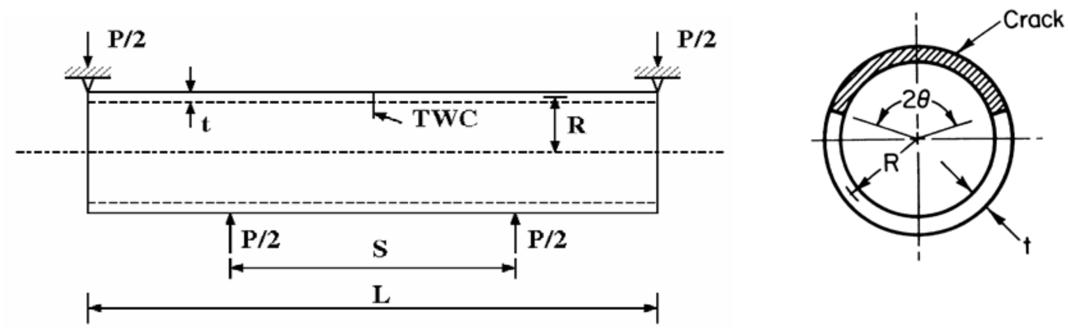
THE UNIVERSITY OF IOWA
Department of Mechanical Engineering

Fracture Mechanics	**Computer Project #4**	**Assigned: April 03, 2020**
ME:5159	**Total Points: 20**	**Due: April 13, 2020**

Consider a pipe with mean radius, *R*, wall thickness, *t*, and a circumferential through-wall crack of total angle, 2θ. The pipe is subjected to a pure bending moment, *M*, via four-point bending loads and is shown in the figure below.



The pipe and crack geometric parameters, material properties, and load are given below.

Pipe and Crack Geometry:

Mean Radius, *R*	=	50.8 mm
Wall Thickness, *t*	=	5.08 mm
Normalized Crack Angle, θ/π	=	0.125
Inner Span, *S*	=	600 mm
Outer Span, *L*	=	1500 mm

Material Properties:

The stress-strain curve (Ramberg-Osgood Model): $$\frac{\varepsilon}{\varepsilon_0} = \frac{\sigma}{\sigma_0} + \alpha\left(\frac{\sigma}{\sigma_0}\right)^n$$

Elastic Modulus, *E*	=	207×10^3 N/mm^2 or 207 GPa
Poisson's Ratio, ν	=	0.3
Reference Stress, σ_0	=	344.8 N/mm^2 or 344.8 MPa
Ramberg-Osgood exponent, *n*	=	5
Ramberg-Osgood coefficient, α	=	1

Load:

Total load, *P*, varies from 0 to 126,878 N. Correspondingly, the bending moment, *M*, varies from 0 to 28.548 kN-m.

Using ABAQUS/CAE at CSS/ICAEN,

1. Start ABAQUS/CAE and create a solid model of a quarter pipe from scratch. Generate various subregions of the quarter pipe, including crack-tip regions, to facilitate adequately refined finite-element mesh. Note, due to symmetry, only a quarter of the pipe needs to be modeled. Save your solid model as a CAE file (*e.g.*, `cp4.cae`) in a local directory (e.g., `c:\temp`) of your computer.

2. Start ABAQUS/CAE and import `cp4.cae`, if required. Pre-process the CAE file by defining material properties and enforcing load and boundary conditions. Create a finite-element mesh of the pipe. Use 20-noded hexahedral elements, except for the 20-noded wedge-shaped elements representing the crack-tip elements. Attach snapshot(s) showing mesh geometry, boundary conditions, and load.

3. Conduct a nonlinear, three-dimensional, elastic-plastic (deformation theory of plasticity) finite-element analysis to compute the J-integral for the range of total applied load varying from $P = 0$ ($M = 0$) to $P = 126{,}878$ N ($M = 28.548$ kN-m). Use a step increment of 0.1 or lower. Post-process your finite element results and hence create/attach a snapshot of the deformed pipe geometry superimposed with the undeformed pipe geometry.

4. Calculate J at several contours and according to

$$J = \frac{J_1 + 4J_2 + J_3}{6},$$

   where J_1, J_2, and J_3 are J-integral values at the inside, middle, and outside crack-tip nodes, respectively. The weighted-average J is commonly employed in pipe fracture evaluations. Develop plots of J (kJ/m^2) vs. M (kN-m). Evaluate sensitivity of J to at least three distinct farthest contours (*e.g.*, 6th, 7th, and 8th contours if opted for 8 of them).

5. To examine the GE/EPRI method, which is extensively used in the industry, compute J for the above problem by the GE/EPRI method. Compare the GE/EPRI results of J with your own ABAQUS/CAE nonlinear finite element results. For the GE/EPRI influence functions, use $F(\theta/\pi, R/t) = 1.219$ and $h_1(\theta/\pi, n, R/t) = 6.311$.

Show all work and enclose relevant plots and/or tables to justify your answers.