CEE 570

Finite Element Method

Term Project

Spring 2019

Blind Prediction of the Critical Load Applied to a Cracked Beam

OBJECTIVES:

- To get familiar with the commercial FEM software Abaqus;
- Study the performance of finite elements available in Abaqus;
- Get introduced to the application of the FEM to fracture problems: Determine the critical load of a simply supported beam with a crack at mid span.

Submitted By:

Vardhan Dongre,

Graduate Student,

Department of Civil & Environmental Engineering

University of Illinois at Urbana Champaign

vdongre2@illinois.edu

CONTENTS

1. ABSTRACT

The three-point bending test is a classical experiment in mechanics that is usually used to measure the Young's modulus of a material of a beam. The beam, of length L, rests on two roller supports and is subject to a concentrated load P at its center. In our case we consider a pre-cracked beam with a crack at its mid-span that propagates when the load applied to be beam reaches a critical value **Pcr.** In this work we predict this critical load using Finite Element Modelling of the structure (Fig. 1) in ABAQUS. The FE Analysis is performed using 4 different element types, viz. QUAD4, QUAD4R, QUAD8, QUAD8R to perform a comparative study of the performance of each element. The predicted value of Critical Load is then compared with experimentally obtained value of Critical Load for the given structure.

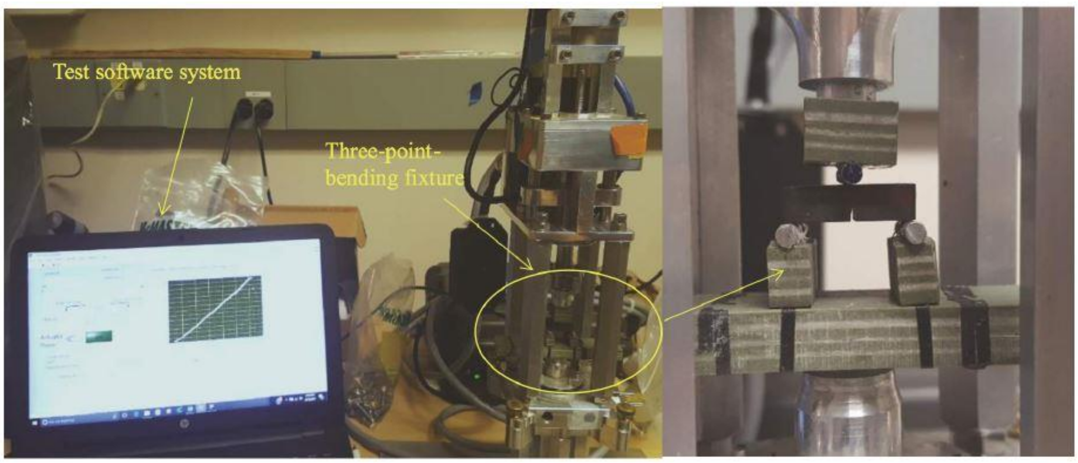

Figure 1 Setup for loading and data acquisition system

2. INTRODUCTION

In this work we aim to perform a blind prediction of critical load that is necessary to propagate the crack in a pre-cracked beam. We model the beam in ABAQUS considering a Plane-Stress problem. The beam is supported on roller supports on each end and is subjected to a concentrated load at the mid-span. We initially apply a unit load (1N) and perform analysis using each element type to determine the J-integral values. We evaluate the contour integral using the conventional finite element method, which typically requires to conform the mesh to the cracked geometry, to explicitly define the crack front, and to specify the virtual crack extension direction. A detailed focused mesh is created to obtain accurate contour integral results for the crack (Fig.2) The J-Integral values are related to the Stress Intensity Factor as follows:

$$K_I = \sqrt{E'J} \qquad \text{(For Plane Stress E' = E)} \qquad (1)$$

KI values contain information about all remote loading types, remote loading levels, and remote boundary conditions. Since the KI value is directly proportional to the applied load (Eq. 2), we evaluate the Critical Load Pcr using the KIc of the material that is given to us.

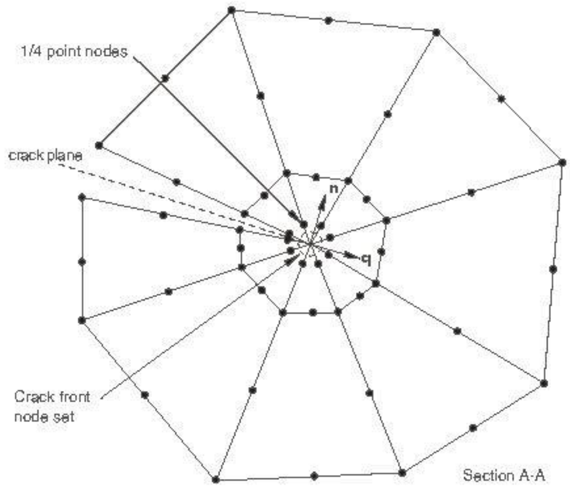

Figure 2 Typical focused mesh for fracture mechanics evaluation

3. METHODOLOGY

3.1 PROBLEM FORMULATION

Assuming that the beam is in a state of Plane Stress and a 2-D model can approximate the solution, the parameters of the given problem are as follows:

MATERIAL PARAMETERS:

Table 1 Material Parameters

E	37.7 MPa
ν	0.25
KIc	0.967 MPa√(m)

DIMENSIONAL PARAMETERS:

Table 2 Dimensional Parameters

L = S	74 mm
D	23.9 mm
t	13.86 mm
a_o	7.6 mm

Since we are using all dimensions in mm, we use KIc in units MPa$\sqrt{mm}$, Thus,

KIc = 30.579 MPa$\sqrt{mm}$

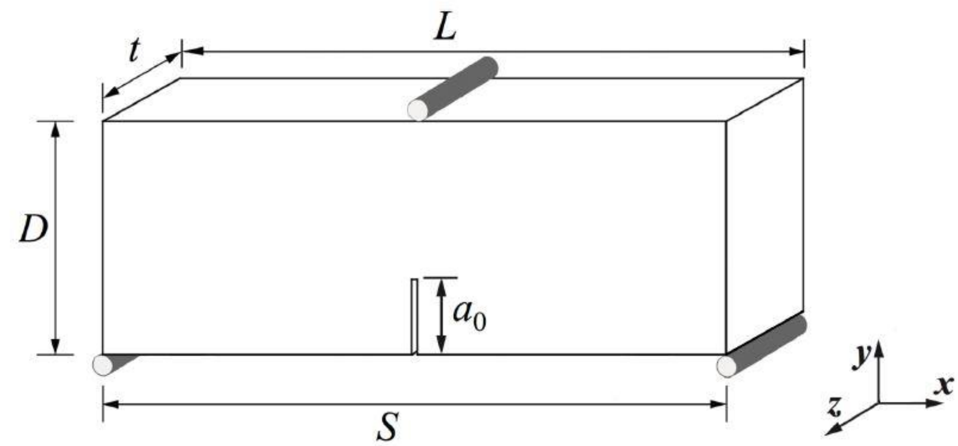

Figure 3 Sketch of the three-point bending (TPB) specimen and loading condition

3.2 MODELING

The Finite Element Model is created in ABAQUS using the following element types:

- QUAD4
- QUAD4R
- QUAD8
- QUAD8R

For each element type listed above, a mesh with element edge length of 1 mm is used. Initially a load of 1N is applied at the mid-span. A focused mesh is created around the crack-tip with 5 rings of elements (Fig.4)

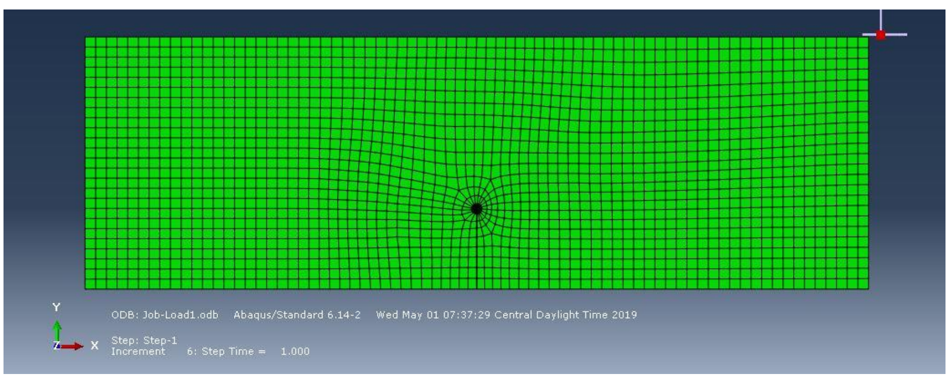

Figure 4 Abaqus FEM of cracked beam

4. RESULTS & CONCLUSION

In this problem we adopt a simply supported boundary condition for the both left and right beam supports. (Uy = 0; since we are considering roller supports on both ends).

We solve the problem for each element type for point load P = 1N at mid span and obtain the contour plots for Von-Mises Stress in the deformed configuration that are shown in Fig. 5 to Fig. 8.

We evaluate the KI at the crack tip using the J-integral method (J-integral is a quasi-static fracture mechanics parameter for linear material response). We also evaluate the Crack Mouth Opening Displacement (CMOD) for each case. The results are tabulated in Table 3 where we take J integral values corresponding to 5 contours at the last time step and take average of the five values.

We know that KI is proportional to the applied load

$$KI \propto P \tag{2}$$

$$KI_{cr} \propto P_{cr}$$

$$KI_{cr} / KI = P_{cr} / P$$

$$P_{cr} = P \times (KI_{cr} / KI) \tag{3}$$

The KI value for each element type is evaluated and tabulated in Table 4, CMOD (P=1) in Table 5, Pcr in each case is calculated and tabulated below in Table 6.

Now the model is subjected to the calculated Pcr for each element type and analyzed to compute the the J-Integral values. They are tabulated in Table 7, KI at the crack tip are calculated using Eq. 1 and tabulated in Table 8. The relative error between calculated KIc and given KIc is calculated and Tabulated in Table 9. Table 10 contains the CMOD (Pcr) and Table 11 has data for Strain Energy U(Pcr) and Number of Degrees of Freedom (NDOF)

Contour Plots for Von-Mises Stress for Pcr are shown in Fig. 9 to Fig. 12.

On analyzing the results of analysis following conclusions can be drawn:

- From the available data we can see that the CMOD values do not seem to be directly proportional to the applied load, Although, it appeared to be a feasible result if crack mouth opening would have been directly proportional to applied load at mid-span.
- The Strain Energy shows an inverse proportionality with applied load. As the values seem to decrease with increasing Pcr value. This might be happening as for higher load more energy is released due to increasing crack size thereby reducing the energy of the system.
- Out of all the four element types used CPS8 gives the least relative error in calculation of KIc thus, we can say CPS8 performs the best in this case. To capture the cracking effect we need a $1/4^{th}$ node in our elements that is shifted towards the crack tip. In case of CPS4 and CPS4R which have only 4 corner nodes this is unlikely.
- From the results it can be concluded that the Critical load for cracking the given pre-cracked beam is in the range of 430 N – 433 N as all the elements have approximately calculated this value (except CPS4R) with a certain degree of accuracy. It can also be observed from the results that Energy values are directly proportional with KI values

Sources of Error in the adopted process:

Discretization error: This error is intrinsic to the polynomial form of the Finite element approximation. The order of approximation and refinement of mesh might be inadequate to capture the actual crack load Pcr. This error can be handled by either reducing the size of element (increasing the number of elements) or increasing the polynomial order of the approximation

Error in the computation of the element integrals: The approximation of the exact value requires a large number of integration points, which is very expensive. Since exact integration might not be possible in our case, we use numerical integration to approximate our solution which is a potential cause of error in comparison to values obtained experimentally.

Errors in the solution of the global equation system: This includes errors due to the *ill-conditioning* of the equations; *truncation* errors and *round-off* errors. The problem of ill-conditioning occurs when there exists an element, or a group of elements, of large stiffness connected to elements of much smaller stiffness. Round-off errors, such as those in some parameters like the coordinates and final reporting of results also affect the quality of results.

Error due to Averaging J-Integrals: The J-integral Values evaluated from ABAQUS are used to calculate the Stress Intensity Factor using Eq. 1. The calculation can be performed either by taking average J value to evaluate KI or Calculating KI for each J value and taking their average. We see that this causes a significant change in our results for CPS4 and CPS4R.

Error due to approximation to 2D geometry: The experiment in the lab is performed on 3D beam while the results presented here are for a 2D approximation of the geometry to consider it as a plane stress problem. However, it might be possible that this mathematical model might turn out to be inaccurate to model fracture problems.

Errors associated with the constitutive equation: We consider a linear elastic material thus, for the homogeneous and isotropic material the displacements are proportional to the Young modulus. However, this is not ideal case as the specimen to be tested in the laboratory might have some material non-linearities that we might be neglecting in our mathematical model.

FIGURES AND TABLES

CPS4

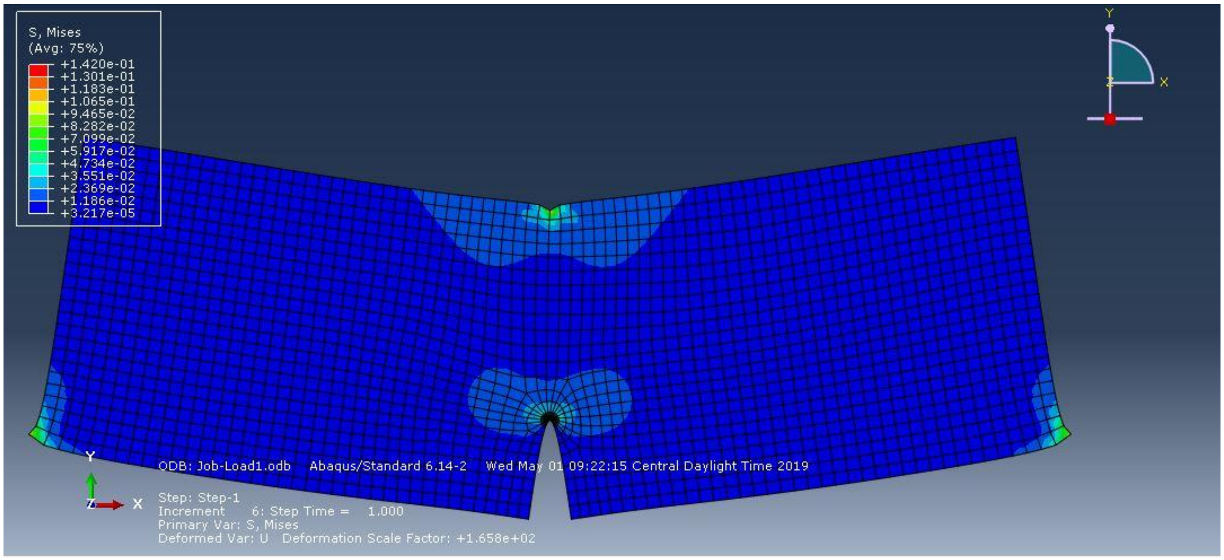

Figure 5 Contour Plot for Von Mises Stress Distribution for CPS4 element type

CPS4R

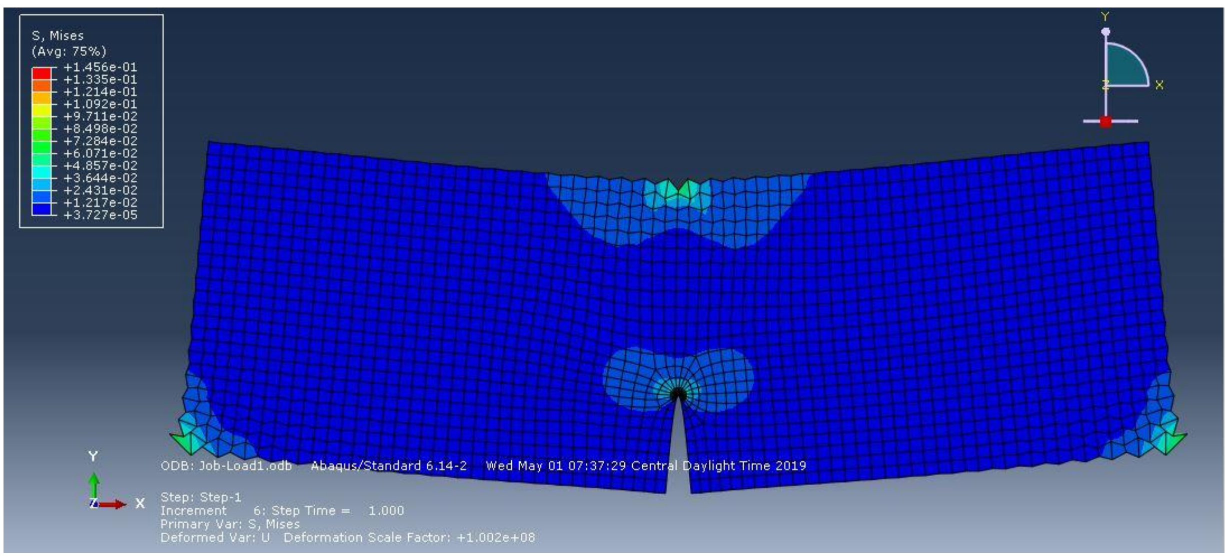

Figure 6 Contour Plot for Von Mises Stress Distribution for CPS4R element type

CPS8

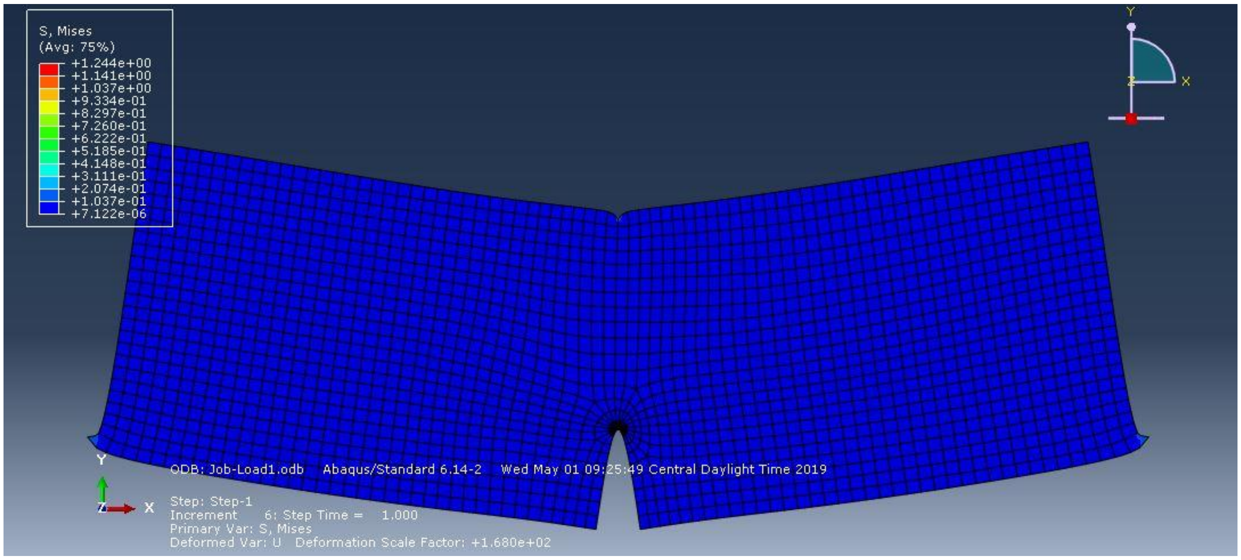

Figure 7 Contour Plot for Von Mises Stress Distribution for CPS8 element type

CPS8R

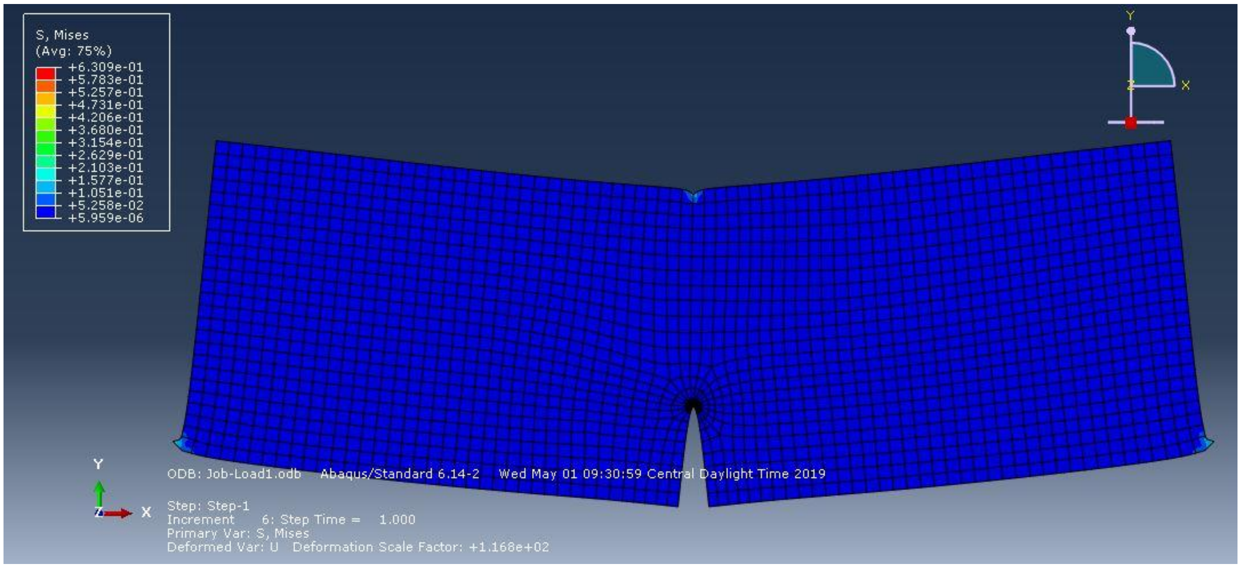

Figure 8 Contour Plot for Von Mises Stress Distribution for CPS8R element type

CONTOUR PLOTS FOR P = Pcr

CPS4

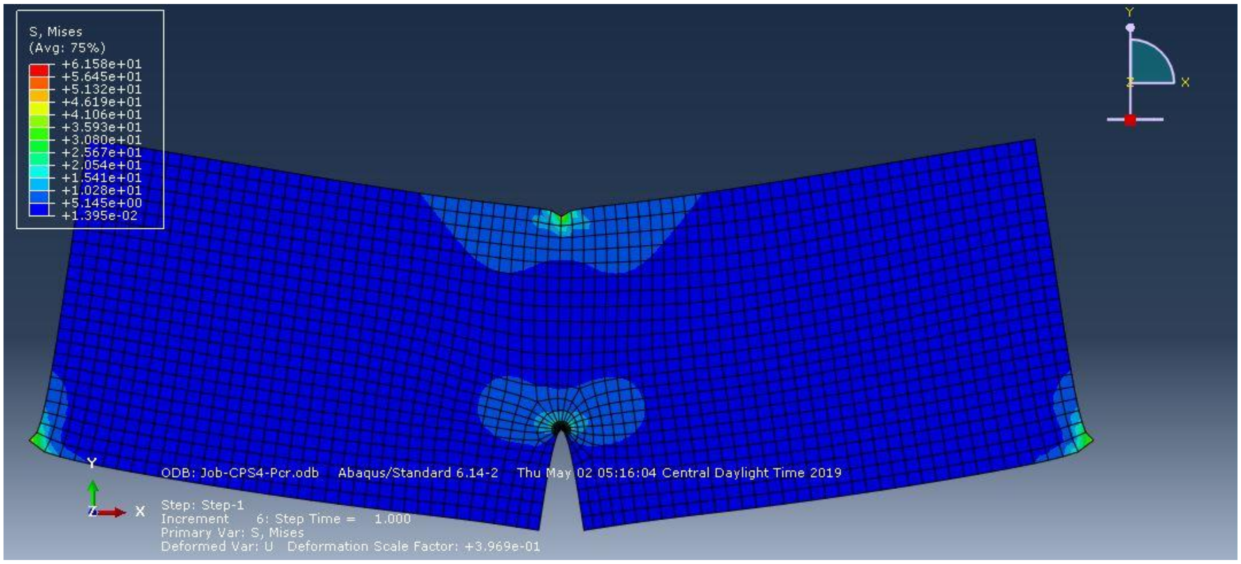

Figure 9 Contour Plot for Von Mises Stress Distribution for CPS4 element type

CPS4R

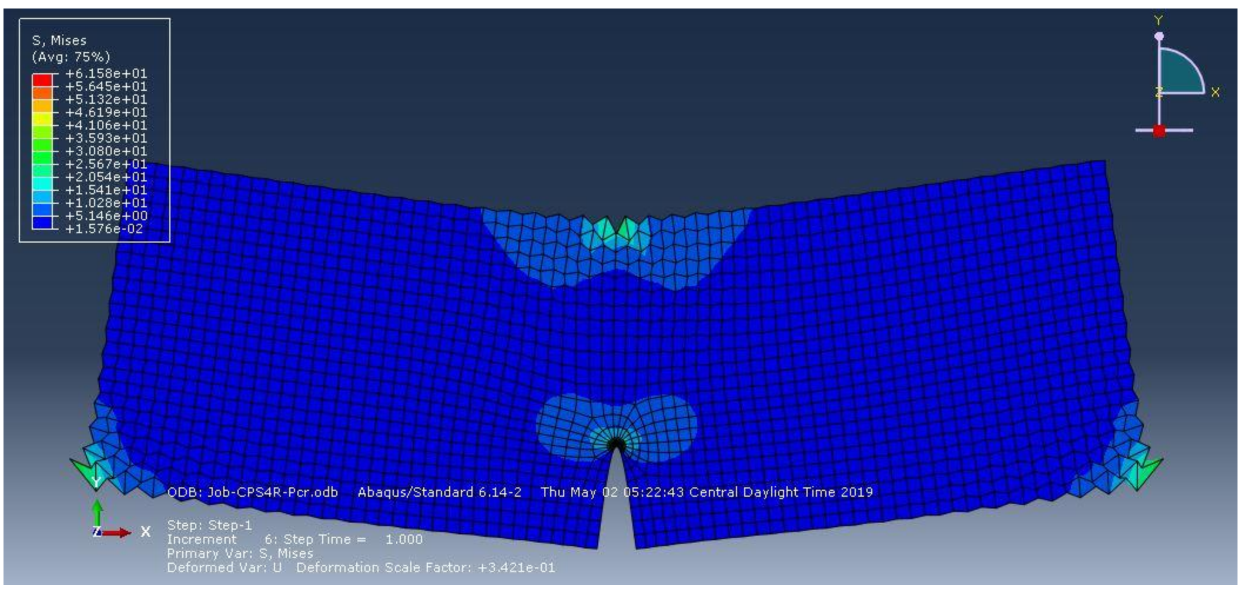

Figure 10 Contour Plot for Von Mises Stress Distribution for CPS4R element type

CPS8

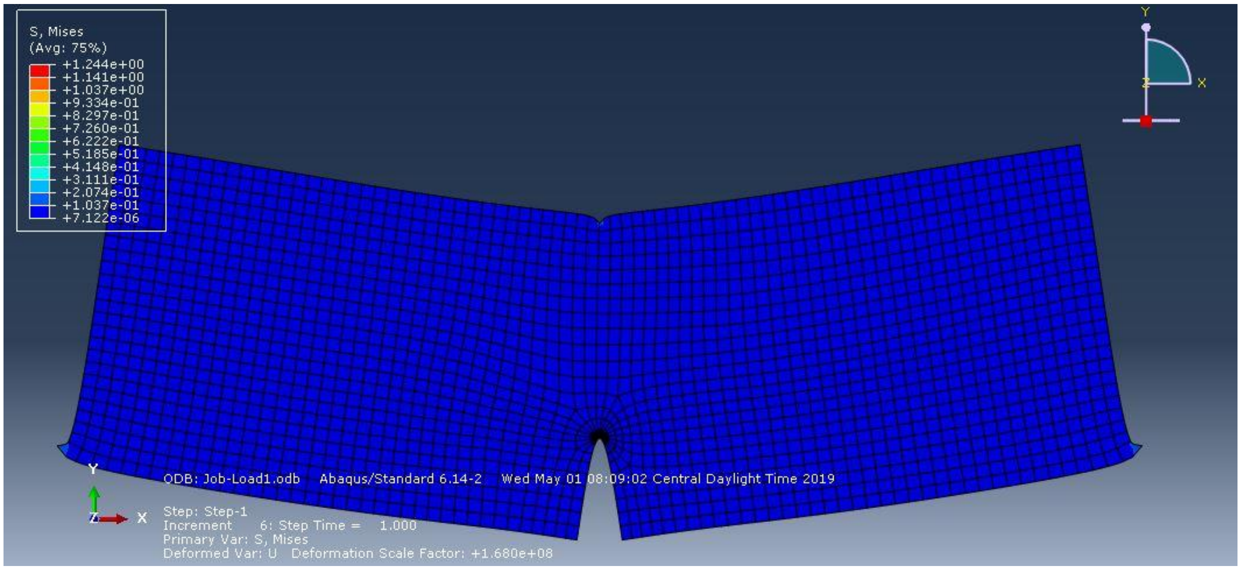

Figure 11 Contour Plot for Von Mises Stress Distribution for CPS8 element type

CPS8R

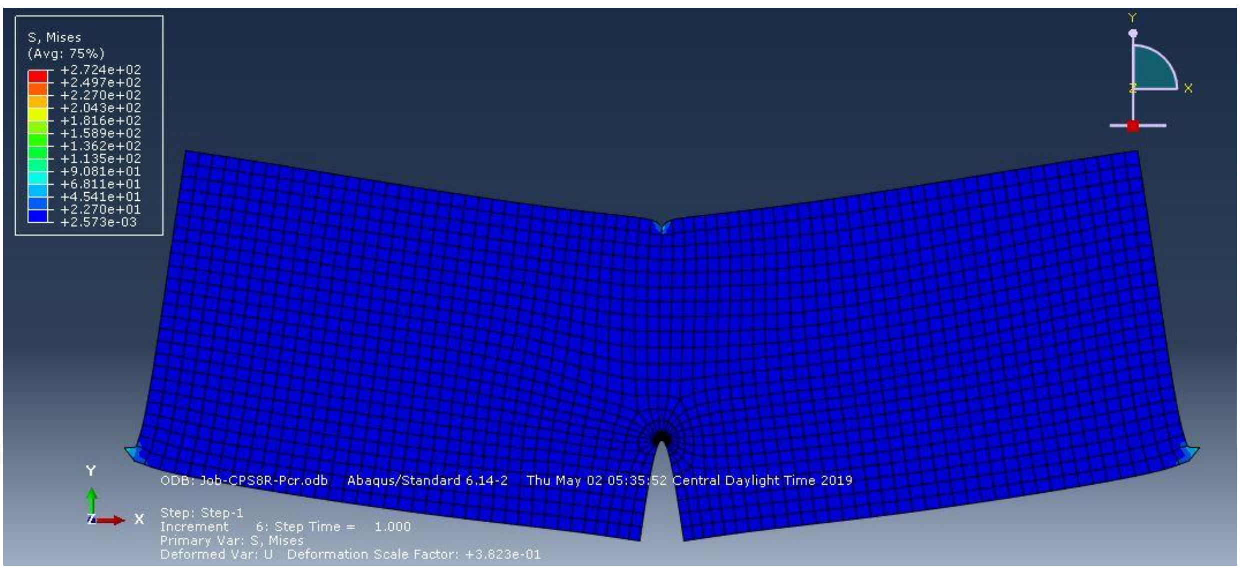

Figure 12 Contour Plot for Von Mises Stress Distribution for CPS8R element type

TABLE 3 J Integral Values

Element Type	**J Integral Values** MPa$\sqrt{mm}$				
CPS4	1.226900E-04	1.326600E-04	1.343100E-04	1.346400E-04	1.348700E-04
CPS4R	1.299300E-04	1.387600E-04	1.395500E-04	1.395800E-04	1.394800E-04
CPS8	1.329400E-04	1.328300E-04	1.328800E-04	1.328800E-04	1.328800E-04
CPS8R	1.330900E-04	1.329400E-04	1.330100E-04	1.330200E-04	1.330100E-04

TABLE 4A KI Stress Intensity Factor Values for P = 1 N (for each J value)

Element Type	KI					Avg KI
CPS4	6.801039E-02	7.071974E-02	7.115818E-02	7.124555E-02	7.130637E-02	7.048805E-02
CPS4R	6.998829E-02	7.232739E-02	7.253299E-02	7.254079E-02	7.251480E-02	7.198085E-02
CPS8	7.079434E-02	7.076504E-02	7.077836E-02	7.077836E-02	7.077836E-02	7.077889E-02
CPS8R	7.083426E-02	7.079434E-02	7.081297E-02	7.081563E-02	7.081297E-02	7.081404E-02

TABLE 4B KI Stress Intensity Factor Values for P = 1 N (For Average J Values)

J Avg.	KI
1.318340E-04	7.049923E-02
1.374600E-04	7.198779E-02
1.328820E-04	7.077889E-02
1.330140E-04	7.081404E-02

TABLE 5 CMOD values for P = 1N

Element Type	U1 Left	U1 Right	|CMOD (x)|
CPS4	0.03577	0.016042	0.019728
CPS4R	0.020701	0.000818	0.019883
CPS8	0.012851	-0.00712	0.019972
CPS8R	0.007893	0.027866	0.01997

TABLE 6 Pcr Values for all Elements

Element Type	KI(P=1)	Pcr (N)	KIc
CPS4	0.070499	433.752593	30.579
CPS4R	0.071988	424.783490	30.579
CPS8	0.070779	432.038770	30.579
CPS8R	0.070814	431.824344	30.579

TABLE 7 J Integral Values for P = Pcr

Element Type	**J Integral Values** MPa$\sqrt{mm}$				
CPS4	23.090	24.970	25.280	25.340	25.380
CPS4R	23.230	24.810	24.950	24.950	24.940
CPS8	24.810	24.790	24.800	24.800	24.800
CPS8R	24.820	24.790	24.800	24.800	24.800

(Length units in mm, Pressure units in MPa, Force units in N, Energy units in N-mm)

TABLE 8 KI Stress Intensity Factor Values for P = Pcr

Element Type	**KI**				
CPS4	29.504118	30.681737	30.871605	30.908219	30.932604
CPS4R	29.593428	30.583280	30.669447	30.669447	30.663301
CPS8	30.583280	30.570950	30.577116	30.577116	30.577116
CPS8R	30.589443	30.570950	30.577116	30.577116	30.577116

TABLE 9 Relative Error between material KIc and Kic from FEM

Element Type	**KI(Pcr)**	**KIc**	**|(KI(Pcr)-KIc))/Kic|**
CPS4	30.5797	30.579	0.000014
CPS4R	30.4358	30.579	0.004691
CPS8	30.5771	30.579	0.000069
CPS8R	30.5783	30.579	0.000029

TABLE 10 CMOD values for P = Pcr

Element Type	**|CMOD(Pcr)|**
CPS4	8.540760
CPS4R	8.407050
CPS8	8.628520
CPS8R	8.624920

TABLE 11 Strain Energy U(Pcr) and Number of DOFs

Element Type	U(Pcr)	NDOFS
CPS4	3703.83	4146.00
CPS4R	4446.76	4146.00
CPS8	4110.51	12228.00
CPS8R	4179.68	12228.00

(All Length units in mm, Pressure units in MPa, Force units in N, Energy units in N-mm)

APPENDIX

ABAQUS files are available at the following link:

https://drive.google.com/open?id=1VIHUHPxuSjQxNS60Q0a9hH6QDgUA9JfX

There are 2 folders:

1.) Model: Contains the ABAQUS model of the given problem
2.) Load P1: This contains files for each element type for the model subjected to load P = 1N
3.) Load Pcr: This contains files for each element type for the model subjected to respective critical load.
4.) An excel file containing all the results

A detailed description of each file is as follows:

File Name	Task Performed
Job-CPS4-P1	A FE model of Fig2 in ABAQUS with CPS4 element type, focused mesh around crack tip with 5 element rings, subjected to Load P = 1N at mid-span
Job-CPS4R-P1	A FE model of Fig2 in ABAQUS with CPS4R element type, focused mesh around crack tip with 5 element rings, subjected to Load P = 1N at mid-span
Job-CPS8-P1	A FE model of Fig2 in ABAQUS with CPS8 element type, focused mesh around crack tip with 5 element rings, subjected to Load P = 1N at mid-span
Job-CPS8R-P1	A FE model of Fig2 in ABAQUS with CPS8R element type, focused mesh around crack tip with 5 element rings, subjected to Load P = 1N at mid-span
Job-CPS4-Pcr	A FE model of Fig2 in ABAQUS with CPS4 element type, focused mesh around crack tip with 5 element rings, subjected to Load P = 433.809 N at mid-span
Job-CPS4R-Pcr	A FE model of Fig2 in ABAQUS with CPS4 element type, focused mesh around crack tip with 5 element rings, subjected to Load P = 424.824 N at mid-span
Job-CPS8-Pcr	A FE model of Fig2 in ABAQUS with CPS4 element type, focused mesh around crack tip with 5 element rings, subjected to Load P = 432.039 N at mid-span
Job-CPS8R-Pcr	A FE model of Fig2 in ABAQUS with CPS4 element type, focused mesh around crack tip with 5 element rings, subjected to Load P = 431.824 N at mid-span

CEE 570 FINAL PROJECT DATASHEET

Element Type	KI(P=1)	|CMOD(P=1)|	Pcr (N)	KI(Pcr)	|(KI(Pcr)-KIc))/Kic|	|CMOD(Pcr)|	U(Pcr)	NDOFS
CPS4	0.070488	0.019728	433.809000	30.57965676	0.000014	8.540760	3703.83	4146.00
CPS4R	0.071981	0.019883	424.824431	30.43578066	0.004691	8.407050	4446.76	4146.00
CPS8	0.070779	0.019972	432.038774	30.57711538	0.000069	8.628520	4110.51	12228.00
CPS8R	0.070814	-0.019973	431.824351	30.57834795	0.000029	8.624920	4179.68	12228.00

Element Type	KI(P=1)	|CMOD(P=1)|	Pcr (N)	KI(Pcr)	|(KI(Pcr)-KIc))/Kic|	CMOD (mm)	U (Pcr)	NDOFS
CPS4	0.070499	0.019728	433.752593	30.57852978	2.273403E-05	8.540740	3884.59	4146.00
CPS4R	0.071988	0.019883	424.783490	30.57771219	4.947085E-05	8.394651	4149.83	4146.00
CPS8	0.070779	0.019972	432.038770	30.57921181	4.303976E-07	8.628540	4107.3	12228.00
CPS8R	0.070814	-0.019973	431.824344	30.5792652	1.315328E-06	8.623160	4173.75	12228.00

The above two tables correspond to two different ways of calculating the KI from the J-integral values obtained from ABAQUS for each 5 contours.

The upper table values are obtained by finding KI for each J value of each contour then averaging it to obtain the KI for each element type.

The lower table values are obtained by taking an average J value that is used to find the KI for each element type.

We see that for each methodology there is a different “Best” performing element on the basis of relative error in KI values.

In the first table CPS 4 performs the best which is very unexpected in case of a fracture mechanics problem as we need a focused mesh with nodes that are moved $1/4^{th}$ towards the crack tip which is not possible for CPS4.

In the second table we see that CPS8 is the best performing element type.

(Length units in mm, Pressure units in MPa, Force units in N, Energy units in N-mm)