XFEM BASED FRACTURE ANALYSIS

Objective

To study the crack propagation using FEAST for an infinitely long solid with cross section as shown in Figure 1 using plane strain elements

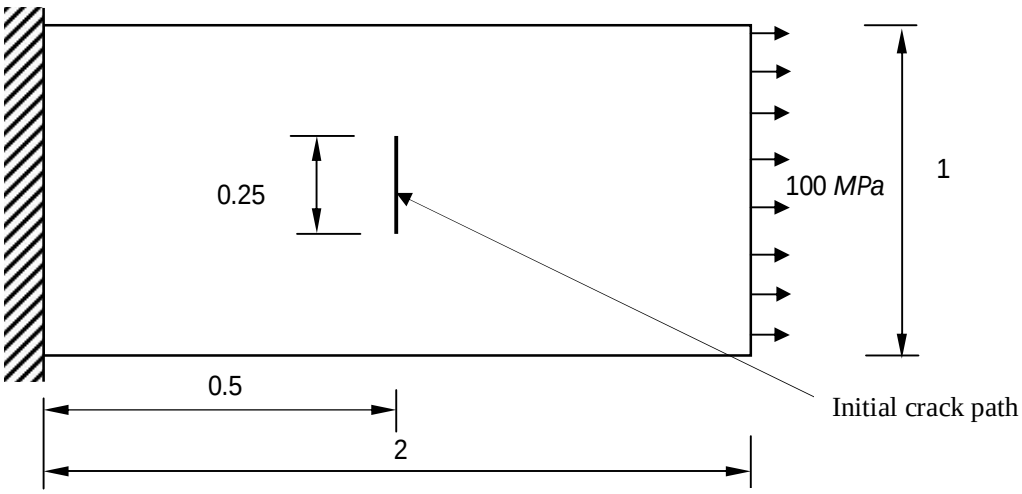


Figure 1: Dimensions and initial crack path

All dimensions are in *m*

Problem definition

Analysis Type: XFEM based fracture analysis

Modulus of elasticity, E	=	200 *GPa*
Poisson's Ratio, υ	=	0.3
Specific gravity	=	7.8

Loads and boundary conditions:

- Pressure load .1kPa is applied normal to the plate.
- Left edge of the plate is assumed to be fixed.

PROCEDURE

1. Creation of geometry

- **Create 4 points at (0,0,0) (2,0,0) (0,1,0) (2,1,0) (1/0.375,0) and (1/0.625/0)**

Commands : POINT, ADD

Menu : Geometry → Key point → Create → By X/Y/Z

Parameters : (To be filled by the user)

Coordinate Data	0/0/0
Coordinate Type	0
Entity ID	1

- **Create quadrilateral surface on four points**

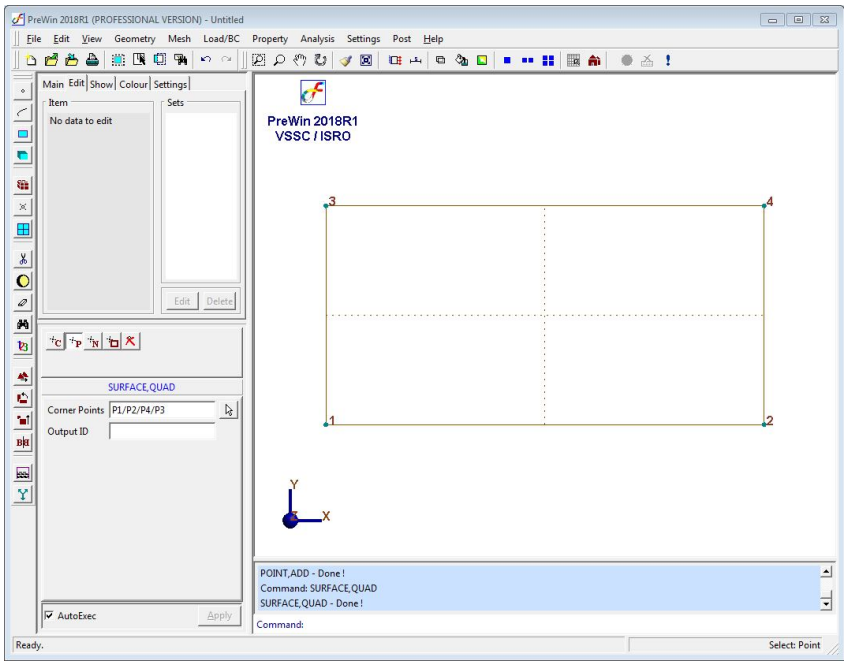
Command : SURFACE, QUAD

Menu : Geometry → Surface → Create → Quad Surface

Parameters : (To be filled by the users)

Point Data	Use mouse to pick the points
Entity ID	1

At the end of the above operations your screen should looks like this.



2. Meshing using quadrilateral elements

Command : FEM, QUAD

Menu : Mesh → FE Mesh → QUAD

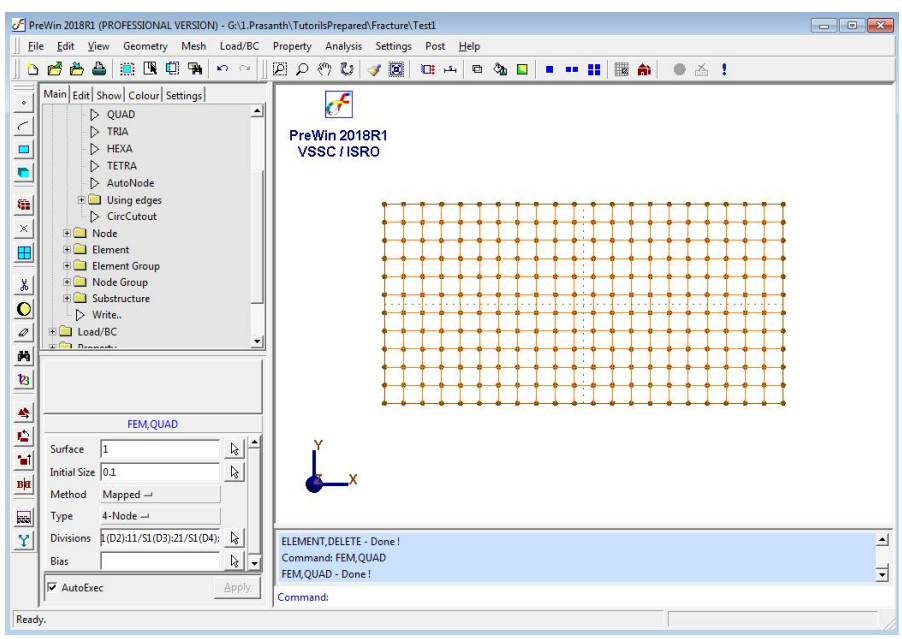
Parameters :

Surface	Use mouse to select the surface
Initial size	0.1
Method	Mapped
Type	4-Node
Divisions	S1(D1):21/S1(D2):11/S1(D3):21/S1(D4):11

Note:

Divisions along each edge can be varied by selecting the corresponding edges and right/left clicking the mouse point to adjust the subdivisions. Eg: S1(D1)/21 shows 21 sub divisions along edge1 & S1(D2)/11 shows 11 sub divisions along edge2 of surface1.

At the end of the above operations your screen should looks like this.

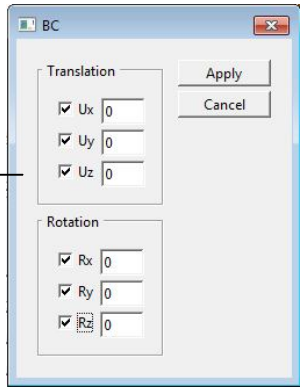


3. Specify displacement boundary conditions

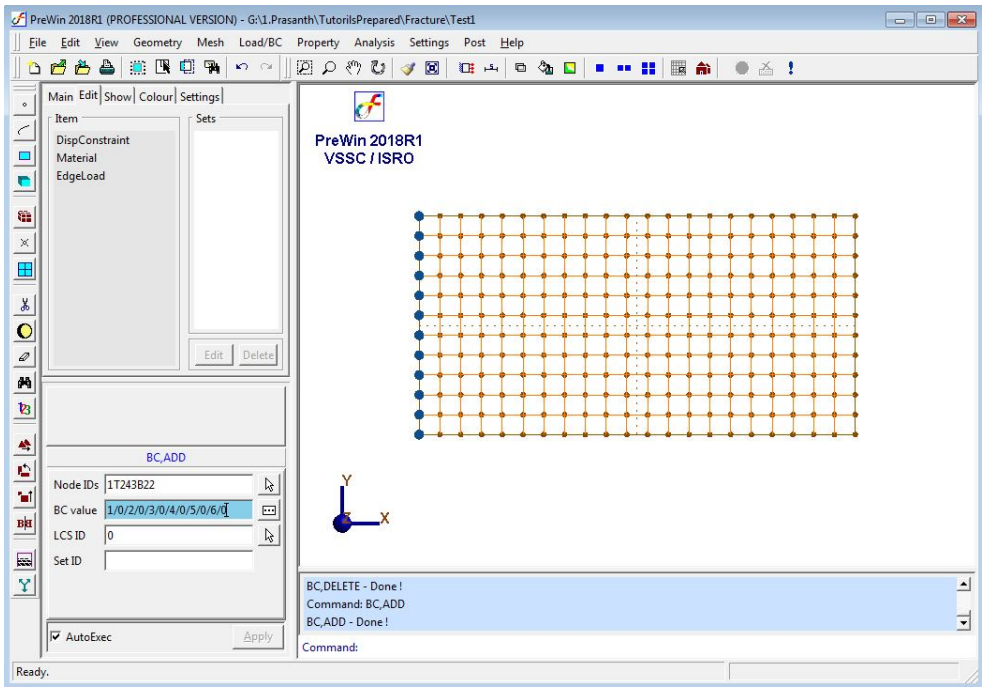
Command : BC, ADD

Menu : Load/BC → Displacement BC → Add

Node ID	Select the nodes on the left side boundary edges
BC Value	1/0/2/0/3/0/4/0/5/0/6/0
LCS ID	0
Set ID	1



At the end of the above operations your screen should looks like this.



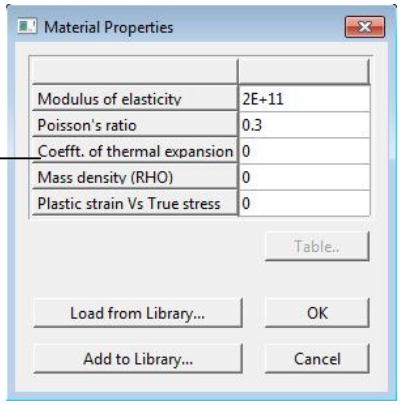
4. Specify material properties

Command : MATERIAL, ISO

Menu : Property → Material → Isotropic → Add

Parameters

Element IDs	All
Material Data	2E+11/0.3/0/0/0
Material ID	1



5. Convert the element type to plane strain

Command : ELEMENT, TYPE

Menu : Mesh → Element → Modify → Type

Parameters

Element IDs	All
Element Type	Plane Strain

6. Specify edge load

Command : EDGE, ADD

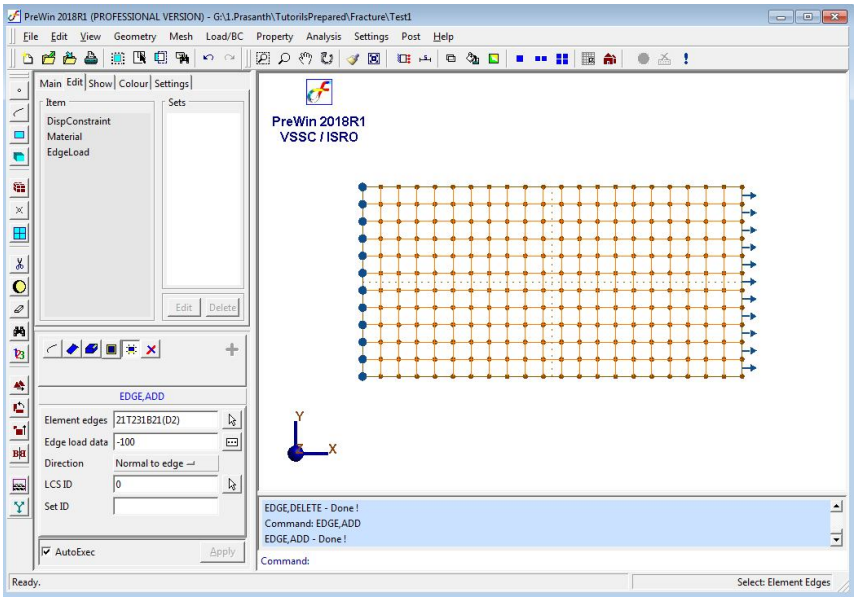
Menu : Load/BC → Edge loads → Add

Element face	21(D2)/42(D2)/63(D2)/84(D2)/105(D2)/126(D2)/147(D2)/168(D2)/189(D2)/210(D2)/231(D2
Pressure value	-100 *N*
Direction	Normal to edge

Parameters :

LCS ID	0
Set ID	1

At the end of the above operations your screen should look like this



7. Specify analysis type

Commands : ANTYPE, SET

Menu : Analysis → Analysis Type

Parameters : (To be filled by the user)

Analysis Type	Fracture

8. Specify analysis options

Commands : ANOPTION, SET

Menu : Analysis → Analysis Options

Parameters : (To be filled by the user)

Linear Solver	Multi Frontal
Eigen Solver	Lanczos

9. Specify fracture general data

Commands : XFEMFRACTURE,ADD

Menu : Analysis →Fracture →General →Add

Parameters : (To be filled by the user)

No. of time steps	10
Crack increment	0.01

10. Specify crack path

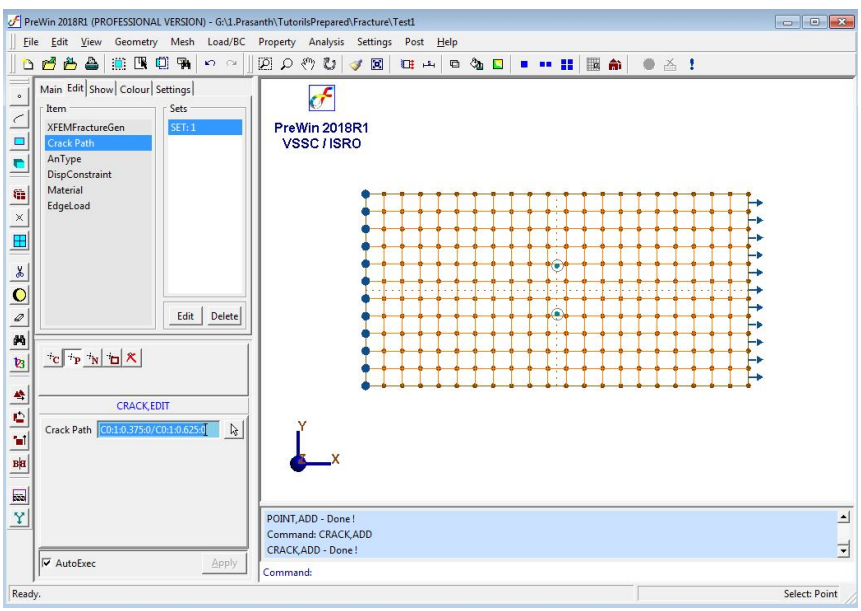
Commands : CRACK,ADD

Menu : Analysis → Fracture →Crack definition →Add

Parameters : (To be filled by the user)

Crack path	P5/P6
Output Id	1

At the end of the above operations your screen should look like this



11. Save the data file

Menu : File→ Save

12. Submit the job into FEAST

Menu : Analysis →Run Solver

```
Writing Results...
ELAPSED TIME   : 0 H :: 0 M :: 1 S
Finished successfully
```

After the solution is completed the message *"successfully completed"* appears in the *message box.*

13.Post processing

a) Displacement Contour

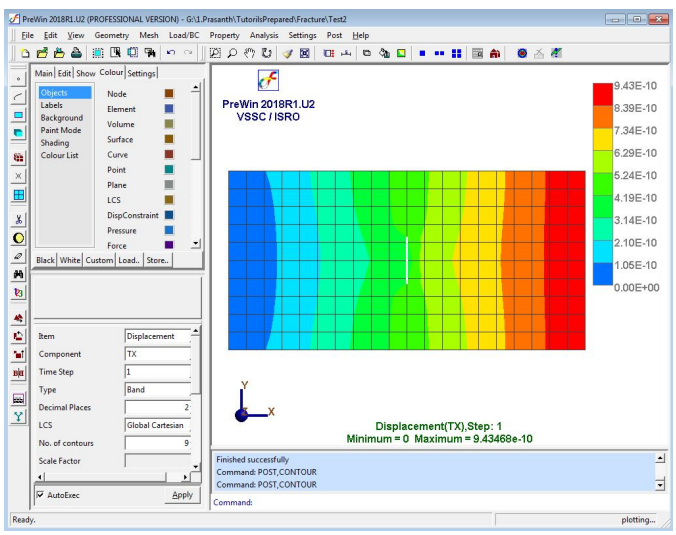
Command : POST, CONTOUR

Menu : Post → History plot

Parameters :

Item	Displacement
Component	TX
Time Step	1

At the end of the above operations your screen should looks like this.



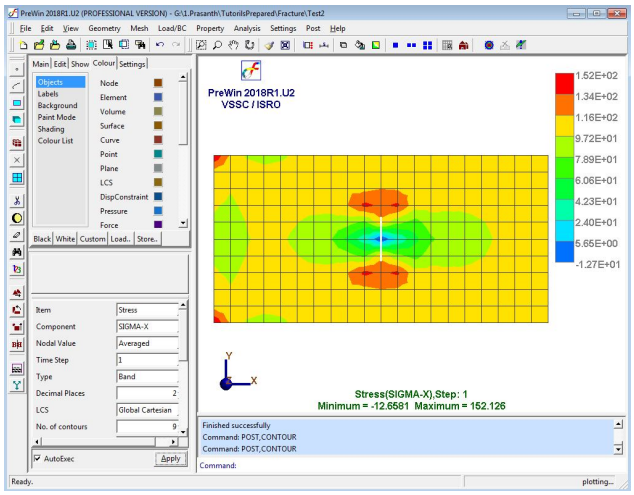
b) Stress Contour

Command : POST, CONTOUR

Menu : Post → History plot

Item	Stress
Component	SIGMA-X
Time Step	1

At the end of the above operations your screen should looks like this.



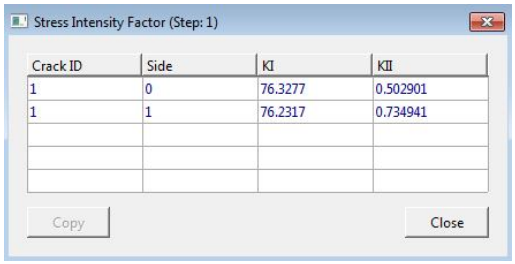
c) View Results

Command : POST, VIEWRESULTS

Menu : Post → History plot

Item	Stress Intensity Factor
Time Step	1

The following table will be displayed



Note:
For more accurate results use refined mesh.
Output file can be seen in *.OUT