Course Name

Fracture Mechanics KM

Release 2023 R2

Please note:
- These training materials were developed and tested in Ansys Release 2023 R2. Although they are expected to behave similarly in later releases, this has not been tested and is not guaranteed.
- The screen images included with these training materials may vary from the visual appearance of a local software session.

Ansys

Content

- Introduction
  - Objective
  - Prerequisites
- Theory
  - Static vs. Fatigue Crack Growth
  - Crack Growth Modeling
  - Crack-Closure
- Step-by-Step Procedure
- Observations

Introduction

Ansys

Objective

- To understand how Crack Closure affects Crack Growth in case of fatigue.
- To Understand the workflow of how Crack-Closure is modeled in Ansys Mechanical.
- Comparing the results how Crack-Closure affects the No. of Load cycles for the specimen.

Prerequisites

- Working knowledge of the Workbench Mechanical environment and familiarity with fracture mechanics applications.

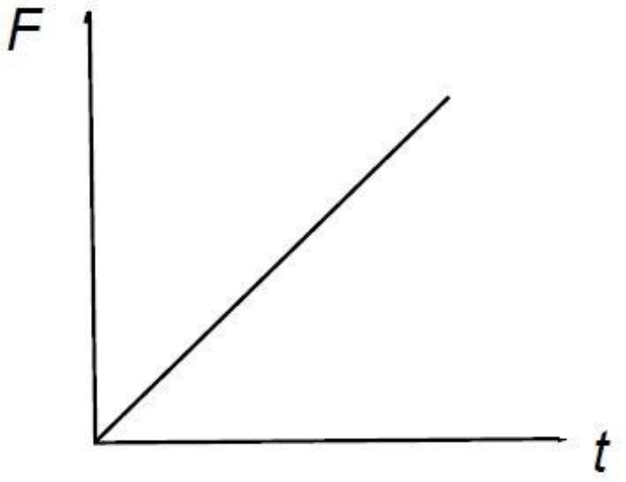
- Geometrical dimensions, loads, BCs, etc. chosen might not be completely realistic. Focus is given on the **workflow** and not on design considerations.

Theory

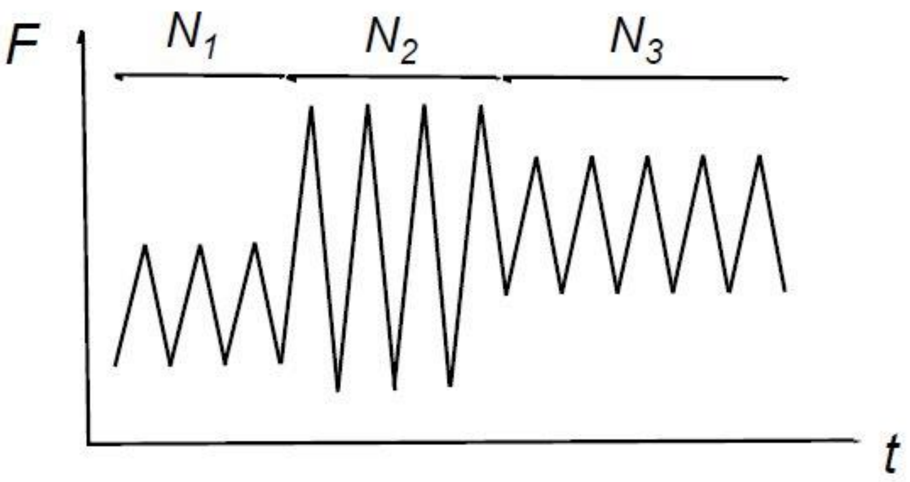
Static Vs Fatigue Crack Growth

- Static Crack Growth: Preexisting cracks may propagate when certain loading conditions are reached or when certain localized conditions are met.

- Fatigue Crack Growth: In cases when structures are subject to cyclic loading, it is of interest to know the interaction between crack extension rate and the number of load cycles.



Static



Fatigue

Crack growth Modeling

- A common fatigue crack-growth model is Paris' Law, which relates the stress-intensity-factor range to the crack-growth rate for region II under a fatigue crack-growth regime:

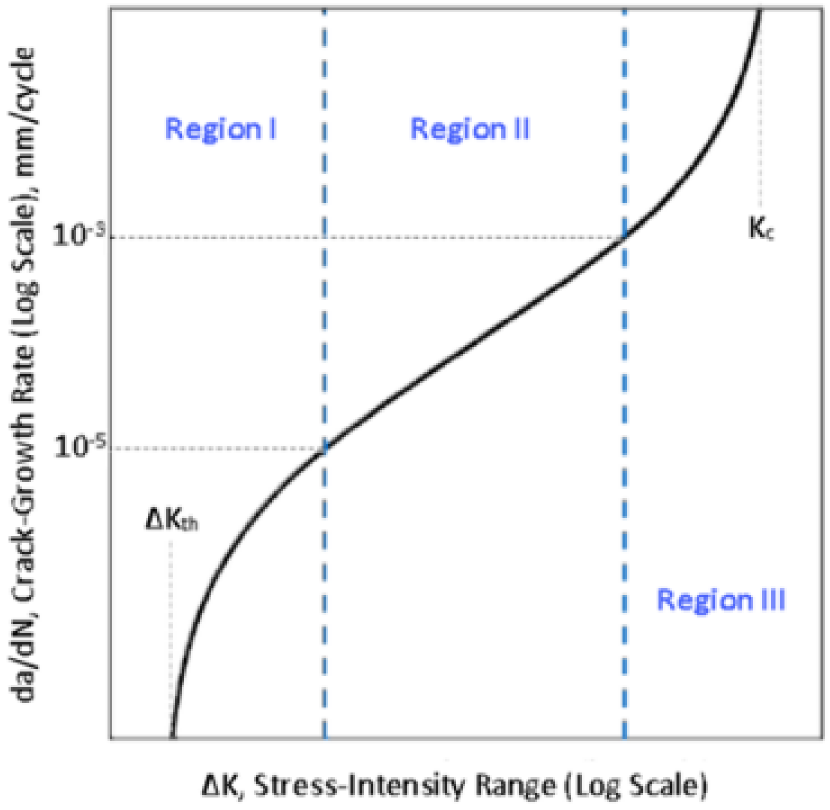
$$\frac{da}{dN} = C(\Delta K)^m$$

- The stress-intensity-factor range ΔK can be written as:

$$R = \frac{K_{min}}{K_{max}}$$

$$\Delta K = (1 - R)\, K_{max}$$

- Where R is the stress ratio and $Kmin$, $Kmax$ represent the stress-intensity factors at the minimum and maximum loads in a load cycle, respectively.

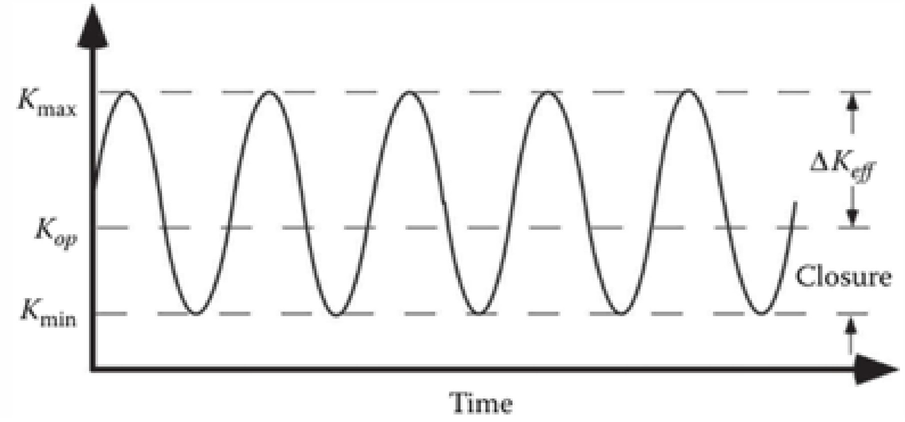


Crack-Closure

- The concept of Crack-Closure is that during a fatigue cycle, the crack remains closed below the load level corresponding to K_{op} and, consequently, the effective stress-intensity-factor range ΔK_{eff} driving the crack-growth is less than ΔK. A measure of crack closure is the U ratio, defined as:

$$U = \frac{\Delta K_{eff}}{\Delta K}$$

$U = Crack\ Closure\ Parameter$



- Ansys supports the following Crack-Closure Models:-

Fatigue crack closure models	
Elber function	$U = 0.5 + 0.4R$
Schijve function	$U = 0.55 + 0.33R + 0.12R^2$
Newman function	$U = \frac{1 - f(R)}{1 - R}$ $f(R)$ is Newman crack opening function
Polynomial function	$U = A_0 + A_1R + .. + A_nR^n$

$R = Stress\ Ratio$

Crack-Closure

- The following graph shows the relation between crack-closure and stress ratio R
  - When we have a positive value of R we have low closure as it produces a lower alternating K
  - In the case of negative R we observe high closure

$$\frac{da}{dN} = g(\Delta K_{eff})$$

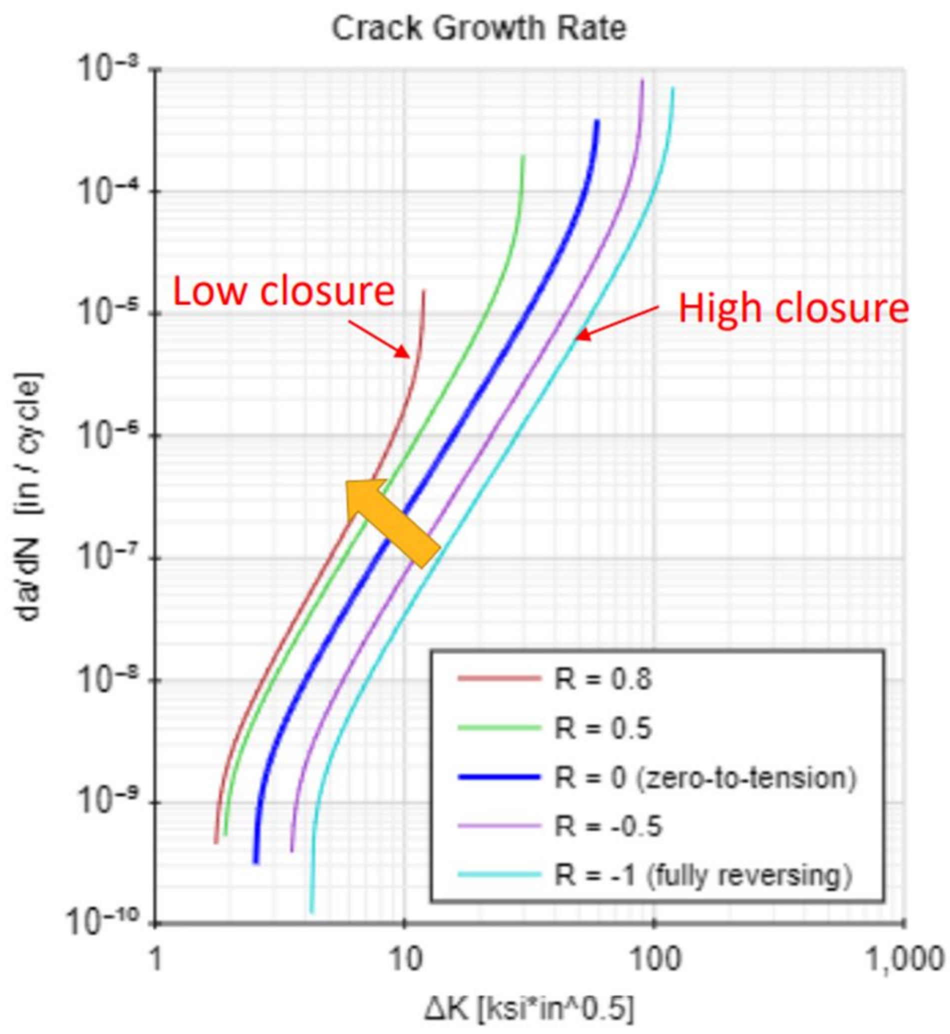
Higher crack closure at lower R

↓

Lower ΔK_{eff} for same ΔK

↓

Lower da/dN

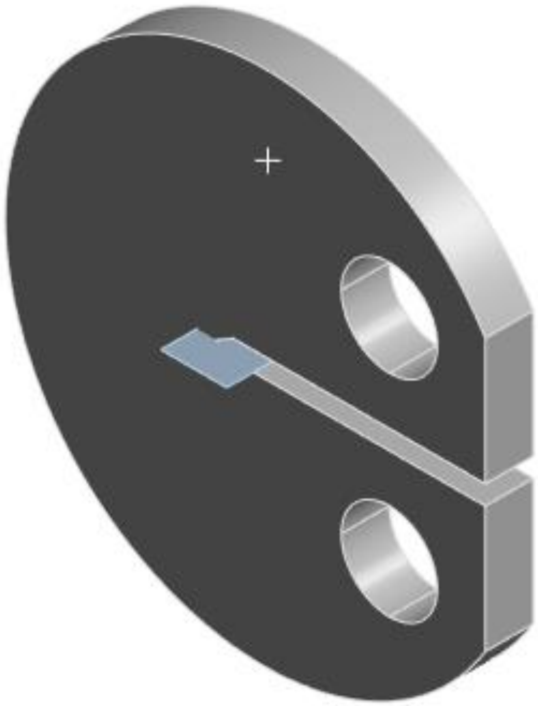


Ansys

Step-by-Step Procedure

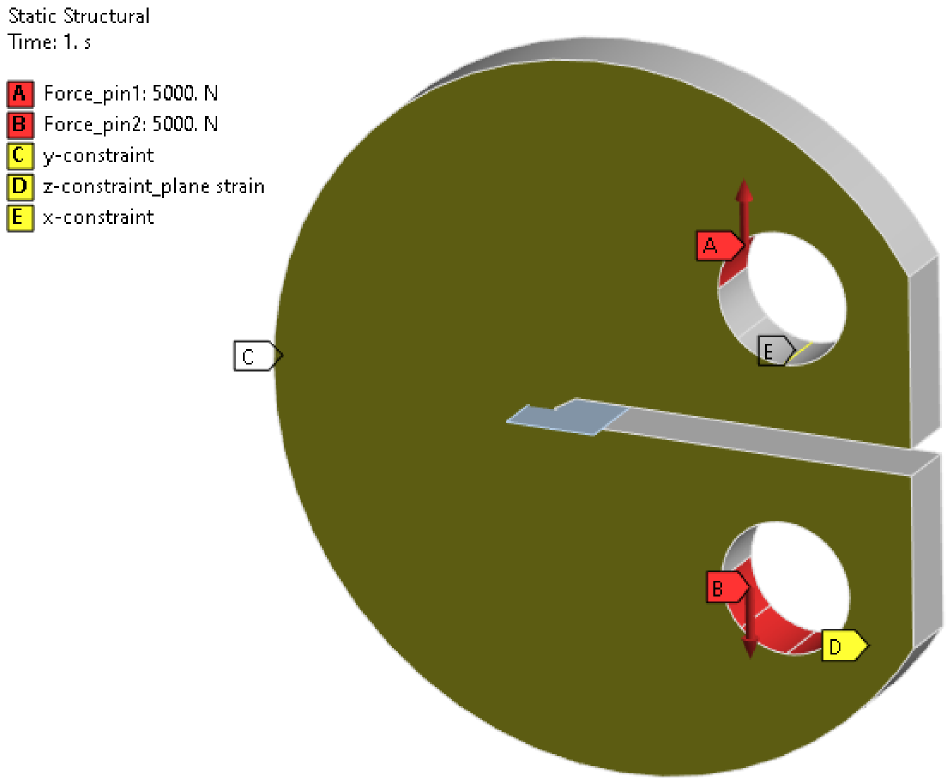
Geometry

- Begin by opening **Crack_Closure_2023R2_Start.wbpz**
- The geometry of a Disk-shaped CT specimen is already imported into Mechanical as shown
  - includes a construction surface body for crack definition



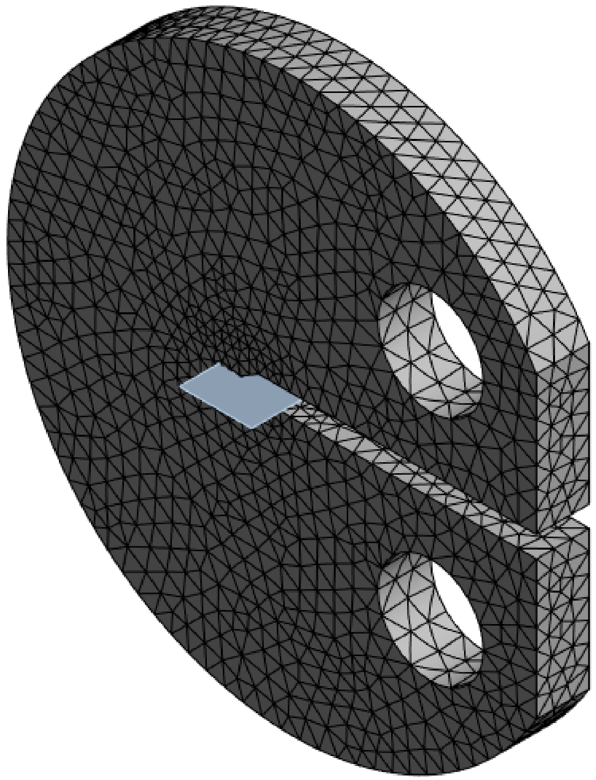
Boundary Conditions

- The boundary conditions are already applied to the geometry.



Mesh

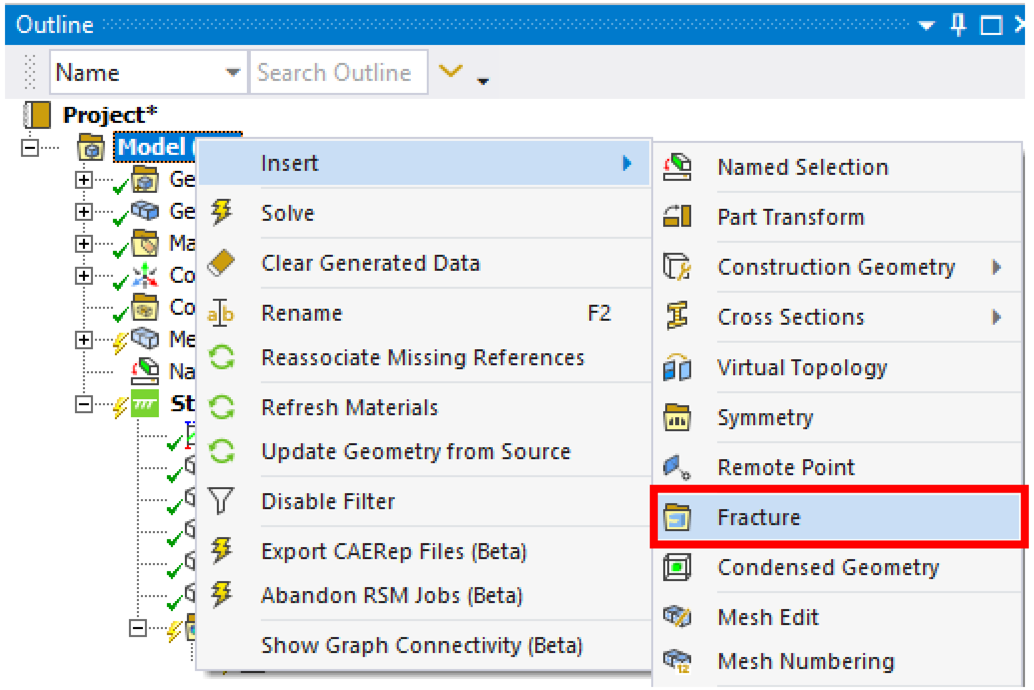
- The meshing controls are already pre-defined in the model.
- The mesh is more refined where the crack is inserted.



SMART is only applicable to quadratic, tetrahedral meshes (SOLD187) and linear, elastic material models.

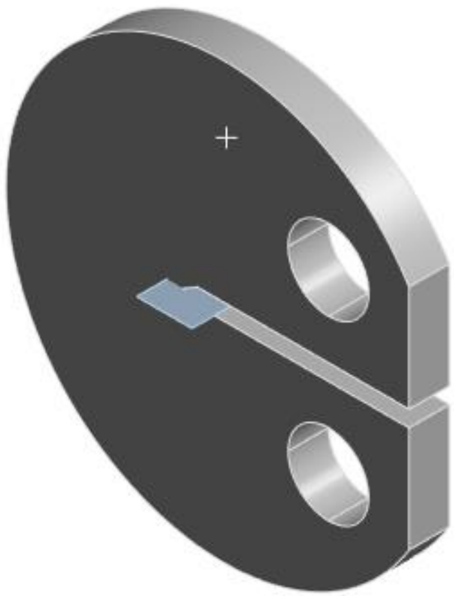
Fracture

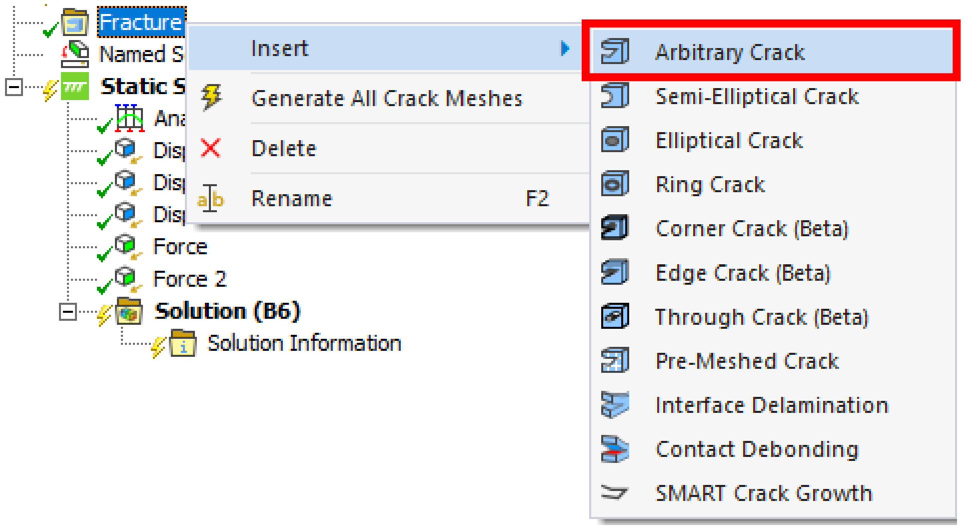
- For the Crack growth study, we first must insert a Fracture object in the Model.
- To insert a Fracture object
  - Click RMB on Model > Insert > Fracture.



Fracture

- We first need to insert a crack into the model to simulate the growth under loading.
- To insert a crack by
  - Clicking RMB on Fracture > Insert > Arbitrary Crack
- The arbitrary crack option uses a surface to create a crack in a solid body.



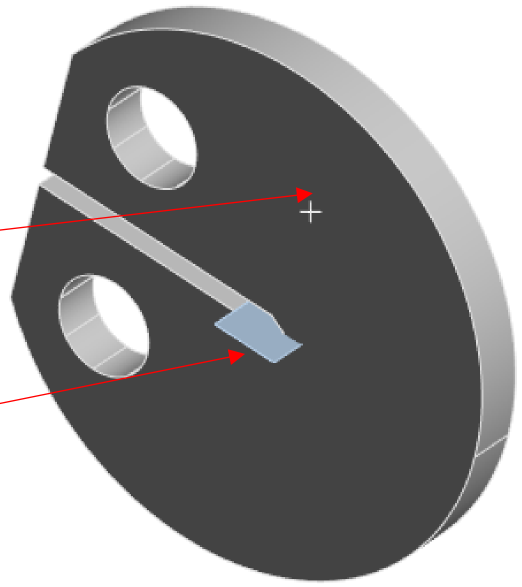


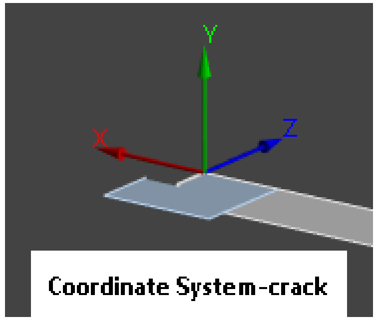
Ansys

Arbitrary Crack

- Enter the following options under the details of Arbitrary crack.
  - Select the solid body and scope it to the arbitrary crack.
  - Select the Local Coordinate System which has been defined at the crack tip.
  - Select the Surface body as the crack surface.
- There are a few requirements for the crack Coordinate system-
  - The x-axis of the crack coordinate system must point in the direction of the crack extension
  - The y-axis of the crack coordinate system must point normal to the crack surface
  - The z-axis of the crack coordinate system must point tangential to the crack front

Details of "Arbitrary Crack"	
Scope	
Source	Arbitrary Crack
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Crack ID	41
Coordinate System	Coordinate System-crack
Crack Shape	Arbitrary
--Crack Surface	1 Body
Mesh Method	Tetrahedrons
Growth Rate	Default (1.2)
Front Element Size	1.0 mm
Solution Contours	6
Suppressed	No
Buffer Zone Scale Factors	
X Scale Factor	2.
Y Scale Factor	2.
Z Scale Factor	2.
Named Selections Creation	
Crack Front Nodes	NS_ArbCrack_Front
Crack Faces Nodes	On
--Top Face Nodes	NS_ArbCrack_TopFace
--Bottom Face Nodes	NS_ArbCrack_BottomFace

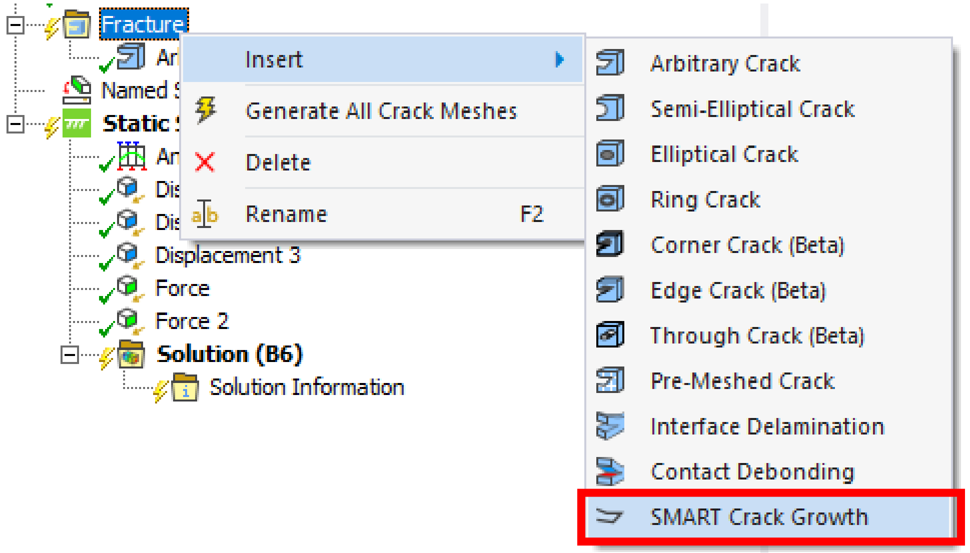




SMART Crack Growth

- SMART is a fatigue or static Crack growth simulation using Separation, Morphing, Adaptive, and Remeshing Technology.

- Insert a SMART Crack Growth object by
  - RMB on Fracture > Insert > SMART Crack Growth



Ansys

SMART Crack Growth

- Enter the following options under SMART Crack Growth.
  - Select the Arbitrary Crack defined earlier for Initial Crack.
  - Select Fatigue as the Crack Growth Option.
  - Select the Mat1 as the Material.
  - Enter -0.2 as the Stress ratio

- This example uses 30 Substeps.
- Paris Law is used by SMART as the Crack Growth Law. It has been added to the Engineering Data of Mat1.

Details of "SMART Crack Growth"

Definition	
Analysis	Crack Growth
Method	SMART
Suppressed	No
Options for Crack Growth	
Initial Crack	Arbitrary Crack
Crack Growth Option	Fatigue
Failure Criteria Option	Material Data Table
Material	Mat1
Crack Growth Law	Paris Law
Crack Growth Methodology	Life Cycle Prediction
Min Increment of Crack Extension	Program Controlled
Max Increment of Crack Extension	Program Controlled
Stop At Max Crack Extension	None
Stress Ratio	-0.2
Remeshing Controls	
Mesh Coarsening	Conservative

$$\frac{da}{dN} = C(\Delta K)^m$$

Properties of Outline Row 3: Mat1

	A	B
1	Property	Value
2	Material Field Variables	Table
3	Isotropic Elasticity	
9	Paris' Law	
10	Reference Units (Length, Force)	mm, tonne mm s^-2
11	Material Constant C	1.3E-10
12	Material Constant m	2.08

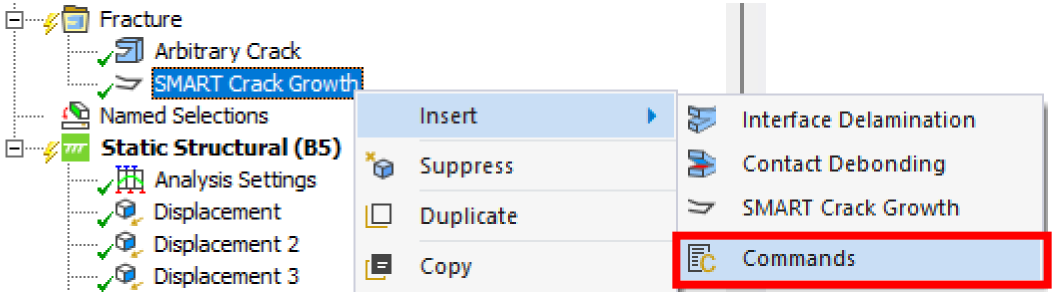
Crack-Closure

- SMART Crack Growth object by default only includes the Fatigue crack growth rate model. We must separately include the Crack-closure model in SMART.
- There are many crack-closure models available. We will be using the Polynomial function in this example.
- The Crack-Closure model modifies the Paris law into –

$$\frac{da}{dN} = \mathrm{C}(\Delta K_{eff})^m = C\,(\mathrm{U}\Delta K)^m$$

Crack-Closure

- To insert the Crack closure model into SMART
  - RMB on SMART Crack Growth > Insert > Commands



- Write the given MAPDL script into the commands window or import them from – **Crack Closure Commands.txt**

Commands

```
!   Active UNIT system in Workbench when this object was created:  Metric (mm, kg, N, s, mV, mA)
!   NOTE:  Any data that requires units (such as mass) is assumed to be in the consistent solver unit system.
!                See Solving Units in the help system for more information.

!   Commands inserted into this file will be executed just after the SMART crack growth definition.
!   The crack growth set id is "cgid"
!   Enter SOLU commands here


tb,cgcr,4,,3,upoly
tbdata,1,0.6,0.35,0.05

cgrow,fcop,mtab,4

```


Ansys

Crack-Closure

- After adding the MAPDL commands into SMART Crack Growth the fundamental crack Growth equation changes to the following equation-

$$\frac{da}{dN} = C\,(\mathrm{U}\Delta K)^m \qquad U = 0.6 + 0.35\,R + 0.05\,R^2$$

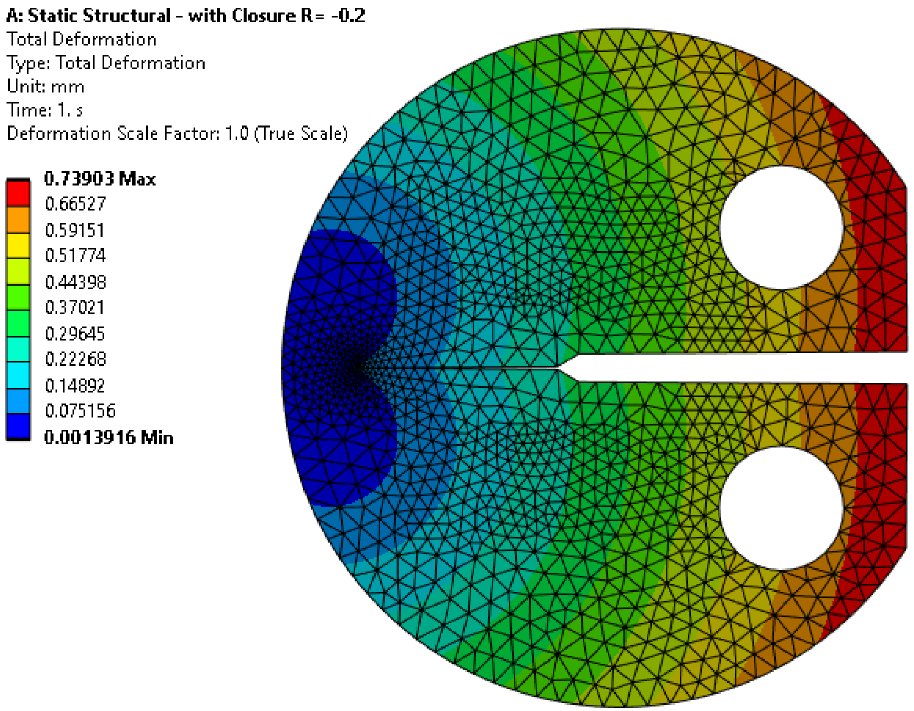
- The polynomial function is defined using the MAPDL command in lines 10 & 11.

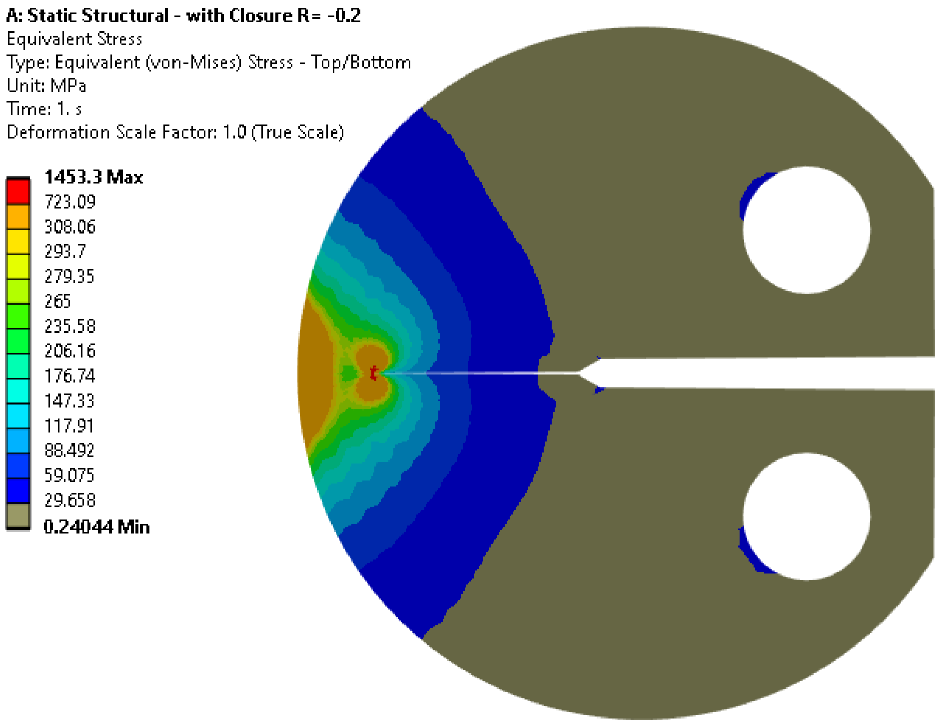
```
10  tb,cgcr,4,,3,upoly
11  tbdata,1,0.6,0.35,0.05
12
13  cgrow,fcop,mtab,4
14
```

4 is the Material ID
3 is the number of Coefficients in the Polynomial Function

Solution

- Solve the model
- Insert Total Deformation & Equivalent Stress (shown with increased # of contours and independent bands-bottom) results.
- We can see the crack growth in the model as well as the changing mesh as the crack grows.

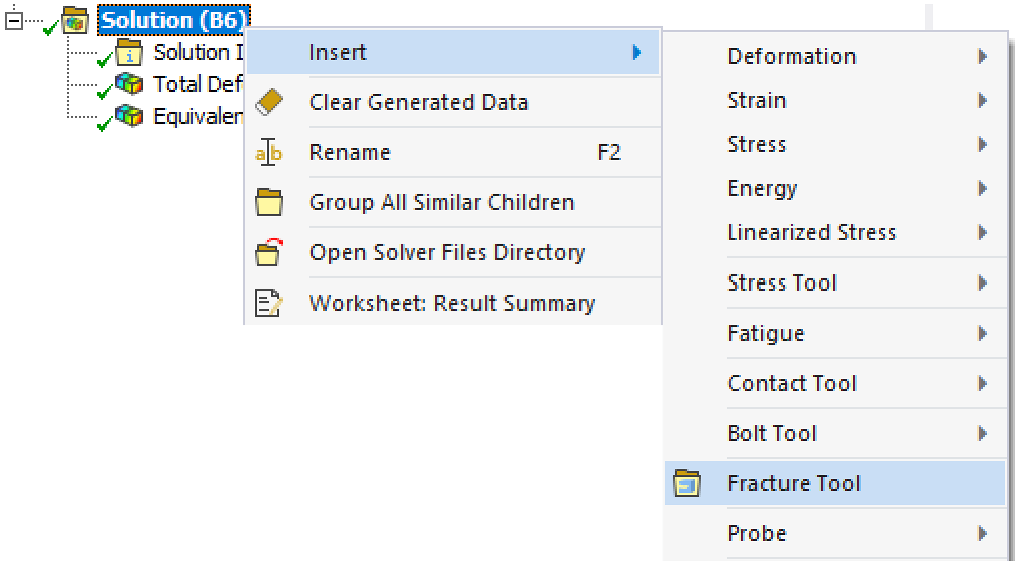


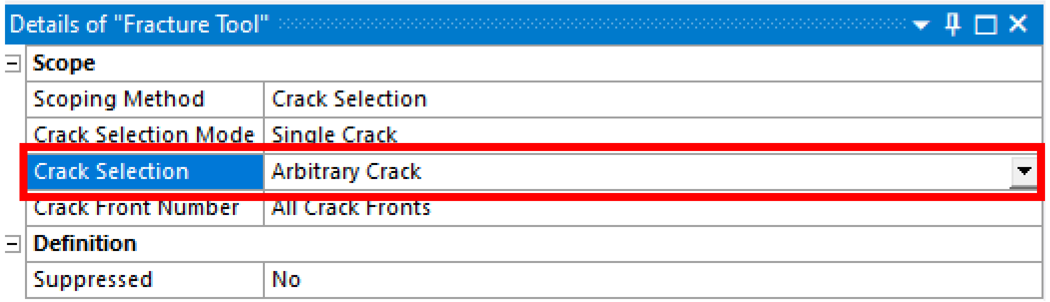


Fracture Tool

- Insert a Fracture Tool by :
  - RMB on Solution > Insert > Fracture Tool

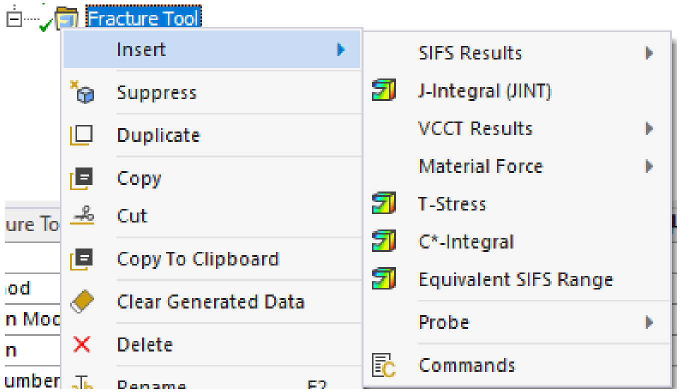
- Select the Arbitrary Crack as the Crack Selection.



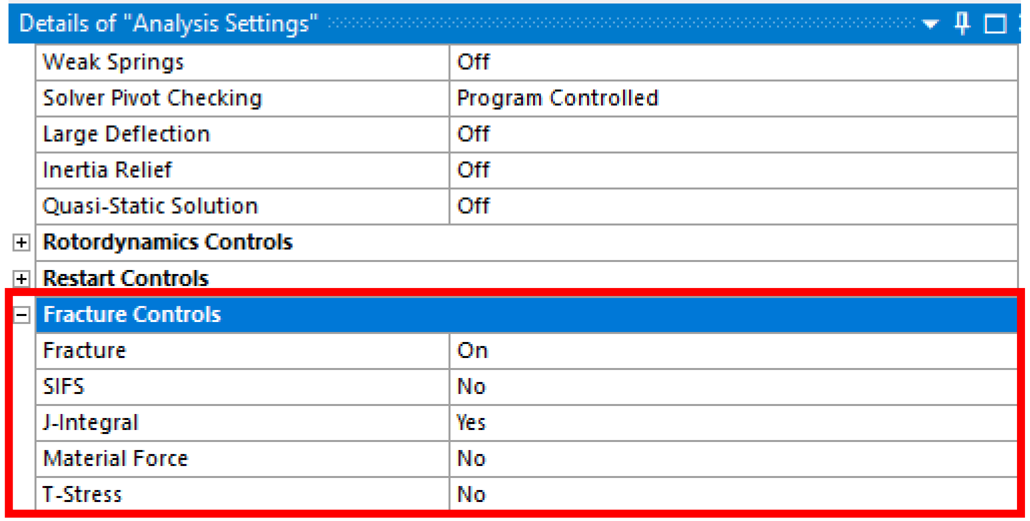


Fracture Tool

- Following Fracture Parameters results can be inserted using the Fracture Tool-
  - SIFS Results
  - J-Integral
  - VCCT Results
  - Material Force
  - T-Stress
  - C*-Integral
  - Equivalent SIFS Range



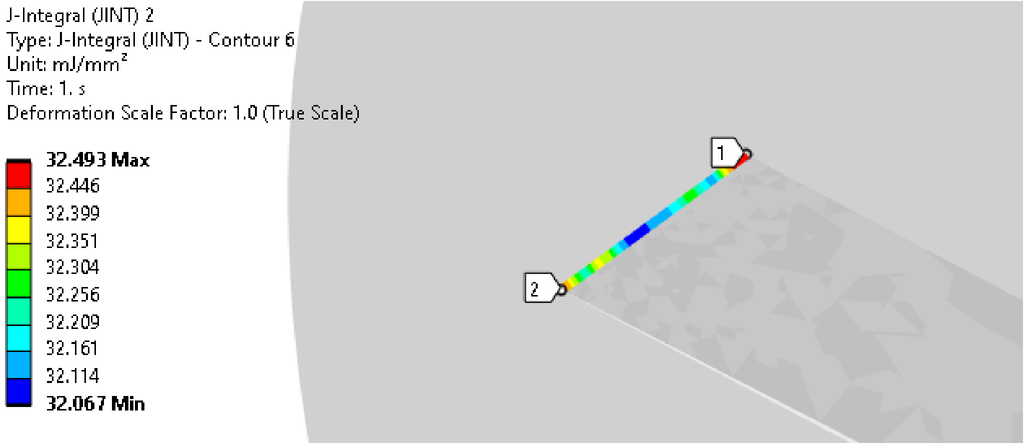
- But remember to request these results in Fracture Controls under Analysis Settings.



Note:- SIFS and J-Integral Results can't be requested simultaneously while using SMART Crack Growth.

Fracture Tool

- There are two types of results – Contour & Probe Results.
- Contour Results show the fracture parameter results along the crack front.



- Probe Results enable you to view the time history of a fracture parameter for a specific crack front node along the crack front only.

Tabular Data

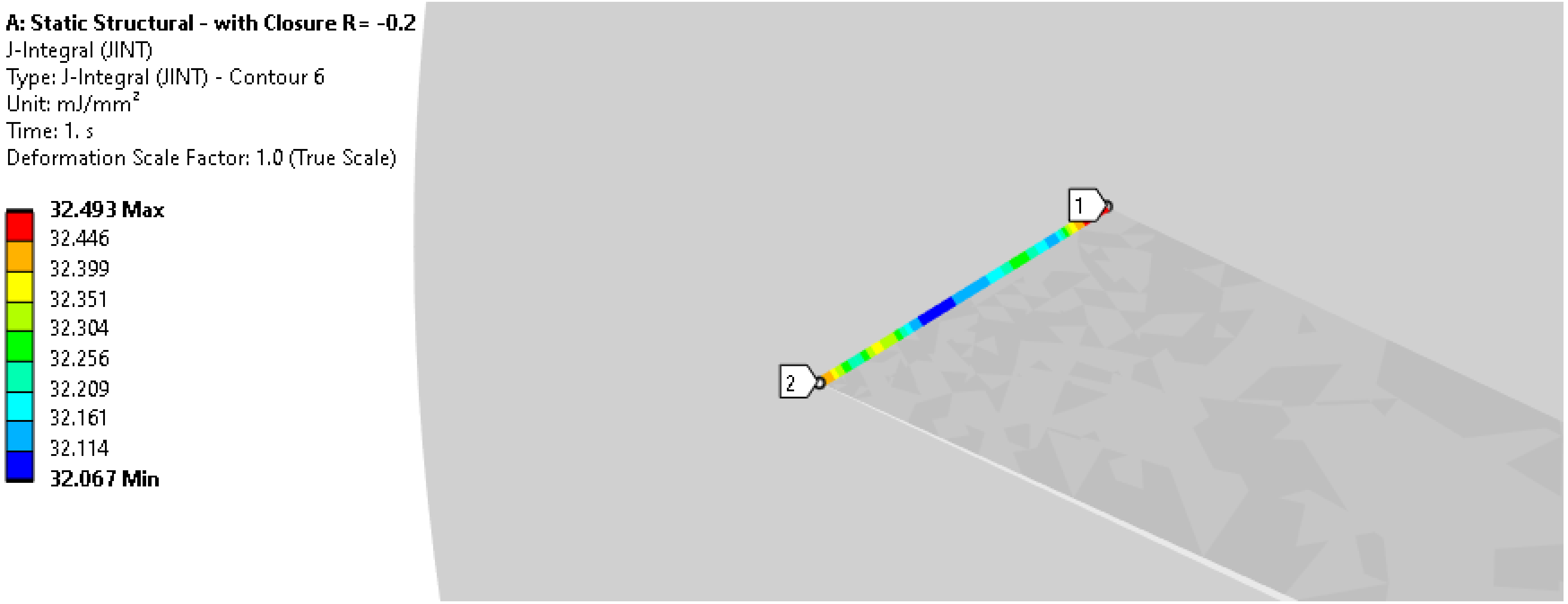
	Time [s]	Crack Extension Probe [mm]	Changed Mesh
1	3.3333e-002	1.3209	
2	6.6667e-002	2.6617	Yes
3	0.1	3.9944	Yes
4	0.13333	5.3152	Yes
5	0.16667	6.6276	Yes
6	0.2	7.9646	Yes
7	0.23333	9.2912	Yes
8	0.26667	10.638	Yes
9	0.3	11.986	Yes
10	0.33333	13.33	Yes
11	0.36667	14.671	Yes
12	0.4	16.001	Yes
13	0.43333	17.336	Yes
14	0.46667	18.659	Yes
15	0.5	19.978	Yes

Ansys

Observations

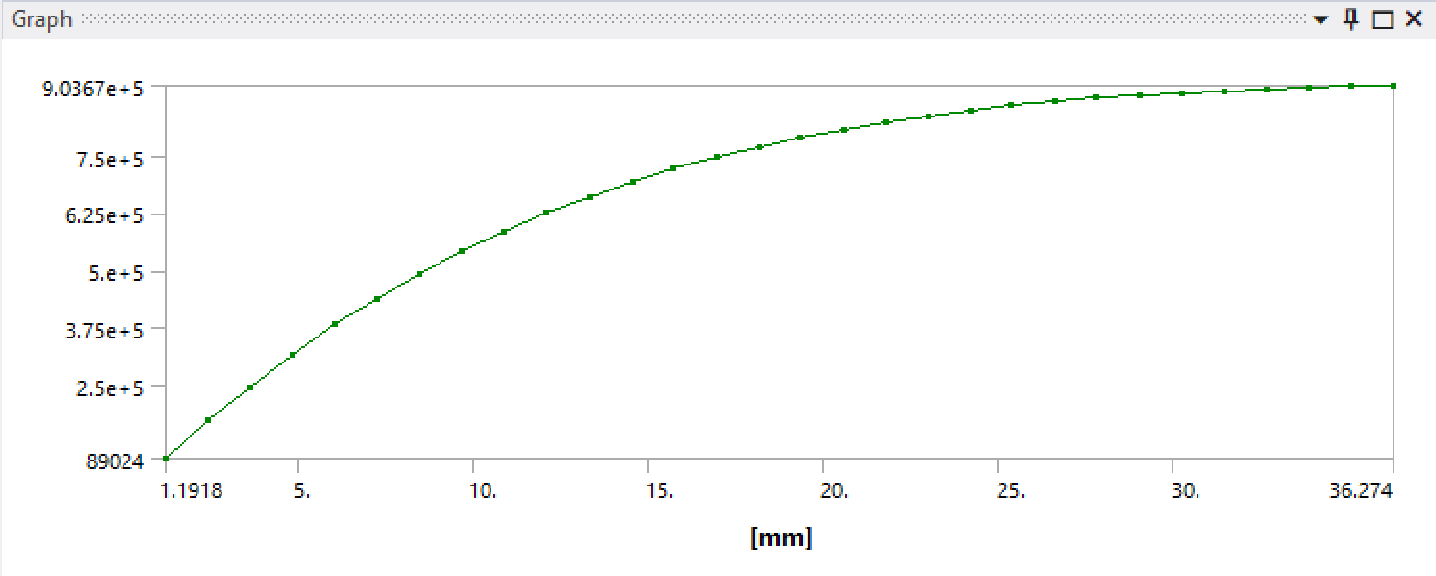
Results

- We can use the J-Integral results to understand about the Strain energy during fracture.
  - RMB on the Fracture Tool and insert J-Integral (JINT)
  - The J-integral represents the strain energy release rate, or Work (energy) per unit fracture surface area, in a material.



Results

- We can use the Chart option to plot the Total Number of Cycles Probe Vs Crack Extension Probe
  - RMB on the Fracture Tool and insert Crack Extension Probe and Total Number of Cycles Probe
  - Select both objects and create a Chart from them

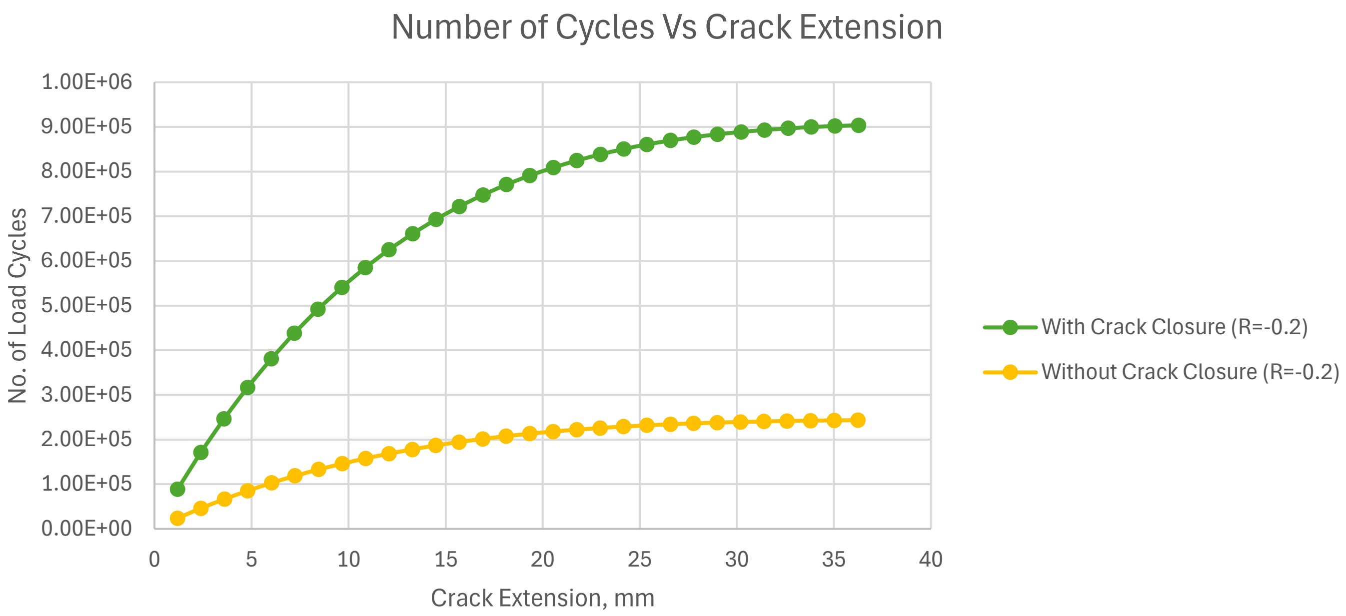


Tabular Data

	Steps	Time [s]	Crack Extension Probe [mm]	[A] Total Number of Cycles Probe
1	1	3.3333e-002	1.1918	89024
2	1	6.6667e-002	2.3993	1.7152e+005
3	1	0.1	3.5823	2.4601e+005
4	1	0.13333	4.8032	3.1677e+005
5	1	0.16667	6.0258	3.8088e+005
6	1	0.2	7.2138	4.3832e+005
7	1	0.23333	8.4371	4.9207e+005
8	1	0.26667	9.6574	5.4081e+005
9	1	0.3	10.867	5.8496e+005
10	1	0.33333	12.084	6.249e+005
11	1	0.36667	13.299	6.6073e+005
12	1	0.4	14.508	6.93e+005
13	1	0.43333	15.71	7.2171e+005
14	1	0.46667	16.93	7.4776e+005
15	1	0.5	18.134	7.7083e+005

Results

- Let's plot the No of Load Cycles Vs. Crack Extension for this model with Crack-closure and without Crack-closure:
  - Duplicate Schematic A on the Workbench Project Schematic and in Schematic B, suppress the Crack Closure commands object
  - We can see how using crack-closure drastically affects the No. of Load Cycles.
  - Crack-Closure helps in capturing a more realistic behavior of the model under fatigue crack growth.



This graph has been created in Excel

End of presentation

Appendix

Solution

- This is the Run Time & the Result File Size –
- The Run time and Result file size can be optimized by using the following script –
  - RMB on Static Structural > Insert > Commands **(Reduce Run time Commands.txt)**

Details of "Solution (B6)"

Solution	
Number Of Cores to Use (Beta)	Solve Process Settings
Adaptive Mesh Refinement	
Max Refinement Loops	1.
Refinement Depth	2.
Information	
Status	Done
MAPDL Elapsed Time	6 m 32 s
MAPDL Memory Used	2.5596 GB
MAPDL Result File Size	839.75 MB
Post Processing	
Distributed Post Processing (Beta)	Program Controlled
Mesh Source (Beta)	Program Controlled
Beam Section Results	No
On Demand Stress/Strain	No

Commands

```
!   Commands inserted into this file
!   These commands may supersede comm

!   Active UNIT system in Workbench w
!   NOTE:  Any data that requires uni
!                 See Solving Units in


! redefine results output to save rst
outres,erase
outres,all,none
outres,nsol,2,
outres,strs,2,
outres,cint,all

```

The Script saves results every 2 Substeps.

Solution

- The Simulation Run Time & Result File Size is reduced significantly after running the script-

Details of "Solution (A6)"

Solution	
Number Of Cores to Use (Beta)	Solve Process Settings
Adaptive Mesh Refinement	
Max Refinement Loops	1.
Refinement Depth	2.
Information	
Status	Done
MAPDL Elapsed Time	4 m 45 s
MAPDL Memory Used	1.3652 GB
MAPDL Result File Size	275.56 MB
Post Processing	
Distributed Post Processing (Beta)	Program Controlled
Mesh Source (Beta)	Program Controlled
Beam Section Results	No
On Demand Stress/Strain	No

We saved around 2 min in Run Time and 500 MB in file size. Your results may vary.

- We can see that the results are saved after every 2 substeps.

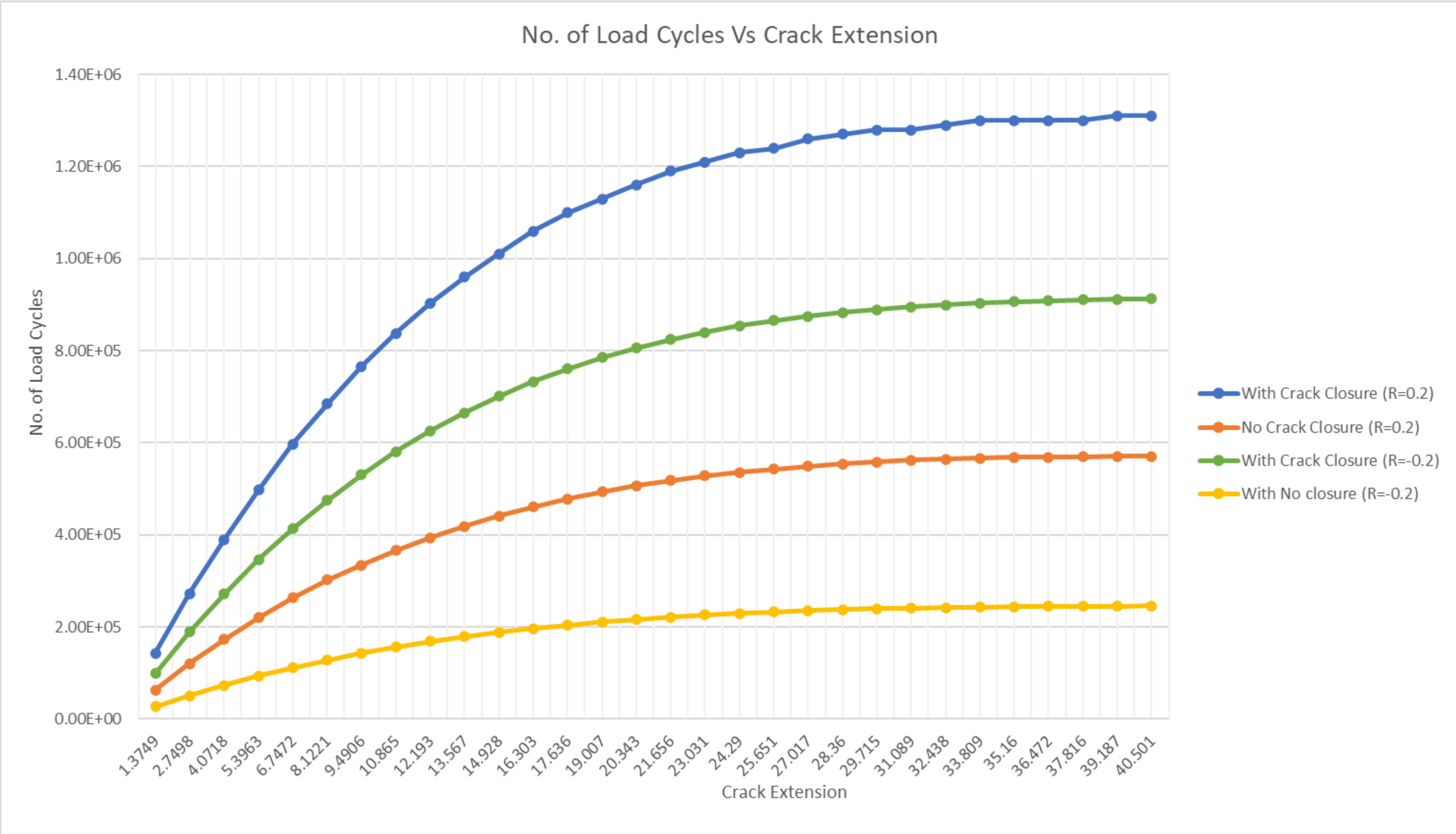
Tabular Data

	Time [s]	Minimum [mm]	Maximum [mm]	Average [mm]	Changed Mesh
1	3.3333e-002				
2	6.6667e-002	1.3373e-005	5.5039e-002	1.4366e-002	Yes
3	0.1				
4	0.13333	2.2288e-005	6.2925e-002	1.7012e-002	Yes
5	0.16667				
6	0.2	4.7271e-005	7.242e-002	2.0241e-002	Yes
7	0.23333				
8	0.26667	7.8304e-005	8.3757e-002	2.4246e-002	Yes
9	0.3				
10	0.33333	8.1713e-005	9.7643e-002	2.9045e-002	Yes
11	0.36667				
12	0.4	1.4387e-004	0.11503	3.5093e-002	Yes
13	0.43333				
14	0.46667	2.1892e-004	0.13677	4.2848e-002	Yes
15	0.5				
[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]

Crack Closure with R = + 0.2 and – 0.2

- The Crack Closure works even when the R-value is positive, it's just that the extent of the closure is not as big as when we have a negative R-value.

- Please refer to the graph on slide 10.



This graph has been created in Excel

New Feature in 24R1

- In the latest release SMART Crack Growth got a new feature called Stop Criterion -
  - We can enter a stop criterion to stop the crack growth simulation.
  - The simulation of crack growth is terminated when the entered criterion is met.
  - Following are the criteria available
    - Max Crack Extension
    - Free Boundary
    - Max Stress Intensity Factor
    - Max Total Number of Cycles