
    i. Coordinates for each key point is shown at each key point
    ii. Each line is numbered
    iii. Constraints are communicated (with symbols and/or words)
    iv. Loads are shown
c. APDL Script, adequately commented

d. ANSYS printout showing elements, constraints, loads
e. Use the "Check Element Shape" function to help identify poorly shaped elements (high aspect ratio, non 90 degree angles.) Include a printout (screen shot) if there are problematic elements.
f. ANSYS nodal stress printout showing first principal stresses (σ_1) using both SMRTSIZE,3 and using SMRTSIZE,1. The maximum value from both plots (even they are near a boundary condition) is the appropriate value to report for criterion 6 (Table 1).

Grading

A: All required elements professionally done. Communication is clear and concise
B: Pretty darn good, but less than A
C: Adequate, but less than B
D: Less than adequate
F: Less than less than adequate

<u>Some addition information that may be useful:</u>

How to check for element shape warnings:

- After running the model, select from the GUI left side menu:
  - >General Postproc > Check Elem Shape > Plot Warning/Error Elements
- You may need to zoom in to see elements flagged as problematic

How to create a "path" and determine stresses along it:

- When creating the geometry, consider placing key points at the ends of the paths you wish to analyze. To create a path, you need to identify specific nodes, and ANSYS usually places node #5 at key point #5, etc. – *usually* – you should always confirm it when you create the path. Type NPLOT,1 into the command line and zoom in to see specific node numbers. To create a path:
  - >General Postproc > Path Operations > Define Paths > By nodes…enter the node numbers for both ends of the path (hit enter after each), select OK, give the path a name
  - >Map onto path…select desired stress and **<u>enter a *Desired Label*</u>** in the dialog box
  - >Plot path item > plot onto graph>*Desired Label* name you gave it.
  - To cut and paste actual values: >Plot path item > list path items>*Desired Label*

APDL Commands that create circular surfaces:

CIRCLE (see HW 8 "Hole in plate script")
LARC (see HW 8 "Symmetry – hole in plate script")
LFILLT (see "FILLT command example" on course page).