

# Fracture Analysis of Compressor Disk

Ch Nithin Chakravarthy  
Mechanical Department,  
Chaitanya Bharathi Institute of Technology,  
Hyderabad, India

K Ashok,  
Mechanical Department, CBIT.

Dr. A. Venugopal Rao  
Scientist G, DMRL

B. Anil Kumar  
Mechanical Department,  
Aurora's Technological and Research Institute.

Dr. P. Ramalakshmi,  
Mechanical department, CBIT.  
Hyderabad, India

**Abstract**— In this paper, stress intensity factor  $SIF_s$  ( $K_I$ ) and J-integral (energy distribution) for a compressor disk in a gas turbine engine is analyzed by using ansys workbench 19.2. Compressor disk is fabricated by titanium alloy and surface crack occurs between the disk and blade attachment. Damage accumulation of a turbine blade is subjected to static load. FEA is used for the analysis and prevention of operation failures of a compressor disk. Finite element analysis is carried out to study the stresses and strains for the basic configurations of a compressor disk and their operational damage. Proposed design modifications and repair technologies to existing in-service compressor disk is analyzed.

**Keywords**:- Finite element analysis, stress intensity factor and J-integral.

## I. INTRODUCTION

Fracture failures of rotating compressor disk with turbine blade of aircraft engines were detected. Their fracture was the result of crack initiation and propagation until a critical crack length has been reached. In all of these failures, crack propagation started from a part-through surface crack defect. Initiation of cracks in compressor disks made from two-phase titanium alloy has occurred in a disk and blade attachment (Figure 1.1(a), 1.1(b)). This attachment of a dovetail type has the  $26^\circ$  broach angle at the axis of rotation. Quarter-ellipse corner cracks developed in slot fillets under the blades near the disk outer surface. Damage accumulation and growth for turbine disks made from steel took place on the inner surface of hole in a hub of wheel (Figure 1.1b). In this case the crack front was nearly half-elliptical shape.

## II. ABBREVIATIONS AND ACRONYMS

- $SIFs$  ( $K_I$ ): Stress intensity factor
- JINT: J-integral (energy distribution)
- $U_x$   $U_y$  &  $U_z$ : Total deformation
- GUI tabs: Set of analysis systems for solving the problems
- CFD: Computational fluid dynamics
- CAD: Computer aided design
- Pie-cut shape: This shape is modeled to see the stress distributions for the disk

- $R=1.2$  and  $R=2$ : These are the radius of the cross-sections at the U-tip of the disk
- CAM: Computer aided manufacturing
- CAFEA&CAFEM: Computer aided finite element analysis and modeling

## III. AUTHORS AND AFFILIATIONS

[1] V.N. Shlyannikov, B.V. Iltchenko, N.V. Stepanov conducted experiment on “**Fracture analysis of turbine disks and computational experimental background of the operational decisions**” Fracture failures of rotating compressor disk and turbine blade of civil aircraft engines were detected. Their fracture was the result of crack initiation and propagation until a critical crack length has been reached. In all of these failures, crack propagation started from a part-through surface crack defect.

[2] Newman JC, Raju examined on “**Stress intensity factor equations for cracks in three-dimensional-finite bodies**” The 2D solution by Newman and Raju to estimate the stress intensity factors (SIF) for corner quarter-elliptical and half-elliptical part through surface cracks with complex configuration in variable stress fields has been shown to be quite useful.

## IV. PROBLEM STATEMENT AND DESIGN OF DISK

The statement involves changes in geometries of compressor disc, and FEA is used to find the solution which involves

- 1) Stress-strain analysis and
- 2) Fracture parameters i.e. stress intensity factor and J-integral.

### 4.1 Changes in geometries of Compressor disk:-

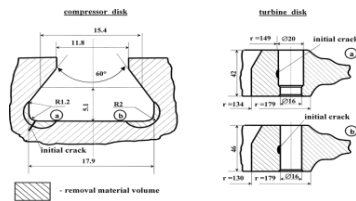


Fig 1.2 (a) Basic configurations of compressor disk and turbine blade.

(b) Changed configurations of compressor disk and turbine blade.

The modification for compressor disk consisted in magnification of coupling radius between a contact surface and slot bottom up to 2 mm is shown in Figure 1.2(a), (b).

This variant was recommended as a method of repair technology for deleting volume of a material containing surface microdefects.

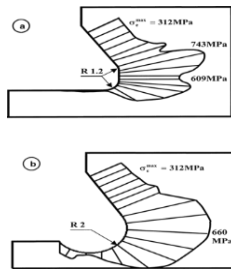


Fig 1.3 2D effective stress distributions for (a) basic and (b) changed geometry of compressor disk.

#### 4.2 Finite-element analysis of Compressor disk:-

The initial step is full 3D analysis of the basic configuration. Appropriate loads and displacements are selected to be used as boundary conditions for 2D parametric study.

Secondly 3D analysis was conducted on a design incorporating the improved contour. This design also included a more generous fillet radius to achieve better stress distribution along the broach.

Throughout this part of the numerical study, the primary design criterion was the maximum effective stress being defined.

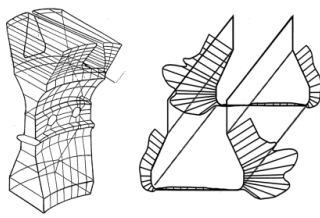


Fig 1.4 A typical section of compressor disk (a) and 3D effective stress distributions (b) for the basic configuration at a dovetail slot.

The geometric modeling process design is to be modeled into a few natural blocks with each block containing several finite elements.

The blocks are defined by one or more plane sections with interpolation employed to develop a three dimensional block from the plane sections.

A model generated by this procedure is shown in Figure 1.3. A single disk segment was modeled.

The “pie-cut” shaped disk segment has complex boundary surfaces formed by radial lines originating at the axis of rotation and following the 26 degree broach angle at the rim.

#### 4.3 Fracture parameters:-

#### 1) Stress intensity factor:

The stress intensity factor, K, along the crack front of a part-through surface crack emanating from hole and slot fillet is a complex 3D problem.

Numerical as well as analytical analysis of the 3D crack problems are, of course, much more difficult than analysis of 2D problems.

#### 2) J-integral:

This approach involves the crack which is taken place in the compressor disk at radius R=1.2 the stress distribution at the crack tip is about 723MPa and this is done as part of experimental data.

So the change in the geometry in compressor disk of radius R=2 at this slot the stresses can be reduced to 14% as compared with basic configuration.

#### 4.4 DESIGNING PROCESS OF COMPRESSOR:

For creating anybody we have to open the Solid works software from the desktop. In solid works window, go to the file and select new. Then it will show the module called part, assembly and drafting Or drawing. Select part and select ok.

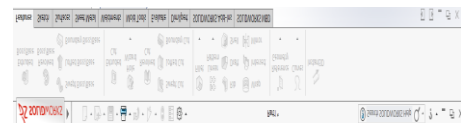


Fig 3.2 Tool box ribbon

#### 4.4.1 Compressor disk:

To create compressor go to sketch and select sketch tool then select top view, then by using sketch tools called line, circle, rectangles, etc. We created the profile shown in below with the dimension by using smart dimensions tool.

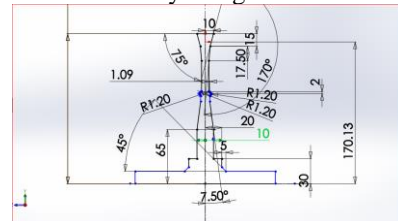


Fig 3.3 Sketch of compressor

Go to sketch and select sketch tool then select top view. Then by using sketch tools called line, circle, rectangles, etc. We created the profile shown in below with the dimension by using smart dimensions tool.



Fig 3.5 Sketch

Later go to the extrude cut boss/base tool and select the above profile as a extrude profile and specify the up to next in direction 1 and select merge results as shown in figure.

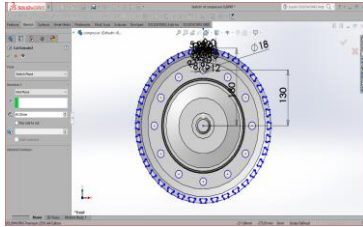


Fig 3.6 Extrude cut boss

#### 4.4.2 Turbine blade:

Go to sketch and select sketch tool then select top view. Then by using sketch tools called line, circle, rectangles, etc. We created the profile shown in below with the dimension by using smart dimensions tool.

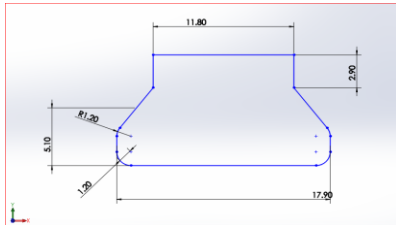


Fig 3.7 Sketch

Go to sketch and select sketch tool then select top view. Then by using sketch tools called line, circle, rectangles, etc. We created the profile shown in below with the dimension by using smart dimensions tool.

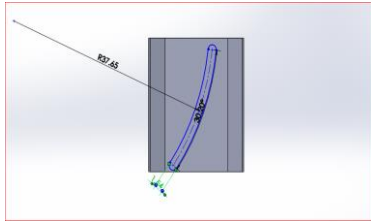


Fig 3.9 Sketch

Later go to the extrude boss/base tool and select the above profile as a extrude profile and specify the up to next in direction 1 and select merge results as shown in figure.

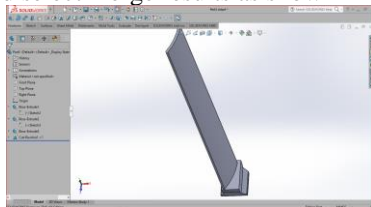


Fig 3.10 Extrude boss

#### 4.5 Assembly of compressor disc and turbine blade:-

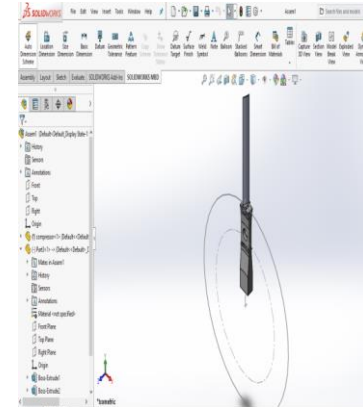
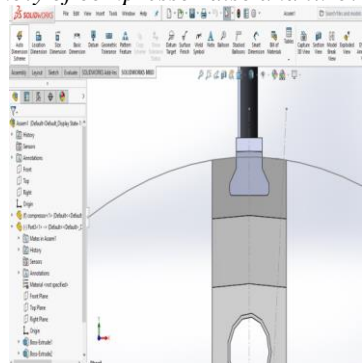


Fig 3.11 Assembly of model 1 at radius=1.2

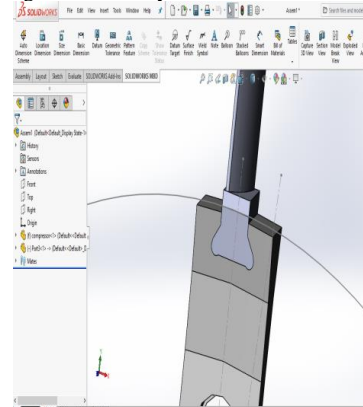


Fig 3.12 Assembly of model 2 at radius R=2

#### 4.6 Drafting:

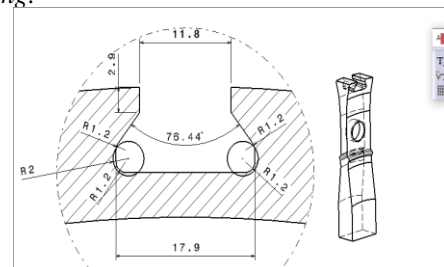


Fig 3.13 Drafting for modal 1

So in this project to the analysis for the compressor and turbine in gas turbines will be taken as:

Name of the analysis

- 1 Explicit dynamics
- 2 Static structural

## Name of loads

- 1 Loads with respect to time
- 2 Static load conditions

## Solution determines

- 1 Total deformation
- 2 Deformation, stresses, strains, fatigue tool, life, damage and safety factor.

## 1 EXPLICIT DYNAMICS:

### Introduction

The Explicit Dynamics analysis is one of the important analyses in ANSYS Workbench. It is available as Explicit Dynamics analysis system under the Analysis System toolbox in the Toolbox window, this system analyses the structural components for displacements (deformation), stresses, strains, and pressures under different loading conditions.

The loads in this analysis system are assumed not to have damping characteristics (time dependent). To start a new Explicit Dynamics analysis system, double-click on Explicit Dynamics in the Analysis Systems toolbox in the Toolbox window; the Explicit Dynamics analysis system will be added to the Project Schematic window.

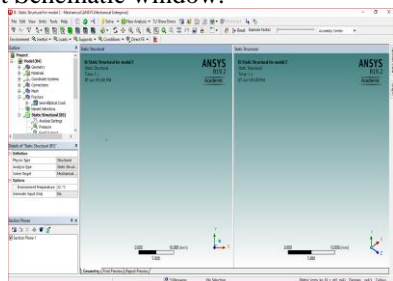


Fig 4.4 The mechanical window

## The Composite Material and General Materials

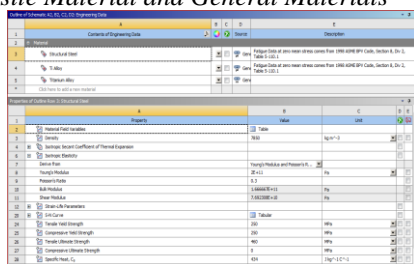


Fig 4.6 Specifying a titanium material

## Generating the Mesh

After the model is created in the Design Modeler window, you need to generate the mesh for the model in the Mechanical window.

- Choose the Generate Mesh tool from the Mesh drop-down in the Mesh contextual toolbar; the mesh is generated shown in figure 4.7.

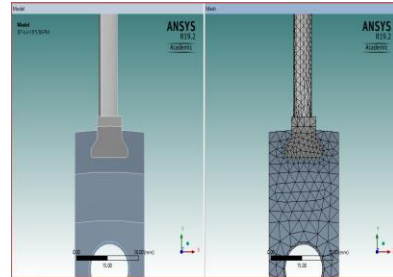


Fig 4.7 Mesh generated with default mesh controls

## Specifying the Boundary Conditions

After you mesh the model, it is required to specify the boundary and loading conditions.

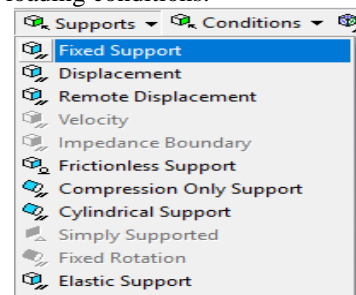


Fig 4.8 Choosing the fixed support tool from the supports drop-down

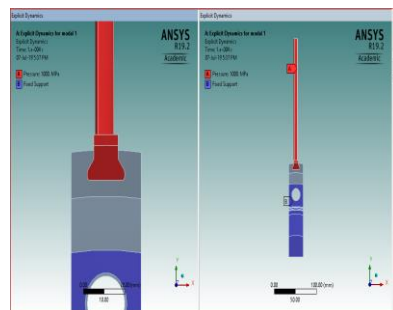


Fig 4.10 The violet and red color face of the model displaying the fixed support and pressures

## Solving the FE Model and Analyzing the Results:

After the boundary and load conditions are specified for the model, you need to solve the analysis. After solving, you will get the Total and Directional Deformations due to the given condition. Also, you will get Equivalent Stress, life, and damage.

## 2 STATIC STRUCTURAL ANALYSES:

### Introduction

The Static Structural analysis is one of the important analyses in ANSYS Workbench. It is available as Static Structural analysis system under the Analysis System toolbox in the Toolbox window, This system analyses the structural components for displacements (deformation), stresses, strains, and pressures under different loading conditions.

The loads in this analysis system are assumed not to have damping characteristics (time dependent). Steady loading and damping conditions are assumed in this type of analysis system.



### Generating the Mesh

After the model is created in the Design Modeler window, you need to generate the mesh for the model in the Mechanical window.

- Choose the Generate Mesh tool from the Mesh drop-down in the Mesh contextual toolbar; the mesh is generated.

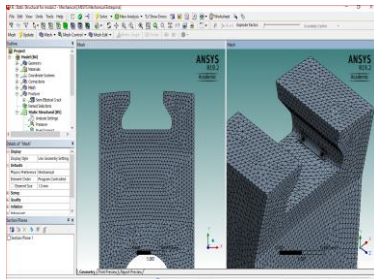


Fig 4.15 Mesh generated with default mesh controls

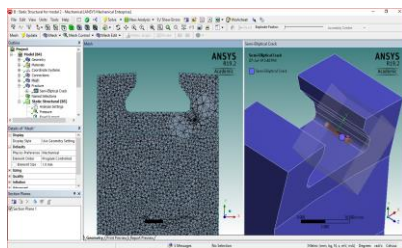


Fig 4.16 Mesh crack zone

### Specifying the Boundary Conditions:

After you mesh the model, it is required to specify the boundary and loading conditions

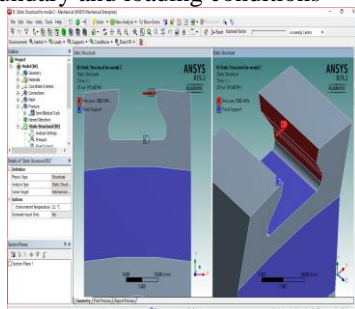


Fig 4.17 The violet and red color face of the model displaying the fixed support and pressures

### Solving the FE Model and Analyzing the Results:

After the boundary and load conditions are specified for the model, you need to solve the analysis. After solving, you will get the Total and Directional Deformations due to the given condition. Also, you will get Equivalent Stress, life, and damage.

## V RESULTS AND DISCUSSIONS

### 5.1 Total Deformation:

#### 5.1.1 Titanium alloy for modal 1:

For explicit dynamics analysis

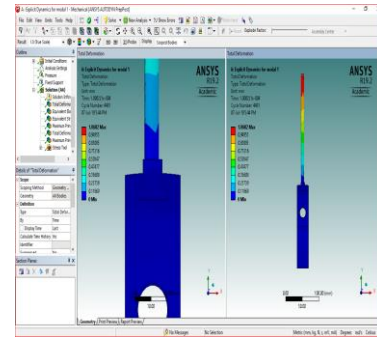


Fig 5.1 Total deformation for the model 1(radius 1.2)

In model 1 at  $R=1.2$  within the load application of pressure on the blade so stress zones are seen in between disk and blade i.e at the upper surface of disk.

The red zone indicates the maximum load of 1.0682 mm, which has taken place at the U-tip of disk.

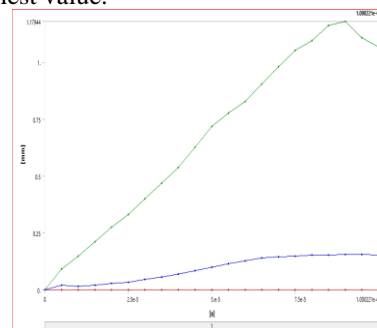
The other colors i.e. green, sky blue, yellow indicates at the upper part of disk (expanding with respect to time).

At last blue area indicates no load takes place i.e. 0.

	Time [s]	Minimum [mm]	Maximum [mm]	Average [mm]
1	1.175404e-038	0	9.071572e-002	2.003002e-002
2	5.022091e-006	0	0.1467087	1.568236e-002
3	1.000962e-005	0	0.2119239	2.022683e-002
4	1.500259e-005	0	0.2750567	2.847657e-002
5	2.001727e-005	0	0.3319598	3.376733e-002
6	2.500133e-005	0	0.3990918	4.610453e-002
7	3.002081e-005	0	0.4686317	5.706886e-002
8	3.50026e-005	0	0.5362404	6.991074e-002
9	4.001934e-005	0	0.6256528	8.379069e-002
10	4.502206e-005	0	0.7117302	0.1002179
11	5.00203e-005	0	0.7773978	0.1143298
12	5.501214e-005	0	0.827623	0.1274322
13	6.001126e-005	0	0.9033666	0.1390979
14	6.500488e-005	0	0.981828	0.1446668
15	7.002141e-005	0	1.053148	0.1483796
16	7.501191e-005	0	1.095741	0.1532254
17	8.000394e-005	0	1.161836	0.1537375
18	8.501555e-005	0	1.17944	0.1554882
19	9.000985e-005	0	1.107269	0.1563545
20	9.502198e-005	0	1.068241	0.152866
21	1.000221e-004	0		

Table 5.1: The max&min values of total deformation taking place on the model 1 at radius  $R=1.2$

The table involves the maximum and minimum deformation of the model 1 at radius  $R=1.2$  with the change in time and to plot a graph. The ratios of 0.5s are taken b/w graph to get the required highest value.



Graph 5.1: The graph shows between Time(s) and maximum & minimum deformation for model 1( $R=1.2$ )

The above graph explains maximum values at ratios of 0.5s are taken and up to 1.002e-4 the increase is seen and the value at 1.07mm the decrease in time taken place i.e. 0.5e-01 thier the deformation value decreased because the tip is flat and sudden decrease is seen.

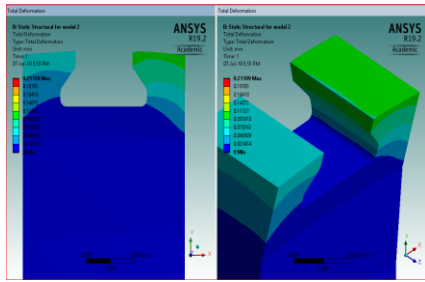


Fig 5.2 Total deformation obtained from the legend display in color bands for model 1(R=1.2)

Here it shows the bands at the U-tip it is high as 0.21 mm and other colors shown decrease in value at that area so model 1 is taken place.

#### 5.1.2 Titanium alloy for modal 2:

For explicit dynamics analysis

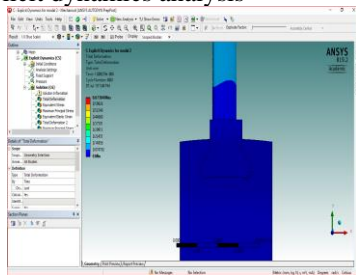


Fig 5.3 Total deformation for the model 2(r=2)

Then for R=2 the cross section changes so when the load is taken at that U-tip there is no much difference in shape so with small changes the deformation is shown.

At Red zone i.e. at the right side of the U-tip as 0.6730407 mm. is the critical zone which states that at this boundary condition i.e. fixed at upper part of disk. The load directly falls at U-tip at that zone.

At yellow, sky blue and green zones takes place at near to tip.

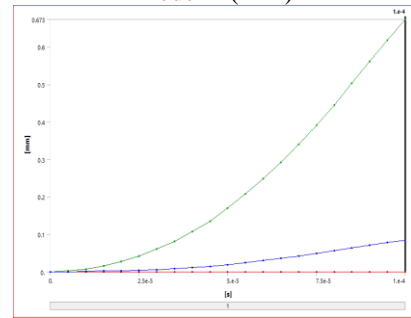
At blue zone there is no change in shape i.e 0.

Table 5.2: The max&min values of total deformation taking place for model 2 at radius (R=2)

Time [s]	Minimum [mm]	Maximum [mm]	Average [mm]
1 1.175494e-038	0.	2.658316e-003	6.665527e-004
2 5.010517e-006	0.	7.236466e-003	1.33279e-003
3 1.001557e-005	0.	1.597049e-002	2.297216e-003
4 1.502059e-005	0.	2.712481e-002	3.291305e-003
5 2.00072e-005	0.	4.237769e-002	4.765985e-003
6 2.50054e-005	0.	6.033407e-002	6.479562e-003
7 3.000985e-005	0.	8.203827e-002	8.807687e-003
8 3.501424e-005	0.	1.069102	1.175131e-002
9 4.001846e-005	0.	0.1356815	1.525811e-002
10 4.502245e-005	0.	0.1694851	1.956345e-002
11 5.00014e-005	0.	0.207434	2.4535e-002
12 5.500427e-005	0.	0.2488115	3.008479e-002
13 6.000647e-005	0.	0.2926411	3.611182e-002
14 6.500803e-005	0.	0.339985	4.25391e-002
15 7.00089e-005	0.	0.390777	4.930266e-002
16 7.500913e-005	0.	0.4450614	5.633534e-002
17 8.000865e-005	0.	0.502393	6.352764e-002
18 8.500742e-005	0.	0.5605618	7.071333e-002
19 9.000544e-005	0.	0.6183272	7.795908e-002
20 9.50027e-005	0.	0.6730407	8.501381e-002
21 1.000239e-004	0.		

The table involves the max&min values of deformation for the model 2 at R=2 with respect to time as plotted in graph. The ratios were taken as 0.5s.

Graph 5.2: Time and maximum & minimum deformation for model 2 (R=2)



Here in above graph because the change in radius R=2 is taken and there is constant increase in values at ratio 0.5s and the tip changed to smooth surface and the value is 1e-0.

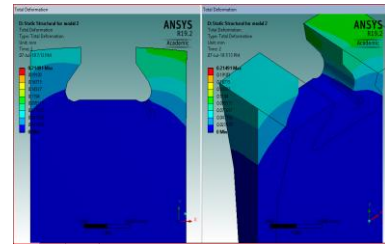


Fig 5.4 Total deformation obtained from the legend display in color bands for model 2 (R=2)

In this required region within the change in cross section at U-tip so there is no much difference in shape and the expansion takes place with less time.

#### 5.2 Equivalent stress

In the Details of "equivalent stress" window, expand the Results node, if it is not already expanded. Note that the maximum and minimum stresses displayed respectively.

#### 5.2.1 Titanium alloy for modal 1:

For explicit dynamics analysis

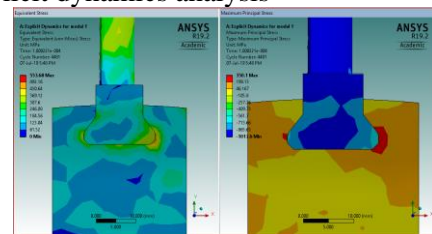


Fig 5.5 Equivalent stress for model 1(R=1.2)

Here when the load applied on the blade the resultant stress is shown. So at the tip were the cross section is flat at that region the stress is high up to 553 MPa. So the expansion takes place high so change as seen in upper part of disk.

At red zone: this zone indicates that stresses are high and at that surface is start to damage the material.

At orange, yellow and green zone indicates the value of stress is near to the U-tip.

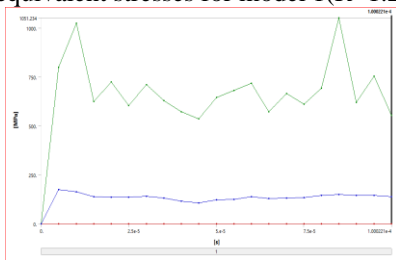
At blue zone represents there is no load i.e. 0.

Table 5.3: Table shows the max&min values of equivalent stresses for model 1 (R=1.2):

Time [s]	Minimum [MPa]	Maximum [MPa]	Average [MPa]
1 1.175494e-038	0.	798.4502	174.6028
2 5.022091e-006	0.	1023.061	163.3258
3 1.000962e-005	0.	624.7929	139.0319
4 1.500219e-005	0.	724.7954	135.5454
5 2.001727e-005	0.	603.3276	135.5769
6 2.500133e-005	0.	711.2367	139.9102
7 3.002081e-005	0.	629.425	132.0358
8 3.50026e-005	0.	571.0331	115.8625
9 4.001934e-005	0.	535.7581	107.7567
10 4.500259e-005	0.	644.5247	122.3273
11 5.00203e-005	0.	680.5835	124.9159
12 5.501214e-005	0.	716.9439	138.8675
13 6.001125e-005	0.	571.1414	128.4951
14 6.500406e-005	0.	665.3647	132.7214
15 7.002141e-005	0.	610.7561	133.565
16 7.501191e-005	0.	691.3971	145.3472
17 8.000394e-005	0.	1051.224	150.0433
18 8.501555e-005	0.	619.7263	146.242
19 9.000985e-005	0.	733.94	145.0637
20 9.502198e-005	0.	553.6799	137.9111
21 1.000221e-004	0.		

Table shows the required values of max&min values of equivalent stress for model 1 (=1.2) and graph is plotted between time and stresses. And the ratios are taken as 0.5s.

Graph 5.3: The graph shows between Time and Max & Min equivalent stresses for model 1(R=1.2)



The above graph shows the stress points within that concentration at R=1.2 the sudden decrease and increase takes place because the surface defects at that area is more because of not removal of volume of material.

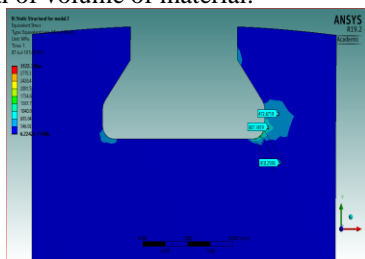


Fig 5.6 Equivalent stress obtained from the legend display in color bands for model 1(R=1.2)

Here at the tip which is flat shape i.e. at that region the max stress of 3122.3 MPa takes place because it is the lowest stress to take to predict some extent of crack at that region.

## 5.2.2 Titanium alloy for modal 2

For explicit dynamics analysis

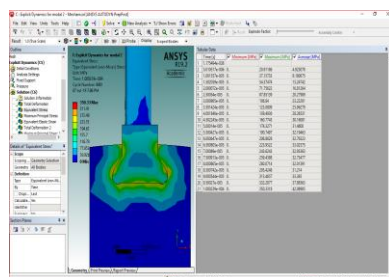


Fig 5.7 Equivalent stress for model 2(R=2)

The table shown as maximum and minimum values at required time which is taken as ratio of 0.5s. So the graph is plotted between stress and time.

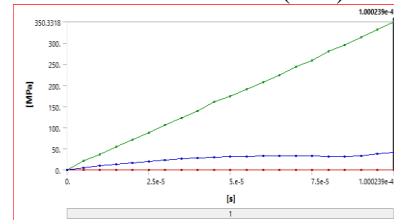
Here within change in cross section of radius R=2 the stresses takes place at U-tip is almost same which is 350.33 MPa so the stresses takes place at that region is near to it.

At Red zone: It takes place throughout the cross section so the load is distributed in that region.

At orange, yellow and green zones the same condition takes place throughout the cross section.

At blue zone there is no stress i.e. 0.

Graph 5.4: The graph shows between time and max & min stresses for model 2(R=2)



Here in the above graph because of change in radius as R=2 so the surface is smooth and there are no material defects so the stress points in that concentrated area is almost same. So within the change in ratio there is instant increase up to 350.33Mpa.

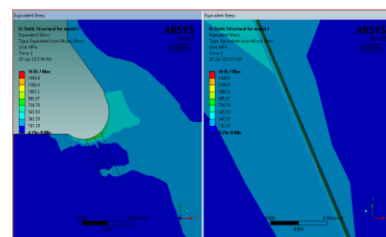


Fig 5.8 Equivalent stress obtained from the legend display in color bands for model 2(R=2)

Here the color bands shown are at U-tip the stress value is reduced i.e. 1630.7 MPa this is because of change in shape.

## 5.3 Strain

In the Details of “strain” window, expand the Results node, if it is not already expanded. Note that the maximum and minimum strains displayed respectively.

### 5.3.1 Titanium alloy for modal 1

For explicit dynamics analysis

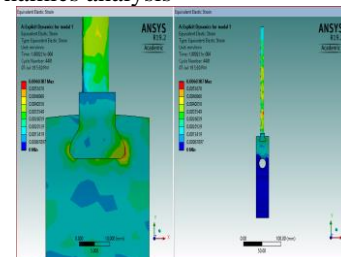


Fig 5.9 Equivalent strain for model 1(R=1.2)

So here at the U-tip region the place at which it cannot regain to that position because of plasticity. So change is seen in the model.

At red zone: It takes place at the two regions of upper part of disk were the plasticity at that region is not regained under load.

At orange, yellow and green zones the values are near to the red zone within change in ratio 0.5s.

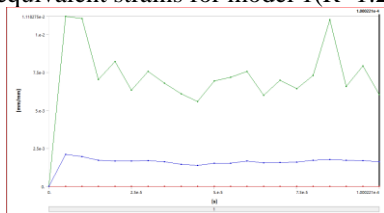
At blue zone there is no strain i.e. 0

Table 5.4: Table shows the max&min strain of the model 1 at R=1.2 with respect to the time and plotted a graph

Time [s]	Minimum (mm/mm)	Maximum (mm/mm)	Average (mm/mm)
1	1.175484e-003	1.118275e-002	2.113016e-003
2	5.022091e-006	1.102949e-002	1.949975e-003
3	1.000962e-005	7.031853e-003	1.707875e-003
4	1.500259e-005	8.20160e-003	1.666701e-003
5	2.001727e-005	6.334615e-003	1.657216e-003
6	2.500133e-005	7.555399e-003	1.686275e-003
7	3.000081e-005	6.795554e-003	1.613529e-003
8	3.50026e-005	6.078211e-003	1.452586e-003
9	4.001934e-005	5.580814e-003	1.382454e-003
10	4.502206e-005	6.921335e-003	1.511242e-003
11	5.002039e-005	7.169696e-003	1.530294e-003
12	5.501214e-005	7.565581e-003	1.655932e-003
13	6.001126e-005	5.982385e-003	1.554676e-003
14	6.500488e-005	6.974934e-003	1.581011e-003
15	7.002141e-005	6.435236e-003	1.590483e-003
16	7.501191e-005	7.301869e-003	1.713215e-003
17	8.000394e-005	1.097171e-002	1.761605e-003
18	8.501555e-005	6.567613e-003	1.710457e-003
19	9.000985e-005	7.812699e-003	1.681004e-003
20	9.502198e-005	6.03874e-003	1.679698e-003
21	1.000221e-004		

The table involves the max&min strains taking place for the model 1 at R=1.2 so at max the value is 6.03874e-003.

Graph 5.5: The graph shows between the time and max & min equivalent strains for model 1(R=1.2)



The graph shows the strain values at change in time of ratio 0.5s so here at R=1.2 the cross section at tip is flat at some region and some is smooth so the plasticity takes place at smooth region increases up to begin of flat area from there increase and decrease behaviour is seen at end of flat area. And again smooth at other end. So when values are plotted there is sudden increase and decrease are seen at that flat region.

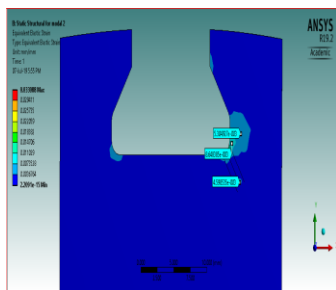


Fig 5.10 Strain obtained from the legend display in color bands for model 1(R=1.2)

Here because at the U-tip the plasticity regain at that region is almost near which is seen in the ratio of values.

### 5.3.2 Titanium alloy for modal 2

For explicit dynamics analysis

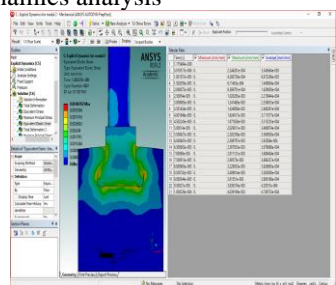


Fig 5.11 Equivalent strain for model 2(R=2)

The table shows the values of max and min strains with change in time as ratio of 0.5s. So the values are taken with respect to time and graph is plotted.

Here the strain is almost same at U-tip region i.e. regain of plasticity is near as seen in the ratios.

At red zone: this indicates throughout the cross section of disk the load takes place is same.

At orange, yellow and green zones is same near to the red zone value.

At blue zone there is no strain i.e. 0

Graph 5.6: The graph shows between time and max & min equivalent strains for model 2(R=2)

The above graph shows the constant increase in the plot because the radius at R=2 there is smooth surface with U-tip with no change in cross section, so the plasticity at the tip is constant. And with change in ratio the values are plotted, and increase is seen up to 4.039mm/mm.

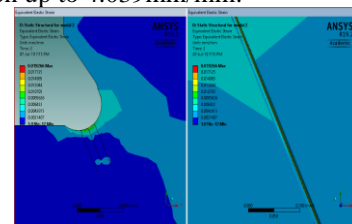


Fig 5.12 Equivalent strain obtained from the legend display in color bands for model 2(R=2)

Here the zones at that region is almost same and reduced as seen in modal 1 at R=2.

### 5.4 Safety factor

In the Details of "safety factor" window, expand the Results node, if it is not already expanded. Note that the maximum and minimum FOS displayed are respectively

#### 5.4.1 Titanium alloy for modal 1

For explicit dynamics analysis for max factor of safety

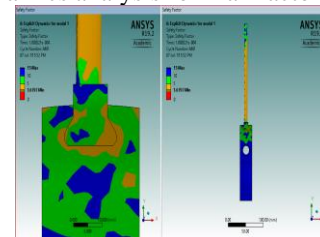


Fig 5.13 Safety factor for model 1(R=1.2)

At the red zone indicates at that area it is safe in condition

At orange and green zones the time is increasing up to maximum value.

At blue zone the value is 15

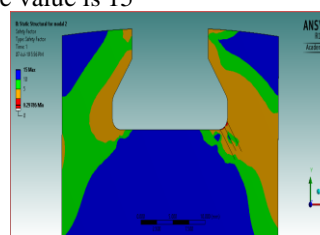


Fig 5.14 Safety factor obtained from the legend display in color bands for model 1(R=1.2)



#### 5.4.2 Titanium alloy for modal 2

For explicit dynamics analysis for max factor of safety

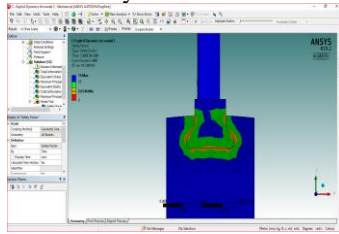


Fig 5.15 Safety factor for model 2(R=2)

In this case the same safe condition takes place as model 1 R=1.2. So at blue zone it is 15 and at red zone is 0.

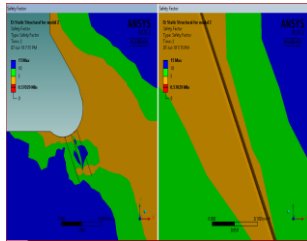


Fig 5.16 Safety factor obtained from the legend display in color bands for model 2(R=2)

#### 5.5 Fracture results

##### 5.5.1 Titanium alloy for modal 1

Form static structural analysis

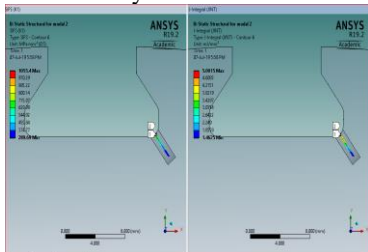


Fig 5.17 Crack growth by stress for model 1(R=1.2)

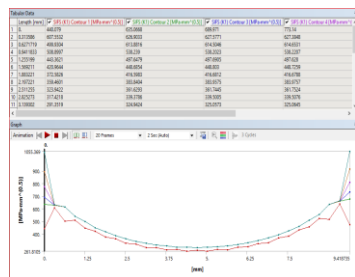


Fig 5.18 Graph shows between crack growth by stress  $SIF_s(K_1)$  and length (mm) at R=1.2

Here the SIFS is taken place at the cross section is smooth at some region and after that flat is taken place and sudden smooth surface so when the crack starts at smooth surface there is increase in value up to flat area from there to predict stress state at the tip sudden decrease and increase again up to the flat end and decreases in smoother area.

So based on the cross section of model 1 the graph is plotted. So here we had taken 6 contours because at the sixth contour there is slight increase in value.

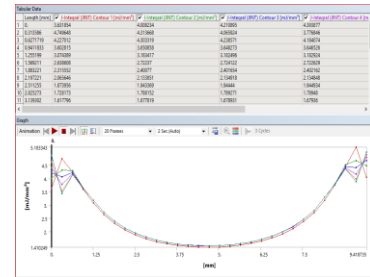


Fig 5.19 The graph shows between J-integral (energy distribution) and length (mm) at R=1.2

The above graph shows for model 2 at radius R=2 the cross section is smooth at U-tip, so there is same plot taken place as slight increase up to flat region from there to end of flat region there is constant u shape and from there sudden decrease with change in cross section.

##### 5.5.2 Titanium alloy for modal 2:

Form static structural analysis

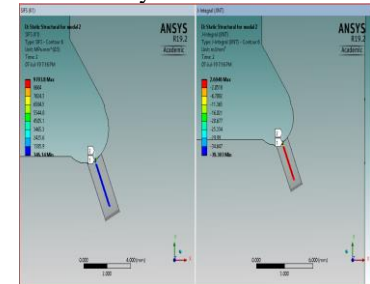


Fig 5.20 Crack growth by stress for model 2(R=2)

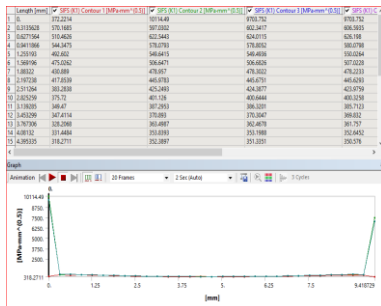


Fig 5.21: The graph shows between crack growth by stress  $SIF_s(K_1)$  and length (mm) at (R=2)

The graph shows at radius R=2 based on cross section here the predicted stress state near the crack tip by applying the load it will decrease to some extent and constant load is taken place within change in ratio of length 1.25mm i.e. up to 9.418mm from there again increases with change in cross section.



Fig 5.22: The graph shows between J-integral (energy distribution) and length (mm) at (R=2)

The graph shows at radius R=2 the cross section is smooth surface with U-tip here the crack takes place within decreased value i.e. negative value. The stress state condition at that region is 2.725 mJ/mm<sup>2</sup>.

Close the existing Mechanical window; the Workbench window is displayed.

A body is called to be deformed if its shape is changed temporarily or permanently. The temporary change of shape is known as elastic deformation and a permanent change of shape is known as plastic deformation. In ANSYS Workbench, You can determine deformation in terms of Total and Directional Deformations

Total Deformation is the total change of shape in a given working condition. You can view the Total Deformation induced in any component by using the Total tool from the Deformation drop-down in the Solution contextual toolbar.

Directional deformation is the total change of shape in a particular axis, due to given working conditions. You can view Directional deformation by using the Directional tool from the Deformation drop-down in the Solution contextual toolbar.

Total Deformation is the summation of all directional deformations produced in a certain region of the model. The following equation describes the Total Deformation:

$$U = \sqrt{U_x^2 + U_y^2 + U_z^2}$$

If Deformation in the X-axis  $U_x$   
Deformation in the Y-axis  $U_y$   
Deformation in the Z-axis  $U_z$

Then Total Deformation U will be given as follows:

$$U = (U_x^2 + U_y^2 + U_z^2)^{1/2}$$

The Legend has colors arranged in a band from top to bottom. Depending upon the type of analysis and the parameters evaluated, each color will indicate a different value. A typical Legend displayed when Total Deformation is selected from the Tree Outline.

The blue color in the Legend indicates the minimum value of Total Deformation. In this case, it displays 0 which means there is no deformation at that region.

Exit the static structural analysis - Mechanical window

### 5.6 Comparison of results for model 1 and 2:-

So the required data from experimental values the model 1 of radius 1.2 is been compared with model 2 of radius 2

Table 5.5: Table shows the comparison for model 1 and model 2

SNO	ANALYSIS VALUES		
	Parameters	Model 1	Model 2
1	Total deformation (mm) max	1.0648	0.60347
	Total deformation (mm) min	0	0
2	Equivalent stress (MPa) max	553.68	350.33
	Equivalent stress (MPa) min	0	0
3	Equivalent strain (mm/mm) max	0.0060387	0.0040392

SNO	ANALYSIS VALUES		
	Parameters	Model 1	Model 2
4	Equivalent strain (mm/mm) min	0	0
	Safety factor max	15	15
5	Safety factor min	0	0
	Fracture for SIFs(K <sub>I</sub> ) MPa-mm <sup>1/2</sup> max	1055.4	9703.8
	Fracture for SIFs(K <sub>I</sub> ) MPa-mm <sup>1/2</sup> min	289.69	346.14
	Fracture for J-integral(energy distribution) mJ-mm <sup>2</sup> max	5.0015	2.6046
	Fracture for J-integral(energy distribution) mJ-mm <sup>2</sup> min	1.465	-39.303

By the comparison of results model 1 at (R=1.2) the stresses, strains and total deformation we obtained the change in the radius for the compressor so to that extent at R=2 the solution is compared with model 1 with this comparison the values of stress strain and total deformation we have got min value.

### V CONCLUSION

So as previous mentioned, the changes in design parameters i.e radius has done using modeling software and based on this the six parameters what is the best to that design is incorporated.

Based on this crack growth by stress SIFs (K<sub>I</sub>) and J- integral energy distribution has been found out in model 1 and 2.

The comparison for model 1 parameters to model 2, based on the radius changes i.e R=2 the value of SIFS is seen as increases to 9703.08 MPa-mm<sup>0.5</sup> and also J-integral also increases. So SIFS is measured to predict the stress state near crack tip caused by remote load.

The comparison for stresses, strains and deformation for model 2 at R=2 decreases with increase in model 1 parameters.

### REFERENCES

- [1] V.N. Shlyannikov, B.V. Iltchenko, N.V. Stepanov. "Fracture analysis of turbine disks and computational experimental background of the operational decisions", accepted in year 2000, Engineering failure analysis 8 (2001); 461-475.
- [2] Newman JC, Raju IS. "Stress intensity factor equations for cracks in three-dimensional-finite bodies". Proceedings of the 14th National Symposium on Fracture Mechanics, Los Angeles, 1981. ASTM STP 791, 1983. p. 238-65.
- [3] Shlyannikov VN, Iltchenko BV. "Elastic stress intensity factors for disk and blade attachment of gas turbine engine". Problems of Strength 1993; 2,,65-70.
- [4] Shlyannikov VN. "Modelling of crack growth by fracture damage zone". Theory of Applied Fract Mech 1996;25,,187-201.
- [5] Stepanov NV, Shkanov IN, Omelchenko VV. "Similarity of disk stress states and damage on multiaxial hydraulic test bed to those in service operation". Information Institution of USSR, Soviet Aeronautics 1985;2,,95-8.
- [6] Shlyannikov VN, Tchadaev AV, Kalatchev VA, "Fatigue fractures of power engineering structures. Fracture from defects." ECF-12 1998;1:375-80.
- [7] Hutchinson JW, "Singular behaviour at the end of a tensile crack in a hardening of a material." J Mech Physics Solids 1968;16:13-31.
- [8] Hutchinson JW, "Plastic stress and strain fields at a crack tip" J Mech physics Solids 1968;16:337-47.
- [9] Rice JR, Rosengren GF, "Plain strain deformation near the crack tip in a power-law hardening material." J Mech physics Solids 1968;16:1-12.