FCPAS

Fracture and Crack Propagation Analysis System

Version 1.0

Software & Tutorial Document

March 2011

©2011 FCPAS



Tutorial Document for Fracture Analysis with FCPAS

Introduction

In this tutorial, some basic linear elastic fracture mechanics examples are included that demonstrate usage of FCPAS (Fracture and Crack Propagation Analysis System) to solve threedimensional fracture problems. The generation of models, meshing and application of boundary conditions and loads are done using the commercially available finite element software, ANSYSTM. Then, a converter program is used to convert the finite element model information, boundary conditions and loads data from ANSYSTM [1] format into FCPAS format. Finally, FCPAS Solver is run to solve the crack problem and determine the stress intensity factors for the problem of interest. The following chart shows the general algorithm and file structure of process (Figure T.1).



The steps from ANSYS[™] model generation to solving the crack problem in FRAC3D are explained in detail. The problems included in this tutorial are:

- 1. Two-dimensional mode-I central crack in a large isotropic medium,
- 2. Mode-I crack in a Compact Tension, C(T) test specimen,
- 3. Mode-I central crack in a finite-thickness plate.



Figure T.2 General algorithm of FCPAS

FCPAS's GUI (Graphical User Interface) allows the user to follows the process in Figure T.3 in an orderly and user-friendly manner. This is the first version of the software and currently linear fracture analysis is available.



Cracked Model Developed using ANSYS : If you click this button, you can work with **Cracked Model Developed with ANSYS**.

Crack Insertion and Fracture Analysis : If you click this button, you can work with **Crack Insertion and Fracture Analysis.**



Figure T.3. FCPAS graphical user interface

When you click first button, the above form comes up.

EXAMPLE.1. Two-Dimensional Mode-I Central Crack in a Large Isotropic Plate

T.1.1. Problem Description

Consider the infinite domain in Figure T.4a containing a central crack and subjected to uniform tensile pressure loading perpendicular to the crack plane. We can model this problem as a plate in tension with a central crack as shown in Figure T.4b. Due to symmetry in the problem; only a quarter model is analyzed as shown in Figure T.4c. The plate is made of steel with Young's modulus $E = 200 \ GPa$ and Poisson's ratio v = 0.33. Let width to be $2w = 20 \ m$, height is $2h=20 \ m$, $a = 1 \ m$ and $\sigma_0 = 1 \ Pa$. The objective is to compute the mode-I stress intensity factor (SIF).



Figure T.4 Through-thickness crack in a large plate.

Note that for this problem, analytical solution is given by;

 $K_1 = \sigma_0 \sqrt{\pi a}$, where σ_0 = Stress (1 Pa), *a*: Half of crack length (1 meter). Use of this solution yields $K_1 = 1.77 Pa \sqrt{m}$.

T.1.2 Assumptions

- Linear elastic fracture mechanics (LEFM).
- Plane strain problem.

$$\varepsilon_{zz} = 0 = \frac{\partial w}{\partial z} = 0$$
$$\varepsilon_{zz}(x, y) = 0$$

In the ANSYSTM tab of the FCPAS, we browse ""C:\Program Files\ANSYS Inc\v120\ansys\bin\intel\launcher120.exe".

The above directory location may change depending on the version of Ansys being used.

FCPAS Tutorial – Version 1.0

1.	First, we browsed	ANSYS TM exec	utable file with its pa	rameters.
Browse C:\Pr	igram Files\ANSYS Inc\v120\ansys\bin\int 2-	^{elVauncher120.exe} To run, press it.	Run ANSYS	Change Working Direct
Also, we ma	y want to run ANS	SYS TM from its j	Run Product Launcher	ANSYS(TM) Geo File
				Frac 3D

Figure T.5 ANSYS[™] tab of FCPAS.

T.1.3 Generation of the Finite Element Model within ANSYS[™]

We will model this problem as a two-dimensional model plane strain model by taking into account the symmetries in horizontal and vertical directions. Also, in the out-of-plane direction, we will use one layer three-dimensional elements. To do this, we will first mesh the back face of the domain with area (2D) elements and extrude the mesh into the third direction. To do this, we will use PLANE82 and SOLID95 elements from the ANSYSTM element library. Note that ANSYSTM Help is very useful tool to identify and select the suitable elements for the problem of interest.

Preprocessing

Change Directory

Before starting modeling, create a folder in which you would like to perform analyses & change directory to this folder.

FCPAS Tutorial – Version 1.0



Give the Job a Name

Utility Menu>File>Change Jobname ...

Enter a name, for example `CC3', and click on OK.

ANSYS Academic Teaching Advance	∧Change Jobname		×
<u>File Select List Plot</u> Plo	[/FILNAM] Enter new jobname	5 C3	
Clear & Start New	New log and error files?	No No	
Change Jobname			
Change Directory	ок	Cancel	Help
Change Title			

Define Element Type

Main Menu>Preprocessor>Element Type>Add/Edit/Delete

This brings up the 'Element Types' window. Click on the Add... button. The 'Library of Element Types' window appears. Highlight Plane 82 `Plane-8node 82' and solid95 `Solid-20node 95'. Click on OK or in command line, use (ET,1,82)¹, (ET,2,95).

ANSYS Academic Teaching Advanced Utility Menu (CC3)							
Eile Select List ANSYS Command Line	<u>n</u> uCtrls <u>H</u> elp						
□ ☞ ■ ◙ ⊜ ☞ ? ▦ /							

Define Material Properties

Main Menu>Preprocessor>Material Props>Material Models

On the right side of the `Define Material Model Behavior' window that opens, double click on `Structural', then `Linear', then `Elastic', finally `Isotropic'. Enter in values for the Young's modulus (EX = 200E9) and Poisson's ratio (PRXY = 0.33) of the plate material.

¹ PLANE82 element provides us both "plane strain or stress" options.

ANSYS Toolbar	erial Model Behavior	
SAVE_DB RESUM_E Materia	Favorite Help	
ANSYS Main Menu Preferences Preprocessor Element Type Real Constants Material Props Material Library	erial Model Number 1 Serial	
 □ Temperature Uni □ Electromag Unit: □ Material Models □ Convert ALPx □ Change Mat Nun □ Failure Criteria □ Write to Eile 	Linear Isotropic Properties for Material Number 1	·]
	EX 200e9 PRXY 0.33	
 B Numbering Ctrls Archive Model B Coupling / Ceqn ■ FLOTRAN Set Up 	Add Temperature Graph OK Cancel	

Define Keypoints

Main Menu>Preprocessor>Modeling>Create>Keypoints>In Active CS We are going to create 5 keypoints given in the following table:

Keypoints	X [m]	Y [m]	Z [m]
1	0	0	0
2	10	0	0
3	10	10	0
4	1	10	0
5	0	10	0

Create Keypoints in Active Coordinate System	×						
[K] Create Keypoints in Active Coordinate System							
NPT Keypoint number	1						
X,Y,Z Location in active CS							
OK Apply	Cancel Help						

8

Define Line Segments

Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Straight Line This is required to create the models boundary lines, successively like first 1 to 2, 2-3 and finally 5to 1.

ANSYS Main Menu	Create Straight Line				
Preprocessor	• Pick C Unpick	_	_	_	
Real Constants	© Single C Box	•	•	3	Noncommercial Use Only
Material Props	C Polygon C Circle				.ПП. 2 2008
Sections	C Loop				13:18:35
Modeling					
□ Create	Count = 0				
⊞ Keypoints	Maximum = 2				
🗆 Lines	Minimum = 2				
⊟ Lines	KeyP No. =			+	
🖉 Straight Line					
➢ In Active Coor	List of Items			+	
➢ Overlaid on Ar	🔿 Min, Max, Inc				
		5			
➢ Normal to Line					
Norm to 2 Line					
At angle to line	OK Apply				
Angle to 2 Line	Reset Cancel	v			
⊞ Arcs		L.			
	Pick All Help	F	~		
│					

Create the Area

Main Menu>Preprocessor>Modeling>Create>Areas>Arbitrary>By Lines Pick all lines (Click OK in the picking window.

ANSYS Main Menu	8		
Preprocessor		4	
Element Type ■	Create Area by Lines		
Real Constants	• Pick C Unnick		Noncommercial Use Only
		li -	JUL 3 2008
Sections	🖲 Single 🔿 Box	I	13:13:40
Modeling	C Polygon C Circle	I	
□ Create	O Loop		
⊞ Keypoints		8	
⊞ Lines	Count = 15		
⊟ Areas	Maximum = 15	+	
Arbitrary	Minimum = 2		
🖉 Through KPs	Line No. = 14	+	
	• List of Items	li.	
🖉 By Skinning	🔿 Min, Max, Inc	3	
≫ By Offset			
⊞ Rectangle		1	
Circle			
Polygon		W	
	OK Apply		
⊞ Volumes	Reset Cancel	 X	
⊞ Nodes	Cancer		
⊞ Elements	Pick All Help		
🔤 Contact Pair		J	
🕀 Pining Models			

ARE AS				A BIOTZO
TYPE NUM	3 3	3		
PRES			Noncomme	rcial Use Only
-1				JUL 3 2008
				13:16:28
	8			
	s			
	2 <u>X</u>			

EXTRUSION

To create volume, we can use easily extrusion property by 0.1 unit through normal direction (z axes)

Main Menu>Preprocessor>Modeling>Operate>Extrude>Areas>By XYZ Offset

Firstly model is selected then entered the offset values.

ANSYS Main Menu	(A)			~	
					77.70
Element Type	Extrude Area by Offset		_		5YS
Real Constants	• Pick C Unpick	3 3	3	Noncommercial Us	e Only
					ΠΠ 2 2008
Sections	💿 Single 🔿 Box				13:21:09
Modeling	🔿 Polygon 🔿 Circle				
Create	C Loop				
Operate	Count = 0	8			
⊟ Extrude	Mevimum = 7	F			
Elem Ext Opts	Minimum = 1		+		
□ Areas	Area No. =				
Along Normal			+		
➢ By XYZ Uffset	List of Items				
About Axis	C Non Non Tra	3			
Along Lines	O MIN, Max, INC				
E Keypointe					
➢ Extend Line	p				
H Booleans		•			
⊞ Scale	OK Apply	Y			
Calc Geom Items	Reset Cancel	<u>x x</u>			
Move / Modify					
🖽 Сору	Pick All Help				
⊞ Reflect		-			
🖽 Piping Models					

FCPAS Tutorial – Version 1.0



Apply Boundary Conditions

Because of the symmetry, our system has the following BC's:

- On symmetry areas
 - Left Area [from (0,0) to (0,10)] $U_x = 0$
 - Top Area [from (10,10) to (1,10)] $U_y = 0$
- On back area, constrain U_z (U_z=0)
- On front area, constrain U_z (U_z =0)

Apply the displacement constrains using; Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>on <u>Areas</u>> on back area and front area or **DA**, **p** for area BC.



Be careful in selecting. To get accurate selection, you can use perspective views using **ctrl+right buttons**.



To check the applied boundary conditions on areas, **DALIST** is used in the command line

ADALIST Com	nmand		×
Eile			
LIST CONST	RAINTS ON ALL SELE	CTED AREAS	
AREA 1 2 3 6	LOAD LABEL UZ UZ SYMM SYMM	VALUE <s> 0.0000 0.0000 0.0000 0.0000 0.0000</s>	0 - 0000 0 - 0000 0 - 0000 0 - 0000

Apply Loads

Now we will apply the distributed surface forces (pressure).

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Areas Carefully pick the bottom area (at z=0 and y=0) and then click OK in the picking window or sfa, p to apply area pressure.

Select `Constant value' enter `-1' for `Load PRES value', then click OK (Negative sign is for tensile loading).



	Apply PRES on areas	×	1
	[SFA] Apply PRES on areas as a	Constant value	
	If Constant value then:		1
	VALUE Load PRES value	-1	
	LKEY Load key, usually face no.	1	
	(required only for shell elements)		
		Cancel Help	
			_
Meshing the Mo	del		
Back area has to ASEL command u	be meshed first. Before meshing, we sing the coordinate (location) option	e can select the back area (te n.	mplate area) with

ANSYS Academic Teaching Advanced Utility Menu (CC2)							
<u>Eile Select List Plot PlotC</u>	ASEL, Type, Item, Comp, VMIN, VMAX, VINC, KSWP						
D 🚄 🖬 🗗 🎒 🖉 📲	asel, , loc,z,-0.001,0.001						

In the above command, Asel is used to select a subset of areas that are located in the coordinate range specified. To be able to select the back area only, very small distance is given between the minimum and maximum coordinate in *z* direction only.

aplot

Meshing

To obtain accurate fracture solution, we need to generate fine mesh near the crack tip. For this, we can use the KESIZE command to specify element size at the crack tip keypoint. First zoom into the crack tip region. Then issue the command

kesize, p

and pick the crack tip keypoint and write 0.01 as the element size value SIZE in the window.



Amesh,p 'Select the back area exactly. Be careful in selecting. Use, zoom in, perspective view or rotate the model or issue other viewing commands to select the back area.



First, zoom into the crack region and use **Ksel** to locate the exact location of the crack tip, since we will measure the element edge sizes ahead of and behind the crack tip. To do this, we use the *ndist*

command (ndist,p) and measure the crack tip edge size. This gives us 6.67e-1, which is coarser than the required size. Therefore, we require finer mesh. Now, we try a specified global element size. Esize,0.5 global size value is entered as 0.5.



Mesh again the back area; Amesh,p



We may check the distance again using the **NDIST** command. This gives us 9.972e-3, which is fine enough (i.e., $1/100^{th}$ of the crack length).

FCPAS Tutorial – Version 1.0



As can be seen above, the specified element size is achieved.

Volume Sweeping

Now we will extrude the 2D mesh in to the third dimension by one element in this direction.

Main Menu>Preprocessor>Meshing>Mesh>Volume>Volume sweep>Sweep





Volume scraping gives us prismatic mesh in the bounded volume, with respect to area mesh (template mesh). After sweeping the 2D mesh to generate the 3D mesh, we need to delete 2D elements, since FRAC3D requires 3D elements only.

To do this, we use Aclear, all 'Area mesh is deleted. Because of this, a gap occurs in the sequence of element numbers. To remove the gap we use, Numcmp, elem

📾 ANSYS 11.0 Output Window	_ 🗆 🗙
CLEAR NODES AND ELEMENTS FROM ALL SELECTED AREAS	
CLEARED 1 AREAS, Ø LINES, Ø KEYPOINTS	
PRODUCE ELEMENT PLOT IN DSYS = Ø	
COMPRESS ELEMENT NUMBERS MAXIMUM ELEMENT NUMBER COMPRESSED FROM 1346 TO 673	-
	• //

To make sure no gap exists in the node numbers as well, we use Numcmp, node



Definition of Crack for FRAC3D

Now, we need to provide crack tip element and node number information for FRAC3D analysis. Zooming into the crack region, we can find the elements and nodes located at the crack tip. (See detailed crack tip definition requirements in this tutorial for which elements and nodes to be selected). We need to identify the crack tip element on the bottom crack surface (with respect to the chosen local coordinate system) immediately behind the crack tip. Using, **Select-Entities-Elements**, form the main menu (or Esel, p command) try to select the elements at the crack tip. For this, move your mouse pointer near to crack tip region. We have to select the element which is both at crack tip and on the bottom crack surface (as defined by the local coordinate system at the crack tip). You can see that the crack tip element number is **1**.



FCPAS Tutorial – Version 1.0

S Academie	: Teaching	Advanced Utility
<u>S</u> elect <u>I</u>	<u>L</u> ist <u>P</u> lo	ot Plot <u>C</u> tris <u>V</u>
<u>ا ا ا ا ا</u>	<mark>∧</mark> Select E	intities 🛛 🔀
'S Toolt	Nodes	•
	Nodes	
/S Main	Areas	
ferenc	(Keypoi	ints Select
lution		lect
neral P	Sele Al	I Invert
pologic	Sele No	ne Sele Belo
M Tool	OK	Apply
sign Op ob Desi	Plot	Replot
diation	Cancel	Help

After element selection, we list the selected element and save the file FRAC3D analysis. Use **Elist** to see and save the list as a file.

	LIST	Comm	and												X
File															
LI	IST AI	L SE	LECT	ED EI	LEMEI	NTS.	(LIST	NODES>							
	ELE	I MAI	ТҮР	REL	ESY	SEC		NODES							
	t	. 1	. 2	1	Ø	1	1977 1170 4455	1976 394 4877	122 264 4454	265 1171 4305	4040 2159	3618 2160	4299 4306	4307 2161	

The element information (for the crack tip elements) is saved from the Elist window as cc3.crelems

In addition to crack tip elements, we also need to select nodes on the crack front. To be able to select the nodes of the crack front, it is required to select the crack tip line and then select the nodes associated with this line. Using, **Select-Entities-Lines**, crack tip line is selected. Then, **NSLL**, **S**, **1** is used to select all the nodes along the selected crack front line. Using **Nlist**, we can see that the crack front nodes numbers are: **122**, **4299**, **4454**.

FCPAS Tutorial – Version 1.0



M NLIST Command					×
Eile					
LIST ALL SELECTED	NODES. DSYS=	0			
NODE X 122 1.0000 4299 1.0000 4454 1.0000	Y 10.000 10.000 10.000	Z 0.0000 0.10000 0.50000E-01	THXY 0.00 0.00 0.00	THYZ 0.00 0.00 0.00	THZX 0.00 0.00 0.00

The coordinates of the selected nodes (the crack tip nodes) are saved from the **Nlist** window as *cc3.crnodes*

AELIST Command LIST ALL SELECTED ELEMENTS. (LIST NODES) ELEM MAT TYP REL ESY SEC NODES 1977 1170 4455 1976 400 4822 1976 394 4877 1975 395 1 1 2 1 0 1 122 264 4454 262 263 265 1171 4305 122 394 4040 3618 4299 4307 2159 2160 4306 2161 0 1 2 1 3618 3617 4441 4299 2162 2163 4442 2160 ? × - 🗧 🖻 🚽 Save in: 1 2 ansuitmp CC2.page cc3.crelems cc3.crnodes à 1 2 1 6 CC3.chodes
CC3.db
CC3.cls
CC3.node
CC3.node
frac3d.exe
frac3d.exe B 7 2 1 1 esher Opts oncatenate lesh ¹Keypoints movieassembly.exe ¹Lines 1 Areas I Volumes I Volume Sw∈ III Sweep O I≫ Sweep cc3.elis • File name <u>S</u>ave 1 Tet Mesh Fr Ŧ Interface Me Lister Files (".lis) Cancel C auto

Also using **Select-Everything**, all elements in model are saved in file, *cc3.elis*.

Also using **nwrite** all nodes are saved as *cc3.node* automatically in current working directory.

<u>F</u> ile	<u>S</u> elect	<u>L</u> ist	<u>P</u> lot	Plot	NWRITE, Fname, Ext,, KAPPND
	2 🖬 🖸	9 6	£7	? ₩	nwrite

SBCTRAN is used to transfer solid model loads and boundary conditions to the FE model. Loads and boundary conditions on unselected keypoints, lines, areas, and volumes are not transferred. **sbct**





Using **dlist** displacement BC's are saved as *cc3.dlis*.



Using sflist pressure loads on elements are saved as cc3.sflis.

∧ sflist c	ommand						
Ele							
LIST NOD	AL SUR	FACE LOAD PRE	S FOR ALL S	ELECTED NODE	s		
ELEMENT 65	LKEY 5	FACE NODE 2 44 4101 4090	3 -1 -1 -1	REAL 00000000 00000000 00000000 000000000	IMAGI 0.000000 0.000000 0.000000 0.000000	NARY 100 100 100	
66	4	80	î	000000000	a aaaaaa	00 00	
		Save As					? ×
85	4	Save in:	🗁 Center_crack	(•	+ 🗈 💣 📰	
86	4	My Recent Documents					
caterrate h eypoints		Desktop					
nes							
olumes							
plume Swe	eep	My Computer					
Sweep O Sweep	pts	S					
et Mesh Fr	om	My Network	File <u>n</u> ame:	cc3.sflis		-	Save
terface Me	sh	Places	Save as tupe:	Lister Files (* lis)		-	Cancel
ify Mesh	-1		care as gpc.	Transform lifes (.ilis)			

Now, we completed all modeling steps in ANSYS[™]. We are ready to convert all model information into FRAC3D format using the converter program.

T.1.4 Converting ANSYS Model into FCPAS Format (Generation of *cc3.geo* File)

FCPAS requires its model information in a specific format. To convert ANSYSTM model files into FCPAS format, we can use the *convert_ansys_fcpas.exe* program. The converter program can be run by typing, its path in MSDOS prompt or from the "Geo File" tab from FCPAS GUI.

Input file names can be selected by "Browse" buttons. "Generate Geo file" creates *cc3.elis_3d.geo* file. To go to Run File preparation, press "Next Step".

T.1.5 Generation of *.run File

Now, we need to generate a run file which is also required for fracture analysis. We use *writerun_fcpas.exe* or FCPAS GUI to generate *.run file (*cc3.elis_3d.run* file). The *.*run* file contains analysis type, material properties, solver type tolerances, body forces and local coordinate systems data.

Parameters can be selected by clicking the objects in the tab. "Generate Run file" creates *cc3.elis_3d.run* file.

CPAS 1.0-1)Cracked Model Devoloped using ANSYS		
Help		
YS (TM) GEO File RUN File Fracture Analysis Fracture Info I	Post Processing Visualization	
Run File Name without ".run" cc3.elis		Control Panel
Analysis Type C Linear C Nonlin	near Thermal Stress 🔽 Fracture T FGM	CARA MATERIAL INFORMATION
Solver Type C. Eccelul C. BCC Te		0.10000000E-07 0.30000000E+08 0.3000000
Material Properties	serance> .e-8	0.30000 0.30000 0.
Number of different Material in The Model 1		C*** THERMAL LAODING
Ex Ey Ez Gxy Gyz	Gxz Nuxy Nuyz Nuxz Behaviour	
3.e7 3.e7 3.e7 1153846 1153846 1	153846 J0.3 J0.3 J Orthotropic J Elasto Plastic	
	Crithotropic Elasto Plastic	
FGM Properties(E(x)=E0 exp(Beta x) Nu(x)=	Nu0 exp(eta x) Alpha(x)=Alpha0 exp(w x))	ANSYS(TM)
		Geo File
		RUN File
		Frac 3D
		Fracture Info
How many DOE do you have?	CTE Values (XX,YY,ZZ) and Tref 1112	Post Processing
How many nodes	Temp Dependet Material Property	Visualisation
Node number and direction	Initial Temperature	
1 X 1 Y 1 Z	Temp. Points T Ex,Ey,Ez,Gx,Gy,Gz 13e73e7	
Have Body Force Loading	Nu of Material 1 is temperature dep.	
Type of Body Force Loading Gravity C Centrifugal	Temp. Points 1 Nux, Nuy, Nuz 1 0.3 0.3 0.3	
The Acceleration 9.81	CTE temperature depended	Generate RUN File
Direction (wrt Global Coord. Sys.)	The Number of temperature points	
Second on Axis 1111 Density of Material 1	CTE Values(x,y,z points) for point 1	**
Have Local Coordinate Systems	Integration Order 24	
How many coordinate system	✓ Use Transition Element Generalized Plane Strain Problem	
	Input Number of Increments ,Max. Number of	
CSYS ID no, THXY THYZ THXZ angles 11 30 0 0	Iterations and Tolerance for Convergence 10 20 1.0e-6	

To pass into "Frac3D" tab, press "Next Step". T.1.6 Running FRAC3D (FCPAS Solver)

To run the FRAC3D, three kinds of input files are required;

*.run (compulsory)

*.geo (compulsory)

*.tem (optional)

FRAC3D gives the results in the following output files;

*.out

*.str

*.stn

*.crk

Now, we are ready to run FRAC3D. To do this we can use *frac3d.exe*. When running FRAC3D, geo and run files names have to be entered. The following table shows the steps and input for this specific problem.

Select the *cc3.elis_3d.geo, cc3.elis_3d.run*, and *cc3.elis_3d.tem* (if required) files by browsing and press run button to run the *Frac3D.exe* in the background.

e Help					
NSYS (TM) GEO File RUN File	Fracture Analysis Fracture Info Post	t Processing Visualization			
R G T	UN File Name SEO File Name Thermal Analysis?	cc3.elis_3d cc3.elis_3d	Browse Browse		Control Panel
N	LIM FILE Name	2	Browse RUN		ANSYS(TM) Geo File
D	lo you accept License Agreement?	© Yes ⊂ No	*	Review Error File	RUN File Fracture Analysis Fracture Info
	FCPAS Solver Fracture and Crack Propagation Analy LICENSETERMS Copyright 2010-AirO. Ayhan, ayhan (ysis System ≌sakanya.edu.tr			Post Processing Visualization
per sub (th an - U U) IN O O D	Permission is hereby granted, free of c rson obtaining a copy of this software, is boonstituents and the associated docum e "Software"), for the rights to use, copy, d publish resultafrom the Software subject Isage of the Software is permissible and f NDER THE CONDITION THAT THE SO ICLUDING APPROPRIATE SUBCONST F THE RESPECTIVE AUTHOR(S) AND ISTRIBUTED TO ANY OTHER THIRD F	sharge, to any ts all entration files merge software, ted to the following conditons: reely available FTWARE, IFTWARE, ITUENTS, REMAIN UNDER COPYRIGHT THE SOFTWARE IS NOT PARTIES.			

REVIEW EROR FILE toolu olan yeni görüntüyü al

After finishing the process, output files can be seen in the "Fracture Info" tab. In the "Fracture Info" tab you can browse anyfile to see its content and plot the K_1 , K_2 and K_3 data in an x-y graph.

this section, you can view contents of output files and p	plot stress intensity factors	along crack front.					
Open File: C:Documents and Settings Admin/Desk	Aç	_3d.crk		P	IOT SIFS :	? 🗙)
	Konum	CC3		▼ ← 6	* 🗐 •		≥ Working Directo
FRACTURE MECHANICS INFORMATION	N En Son Kullandiklanm	inputs cc3.elis_3d.crk					_
24 X 24 X 24 INTEGRATION IS USED FOR ENRICH TRANSITION ELEMENTS ARE INCLUDED	IED CR/ IN THE Masaüstü						ANSYS(TM)
CRACK NO: 1 CRACK TIP NODES: 82 4605 4412	Belgelerim						Geo File RUN File
CRACK IN AN ORTHOTROPIC MATERIAL	- 						Frac 3D Fracture Info
82 0.1878083E+00 0.0000000E+00 0.00000 4605 0.1877509E+00 0.0000000E+00 0.00000 4412 0.1860791E+00 0.000000E+00 0.00000	Bilgisayanm 0E+00 00E+00 00E+00						st Processing /isualisation
G1 G2 G3 GTOT	Ağ Bağlantılanm	Dosya adı:	cc3.elis_3d.crk		-	Aç	• →
82 0.1175731E-08 0.0000000E+00 0.0000000	DE+00	Dosya türü:	"Crack Files(*.crk)		•	İptal	
4605 0.1175013E-08 0.0000000E+00 0.000000 4412 0.1154182E-08 0.0000000E+00 0.000000	0.000000000000000000000000000000000000						-

To plot the K_1 , K_2 and K_3 data in a graph, just press "Plot SIF's" button.





We can see that the computed K_1 (1.814) is ~%2 higher than the analytical solution.

T.1.7 Post-processing of FRAC3D Results

Help	I Ele [Emature Anaburie] Emature late Post Processing]	Manualization 1		
	rie riduuie Analysis Fracture into i ost riduessing	visualization		Control Panel
	RUN File Name	cc3.elis_3d	Browse	
	GEO File Name	cc3.elis_3d	Browse	
	How Many Different Materials Do You Want To Se	æ? 1	Create	
	Input 1 Material ID Numbers	1	÷	
	Do You Want to Prepare Animation File?	No	T	
				ANSYS(TM)
				RUN File
				Frac 3D Fracture Info
				Post Processing
				Visualisation
				\leftarrow \rightarrow

T.1.8 Visualization of FRAC3D Results

Help						
YS (TM) GEO File RUN File Fracture Analysis	Fracture Info Post Process	sing Visualization				
· · ·				Control Panel		
		Yee w				
Choose whether to draw an outline before	deformation	105		78.		
Input the scaling factor for displacement		1000				
Which contacts he displayed?		VV Strang				
which scalar to be displayed?		TT Stress				
Do you want to change range?		Yes 💌				
from 0 ÷ to 2 ÷						
Input the geoming coole		1		ANSYS(TM)		
input the zooming scale		1		Geo File		
			Show Results	DUN SI		
				KUN File		
			يغلاد	Frac 3D		
			**	Fracture Info		
				Post Processing		
VTK software Usage is Lir	nited To This Conv	vright		Visualisation		
· · · · · · · · · · · · · · · · · · ·						
VTK is an open-source toolkit licensed under the BSD license. http://www.ytk.org/VTK/project/license.htm Copyright (c) 1993-2008 Ken Martin, Will Schroeder, Bill Lorensen						
	All rights reserved.	aurce and binary forms, with or without	modification, are permitted provided that the fo	llowing conditions are met		
			indunication, are permitted provided that the re			
	Redistributions of source	code must retain the above copyright i	notice, this list of conditions and the following dis	sciaimer.		
	* Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.					
	* Neither name of Ken Mar	tin Will Schroeder, or Bill Lorensen por	the names of any contributors may be used to a	endorse or		
	promote products derived	from this software without specific prior	r written permission.	andorac of		
	THIS SOFTWARE IS PROV	IDED BY THE COPYRIGHT HOLDERS AN	D CONTRIBUTORS ``AS IS" AND ANY EXPRESS	OR IMPLIED WARRANTIES,		
	INCLUDING, BUT NOT LIMI	TED TO, THE IMPLIED WARRANTIES O	F MERCHANTABILITY AND FITNESS FOR A PART	ICULAR PURPOSE ARE		
	EXEMPLARY, OR CONSEQU	JENTIAL DAMAGES (INCLUDING, BUT N	OT LIMITED TO, PROCUREMENT OF SUBSTITUT	E GOODS OR SERVICES;		
	CONTRACT, STRICT LIABI	LITY, OR TORT (INCLUDING NEGLIGEN	CE OR OTHERWISE) ARISING IN ANY WAY OUT	OF THE USE OF THIS		
	ISOFTWARE, EVEN IF ADVI	SED OF THE POSSIBILITY OF SUCH DAI	MAGE.			

To see the Cracked Model results, choose the parameter you would like to contour plot and press "Show Results" button.



EXAMPLE.2. Three-Dimensional Mode-I C(T) Specimen

T.2.1. Problem Description

Toughness is the ability of a material to resist fracture. The general factors, affecting the toughness of a material are temperature, strain rate, relationship between the strength and ductility of the material and presence of stress concentration (notch) on the specimen surface. Compact Tensile (CT) specimen is one specimen type to measure fracture toughness of a material. In this example, we will model a crack of length *a*=27.5mm in a CT specimen and compute the mode I stress intensity factor (SIF) along the crack front. The material is Al-7075 with Elasticity Modulus = 70 GPa, σ_{Y} = 95 MPa, *P*= 1 N, *v*= 0,33, ρ = 2,81 g/cm³. The dimensions are given in Figure D.38. These dimensions are in mm.



Figure D. 2 CT specimen and its dimensions

In FCPAS ANSYS[™] tab, we run ANSYS[™] program.

Help SYS (TM)	Ela DIN Ela Ersetura Anskaia Ersetura lafa Daet Processia Maustratian	
		Control Panel
Browse	C:\Program Files\ANSYS Inc\v120\ansys\bin\intel\auncher120.exe	
	Run ANSYS	Change Working Directory
	1	
Browse		
	Run Product Launcher	
		Geo File
		RUN File
		Frac 3D
	Working Directory	Fracture Info
		Visualisation
	FCPAS Software Path	
	C:\	\leftarrow
	Current Berth	
	Curent Path	
	C:\Documents and Settings\Admin	

Figure T.7. FCPAS graphical user interface

T.2.2. Generation of the Finite Element Model within the ANSYS[™] Preprocessor

First of all, we must take into account the problem type, we will model this as a 3D problem. Also, due to the symmetry of the problem, only analysis of a half model is needed. We will model this three-dimensional problem using multi-layers of 3D elements in the out of plane direction. To do this, we will first mesh the back face of the domain with area (2D) elements and extrude the mesh into the third direction. We will use PLANE82 and SOLID95 elements from the ANSYSTM element library [3]. Note that ANSYSTM Help is very useful tool to identify and select the suitable elements for the problem of interest.

Preprocessing Change Directory

Before starting the model, create a folder in which you would like to work&change directory to this folder.



Give the Job a Name

Utility Menu>File>Change Jobname... Enter a name, for example `*CT2*', and click on OK.

ANSYS Academic Teaching Advance	∧Change Jobname				
<u>File Select List Plot Plc</u>	[/FILNAM] Enter new jobname	¦≓T2			
Clear & Start New	New log and error files?	No No			
Change Jobname					
Change Directory	ОК	Cancel	Help		
Change Title					

Define Element Type

Main Menu>Preprocessor>Element Type>Add/Edit/Delete

This brings up the 'Element Types' window. Click on the Add... button. The 'Library of Element Types' window appears. Highlight "PLANE82-8 node 82" and "SOLID95-20 node 95". Click on OK or in command line, use (ET,1,82)², (ET,2,95).



Define Material Properties

Main Menu>Preprocessor>Material Props>Material Models

² PLANE82 element provides us both "plane strain or stress" options.
On the right side of the `Define Material Model Behavior' window that opens, double click on `Structural', then `Linear', then `Elastic', finally `Isotropic'. Enter in values for the Young's Modulus (EX = 70E9) and Poisson's ratio (PRXY = 0.33) of the plate material. Or in command line, use

MP,EX,1,70e9			
MP,PRXY,1,0.33			
ANSYS Main Menu	N Define Material Model Behavior		
📰 Preferences 🗾	Material Edit Favorite Help		
Preprocessor Element Type	Material Models Defined	Material Models Available	
Real Constants	8 Material Model Number 1	Favorites	_
Material Props		Structural	
Temperature Units		🔒 Linear	
📰 Electromag Units		Blastic	
Material Models Convert ALPx		8 Isotropic	
📰 Change Mat Num		Linear Isotropic Properties for Material Number 1	
		Linear Isotropic Material Properties for Material Number 1	
Read from File			
Sections Modeling		T1	
		Temperatures	-
Checking Ctrls		PRXY 0.3	
Archive Model			
Coupling / Ceqn			
			2
Loads Loads		Add Temperature Delete Temperature Graph	ř.
Physics Path Operations		OK Cancel Help	

Define Keypoints

Main Menu>Preprocessor>Modeling>Create>Keypoints>In Active CS We are going to create 5 keypoints given in the following table:

X [m] Y [m] Z [m] 1 0 0 0						
1 0 0 0						
2 0.0625 0 0						
3 0.0625 0.03 0						
4* 0.04 0.03 0						
5 0.035 0.03 0						
6 0.0325 0.0275 0						
7 0 0.0275 0						
8** 0.0125 0.0165 0						
*: If sharp crack length is 2 mm, then coordinates must be (0.037, 0.03, 0). In this case it is 5 mm.						

**: This keypoint is the center of the hole and going to be used in Hole (circle) placing section.

K,1,0,0,0, K,2,0.0625,0,0, K,3,0.0625,0.030,0,

K,4,0.04,0.03,0,

K,5,0.035,0 K,6,0.0325, K,7,0,0.027 K.8.0.0125.	0.03,0, 0.0275,0, '5,0, 0.0165.0.		
.,_,_,_,,	Create Keypoints in Active Coordinate System	×	
	NPT Keypoint number X,Y,Z Location in active CS	1 0 0 0	
	OK Apply	Cancel Help	

Define Line Segments

Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Straight Line

Clicking keypoints from 1 to 2, 2 to 3..., lines can be drawn. Last line must be KP7 to KP1. Keypoint 8 is for only creating the hole. For now keypoint 8 will not be used.

			al	
NSYS Ma	iin Men	u 🛞		
nces		-		
essor				
ent Type				
Constan	ts			
ial Prop	S			
ons			5 4 3	
ling			.7 .6	
eate				
Keypoin	ts			
	rking	Fian		
in Acti N On Lin	veus		8	
→ Un Lin N On Lin		otio	, č	
- 2° OH LIP - ⊠ On Mo	ie wiki de	auo		
- 水 OH NU - 网 K P ha	hveen	KPe	Create Keynoints in Active Coordinate System	
Ø Fill be	hween	KPs	W1 Create Keypoints in Active Coordinate System	
⊞ KP at	center		[K] Create Keypoints in Active Coordinate System	
⊞ Hard F	PT on I	ine		
⊞ Hard F	PT on a	area	X,Y,Z Location in active CS	
Lines				
Areas				
Volumes			OK Apply Cancel Help	
Nodes				
Flements	2			
Or,				
LSTR,	1,	2		
LSTR,	2,	3		
LSTR,	3,	4		
LSTR,	4,	5		
LSTR,	5,	6		
	6	7		
	7	1		
LSTK,	7,	T		



Hole (circle) placing

Main Menu>Preprocessor>Modeling>Create>Areas>Circle>Solid Circle Click keypoint 8 and enter radius value as 0.00625m. Or; CYL4,0.0125,0.0165,0.00625

	▲ Solid Circular	Area 🔀	
	Pick	C Unpick	
	WPX =		
	¥ =		
	Global X =		
	¥ =		
	Z =		
	WP X	0.0125	
	WP Y	0.0165	
	Radius	0.00625	
	ОК	Apply	
	Reset	Cancel	
	Help	1	
		-	1
ANSYS Main Menu 🛞			
ences			
Cessor Association Circular Area			
nent Typ © Pick C Unpick			
erial Prog WP X =			
tions Y =		:	5 4
deling Global X =	7	5	
ireate Y =			
uneypoli alines Z =			
Areas			
⊞ Rect; ^{WP Y} 0.0165			
Circle Radius 0.00625			
≫ <mark>So</mark>			
➢ An ОК Аррју			
Pa Reset Cance1			
By Help			
P Polygon			

Subtracting hole area

Main Menu>Preprocessor>Modeling>Operate>Booleans>Subtract>Areas

Firstly select the whole area and hit Ok then select the bigger area and click apply and then hole area that its center point is on keypoint8 is selected and click OK. Or use ASBA, 1, 2 and hit OK.

FCPAS Tutorial – Version 1.0



EXTRUSION

To create volume, we can easily extrude the area by 0.025m in the normal direction (*z* axes) *Main Menu>Preprocessor>Modeling>Operate>Extrude>Areas>By XYZ Offset* First, model is selected then the extrusion distance is entered.



Meshing the Model

Back or front area has to be meshed first. Before meshing we can select the back area (template area) with ASEL command using the coordinate (location) option.

ANSYS Academic Teaching Advanced Utility Menu (CC2)					
Eile Select List Plot PlotC ASEL, Type, Item, Comp, VMIN, VMAX, VINC, KSWP					
🗅 🛩 🖬 🚳 🕼 💡 🧱 asel, , loc,z,-0.001,0.001					

In the above command *Asel* is used to select a subset of areas that are located in the coordinate range specified. To be able to select the back area, very small distance is given between the minimum and maximum coordinate in *z* direction.

asel, , loc,z,-0.001,0.001

aplot



To obtain accurate fracture solution, we need to generate fine mesh near the crack tip. For this, we can use the KESIZE command to specify element size at the crack tip keypoint. First zoom into the crack tip region. Then issue the command

kesize, p

and pick the crack tip keypoint on the back area and write 0.0005m as the element size value SIZE in the window.

FCPAS Tutorial – Version 1.0

	Crack tip (back surface)
Elem Size at Picked KP Pick C Unpick Single C Box Polygon C Circle Count = 0 Maximum = 10 Minimum = 1 KeyP No. = C List of Items C Min, Max, Inc MK Apply Reset Cancel Pick All Help	Back Area
Element Size at Picked Keypoints [KESIZE] Element size at picked keypoints	×
SIZE Element edge length	0.0005
Show more options	No
OK Cancel	Help

Amesh,all 'Select the back area exactly. Be careful, in selecting. Use, zoom in, perspective view or rotate the model or issue other viewing commands to select the back area.

ī



For better meshing of the hole be must refinement using *Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Lines* Use isometric view when you select hole edges. Choose 1(Minimal) value. Note: All meshing and refining processes must be done in this stage. Because volume sweeping gives us hexahedral type volume mesh. Once volume sweeping is done, there is no way to return back to 2D mesh again, unless all mesh is deleted. Therefore all refinement and tuning processes on meshing stage must be finished at this point.

Now, we will refine the hole circumference. When asked by the program select all lines on the hole circumference.



We will do a minimum level of refinement.



Next, let us check the element size near the crack tip. First, zoom into the crack region and use **Ksel** to locate the exact location of the crack tip, since we will measure the element edge sizes ahead of and behind the crack tip. To do this, we use the *ndist* command (**ndist**,**p**) and measure the crack tip edge size. This gives us 4.8237299690E-04, which is fine enough.



Note: If inter-node distances had not been done fine enough, we would have returned back to the refinement process again after cleaning the mesh.

As can be seen above, the specified element size is achieved.

Applying Boundary Conditions

Because of the symmetry, our system has following BC's:

- 1. On symmetry area (Top Area): [from (0.0225, 0.03, 0) to (0.0625, 0.03, 0)] $U_y = 0$
- 2. On corner 1, constrain $U_x=U_y=U_z=0$
- 3. On corner 2, constrain U_x (to avoid the rotation about the Y axis. Note that: we could choose the corner 3 (U_z =0), instead.)

1. Apply the displacement constraint on symmetry area

ALLSEL,ALL

aplot

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>on Area or **DA**, **p** for area BC.

Symmetric area is top area of our model. This area must be constrained in *y* direction. Select the symmetry area carefully.



ctrl+right button.

To check the applied boundary conditions on areas, **DALIST** is used in the command line

Tile	Command			×
LIST CONST ** HARNING ANSYS, INC	RAINTS ON ALL SELE : PRE-RELEASE VERS TESTING IS NOT CO	CTED AREAS ION OF ANSYS 11.D HPLETE - CHECK RE	Beta Sults carefully **	
AREA 5	load label Symh	VALUE(S) 0.0000	0.0000	

2. Apply the displacement constraint on corner 1, constraint $U_x=U_y=U_z=0$

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>on Keypoints



Apply U,ROT on KPs	×
[DK] Apply Displacements (U,ROT) on Keypoints	
Lab2 DOFs to be constrained	All DOF UX UY UZ
Apply as	Constant value
If Constant value then:	
VALUE Displacement value	0
KEXPND Expand disp to nodes?	□ No
ОК Арріу	Cancel Help

To check the applied boundary conditions on areas and keypoints, **DKLIST** and **DALIST** is used in the command line



3. Apply the displacement constraint on corner 2, constraint U_x





[DK] Apply Displacements (U,ROT) on Keypoints	<u>×</u>
Lab2 DOFs to be constrained	All DOF UX UY UZ
Apply as	Constant value
If Constant value then:	
VALUE Displacement value	0
KEXPND Expand disp to nodes?	□ No
OK Apply	Cancel Help

Now let's list the last BC's.

	ADALIST	Commar	ıd							×
	<u>F</u> ile									
	LIST CO	NSTRAII	NTS ON	ALL SELECTED	AREA	s				
	AREA	5	LOAD Symp	LABEL 1	VALU Ø.	E(S) 0000	0.0000			
		Comman	d							×
	LIST SE CURRENT	ELECTED	DOF (ECTED	CONSTRAINTS ON DOF SET= UX	ALL UY	SELECTED UZ	KEYPOINTS			-
	KEYPC)INT 3 3 3 5	LOAD UX UY UZ UX	LABEL	VALU 0. 0. 0. 0.	E < S > 0000 0000 0000 0000 0000	0 - 0000 0 - 0000 0 - 0000 0 - 0000	EXP	KEY 0 0 0	•
Or; DA, FLST FITE FLST FITE !* /GO DK,F FLST FITE	5,SYM (7,2,1,3,OR (2,1,3,OR (7,2,1,3,OR (7,2,1,3,OR (7,2,1,3,OR (7,2,1,3,OR (7,2,15)	M DE,1 DE,1),UX,UY, DE,1	UZ, , , , ,							

!* /GO DK,P51X, ,0, ,0,UX, , , , , ,

Volume Sweeping

First, let's define layer size for the mesh to extruded. Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Layers>Picked Line



Select the on eline in z direction



	x
[ESIZE] Area Layer-Mesh Controls on Picked Line-	long the selected line.
STZE Element edge length	
or	
NDIV No. of line divisions	
KYNDIV SIZE,NDIV can be changed	V Yes
SPACE Spacing ratio (Normal 1)	0
LAYER1 Inner layer thickness	0
Size factor must be $>$ or $= 1$	
Thickness input is:	
	Size factor
	C Absolute length
(LAYER1 elements are uniformly-sized)	
LAYER2 Outer layer thickness	0
Transition factor must be > 1	
Thickness input is:	
	Transition fact.
	C Absolute length
(LAYER2 elements transition from LAYER1 size to global size)	
NOTE: Blank or zero settings remain the same.	
OK Apply	Cancel Help

Now we will extrude the 2D mesh in *z* direction and 20 layers will be generated in this direction. *Main Menu>Preprocessor>Meshing>Mesh>Volume>Volume sweep>Sweep*



Volume sweeping gives us prismatic mesh in the bounded volume, with respect to area mesh (template mesh). After sweeping the 2D mesh to generate the 3D mesh, we need to delete 2D elements, since FRAC3D requires 3D elements only.

To do this, we use Aclear, all 'Area mesh is deleted. Because of this, a gap occurs in the sequence of element numbers. To remove the gap we use, Numcmp, elem

To make sure no gap exists in the node numbers as well, we use Numcmp, node

ex ANSYS 11.0 Output Window	
CLEAR NODES AND ELEMENTS FROM ALL SELECTED AREAS	
CLEARED 1 AREAS, Ø LINES, Ø KEYPOINTS	
COMPRESS ELEMENT NUMBERS MAXIMUM ELEMENT NUMBER COMPRESSED FROM 4599 TO	4380
COMPRESS NODE NUMBERS MAXIMUM NODE NUMBER COMPRESSED FROM 20533 TO	20533
۲ <u> </u>	l DÍ

Applying Loads

Now we will apply the pin loads on a line along inner surface of the hole. To do this, we can apply concentrated forces on nodes located on these lines (We can not apply concentrated forces on line entities).

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>On nodes

Carefully pick the nodes on the bottom line and then click OK in the picking window.

NOTE: To do this in more easy way, we can use select, line and associating nodes on this line.



nsll,s,1

nplot

This select all nodes associated with the line selected). Note that 41 nodes are selected (20 layersx2+1).



Use *menu-pltctrl-numbering* and *switch* the nodes *on* to show the numbers of the nodes.

Plot Numbering Controls	×
[/PNUM] Plot Numbering Controls	
KP Keypoint numbers	C Off
LINE Line numbers	C Off
AREA Area numbers	C Off
VOLU Volume numbers	C Off
NODE Node numbers	🔽 On
Elem / Attrib numbering	No numbering
TABN Table Names	C Off
SVAL Numeric contour values	C Off
[/NUM] Numbering shown with	Colors & numbers
[/REPLOT] Replot upon OK/Apply?	Replot
ОК Арріу	Cancel Help



Now we will apply the force to hole's baseline. Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>On nodes



Select F_y direction, constant value and enter "-1/41" for load value, then click OK. Note that; force is applied on the 41 nodes.

	Apply F/M on Nodes	×
	[F] Apply Force/Moment on Nodes	
	Lab Direction of force/mom	FY
	Apply as	Constant value
	If Constant value then:	
	VALUE Force/moment value	-1/21
	OK Apply Cancel	Help
Select-every	rthing. <mark>Eplot</mark>	

FCPAS Tutorial – Version 1.0



Definition of Crack for FRAC3D

Now, we need to provide crack tip element and node number information for FRAC3D analysis. Zooming into the crack region, we can find which element and which nodes are located at the crack tip. (See detailed crack tip definition requirements in this tutorial for which elements and nodes to be selected). We need to identify the crack tip element on the bottom crack surface (with respect to chosen local coordinate system) immediately behind the crack tip. Using, **Select-Entities-Elements**, form the main menu (or Esel, p command) try to select the elements at the crack tip. For this, move your mouse pointer near to crack tip region. We have to select the element which is both at crack tip and on the crack surface.

eplot



Use *menu-pltctrl-numbering* and switch the nodes off to not show the numbers of the nodes.



After element selection, we list the selected element and save the file for FRAC3D analysis. Use **Elist** to see and save the list as a file.

Λ	ELIST	C	om	mar	nd										x
Eile	9														
LI	ST ALL	SEL	ECTEL) ELE	EHEN	TS.	(LIST	NODES)							_
**	HARNIN NSVS. TI	4G: 1 4C ТТ	PRE-F	RELEA NG TS	ASE (S NOT	VERS) T. COM	ION OF	ansys - Chec	11.0BE K RESU	TA LTS CA	REFIILI	V ***			
"	FIFH P	юп.	TVP		5 110 EQV (, co. SEV		NUDES	N NE00						
L .			1 II F			JEC		NODEO							
L .	1	1	2	1	٥	1	57	263	250	313	2080	6012	6278	5062	
L .							2081	380 6011	- 389 - 6277	153 5061	7890	8184	8139	7212	
L .	2	1	2	1	٥	1	2080	6012	6278	5062	2078	6014	6280	5064	
L .							7896	8184	8139	7212	7897	8185	8140	7213	
	2	1	0	1	а	1	2079	6013 6014	6279 6200	5063	2026	6016	6000	5066	
	3	T	2	Т	U	T	7897	8185	8140	7213	7898	8186	8141	7214	
							2077	6015	6281	5065					-

The element information (for the crack tip elements) is saved from the Elist window as *ct2.crelems*

ELIST Cor	mmand			×
LIST ALL SELECT	TED ELEMENTS. (LIS	r Nodes)		
** HARNING: PR	Farklı Kaydet			
FIEM MAT TV	Konum:	Ct2		💽 🔶 🖻 🖆
1 1	2	🚞 bin		
	Recent			
2 1				
3 1	Masaüstü			
4 1	🄌			
5 1	Belgelerim			
6 1	Bilgisayarım			
7 1	Aă Doğlantılarım			
	Ay Daylankilalini			
		Dosya adı:	ELIST.lis	•
		Kayıt türü:	Lister Files (*.lis)	•

In addition to crack tip elements, we also need to select nodes on the crack front. To be able to select the nodes of the crack front, it is required to select the crack tip line and then select the nodes associated with this line. Using, **Select-Entities-Lines**, crack tip line is selected. Then, **NSLL**, **S**, **1** is used to select all the nodes along the selected crack front line. Using **Nlist**, we can see that the crack front nodes numbers are: **18,2199**,....

Select lines
• Pick C Unpick
• Single C Box
C Polygon C Circle
C Loop
Count = 0
Maximum = 33
Minimum = 1
Line No. = 26
Ist of Items
🔿 Min, Max, Inc
J
OK Apply
Reset Cancel
Pick All Help



NSLL, S, 1 nlist

	r Command				×
Eile					
LIST ALL ** HARNI ANSYS,I	SELECTED NODES. DSYS= 0 NG: PRE-RELEASE VERSION OF ANSYS 11.0BR NC TESTING IS NOT COMPLETE - CHECK RESU	eta Jlts care	FULLY **	ĸ	
NODE 18 1581 1582 1583 1584 1585 1586 1587 1588 1589 1590 2105	X Y Z 0.40000E-01 0.30000E-01 0.0000 0.40000E-01 0.30000E-01 0.25000E-01 0.40000E-01 0.30000E-01 0.25000E-02 0.40000E-01 0.30000E-01 0.25000E-02 0.40000E-01 0.30000E-01 0.25000E-02 0.40000E-01 0.30000E-01 0.37500E-02 0.40000E-01 0.30000E-01 0.50000E-02 0.40000E-01 0.30000E-01 0.50000E-02 0.40000E-01 0.30000E-01 0.75000E-02 0.40000E-01 0.30000E-01 0.75000E-02 0.40000E-01 0.30000E-01 0.75000E-02 0.40000E-01 0.30000E-01 0.75000E-02 0.40000E-01 0.30000E-01 0.7500E-02 0.40000E-01 0.30000E-01 0.1250E-01 0.40000E-01 0.30000E-01 0.1250E-01	THXY 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.	THYZ 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.	THZX 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.	

The coordinates of the selected nodes (the crack tip nodes) are saved from the **Nlist window** as *ct2.crnodes*

	.IST Command				×			
File								
LIST	Farklı Kaydet							<u>? ×</u>
*** H Ans	Konum:	Ct2			•	+ 🗈 🕻	* 🎟 -	
NO	Recent Recent Masaiistii Delgelerim Bilgisayarm Silgisayarm	i bin						
		Dosya adı:	ct2.crnode	s			- [Kaydet
		Kayıt türü:	Lister Files	(*.lis)			•	İptal

Also using **Select-Everything** the whole element list is saved *ct2.elis Allsll,all elist*



Also using **nwrite** all nodes are saved as *ct2.node* automatically in current working directory.

<u>File</u> <u>S</u> elect	<u>L</u> ist <u>P</u> lot	Plot <u>C</u>	NWRITE, Fname, Ext,, KAPPND
🗅 🖻 🖬 🧉		? 🖾	nwrite

SBCTRAN is used to transfer solid model loads and boundary conditions to the FE model. Loads and boundary conditions on unselected keypoints, lines, areas, and volumes are not transferred.



And using flist the nodal loads are saved as ct2.flis

FCPAS Tutorial – Version 1.0

Save DataBase		
Save Database to	<u>D</u> irectories: d:\\ilyas\desktop\ct3	ОК
		Cancel
	Documents and Se ilyas Desktop Ct3	<u>H</u> elp
List Files of <u>Type</u> :	Dri⊻es:	
Database Files (*.db*) 💌	🗖 d: 💽	Net <u>w</u> ork

Now, we completed all modeling steps in $ANSYS^{TM}$. Now, we can save the model and close $ANSYS^{TM}$. We are ready to convert all the model information into FRAC3D format using the converter program.

T.2.3. Using converter codes for FRAC3D (Generation of *ct2.geo* File)

FRAC3D requires its model information in a specific format. To convert ANSYS[™] model files into FRAC3D format, we can use the *convert_ansys_frac3d.exe* program. The converter program can be run by typing, its path in MSDOS prompt or from the "Geo File" tab from FCPAS. Both methods are shown respectively.

T.2.3.1. Using convert_ansys_frac3d.exe

Run *convert_ansys_frac3d.exe*. Using this exe file, we can obtain *ct2.geo* file, which contains element connectivity, nodal coordinates, boundary conditions, loads, and crack information. The following table shows the steps and input for the current problem.

Input Element Connectivity FileName >	ct2.elis
Input Nodal Coordinate FileName >	ct2.node
Input Boundary Conditions FileName >	ct2.dlis
Input Nodal Forces FileName >	ct2.flis
Input Pressure Loaging FileName >	
Input Nodal Temperatures FileName >	If there is any temperature file, it's
	name is entered, otherwise hit return.
Input the corresponding mesh ID from the below list	1
1. 20 node quadratic- Hexahedorn (incl. mixed	
mesh)	
2. 15 node quadratic Pentahedron	
3. 10 node quadratic Tetrahedron	
4. 8 node linear Hexahedron	
5. 6 node linear Pentahedron	
6. 4 node linear Tetrahedron	
How many cracks do you have? < Input 0 if no crack>	1
Select an Option for Crack Front Information Input	1
Input Files for Crack Front Nodes and Elements: 1	
Input Crack Front Nodes and Elements Interactively: 2	
Input file name of file for Cr. Fronts Elements	ct2.crelems
Input file name of file for Cr. Fronts Nodes	ct2.crnodes
Input Coordinate Axis for Crack Front Node Order	3
1 for X, 2 for Y or 3 for Z Coordinate	
Are There SIF Constraints on The Crack Front? (Def: n)	У
Is The Constraint Along The Whole Crack Front or on	f
Specific Nodes?	
Along The Front: F, On Nodes: N	
Input The Type of Constraint/K1: 1, K2: 2, K3: 3	2
Input the Value of SIF	0
Do you have a more SIF constrains? < Def:n>	Y <mark>Bu ifade mevcut değil</mark>
Is The Constraint Along The Whole Crack Front or on	f
Specific Nodes?	

Along The Front: F, On Nodes: N	
Input The Type of Constraint/K1: 1, K2: 2, K3: 3	3
Input the Value of SIF	0
Do you have a more SIF constrains? < Def:n>	n
Generating The FRAC3D .geo File, Please Wait	Finalization Message

T.2.3.2. Using FCPAS

GED File RUN File Fracture Analysis Fracture Info Post Processing Visualization put Element Connectivity FileName> ct2 elis Browse put Nodal Coordinate FileName> ct2 node Browse put Boundary Conditions FileName> ct2 dis Browse put Nodal Forces FileName> ct2 file Browse put Nodal Forces FileName> ct2 file Browse put Nodal Temperatures FileName> browse Browse put Nodal Temperatures FileName> prove Browse put the corresponding mesh ID from the list 20 node quadratic-Hexahedron (nc. mixed mesh) Imput Files for Crack Front Nodes and Elements Imput Files for Crack Front Nodes and Elements es nput file name of file for Cr. Fronts Elements ct2 crelems Browse nput file name of file for Cr. Fronts Nodes ct2 crelems Browse crelems crelems Add Remove	Control Panel
GED File RUN File Fracture Analysis Fracture Info Post Processing Visualization put Element Connectivity FileName> ct2 elis Browse put Nodal Coordinate FileName> ct2 clis Browse put Boundary Conditions FileName> ct2 dlis Browse put Nodal Forces FileName> ct2 flis Browse put Nodal Forces FileName> ct2 flis Browse put Nodal Teorces FileName> ct2 flis Browse put Nodal Teorces FileName> ct2 flis Browse put Nodal Teorces FileName> Donde quadratic-Hexahedron (nc. mixed mesh) Image: Stress and S	Control Panel
put Element Connectivity FileName> ct2 elis Browse put Nodal Coordinate FileName> ct2 node Browse put Boundary Conditions FileName> ct2 dlis Browse put Nodal Forces FileName> ct2 flis Browse put Nodal Forces FileName> ct2 flis Browse put Nodal Forces FileName> ct2 flis Browse put Nodal Temperatures FileName> Browse Browse put the corresponding mesh ID from the list 20 node quadratic-Hexahedron (nc. mixed mesh) I lect an Option for Crack Front Information Input Input Files for Crack Front Nodes and Elements I es es Browse Browse nput file name of file for Cr. Fronts Elements ct2 crelems Browse crelems crelems Add Remove	Control Panel
In the set of th	ANSYS(TM)
Lit Hode Browse put Boundary Conditions FileName> ct2 dis put Nodal Forces FileName> ct2 fils put Nodal Forces FileName> Browse put Nodal Temperatures FileName> Browse put the corresponding mesh ID from the list 20 node quadratic-Hexahedron (inc. mixed mesh) ilect an Option for Crack Front Information Input Input Files for Crack Front Nodes and Elements es es nput file name of file for Cr. Fronts Elements ct2.crelems nput file name of file for Cr. Fronts Nodes ct2.crelems crelems Add Remove M Baraware	E*** 119(10) 4608 119(1) 90(19) 4706 10 9(19) 4706 2 4(19) 4706 2 4(19) 4706 2 4(19) 4706 4 49) 4706 4 49) 4828 0 4827
ct2.dlis Browse put Nodal Forces FileName> ct2.flis put Nodal Temperatures FileName> Browse put Nodal Temperatures FileName> Browse put the corresponding mesh ID from the list 20 node quadratic-Hexahedron (inc. mixed mesh) i i ilect an Option for Crack Front Information Input input Files for Crack Front Nodes and Elements res rt2.crelems nput file name of file for Cr. Fronts Elements ct2.crelems crelems crelems crelems Add	Average of the second s
Image: Contract of the state of the stat	ANSYS(TM)
put Nodal Temperatures FileName>	ANSYS(TM)
Dowse put the corresponding mesh ID from the list 20 node quadratic-Hexahedron (inc. mixed mesh) I les nput file name of file for Cr. Fronts Elements ct2.crelems nput file name of file for Cr. Fronts Nodes ct2.crelems crelems crelems crelems	ANSYS(TM)
elect an Option for Crack Front Information Input Input Files for Crack Front Nodes and Elements es nput file name of file for Cr. Fronts Elements ct2 crelems crelem	ANSYS(TM)
les nput file name of file for Cr. Fronts Elements ct2 crelems	ANSYS(TM)
Input life name of file for Cr. Fronts Letinentis [ct2.crelems] put file name of file for Cr. Fronts Nodes crelems crelems crelems credems c	Geo File
nput file name of file for Cr. Fronts NodesBrowse	Geothe
crelems cmodes Add Remove	RUN File
Lorelems Lorelems Lorelems	Frac 3D
crelems Add Remove	Fracture Info
cmodes Remove	Post Processing
All Demotes	Visualisation
All Remove	
vut Coordinate Axis for Crack Front Node Order Y or Z Coordinate	
e there SIF Constraints on The Crack Front?	
JF Parameters	
Is The Constraint Along The Whole Crack Front or on Specific Nodes? Along The Front	
Input The Type of Constraint:K1,K2,K3	
Input the Value of SIF	
F 2 0 F 3 0	
Remove	

Input file names can be selected by "Browse" buttons. "Generate Geo file" creates *ct2.elis_3d.geo* file. To go to "Run File" preparation, press "Next Step".

T.2.4. Generation of *.run FILE

Now, we need to create a run file which is also required for FRAC3D. We use *writerun_frac3d.exe* or FCPAS to generate *.run file (*ct2.run* file). The *.*run* file contains analysis type, material properties, solver type tolerances, body forces and local coordinate data.

T.2.4.1 Using writerun_frac3d.exe

The following table shows the steps and input for this specific problem.

Input Run File Name without ".run" (Include "_3d")	ct2.elis_3d
Is This A Non-linear Analysis (y,n)?, (Default:n)	Ν
Is This A Thermal Stress Analysis (y,n)?, (Default:n)	N
Do you have temperature dependent material properties (y,n)?,	
(Default:n)	
Is This A Fracture Analysis? (y,n)?, (Default:n)	У
Please Choose a Solver Type	1
Input "0" for Frontal Solver, "1" for PCG Solver'	
Input the Tolerance for PCG Solver (1.E-8 Recommended)'	1.E-8
Input Number of Different Materials in The Model, (Default:1)	1
Input Ex, Ey, Ez, Gxy, Gyz and Gxz for Mat.	Hit Enter
Input Nuxy, Nuyz, Nuxz for Mat.#	Hit Enter
Enter CTE Values (XX,YY,ZZ) and Tref for Mat.#	
Input Number of Different Materials in the Model (Default:1	
Input Integration Order for Enriched Elements	24
Do You Want to Use Transition Elements? (Default: n)	Y
Is This A Generalized Plane Strain Problem? (y,n) (Default: n)	n
Do You Have Other Tied DOF Sets? (y,n) (Default: n)	N
How Many Sets Do You Have?	
How Many Nodes to be Tied in Set#	
Input Node Numbers and Tieing Direction (x:1, y:2, z:3)	
How Many Nodes to be Tied in Set#	
Input Number of Increments, Max. Number of Iterations	
and Tolerance for Convergence(Default: 10, 20, 1.0E-6)	
Does Material#,NM, Exhibit Elasto-Plastic Behavior (y,n)? (Default:n)	
Input Initial Yield Stress and Number of Break-Points/Including The Initial	
Yield Stress	
Do You Want to Output The Results At The End of Every Increment? (y,n)	8888888
(Default:y)	
Input Number of Increments for Which The Results To Be Printed	
Input The Initial Temperature	
Is Elastic Modulus of Material # Temperature Dependent? (y,n)	
Is Poisson',"",'s Ratio of Material # Temperature Dependent? (y,n)	

Input Temperature and The Corresponding Poiss. Ratio Value for Point #	KXXXXXXX
Is CTE of Material # Temperature Dependent? (y,n)	
Input Number of Temperature Points and Tref	
Input Temperature and The Corresponding CTE Values (x,y,z) for Point #	
Is Yield Stress of Material # Temperature Dependent? (y,n)	
Input Number of Temperature Points	
Input Temperature Value for Set #	
Input The Corresponding Yield Stress and Plastic Strain Values for Temp.	
Set #', i3,' Stress Point #	<u> XXXXXXXX</u>
Do You Have Body Force Loading ? (y,n), (Default:n)	n
Input Type of Body Force Loading'1: Gravity, 2: Centrifugal	
Input The Acceleration	
Input Direction Cosines of Grav. Loading Direction	
Input Density for Material #	
Input The Angular Velocity in Rd/Sec	
Input The x,y,z Coord.s of The 1 st Point on The Rot. Axis	
Input The x,y,z Coord.s of The 2 nd Point on The Rot. Axis	
Do You Have Local Coordinate Systems To Be Included In The Analysis (y,n)?, (Default:n)	n
How Many Coordinate Systems Will Be Defined?	*******
Do You Really Want to Exit ? (y,n)	
Input The Initial Temperature'	
Is Elastic Modulus of Material # Temperature Dependent? (y,n)	
Input Temperature and Elastic Moduli (Ex,Ey,Ez,Gxy,Gyz and Gxz) for	8888888
Point #	
Is Poisson',"'",'s Ratio of Material # Temperature Dependent? (y,n)	
Input Temperature and The Corresponding Poiss. Ratio Values Nuxy,	RXXXXXX
Nuyz, Nuxz for Point #	XXXXXXX
Input Temperature and The Corresponding CTE Values (x,y,z) for Point #	

T.2. 4.2 Using FCPAS for *.run FILE

Parameters can be selected by clicking the objects in the tab. "Generate Run file" creates *ct2.elis_3d.run* file.

FCPAS Tutorial – Version 1.0

Due File Nerse without " aus"			Control Panel
Run File Name without .run	ct2.elis		
Analysis Type	 Linear Nonlinear 	Thermal Stress 🔽 Fracture 🗌 FGM	C*** MATERIAL INFORMATIO
Solver Type	C Frontal	nce> 1.e-8	0.3000000E+08 0.300000 0.11538462E+08
Material Properties			0.30000 0.30000 0
Number of different Material in The M	odel 1		0
Ex Ey Ez	Gxy Gyz Gx	z Nuxy Nuyz Nuxz Behaviour	
3.e7 3.e7 3.e7	1153846 1153846 1153	846 0.3 0.3 0.3 Orthotropic Elasto Plastic	
		Corthotropic C Elasto Plastic	
		Orthotropic Elasto Plastic	
FGM Properties(I	E(x)=E0.exp(Beta.x),Nu(x)=Nu0	exp(eta.x),Alpha(x)=Alpha0.exp(w.x))	ANSYS(TM)
			Geo File
			RUN File
			RUN File Frac 3D
			RUN File Frac 3D Fracture Info
Do You Have Other Tied DOF Se	ts	CTE Values (XX,YY,ZZ) and Tref	RUN File Frac 3D Fracture Info Post Processing
Do You Have Other Tied DOF Se How many DOF do you have?		CTE Values (XX,YY,ZZ) and Tref 1112	RUN File Frac 3D Fracture Info Post Processing Visualisation
Do You Have Other Tied DOF Se How many DOF do you have? 1 How many nodes 1	ts	CTE Values (XX,YY,ZZ) and Tref 1112 Temp Dependet Material Property Initial Temperature 1	RUN File Frac 3D Fracture Info Post Processing Visualisation
Do You Have Other Tied DOF Se How many DOF do you have? 1 How many nodes 1 Node number and direction 1	ts	CTE Values (XX,YY,ZZ) and Tref 1112 Temp Dependet Material Property Initial Temperature 1 E of Material 1 is temperature dep.	RUN File Frac 3D Fracture Info Post Processing Visualisation
Do You Have Other Tied DOF Se How many DOF do you have? How many nodes Node number and direction	ts	CTE Values (XX,YY,ZZ) and Tref 1112 Temp Dependet Material Property Initial Temperature 1 E of Material 1 is temperature dep. Temp. Points Ex,Ey,Ez,Gx,Gy,Gz 13e73e7	RUN File Frac 3D Fracture Info Post Processing Visualisation
Do You Have Other Tied DOF Se How many DOF do you have? How many nodes Node number and direction Have Body Force Loading		CTE Values (XX,YY,ZZ) and Tref 1112 Temp Dependet Material Property Initial Temperature 1 E of Material 1 is temperature dep. Temp. Points Ex,Ey,Ez,Gx,Gy,Gz 13e73e7 Nu of Material 1 is temperature dep.	RUN File Frac 3D Fracture Info Post Processing Visualisation
Do You Have Other Tied DOF Se How many DOF do you have? How many nodes Node number and direction Have Body Force Loading Type of Body Force Loading © C	ts	CTE Values (XX,YY,ZZ) and Tref 1112 Temp Dependet Material Property Initial Temperature 1 E of Material 1 is temperature dep. Temp. Points Ex,Ey,Ez,Gx,Gy,Gz 1 3 e7 3 e7 3 e7 Nu of Material 1 is temperature dep. Temp. Points Nux,Nuy,Nuz 1 0 3 0 3 0 3	RUN File Frac 3D Fracture Info Post Processing Visualisation
Do You Have Other Tied DOF Se How many DOF do you have? How many nodes Node number and direction Have Body Force Loading Type of Body Force Loading The Acceleration 9 88	ts	CTE Values (XX,YY,ZZ) and Tref 1112 Temp Dependet Material Property Initial Temperature 1 E of Material 1 is temperature dep. Temp. Points Ex,Ey,Ez,Gx,Gy,Gz 13e73e7 Nu of Material 1 is temperature dep. Temp. Points Nux,Nuy,Nuz 1030303 CTE temperature depended	RUN File Frac 3D Fracture Info Post Processing Visualisation Generate RUN File
Do You Have Other Tied DOF Se How many DOF do you have? How many nodes Node number and direction Have Body Force Loading Type of Body Force Loading The Acceleration Input Direction Cosines of Grav. Los Direction (wr Global Coord. Sys.)	ts	CTE Values (XX,YY,ZZ) and Tref 1112 Temp Dependet Material Property Initial Temperature 1 E of Material 1 is temperature dep. Temp. Points Ex,Ey,Ez,Gx,Gy,Gz 13e73e73e7 Nu of Material 1 is temperature dep. Temp. Points Nux,Nuy,Nuz 1030303 CTE temperature depended The Number of temperature points 11	RUN File Frac 3D Fracture Info Post Processing Visualisation Cenerate RUN File
Do You Have Other Tied DOF Se How many DOF do you have? How many nodes T How many nodes T Have Body Force Loading Type of Body Force Loading Type of Body Force Loading The Acceleration Input Direction Cosines of Grav. Los Direction (wrt Global Coord. Sys.) Second on Axis 111	ts	CTE Values (XX,YY,ZZ) and Tref 1112 Temp Dependet Material Property Initial Temperature 1 E of Material 1 is temperature dep. Temp. Points Ex.Ey,Ez,Gx,Gy,Gz Temp. Points Ex.Ey,Ez,Gx,Gy,Gz Temp. Points Nux,Nuy,Nuz 1030303 CTE temperature depended The Number of temperature points 11 Temperature and corresponding 1111	RUN File Frac 3D Fracture Info Post Processing Visualisation Cenerate RUN File
Do You Have Other Tied DOF Se How many DOF do you have? How many nodes Mode number and direction Have Body Force Loading Type of Body Force Loading Type of Body Force Loading The Acceleration Jan Input Direction Cosines of Grav. Los Direction (wrt Global Coord. Sys.) Second on Axis Density of Material	ts	CTE Values (XX,YY,ZZ) and Tref 1112 Temp Dependet Material Property Initial Temperature 1 E of Material 1 is temperature dep. Temp. Points Ex,Ey,Ez,Gx,Gy,Gz Temp. Points Ex,Ey,Ez,Gx,Gy,Gz Temp. Points Nux,Nuy,Nuz 10.30.30.3 CTE temperature depended The Number of temperature points Tamperature and corresponding CTE Values(x,y,z points) for point 1	RUN File Frac 3D Fracture Info Post Processing Visualisation Cenerate RUN File Cenerate RUN File
Do You Have Other Tied DOF Se How many DOF do you have? How many nodes Mode number and direction Have Body Force Loading Type of Body Force Loading Type of Body Force Loading The Acceleration Jan Input Direction Cosines of Grav. Los Direction (wrt Global Coord. Sys.) Second on Axis Till Have Local Coordinate Systems	ts	CTE Values (XX,YY,ZZ) and Tref 1112 Temp Dependet Material Property Initial Temperature 1 E of Material 1 is temperature dep. Temp. Points Ex,Ey,Ez,Gx,Gy,Gz Temp. Points Ex,Ey,Ez,Gx,Gy,Gz Temp. Points Nux,Nuy,Nuz Temp. Points Nux,Nuy,Nuz Temp. Points Nux,Nuy,Nuz Temperature depended 11 The Number of temperature points 11 Temperature and corresponding 11111 Integration Order 24	RUN File Frac 3D Fracture Info Post Processing Visualisation Cenerate RUN File Cenerate RUN File
Do You Have Other Tied DOF Se How many DOF do you have? How many nodes Mode number and direction Have Body Force Loading Type of Body Force Loading Type of Body Force Loading The Acceleration Jan Input Direction Cosines of Grav. Los Direction (wr Global Coord. Sys.) Second on Axis Till Have Local Coordinate Systems How many coordinate system	ts	CTE Values (XX,YY,ZZ) and Tref 1112 Temp Dependet Material Property Initial Temperature 1 E of Material 1 is temperature dep. Temp. Points Ex,Ey,Ez,Gx,Gy,Gz Temp. Points Nux,Nuy,Nuz Temp. Points Nux,Nuy,Nuz Temp. Points Nux,Nuy,Nuz Temperature depended The Number of temperature points Tamperature and corresponding CTE Values(x,y,z points) for point 1 Integration Order 24 V Use Transition Element	RUN File Frac 3D Fracture Info Post Processing Visualisation Cenerate RUN File Cenerate RUN File
Do You Have Other Tied DDF Se How many DDF do you have? How many nodes Mode number and direction Have Body Force Loading Type of Body Force Loading Type of Body Force Loading Type of Body Force Loading The Acceleration Jan Input Direction Cosines of Grav. Los Direction (wrt Global Coord. Sys.) Second on Axis Density of Material Have Local Coordinate Systems How many coordinate system	ts	CTE Values (XX,YY,ZZ) and Tref 1112 Temp Dependet Material Property Initial Temperature 1 E of Material 1 is temperature dep. Temp. Points Ex,Ey,Ez,Gx,Gy,Gz Temp. Points Ex,Ey,Ez,Gx,Gy,Gz Temp. Points Nux,Nuy,Nuz Temp. Points Nux,Nuy,Nuz Temperature depended The Number of temperature points 11 Temperature and corresponding 1111 Integration Order 24 V Use Transition Element Generalized Plane Strain Problem Input Number of Increments Max. Number of	RUN File Frac 3D Fracture Info Post Processing Visualisation Generate RUN File Č

To go to "Frac3D" tab, press "Next Step".

T.2.5 RUNNING FRAC3D T.2.5.1 Using *frac3d.exe*

To run the FRAC3D, three kinds of input files are required;

- *.run (compulsory)
- *.geo (compulsory)
- *.tem (optional)

FRAC3D gives the results in the following output files;

*.out

- *.str
- *.stn
- *.crk

Now, we are ready to run FRAC3D. To do this we can use *frac3d.exe*. When running FRAC3D, geo and run files names have to be entered. The following table shows the steps and input for this specific problem.

Input Run File Name without ".run"	ct2.elis_3d
Input geo File Name without ".geo"	ct2.elis_3d
Input ter File Name without ".ter"	Hit Enter

As a result, ***.crk** file is created like this:

FRACTURE MECHANICS INFORMATION

ct2.elis_3d.crk

24 X 24 X 24 INTEGRATION IS USED FOR ENRICHED CRACK TIP ELEMENTS TRANSITION ELEMENTS ARE INCLUDED IN THE ANALYSIS

CRACK NO: 1

CRACK TIP NODES:

182200220122022203220422052206220722082209221022112212221322142215221622172218571557165717571857195720572157225723572457255726572757285729573057315732573357342199

CRACK IN AN ORTHOTROPIC MATERIAL

K1 K2 K3

18	0.1643408E+04	0.0000000E+00	0.0000000E+00
2200	0.1789624E+04	0.0000000E+00	0.0000000E+00
2201	0.1939526E+04	0.0000000E+00	0.0000000E+00
2202	0.1971490E+04	0.0000000E+00	0.000000E+00
2203	0.2004858E+04	0.000000E+00	0.000000E+00
2204	0.2033197E+04	0.000000E+00	0.000000E+00
2205	0.2061087E+04	0.0000000E+00	0.0000000E+00
2206	0.2078518E+04	0.0000000E+00	0.0000000E+00
2207	0.2095062E+04	0.0000000E+00	0.0000000E+00
2208	0.2108352E+04	0.0000000E+00	0.0000000E+00
2209	0.2120820E+04	0.0000000E+00	0.0000000E+00
2210	0.2130562E+04	0.0000000E+00	0.0000000E+00
2211	0.2139531E+04	0.0000000E+00	0.000000E+00

2212	0.2146679E+04	0.000000E+00	0.000000E+00
2213	0.2153085E+04	0.000000E+00	0.000000E+00
2214	0.2158029E+04	0.000000E+00	0.000000E+00
2215	0.2162254E+04	0.000000E+00	0.000000E+00
2216	0.2165264E+04	0.000000E+00	0.000000E+00
2217	0.2167572E+04	0.000000E+00	0.000000E+00
2218	0.2168789E+04	0.000000E+00	0.000000E+00
5715	0.2169318E+04	0.0000000E+00	0.000000E+00
5716	0.2168789E+04	0.0000000E+00	0.000000E+00
5717	0.2167584E+04	0.0000000E+00	0.000000E+00
5718	0.2165264E+04	0.0000000E+00	0.000000E+00
5719	0.2162278E+04	0.0000000E+00	0.000000E+00
5720	0.2158029E+04	0.0000000E+00	0.000000E+00
5721	0.2153123E+04	0.0000000E+00	0.000000E+00
5722	0.2146679E+04	0.0000000E+00	0.000000E+00
5723	0.2139585E+04	0.0000000E+00	0.000000E+00
5724	0.2130564E+04	0.0000000E+00	0.000000E+00
5725	0.2120897E+04	0.0000000E+00	0.000000E+00
5726	0.2108353E+04	0.0000000E+00	0.000000E+00
5727	0.2095165E+04	0.0000000E+00	0.000000E+00
5728	0.2078529E+04	0.0000000E+00	0.000000E+00
5729	0.2061253E+04	0.0000000E+00	0.000000E+00
5730	0.2033191E+04	0.0000000E+00	0.000000E+00
5731	0.2004985E+04	0.0000000E+00	0.000000E+00
5732	0.1971658E+04	0.0000000E+00	0.000000E+00
5733	0.1939746E+04	0.0000000E+00	0.000000E+00
5734	0.1789692E+04	0.0000000E+00	0.000000E+00
2199	0.1640899E+04	0.000000E+00	0.000000E+00

G1

G2

GTOT

G3

18	0.3858269E-04	0.000000E+00	0.000000E+00	0.000000E+00
2200	0.4575365E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2201	0.5373944E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
2202	0.5552534E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
2203	0.5742077E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
2204	0.5905559E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
2205	0.6068684E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
2206	0.6171768E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2207	0.6270406E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2208	0.6350212E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
2209	0.6425540E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2210	0.6484707E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
2211	0.6539416E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
2212	0.6583186E-04	0.0000000E+00	0.0000000E+00	0.000000E+00

2213	0.6622536E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2214	0.6652986E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
2215	0.6679059E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
2216	0.6697667E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2217	0.6711955E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
2218	0.6719491E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
5715	0.6722773E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
5716	0.6719491E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
5717	0.6712029E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
5718	0.6697667E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
5719	0.6679210E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5720	0.6652987E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5721	0.6622771E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5722	0.6583189E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5723	0.6539746E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5724	0.6484716E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5725	0.6426004E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
5726	0.6350218E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
5727	0.6271023E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5728	0.6171833E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
5729	0.6069664E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
5730	0.5905523E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
5731	0.5742809E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
5732	0.5553480E-04	0.0000000E+00	0.0000000E+00	0.000000E+00
5733	0.5375165E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5734	0.4575709E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2199	0.3846499E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00

T.2.5.2 Using FCPAS to run FRAC3D

Select the *ct2.elis_3d.geo*, *ct2.elis_3d.run*, and *ct2.elis_3d.tem* (if required) files by browsing and press run button to run the Frac3D.exe in the background.
нер				
ISYS (TM) GEO File F	RUN File Fracture Analysis Fracture Info Po	st Processing Visualization		
				Control Panel
	DUN File News			
	RON File Name	ct2.elis_3d	Browse	
	GEO File Name	ct2 elia 2d	Province	
		[0.2.6IS_30	browse	
	Thermal Analysis?			
	TEM File Name		Browse	
			Diowse	
	Number of Processors to be Used	2	RUN	ANSYS(TM)
				Geo File
	Do you accept License Agreement?	⊙ Yes C No	Beview E	RUN File
				Eracture Analysis
				Erzeture Infe
			<u> </u>	Practure mile
	FCPAS Solver			Post Processing
	Fracture and Crack Propagation And	alysis System		Visualization
	LICENSE TERMS			
	Copyright 2010- Ali O. Ayhan, ayhar	1@sakarya.edu.tr		
	Permission is hereby granted, free of	charge, to any		
	subconstituents and the associated docu	mentation files		
	(the "Software"), for the rights to use, cop and publish resultsfrom the Software, subje	y, merge software, cted to the following condtions:		
	* Usage of the Software is permissible and	freely available		
	UNDER THE CONDITION THAT THE S	OFTWARE,		
	OF THE RESPECTIVE AUTHOR(S) AND	THE SOFTWARE IS NOT		
	DISTRIBUTED TO ANY OTHER THIRD	PARTIES.		
	* Usage of the Software for private use, re	search, publication		
	and education IS PERMISSIBLE.		×	

After FRAC 3D run ends, output files can be viewed in the "Fracture Info" tab. In the "Fracture Info" tab, you can browse anyfile to see its content and plot the K_1 , K_2 and K_3 data in an x-y plot.

To plot the K_1 , K_2 and K_3 data, just press "Plot SIF's" button.





*K*_{*i*} value is Mode –I crack stress intensity factor along the crack front and depends on both load and crack geometry as follow (plane strain conditions) [4];

$$K_I = \frac{P_Q}{B\sqrt{W}} f(\frac{a}{W})$$

where

$$f(\frac{a}{W}) = \frac{\left(2 + \frac{a}{W}\right)\left(0,886 + 4,64\frac{a}{W} - 13,32\left(\frac{a}{W}\right)^2 + 14,72\left(\frac{a}{W}\right)^3 - 5,6\left(\frac{a}{W}\right)^4\right)}{\sqrt{\left(1 - \frac{a}{W}\right)^3}}$$

P_Q=Load as determined in P-v diagram, B=Specimen thickness, W=Specimen width, a=crack length.

a [m]	W [m]	a/W	f(a/W)	B [m]	Pq [MN]	K1 [MPa⊡m]
2.75E-02	5.00E-02	0.55	11.36428629	0.025	1.00E+00	2032.90533

Frac3D gives K_1 =2147 MPa \mathbb{D} m. We can use the K_1 value at the mid thickness location. Difference is $\frac{2147 - 2032.90533}{2022.90533} = 0.056 \cong \%5.6$

2032.90533

T.2.6 POSTPROCESSING of FRAC3D Results Using movieassembly.f E FCPAS 1.0-1)Cracked Model Devoloped using ANSYS File Help ANSYS (TM) GEO File RUN File Fracture Analysis Fracture Info Post Processing Visualization Control Panel RUN File Name ct2.elis_3d Browse GEO File Name ct2.elis_3d Browse Create How Many Different Materials Do You Want To See? Input 1 Material ID Numbers 1 No -Do You Want to Prepare Animation File? ANSYS(TM) Geo File RUN File Frac 3D Fracture Info Post Processing Visualisation

T.2.7 Visualization of FRAC3D Results

FCPAS Tutorial – Version 1.0

e Help			
ISYS (TM) GEO File RUN File Fracture Analysis Fracture Info Post Processing	Visualization		
ISYS (TM) GEO File RUN File Fracture Analysis Fracture Info Post Processing Choose whether to draw an outline before deformation Input the scaling factor for displacement Which scalar to be displayed? Do you want to change range? from 0 ± to 20 ± Input the zooming scale VTK software Usage is Limited To This Copyrig http://www.vtk.org//TK/project/license.html	Yes Visualization	Show Results	Control Panel

To see the Cracked Model results, choose the parameter you would like to contour plot and press "Show Results" button.



Appendix A Definition of Crack in FRAC3D

In fracture analysis of solid structures, FRAC3D uses special 3-D enriched crack tip elements. The enriched elements are defined as the finite elements that have common border with the crack front. In FRAC3D, the crack is defined by the nodes along the crack front and the enriched elements on the bottom crack surface (with respect to the local orientation of the crack tip). The current version of the program interacts with FRAC2D and the converter program determines the crack front nodes automatically. On the other hand, the enriched element numbers as reference elements along the bottom crack surface is needed. These element numbers must be added to next line after the crack tip node numbers (at the end of the *_3d.geo file) in the order that the crack tip node numbers are listed, i.e., from back face of the model to front face. Examples 1 and 2 provided below explain the procedure. Alternatively, if 3-D *.geo file is prepared by translating the external list files from ANSYS (section 2.1.2 in this report), then users should prepare the crack tip node files and reference element files in ANSYS, the ANSYS-to-FRAC3D program would automatically add these fracture information into the *.geo file. Example 3 illustrated the definition of a curved crack tip.







For the latter case, for example, the element number information shown in the rectangle must be added by the user to the "*_3d.geo" file as shown below.

C*** FRACTURE MECHANICS DATA 1 10 120.00000000 4<u>31 2829 2830 1607 3455</u> 3456 3101 4201 4202 3800 106 107 108

C*** J-INTEGRAL PATHS

EXAMPLE.3.Two-Dimensional Mode-I Central Elliptical Crack in a Large Isotropic Medium

T.3.1 Problem Description

Develop a new problem case in the tutorial with the following data: A three-dimensional elliptical surface crack (a/c=0.3) in a finite-thickness plate under uniform tension with 2H x 2W x t (height x width x thickness), where H = W = 5c and t = 2a. compare your results from FRAC3D/FCPAS with those of Newman and Raju's surface crack formula.







In the ANSYS[™] tab of the FCPAS, we browse ""C:\Program Files\ANSYS Inc\v120\ansys\bin\intel\launcher120.exe".

FCPAS 1.0-1)Cracked Model Devoloped using ANSYS	
ile Help	
ANSYS (TM) GEO File RUN File Fracture Analysis Fracture Info Post Processing Visualization	
1- First, we browsed ANSYS TM executable file with its para	Control Panel
Browse C:\Program Files\ANSYS Inc\v120\ansys\bin\intel\auncher120.exe 2- To run, press it. Browse Run ANSYS Run Product Launcher	Change Working Directory
Also, we may want to run ANSYS TM from its product launcher.	ANSYS(TM) Geo File RUN File
Working Directory	Frac 3D Fracture Info
FCPAS Software Path	Visualisation
CA	$\leftarrow \rightarrow$

Figure T. 7 ANSYS[™] tab of the FCPAS

T.3.2 Generation of the Finite Element Model within the ANSYS[™] Preprocessor

First of all, we must take into account the problem type, which is plane strain. Also, due to the symmetry of the problem, only analysis of a quarter model is needed. We will model this two dimensional problem using one-layer (in the out-of-plane direction) three-dimensional elements. To do this, we will first mesh the back face of the domain with area (2D) elements and extrude the mesh into the third direction. To do this, we will use <u>PLANE82</u>, <u>SHELL 281</u> and SOLID95 elements from the ANSYSTM element library [3]. Note that ANSYSTM Help is very useful tool to identify and select the suitable elements for the problem of interest (Figure D.6).

Preprocessing

Change Directory

Before starting the model, create a folder in which you would like to work&change directory to this folder.

FCPAS Tutorial – Version 1.0

Change Working Directory C:\Documents and Settings\Uslu\Desktop\a_c_0.3
C:\Documents and Settings\Uslu\Desktop\a_c_0.3
a_c_0.3
ANSYS Academic Teaching Advance Eile Select List Plot Plot Clear & Start New Change Jobname Change Directory Change Title Resume Jobname.db Resume from

This brings up the 'Element Types' window. Click on the Add... button. The 'Library of Element Types' window appears. Highlight "<u>PLANE82-8 node 95</u>" Shell 281 and "SOLID95-20 node <u>93 95</u>". Click on OK or in command line, use (ET,1,95), (ET,2,93).

FCPAS Tutorial – Version 1.0



Define Material Properties

Main Menu>Preprocessor>Material Props>Material Models

On the right side of the `Define Material Model Behavior' window that opens, double click on `Structural', then `Linear', then `Elastic', finally `Isotropic'. Enter in values for the Young's modulus (EX = 200E9) and Poisson's ratio (PRXY = 0.3) of the plate material.

MP,EX,1,200e9

MP,PRXY,1,0.3



Modeling

Preprocessor-Modeling-Create-Volume-Block-By Dimensions

AND/15 Main Menu	9	A Create Block by Dimensions	×	
Preferences		A create block by Dimensions	<u>ם</u>	
C Preprocessor		[BLOCK] Create Block by Dimensions		
E thement Type		for and more proved even and		
E Real Constants		X1 X2 X-coordinates	5	
E Patena Props		nayna n coordinacaa	2 2	
C Madalana				
III Create		Y1,Y2 Y-coordinates	0 0.6	
III Erypoints				
(E) Lines		na na na secola de s		
El Arman		21,22 Z-coordinates	0 5	
C Volumes				
E Arbitrary				
E thek				
JR By 2 Corners & 2				
J ²¹ By Centr / Lorry J		OK Annhy	Capcel Help	
and the Connection of the		VII 19947	Cancor	
25 Cylinder				
a februar				
III Cope	-			
E form				
(i) Nodes				
(E) thements				
Contact Pair				
(2) Pipeng Hadels				
III Circuit				
E Racetrack Coll				
E Trateducers				
10 Operate				
E Pleve / Heddly				
a chev				
Check Genera				
III Delete				
III Cyclic Sector	-1			
	-			



We must be choose the Workplane-offset WP with - keypoints on upper X Y surface





Preprocessor-Modeling-Create-Lines-Arcs-By Cent & Radius First, we must create a circle r = 0.3



Then we delete the lines by *Delete-lines and below...*

1



Preprocessor-Modeling-Operate-Scale-Lines We must select ¼ crack line and apply scale





We must be delete the quarter part of the circle **ANSYS Command Line /repl.** We offset to the workplane to the centure of the model

Offset WP to Keypoints • Pick O Unpick 🖲 Single C Box C Polygon C Circle WY WX-C Loop et. Count 1 = Maximum = 10 Minimum 1 = KeyP No. = 10 Ist of Items 🔘 Min, Max, Inc 0K Apply Cancel Reset X Pick All Help

And we use Workplane-Offset WP with - keypoints



Create Keypoints on Wp

Preprocessor-Create-Keypoints-On Working Plane...

We are going to create 5 key points given in the following table:

Keypoints	X(m)	Y(m)	Z(m)
11	0	-0.03	0
12	-0.03	-0.03	0
13	-0.03	0	0
14	-0.03	0.03	0
15	0	0.03	0



Create Lines

Preprocessor-Modeling-Creat-Lines-Straight Lines

This is required to create the models boundary lines, successively like first 11 to 12, 12 to 13, 13 to 14, 15 to 10, 10to 11 and finally 10 to 13.



Create Lines

Preprocessor-Modeling-Creat-Areas-Arbitrary-By Lines

Pick all lines (Click OK in the picking window).



Preprocessor-Modeling-Operate-Extrude-Areas-Along lines First, we choose small areas and click Apply Button. After choosing line click Ok.





ANSYS Academic Tea	aching Advanced Utility Menu (a_c_0.3)
<u>File S</u> elect <u>L</u> ist	t <u>Plot</u> Plot <u>C</u> APLOT, NA1, NA2, NINC, DEGEN, SCALE
D 🖻 🖬 🔊 🛎	🖻 🖉 😢 aplot
ANSYS Toolbar	ANSYS Help
1	
nsys Command Pro	<mark>ompt (Isel,,p)</mark> Select line ın cevabı yok Komut ve resmi silinm
Utility Menu (a. c. 0.3	3)
LSEL, Type, Item	n, Comp, VMIN, VMAX, VINC, KSVVP
isei.,p	
eprocessor-Model	ling-Operate-Booleans-Substract-Volumes
rst we choose the c	all volume to pick apply and than select crack area to delete
rst we choose the c	all volume to pick apply and than select crack area to delete
rst we choose the c nsys Command Pro	all volume to pick apply <mark>and than</mark> select crack area to delete ompt (vsbv,p)
irst we choose the c nsys Command Pro ANSYS Academic Tea	all volume to pick apply and than select crack area to delete ompt (vsbv,p) eaching Advanced Utility Menu (a_c_0.3)
irst we choose the c nsys Command Pro ANSYS Academic Tea Eile <u>S</u> elect List	all volume to pick apply and than select crack area to delete ompt (vsbv,p) caching Advanced Utility Menu (a_c_0.3) t <u>Plot Plot</u> VSBV, NV1, NV2, SEPO, KEEP1, KEEP2
rst we choose the c nsys Command Pro ANSYS Academic Tea Eile Select List	all volume to pick apply and than select crack area to delete ompt (vsbv,p) caching Advanced Utility Menu (a_c_0.3) t Plot PlotC VSBV, NV1, NV2, SEPO, KEEP1, KEEP2 S & Vsbv,p
rst we choose the consys Command Pro ANSYS Academic Tea Eile Select List	all volume to pick apply and than select crack area to delete ompt (vsbv,p) caching Advanced Utility Menu (a_c_0.3) t Plot Plot VSBV, NV1, NV2, SEPO, KEEP1, KEEP2 S & VSbV,p
rst we choose the consys Command Process Academic Teals ANSYS Academic Teals Eile Select List	all volume to pick apply and than select crack area to delete ompt (vsbv,p) caching Advanced Utility Menu (a_c_0.3) t Plot Plot VSBV, NV1, NV2, SEPO, KEEP1, KEEP2 Solve Structure VSbV,p
rst we choose the consys Command Process Academic Teals Select List	all volume to pick apply and than select crack area to delete ompt (vsbv,p) caching Advanced Utility Menu (a_c_0.3) t Plot Plot VSBV, NV1, NV2, SEPO, KEEP1, KEEP2 Solve Store
rst we choose the consys Command Process Academic Teals Select List Difference Teals ANSYS Toolbar	all volume to pick apply and than select crack area to delete ompt (vsbv,p) caching Advanced Utility Menu (a_c_0.3) t Plot Plot VSBV, NV1, NV2, SEPO, KEEP1, KEEP2 Solve Store S
irst we choose the consys Command Process Academic Teals File Select List Dials Select List ANSYS Toolbar ANSYS Toolbar	all volume to pick apply and than select crack area to delete ompt (vsbv,p) caching Advanced Utility Menu (a_c_0.3) t Plot Plot VSBV, NV1, NV2, SEPO, KEEP1, KEEP2 Solve VSbV,p with Options s with Options
irst we choose the consys Command Processing Comman	all volume to pick apply and than select crack area to delete ompt (vsbv,p) caching Advanced Utility Menu (a_c_0.3) t Plot Plot VSBV, NV1, NV2, SEPO, KEEP1, KEEP2 with Options with Options with Options with Options with options with options with options with options
rst we choose the or nsys Command Pro- ANSYS Academic Tea Eile Select List D P P P P P ANSYS Toolbar ANSYS Toolbar Subtract Volumes (VSBV] Subtract Volumes SEPO Intersect bodry will KEEP1 Base volumes will	all volume to pick apply and than select crack area to delete ompt (vsbv,p) saching Advanced Utility Menu (a_c_0.3) t Plot Plot VSBV, NV1, NV2, SEPO, KEEP1, KEEP2 with Options with Options with Options with Options il have I be Deleted
irst we choose the consys Command Processing Command Processing Command Processing Command Processing Command Processing Command Processing Command Processing Command Processing Command Processing Command Comma Command Command Com	all volume to pick apply and than select crack area to delete ompt (vsbv,p) saching Advanced Utility Menu (a_c_0.3) t Plot Plot VSBV, NV1, NV2, SEPO, KEEP1, KEEP2 with Options with Options with Options with options will have I be will be Will be Kept
irst we choose the consys Command Processing Comman	all volume to pick apply and than select crack area to delete ompt (vsbv,p) saching Advanced Utility Menu (a_c_0.3) t Plot PlotC VSBV, NV1, NV2, SEPO, KEEP1, KEEP2 with Options with Options with Options with options will have I be Deleted will be Kept
irst we choose the consys Command Procession of the conservation o	all volume to pick apply and than select crack area to delete ompt (vsbv,p) taching Advanced Utility Menu (a_c_0.3) t Plot Plot VSBV, NV1, NV2, SEPO, KEEP1, KEEP2 VSBV,p with Options with Options will have Shared entities I be Deleted Will be Kept Apply Cancel Help



Ansys Command Prompt (allsel)

enu		
Plot <u>⊂</u> tris <u>V</u>	<u>V</u> orkPlane	Pa <u>r</u> ameters į
£ ?	📰 allsel	

Ansys Command Prompt (vplot)





Ansys Command Prompt (lplot)



Ansys Command Prompt (allsel)



Ansys Command Prompt (nummrg,kp) (no keypoint were merged)





To obtain accurate fracture solution, we need to generate fine mesh near the crack tip. For this, we can use the **KESIZE** command to specify element size at the crack tip keypoint. First zoom into the crack tip region.

Image: Constraint of the second se	
Celement Size at Picked Keypoints	×
[KESIZE] Element size at picked keypoints	
SIZE Element edge length	þ.003
Show more options	□ No
or 1	

Ansys Command Prompt (lesize,p)



FCPAS Tutorial – Version 1.0

lotCtric WorkPlana Paramotors Macro	ManuCtric Holp
	медасала Пер
QUIT POWRGRPH	
[1	~
VOLUMES TYPE NUM	
	Noncommercial
the second second second second second second second second second second second second second second second se	TH I
Celement Sizes on Picked Lines	×
[LESIZE] Element sizes on picked lines	
5IZE Element edge length	
NDIV No. of element divisions	80
(NDIV is used only if SIZE is blank or zero)	
KYNDIV SIZE,NDIV can be changed	Ves
5PACE Spacing ratio	
ANGSIZ Division arc (degrees)	
(use ANGSIZ only if number of divisions (NDIV) and	
element edge length (SIZE) are blank or zero)	
Clear attached areas and volumes	I No
	Cancel Help

Size Controls

Preprocessor-Meshing-Size cntrls-Manual Size-Global-Others

FCPAS Tutorial – Version 1.0

ANSYS Main Menu Preferences Preprocessor Element Type Real Constants Material Props Sections Modeling Meshattributes Meshattributes Meshattributes Meshattributes Global Gisze Cntrls Global Gisze Global Gisze Area Cntrls Gither Areas Lines Keypoints Layers Concentrat KPs Mesh Concatenate Mesh Mesh Mesh Mesh Mesh Mesh Mesh Mes	
∧ Other Global Sizing Options	×
[DESIZE] Other Global Sizing Options	
These settings are for mapped meshing and while	e la la la la la la la la la la la la la
not recommended can be used for free meshing w	when
SmartSizing is off.	
Minimum number of divisions per line	
MINL For lower-order elements	Þ
MINH For higher-order element	2
Maximum number of divisions per line	
MXEL For lower + higher-order	60
Maximum spanned angle for arcs	
ANGL For lower-order elements	15
ANGH For higher-order element	28
Element size (edge length)	
EDGMN Minimum edge length	
EDGMX Maximum edge length	
Target aspect ratio for adjacent line	
ADJF For free meshing	2
ADJM For mapped meshing	20
ОК	Cancel Help

Ansys Command Prompt (allsell)
enu
Plot⊆trls <u>W</u> orkPlane Pa <u>r</u> ameters !
Ansys Command Prompt (Iplot)
tels WorkE
Ansys Command Prompt (aplot)
e Workt
APLOT, NA1, NA2, NI
V Bell aplot
Ansys Command Prompt (vplot)
ot⊆trls <u>W</u> orkPlane Pa <u>r</u> ameters <u>N</u>
27 💡 🕎 vplot
Meshing The Areas
Prennacessor-Meshing-Mesh-Areas-Free
reprocessor mesning mesni Areas rice



Areas Sweep For Creating Crack Volume

Preprocessor-Meshing-Mesh-Volume Sweep-Sweep



Ansys Command Prompt (aclear,all)



We use **Aclear**, **all** ' Area mesh is deleted. Because of this, a gap occurs in the sequence of element numbers. To remove the gap we use, **Numcmp**, **elem**

Ansys Command Prompt (eplot)



Preprocessor-Meshing-Mesher Opts



MSHAPE] Set Element Shap	e	
2D Shape key	Quad	
3D Shape key	ET	
1		
<u></u> OK	Cancel Help	
nsys Command Pro	npt (vplot)	
Tris WorkPlane Param	aterc N	
L = 1 —		
🤶 🖭 vplot		
reprocessor-Mesh-V	olume-Free	
SYS Main Menu (Mesh Yolumes
Preterences Preprocessor	ELEMENTS	Pick O Unpick
		G Single C Par
		C Polygon C Circle
		C Loop
Mesh Attributes		Count = 0
■ MeshTool Fisite Cutris		Maximum = 3
Mesher Opts		Minimum = 1 Volu No =
Concatenate Mesh		
又 Keypoints 又 Lines		O List of Items
		C Min, Max, Inc
Volumes Mapped		
➢ Free		
 ➢ Free ☑ Volume Sweep ☑ Tet Mesh From 		0K Ann1v
Tet Mesh From Therface Mesh		OK Apply
Tece Volume Sweep Tet Mesh From Interface Mesh Modify Mesh Check Mesh		OK Apply Reset Cancel
Volume Sweep Tet Mesh From Interface Mesh Modify Mesh Check Mesh Clear		OK Apply Reset Cancel Pick All Help
Tree Tree		OK Apply Reset Cancel Pick All Help
		OK Apply Reset Cancel Pick All Help
Yolume Sweep Tet Mesh From Interface Mesh Modify Mesh Check Mesh Clear Checking Ctrls Numbering Ctrls Archive Model Coupling / Ceqn FLOTRAN Set Up		OK Apply Reset Cancel Pick All Help
Yolume Sweep Tet Mesh From Interface Mesh Godify Mesh Check Mesh Clear Checking Ctrls Numbering Ctrls Numbering Ctrls Archive Model Coupling / Ceqn FLOTRAN Set Up Multi-field Set Up Loads		OK Apply Reset Cancel Pick All Help
Volume Sweep Volume Sweep Tet Mesh From Interface Mesh Check Mesh Check Mesh Clear Checking Ctrls Numbering Ctrls Archive Model Coupling / Ceqn FLOTRAN Set Up Multi-field Set Up Loads Physics		OK Apply Reset Cancel Pick All Help
Volume Sweep Tet Mesh From Tet Mesh From Interface Mesh Check Mesh Check Mesh Clear Checking Ctrls Archive Model Coupling / Crls Archive Model Coupling / Ceqn FLOTRAN Set Up Multi-field Set Up Loads Physics		OK Apply Reset Cancel Pick All Help
Yolume Sweep Yolume Sweep Tet Mesh From Interface Mesh Modify Mesh Check Mesh Clear Checking Ctrls Numbering Ctrls Archive Model Coupling / Ceqn FLOTRAN Set Up Multi-field Set Up Loads Physics		OK Apply Reset Cancel Pick All Help
Volume Sweep Tet Mesh From Interface Mesh Check Mesh Clear Checking Ctrls Numbering Ctrls Archive Model Coupling / Ceqn FLOTRAN Set Up Multi-field Set Up Loads Physics		OK Apply Reset Cancel Pick All Help

Ansys Comm	nand Prompt (dalist)			
To check the	applied bound	lary condition	s on areas, D	ALIST is used in	the command lineFr
rls <u>W</u> orkPlane	Pa <u>r</u> ameters				
💡 🖭 dalis	t				
DALIST Command					×
LIST CONSTRAINTS	S ON ALL SELECTED AR	EAS			
*** NOTE *** No area constra:	ints to list.	CP = 4.5	78 TIME= 11:28:41	L	
1					

Ansys Command Prompt (da,p)

Apply the displacement constrains using



Apply Boundary Conditions

Because of the symmetry, our system has the following BC's:

Select the all left X Y surface... (With small areas)

Lab2 DOFs to be constrained		All DOF UX UY	
		UZ ROTX	
		ROTY	
		UX	
Apply as		Constant value	
If Constant value then:			
VALUE Displacement value			



Ansys Command Prompt (da,p)



Select the Y Z up areas but big area and the crack one part...







FCPAS Tutorial – Version 1.0

Apply Surf Ld on Areas	X
[SFA] Apply Surface Load on Areas	
Lab Type of surface load	Pressure
VALUE Load value	-1
VALUE2 Second value if required	
(bulk temperature or imaginary part)	
LKEY Load key, usually face no.	1
(required only for shell elements)	
	ancei Heip
Ansys Command Prompt (lplot) tris WorkF LPLOT, NL1, NL2, NINC Plot	




Ansys Command Prompt (nlist) a_c_0.3crnodes

The node information (for the crack tip nodes) is saved from the NLIST window as **a_c_0.3.crnodes**



g Advanced U ot Plot⊆ 7 _ ? _ √	tility Menu (a_c /SEL, Type, /sel,.p	
	Select volumes	
Ansys Com dvanced Utili	mand Prompt (eslv)	
PlotCES	LV, Type	
WorkF <mark>ESLN, *</mark>	Type, EKEY	
Ansys Comi	nand Prompt (eplot)	



Ansys Command Prompt (elist) a_c_0.3.crelems

The element information (for the crack tip elements) is saved from the Elist window as $a_c_{0.3.crelems}$



Plot <u>⊂</u> trls	<u>W</u> orkPlane Pa <u>r</u> ameters į
(ET)	
_	
Ansys	Command Prompt (Iplot)
rls <u>W</u> o	Drkfudlot and and annual
9	
<u></u>	
Ansys	Command Prompt (aplot)
; <u>W</u> ork	^F APLOT, NA1, NA2, NI
	aplot
Ansys	Command Prompt (vplot)
t <u>⊂</u> trls	WorkPlane Parameters N
7	vplot
Δηςνς	Command Promnt (enlot)
115 9 5	command i rompt (cplot)
vanced	Utility Menu (a_c_0.3)
Plot <u>C</u>	EPLOT
	eplot



Ansys Command Prompt (sbct)

SBCTRAN is used to transfer solid model loads and boundary conditions to the FE model. Loadsand boundary conditions on unselected keypoints, lines, areas, and volumes are not transferred. **sbct**







Ansys Command Prompt (nwrite)

Also using **nwrite** all nodes are saved as *a*_*c*_*03.node* automatically in current working directory.



Ansys Command Prompt (elis) $a_c_{0.3.elis}$ Also using Select-Everything the whole element list is saved as $a_c_{0.3.elis}$



Ansys Command Prompt (dlis) Using dlist displacement BC's are saved as *a_c_0.3.dlis*



Ansys Command Prompt (sflis) Using sflist pressure loads on elements are saved as *a_c_0.3.sflis*



Now, we completed all modeling steps in ANSYS[™]. Now, we are ready to convert all the model information into FRAC3D format using the converter program.

T.3.3 Using Converter Codes for FRAC3D (Generation of *cc3.geo* File)

FRAC3D requires its model information in a specific format. To convert $ANSYS^{TM}$ model files into FRAC3D format, we can use the *convert_ansys_frac3d.exe* program. The converter program can be run by typing, its path in MSDOS prompt or from the "Geo File" tab from FCPAS. Both methods are shown respectively.

T.3.3.1 Using convert_ansys_frac3d.exe in FCPAS

Run *convert_ansys_frac3d.exe*. Using this exe file, we can obtain *cc3.elis_3d.geo* file, which contains element connectivity, nodal coordinates, boundary conditions, loads, and crack information. The following table shows the steps and input for the current problem.

elp			
(TM) GEO File RUN File Fracture Analysis Fracture Info	Post Processing Visualization		
Inst Element Consertivity EileManes			Control Panel
Input Element Connectivity FileName>	a_c_0.3.elis	Browse	
Input Nodal Coordinate FileName>	a_c_0.3.node	Browse	CANA NIGINODS, NIGILLES CANA 19100 8408 CANA ELEMENT CONNECTIVITY
Input Boundary Conditions FileName>	a_c_0.3.dlis	Browse	66 0 1 0 4788 4795
Input Nodal Forces FileName>		Browse	0 3163
Input Pressure Loading FileName>	a_c_0.3.sflis	Browse	4796 0 0 0 4820 4827 4828 0
Input Nodal Temperatures FileName>		Browse	
Input the corresponding mesh ID from the list	20 node quadratic-Hexahedron (inc. mixed mesh)		
	1		
Select an Option for Crack Front Information Input	Input Files for Crack Front Nodes and Elements		
Files			ANSYS(TM)
Input file name of file for Cr. Fronts Elements	a_c_0.3.crelems	Browse	Geo File
Input file name of file for Cr. Fronts Nodes	a_c_0.3.cmodes	Browse	RUN File
			Frac 3D
			Fracture Info
		Add	Post Processing
a_c_0.3.cmodes		Remove	Visualisation
		All Remove	
Input Coordinate Axis for Crack Front Node Order			
X,Y or Z Coordinate	© X © Y © Z		
Are there SIF Constraints on The Crack Front?	Yes		
SIF Parameters	,		
Is The Constraint Along The Whole Crack Front or o	n Specific Nodes? Along The Front		
Input The Type of Constraint:K1,K2,K3	□ K1 □ K2 □ K3	Generate GEO File	
Input the Value of SIF	0		
	Add	*	
	Bemove		

Input file names can be selected by "Browse" buttons. "Generate Geo file" creates *cc3.elis_3d.geo* file. To go to Run File preparation, press "Next Step".

T.3.3.2 Generation of *.run file (*writerun_frac3d.exe*) using FCPAS

Now, we need to create a run file which is also required for FRAC3D. We use *writerun_frac3d.exe* or FCPAS to generate *.run file (*cc3.elis_3d.run* file). The *.*run* file contains analysis type, material properties, solver type tolerances, body forces and local coordinate systems data.

FCPAS 1.0-1)Cracked Model Devoloped using ANSYS		2
Help		
SYS (TM) GEO File RUN File Fracture Analysis Fracture Info Pos	st Processing Visualization	
Run File Name without ".run" a_c_0.3.elis		Control Panel
Analysis Type C Nonlinea	r Thermal Stress 🔽 Fracture 🔽 FGM	CARA MATERIAL INFORMATION
Saluer Tura		0.10000000E-07 0.3000000E+08 0.3000000
Material Properties	ance>]1.e-8	0.11538462E+08 0.30000 0.30000 0.
Number of different Material in The Model 1		C*** THERMAL LAODING
Ex Ey Ez Gxy Gyz Gi 3.e7 3.e7 3.e7 1153846 1153846 1153846	xz Nuxy Nuxz Behaviour 3846 0.3 0.3 Crthotropic Elasto Plastic	
	Contraction Contra	
	Orthotropic Elasto Plastic	
FGM Properties(E(x)=E0.exp(Beta.x),Nu(x)=Nu	0.exp(eta.x),Alpha(x)=Alpha0.exp(w.x))	ANSYS(TM)
		Geo File
		RUN File
		Frac 3D
Do You Have Other Tied DOF Sets	CTE Values (XX,YY,ZZ) and Tref	Fracture Info
How many DOF do you have?	Temp Dependet Material Property	Post Processing
How many nodes	Initial Temperature	Visualisation
Node number and direction	E of Material 1 is temperature dep.	
	Temp. Points Ex,Ey,Ez,Gx,Gy,Gz 13.e73.e7	
Have Body Force Loading	Nu of Material 1 is temperature dep.	
Type of Body Force Loading © Gravity C Centrifugal	Temp. Points 1 Nux, Nuy, Nuz 1 0.3 0.3 0.3	Generate RUN File
Input Direction Cosines of Grav. Loading	CTE temperature depended	
Direction (wrt Global Coord, Sys.) Second on Axis 111	Temperature and corresponding	
Density of Material	CTE Values(x,y,z points) for point 1	
Have Local Coordinate Systems	Integration Order 24	
How many coordinate system	✓ Use Transition Element Generalized Plane Strain Problem	
CSYS ID no, THXY THYZ THXZ angles	Input Number of Increments ,Max. Number of Iterations and Tolerance for Convergence 10 20 1.0e-6	
	Show The Results At The End of Every Increment?	

To pass "Frac3D" tab, press "Next Step".

T.3.4 Running FRAC3D T.3.4.1 Using *frac3d.exe*

To run the FRAC3D, three kinds of input files are required;

- *.run (compulsory)
- *.geo (compulsory)
- *.tem (optional)

FRAC3D gives the results in the following output files;

- *.out
- *.str
- *.stn
- *.crk

Now, we are ready to run FRAC3D. To do this we can use *frac3d.exe*. When running FRAC3D, geo and run files names have to be entered. The following table shows the steps and input for this specific problem.

Input Run File Name without ".run"	a_c_0.3.elis_3d
Input geo File Name without ".geo"	a_c_0.3.elis_3d
Input ter File Name without ".ter"	Hit Enter

 Image: C:\Documents and Settings\gnc_mhnds\Desktop\a_c_0380\f3d_fpt_sct_32bit.exe

 0

 F R A C 3 D

 Finite Element Analysis Program

 To exit (type E or e)

 0

 0

 0

 F R A C 3 D

 Finite Element Analysis Program

 To exit (type E or e)

 0

 0

 0

 0

 0

 0

 0

 0

 0

 0

 0

 0

 0

 0

 0

 0

 0

 0

 0

 0

 0

 0

 0

 0

 0

 0

 0

 0

 0

 0

 1

 0

 1

 0

 0

 1

 0

 0

 1

 0
 </tr

T.3.4.2 Using FCPAS to run FRAC3D

Select the $a_c_{0.3.elis_{3d.geo}}$, $a_c_{0.3.elis_{3d.run}}$, and $a_c_{0.3.elis_{3d.tem}}$ (if required) files by browsing and press run button to run the *Frac3D.exe* in the background.

FCPAS 1.0-1)Cracked	Model Developed using ANSYS				
Help					
SYS (TM) GEO File RUN	File Fracture Analysis Fracture Info Pos	st Processing Visualization			
					Control Panel
	RUN File Name GEO File Name Thermal Analysis? TEM File Name Number of Processors to be Used Do you accept License Agreement?	a_c_0.3.elis_3d a_c_0.3.elis_3d 2 • Yes No	Browse Browse RUN	Review Error File	ANSYS(TM) Geo File RUN File
			•		Fracture Analysis Fracture Info Post Processing
	FCPAS Solver				Visualization
	Fracture and Crack Propagation Ana	lysis System			
	LICENSETERMS				\leftarrow
	Copyright 2010- Ali O. Ayhan, ayhan	@sakarya.edu.tr			
	Permission is hereby granted, free of person obtaining a copy of this software, subconstituents and the associated docur (the "Software"), for the rights to use, copy and publish resultsfrom the Software.subje- 	charge, to any its all mentation files , merge software, ,ted to the following condtions:			
	* Usage of the Software is pemissible and UNDER THE CONDITION THAT THE SS INCLUDING APPROPRIATE SUBCONS' OF THE RESPECTIVE AUTHOR(S) AND DISTRIBUTED TO ANY OTHER THIRD 	freely available DFTWARE, TITUENTS, REMAIN UNDER COPYRIGHT THE SOFTWARE IS NOT PARTIES.			
	* Llance of the Coffigure for private use, me	earch publication			

After FRAC 3D run ends, output files can be viewed in the "Fracture Info" tab. In the "Fracture Info" tab, you can browse any file to see its content and plot the K_1 , K_2 and K_3 data in an x-y plot.



To plot the K_1 , K_2 and K_3 data, just press "Plot SIF's" button.





T.3.4.2 Post-processing of FRAC3D Results

SYS (TM) GEO File RUN File Reture Analysis Fracture Info Post Processing Visualization RUN File Name a.c.0.3.zlis_3d Browse GEO File Name a.c.0.3.zlis_3d Browse How Many Different Materials Do You Want To See? 1 Input 1 Material ID Numbers 1 Do You Want to Prepare Animation File? No v ANSYS(TM) Geo File RUN File Fracture Info Post Processing Visualization	Help				
RUN File Name	SYS (TM) GEO File RUN File Fracture A	Analysis Fracture Info Post Processing Visu	alization		
RUN File Name a.c.0.3.els_3d Browse GED File Name a.c.0.3.els_3d Browse How Many Different Materials Do You Want To See? 1 Input Material ID Numbers 1 Do You Want to Prepare Arimation File? No ANSYS(TM) Geo File RUN File Fracture Info Poat Processing Vaualisation					Control Panel
GED File Name	RUN Fi	le Name	a_c_0.3.elis_3d	Browse	
How Many Different Materials Do You Want To See? 1 Input 1 Material ID Numbers 1 Do You Want to Prepare Animation File? No Material ID Numbers 1 Do You Want to Prepare Animation File? No Material ID Numbers 1 Fracture Info Post Processing Visualisation Visualisation	CE0 5	la Nazza			
How Many Different Materials Do You Want To See? 1 Input Material ID Numbers Do You Want to Prepare Animation File? No	GEO FI	ie Name	a_c_0.3.elis_3d	Browse	
Input Material ID Numbers I Do You Want to Prepare Animation File? No ANSYS(TM) Geo File RUN File Frac 3D Fracture Info Post Processing Visualisation	How Ma	any Different Materials Do You Want To See?	1	Create	
Do You Want to Prepare Animation File?	Input	1 Material ID Numbers	1	*	
ANSYS(TM) Geo File RUN File Fracture Info Post Processing Visualisation	Do You	Want to Prepare Animation File?	No	Ť	
Geo File RUN File Frac 3D Fracture Info Post Processing Visualisation					ANSYS(TM)
RUN File Frac 3D Fracture Info Post Processing Visualisation Visualisation					Geo File
Frac 3D Fracture Info Post Processing Visualisation					RUN File
Fracture Info Post Processing Visualisation					Frac 3D
Post Processing Visualisation					Fracture Info
Visualisation					Post Processing
					Visualisation

T.3.5 Visualization of FRAC3D Results

To see the Cracked Model results, choose the parameter you would like to contour plot and press "Show Results" button.

FCPAS Tutorial – Version 1.0

nie Help ANSYS (TI	M) GEO File RUN File Fracture Analysis Fracture Info Post Proce	ssing Visualization		Control Panel
	Choose whether to draw an outline before deformation Input the scaling factor for displacement Which scalar to be displayed? Do you want to change range?	Yes 💽 1000 ZZ Stress 💽 Yes 💽		
	from 0 $\stackrel{\bullet}{\underline{}}$ to 2 $\stackrel{\bullet}{\underline{}}$	1	Show Results	ANSYS(TM) Geo File RUN File Frac 3D Fracture Info Post Processing
	VTK software Usage is Limited To This Cop	oyright		Visualisation





EXAMPLE.4. Crack Growth in Plate Using ANSYS Macro for Mode I

In this example the aim is to get crack growth profiles by using written macros. A threedimensional elliptical surface crack in a finite-thickness plate under uniform tension with 2H x 2W x t (height x width x thickness). Plate dimensions are width: 0.175m, height: 0.295m, thickness: 0.03m and also initial crack dimensional are crack length (c): 0.0196m, crack depth (a): 0.0144m.



First we open FCPAS Cracked Model Development with ANSYS after click 3DCPP&C

	3	D Mode-I Crack Propagation in Plates	Control Panel
Browse	C \Program Files\ANSYS Inc\v120\ansys\bin\intel\auncher120 exe	Run ANSYS	Change Working Directo
Browse	J		-
		Run Product Launcher	ANSYS(TM)
			Geo File RUN File
	Working Directory		Fracture Analysis Fracture Info
	FCPAS Software Path		Visualization 3DCPP and C
	CA		$\leftarrow \rightarrow$
	Curent Path [C:\Documents and Settings\Admin		

FCPAS 1.0-1)Cracked Model Developed using A	INSYS		
le Help NSYS (TM) GEO File RUN File Fracture Analysis Fracture	ure Info Post Processing Visualization 3DCPP king Directory nange Working Directory	Copy Macro	Choose Your Macro
$\begin{array}{c} \hline 0 \\ \hline \\ \hline \\ 2 \\ \hline \\ 2 \\ \hline \\ 2 \\ \hline \\ 2 \\ \hline \\ \hline$	e and Crack Dimensions	Creck Propagation Steps Generate ANSYS Model Generate GEO File Generate GEO File Generate RUN File Predict Next Profile Best_ellipse.fit	Control Panel ANSYS(TM) Geo File RUN File Fracture Analysis Fracture Info Post Processing Visualization 3DCPP8C Select propagation Plate Pressure Plate Displacement Cylinder Displacement Cylinder Displacement

Change Working Directory

Before starting to crack growth analysis create a folder in which you would like to work&change directory to this folder.

FC: Plate Pressure		
	Change Working Directory Project Name ANSYS Path ANSYS Path Copy Macro	Choose Axes of the Ellipse. x_ellipse y_ellipse Select y Select y
$\begin{array}{c} & & \\$	Plate and Crack Image: Second Sec	Blocks

Give the Crack Propagation Project Name

Enter a name, for example 'Crack_propagation'

	Change Working Directory C:\Documents and Settings\Adm Project Name Crack_propagation -ANSYS Path Copy Macro	In Desktop (propagation Choose Axes of the Ellipse; X_ellipse Select Y Salect Y
2W	Plate and Crack Dimensions	cks-
\downarrow \downarrow \downarrow σ_{o}	t Start Crack Length (c) Incre Crack Depth (a)	ting Crack Front Number 1 = == ement No Number of Steps Δa(max.) Crack_Lesize 1 1 == Start Crack Growt Analysis
	Material Properties Material Constant (C) Material Constant (n) Modulus of Elasticity (E)	2 1 2 Plot Crack Profile
20	Poisson Ratio (v) Stress Integration Order	
	Number of Processor	8 1 - Life 9 1 - Calculation
		Crack_Lesize: Number of elements along crack from

Choose 'Ansys Pat	th', in this example 'ansys121.exe' should be chos	en.
🖼 Plate Pressure		
σ_{e} σ_{e} σ_{e} σ_{e} σ_{e} σ_{e} σ_{e}	Change Working Directory Chock_propagation Project Name Crack_propagation Ac Image: Control of the second seco	f the Ellipse: y_ellipse Select y Start Crack Growth Analysis
	Ağ Bağlantların Dosya adı: ansys 120 exe Aç Dosya türü: Executable File("exe) Iptal Modulus of Elasticity (E) 5 1 Poisson Ratio (v) 6 1 Stress 7 1	Plot Crack Profiles
	Integration Order 8 1 2 Number of Processor 9 1 2 If Use transition element 10 1 2 Crack_Lesize: Number of element	Life Calculation

In the following picture is shown that, before picking '*Copy Macro*' button, the scheme has got project name and selected working directory.

😭 Plate Pressure		_	
	Change Working Directory C:\Documents and Sets Project Name Crack_propagation ANSYS Path Copy Macro ANSYS Path Copy Macro C\Program Files\ANSYS Inc\v120\ansys\bin\virtel\ansys120 exe	ngs'Admin'Desklop 'propagation Choose Axes of the Ellipse; x_ellipse Select v Select	ie V
	Plate and Crack Dimensions W H t Crack Length (c)	Blocks Number of Crack Growth Increment Blocks	
*	Crack Depth (a)	1 1 2 Start Crack G 2 1 2 Analysis	irowth ;
	Material Constant (C) Material Constant (n)	3 1 Plot Crack Pl	rofiles
\downarrow \downarrow \downarrow \downarrow \downarrow \downarrow \downarrow \downarrow \downarrow \downarrow	Modulus of Elasticity (E) Poisson Ratio (v) Stress		
	Integration Order Number of Processor		
	Use transition element	10 1 Crack_Lesize: Number of elements along crack	tion

When we select copy macro, specific prepared macro is copied from bin directory to working direction folder. At the same time, *'Choose Areas of the Ellipse'* is enabled.

Then 'Plate And Crack Dimensions' and 'Material Properties' are written. If you use transition element we check 'Transition Element' button.

🔛 Plate Pressure					
	Change Working Directory Project Name ANSYS Path ANSYS Path C:\Program Files\ANSYS Inc.\v	Crack_propagation Crack_propagation Copy Macro 1201aneys bin Vitel*aneys 120 exe	ngs/Admin/Desktop/propagation	Choose Axes of x_elipse X_*	the Ellipse: y_ellipse Y
	Plate and Crack Dimensions ↓/ H t Crack Length (c) Crack Depth (a)	0.175 0.295 0.03 0.0196 0.0144	Blocks Number of Crack Growth Increme Starting Crack Front Number Increment No Number of Steps Δ	a(max.) Crack_Lesize	Sulo dont
↓ x y y	Material Properties Material Constant (C) Material Constant (n) Modulus of Elasticity (E) Poisson Ratio (v) Stress Integration Order Number of Processor	7.10-10 1.8500 3.e7 0.3 -1.0 24 2			Start Crack Growth Analysis Plot Crack Profiles
	✓ Use transition element		10 1 🚊 Crack_L	esize: Number of eleme	nts along crack front

Terms of block is related to number of difference maximum crack advancement distance in a step along the crack front.(Δa_{max})If you would like to get more crack profiles, you can also go on by changing *'Start Crack Front Number'*.Increment number is equal number of crack growth increment blocks. We write number of step is each value of (Δa_{max}) to obtain order again. At that time *'Crack Lesize'* number of elements along crack front can be changed.

Plate Pressure						
a_{μ}	Change Working Directory Project Name ANSYS Path ANSYS Path C.\Program Files\ANSYS Inclv1	Crack_propagation Crack_propagation Copy Macro 201anaya*bin'urtel'anays 120 exe	gs VAdmin \Deskt	op'propagation	Choose Axes of t x_ellipse	he Ellipse; y_ellipse Y_
	Plate and Crack Dimensions – W H	0.175	Blocks Number of Cr Starting Crac	ack Growth Increment Block	ks 5 🕂	
	Crack Length (c) Crack Depth (a)	0.0196	Increment No 1 2	Number of Steps ∆a(max.)	Crack_Lesize	Start Crack Growth Analysis
	Material Properties Material Constant (C) Material Constant (n)	7.1D-10 1.85D0	3	3 (0.001) 2 (0.0012)	80	Plot Crack Profiles
\downarrow $($ $($ $($ $))$ $($ $)$	Modulus of Elasticity (E) Poisson Ratio (v)	3.e7 [0.3	5	1 0.0015	80	
	Integration Order	24	7 8			
	Use transition element	h r	9 10			Life Calculation
				Crack_Lesize:	Number of elemer	nts along crack front

For the solve, click to 'Start Crack Growth Analysis'.

😫 Plate Pressure						
	Change Working Directory Project Name ANSYS Path ANSYS Path C.\Program Files\ANSYS Inc\w;	Crack_propagation Copy Macr 120'ansys'bin Vintel'ansys120 exe	ings\Admin\Deskto	pp\propagation are an	Choose Axes of x_ellipse X	the Ellipse; y_ellipse Y_
$\begin{array}{c} \begin{array}{c} \begin{array}{c} \begin{array}{c} \begin{array}{c} a^{2} \\ \end{array} \end{array} \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \begin{array}{c} \begin{array}{c} \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \begin{array}{c} \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \begin{array}{c} \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \begin{array}{c} \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \begin{array}{c} \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \end{array} $	Plate and Crack Dimensions W H t	0.175	Blocks Number of Cr Starting Crac	ack Growth Incremen	t Blocks 5 🔹	
	Crack Length (c)	0.0196	Increment No	Number of Steps ∆a	max.) Crack_Lesize	
C:\WINDOWS\system32\cmd.exe	:k Front Node Order		- - ×	5 0.00	05 80 07 80	Start Crack Growth Analysis
1 for X, 2 for Y or 3 for Z G Are There SIF Constraints on TI Is The Constraint Along The Wh Along The Front: F, On Nodes: M Input The Type of Constraint	pordinate he Crack Front? (Def: Die Crack Front or on V	n) Specific Nodes?		3 1 0.00 2 1 0.00	1 80 12 80	Plot Crack Profiles
Mi: 1, K2: 2, K3: 3 Input The Value of SIF Do You Have More SIF Constraint Is The Constraint Along The Whe Along The Front: F. On Nodes: H Input The Using of Constraint Input, The Value of SIF Do You Have More SIF Constraint	:? (Def: n))le Crack Front or on 4 ;? (Def: n)	Specific Nodes?			15 80	
Generating The FRAC3D .geo Fi	le, Please Wait					Life
C:\Documents and Settings\Admin writerun_frac3d.exe" O<"C:\Doc 2.txt"	h\Desktop\propagation cuments and Settings\	>"C:\Program Files\FC Admin\Desktop\propaga	PAS\bin\ tion\inp ▼			Calculation
				Crack_Le	size: Number of eleme	ents along crack front

'Frac3d Solver'

🕼 Plate Pressure	
Change Working Directory Change Working Dir	ge Vetran' Desktop 'propagation Choose Axes of the Ellipse; x_ellipse X y Y y
Plate and Crack Dimensions W Plate and Crack Dimensions W 0.175 H 0.295 t 0.03 Crack Length (c) 0.0196	Blocks Number of Crack Growth Increment Blocks 5 😒 Starting Crack Front Number 1 😒 Increment No Number of Steps Δa(max.) Crack_Lesize
Example constraints State iteration step 50 max error 0.101804211519043 iteration step 100 max error 2.395231222754796E-002 iteration step 100 max error 2.17563714774719E-002 iteration step 200 max error 2.17563714774719E-002 iteration step 200 max error 2.17563714774719E-002 iteration step 250 max error 1.180085736166636E-002 iteration step 350 max error 1.92824874581081E-002 iteration step 350 max error 1.072924874581081E-002 iteration step 450 max error 1.613986315216640E-003 iteration step 550 max error 1.6139863745206-003 iteration step 550 max error 1.6139863712324E-003 iteration step 550 max error 1.07528074287970E-003 iteration step 650 max error 1.04375383133321E-004 iteration step 650 max error 2.04552203017565E-003 iteration step 650 max error 1.032830520384E-003 iteration step 750 max error 1.232830520384E-003 iteration step </td <td>Image: Second second</td>	Image: Second second
iteration step 850 max error = 1.2730613307573885-003 iteration step = 900 max error = 2.668187933111965E-003 iteration step = 1900 max error = 7.477264807089704E-006 iteration step = 1000 max error = 2.1763823345216E-002 iteration step = 1000 max error = 8.578823145987818E-004 iteration step = 1100 max error = 4.578823145987618E-004 iteration step = 1150 max error = 4.5788231459167167052511E-004 iteration step = 1150 max error = 6.451895809493972E-005	Life Calculation

After the crack growth analysis solution you can plot crack profiles click 'Plot Crack Profiles' button.





133

EXAMPLE.5. Crack Growth in Cylinder Using ANSYS Macro for Uniform Displacement Load (a/D=0.1, a/c=0.2)

In this example, the aim is to get crack growth profiles by using written macros. A threedimensional elliptical surface crack in a cylinder under uniform displacement load with D x H dimensions. Cylinder dimensions are D=1, H=5, and also initial crack dimensions are crack length c=0.5, crack depth a=0.1 unit.



H=5 a=0.1 c=0.5 Δa_{max}=0.1, 0.2, 0.3, 0.4, 0.5

We use symmetry of the fracture model.



FCPAS	Tutorial –	Version	1.0
-------	------------	---------	-----

e Help			
ISYS (TM) GEO File RUN File Fracture Analysis	s Fracture Info Post Processing Visualization 30CPP&C Vorking Directory Change Working Directory	Copy Macro ANSYS Path	Choose Your Macro
$\overbrace{\begin{array}{c} \hline a f^{*} \\ \hline 2c \end{array}}^{a f^{*}} 2c $	Plate and Crack Dimensions	Crack Propagation Steps Generate ANSYS Model	Control Panel
2₩	t Crack Length (c) Crack Depth (a) Crack Depth (a)	Generate GEO File Generate RUN File	ANSYS(TM) Geo File RUN File
	Material Properties Material Constant (C) Material Constant (n)	Fracture Analysis Predict Next Profile	Fracture Info Post Processing Visualization
	Maximum Crack Length Increment Modulus of Elasticity (E) Poisson Ratio	Best_ellipse.fit	
	Applied Stress		Select propagation Plate Pressure Plate Displacement Cylinder Pressure
	Use Transition Element Number of Processors		Cylinder Displacement

Change working directory and write "Project Name".

Change Working Directory		
Project Name		
ANSYS Path	Choose Axes of the Ellipse;	Life Calculation
ANSYS Path	Copy Macro x_ellipse y_ellipse	
	Klasöre Gözat	
Cylinder and Crack Dimensions		-
Crack Length (c)	FCPAS_Setup_64bit_debug	Life
Crack Depth (a)	⊕	Calculation
н		
XR	→ Spain_presentation ⊕ Carl Tez_son_hal	
YR	Thermal_FCPAS_07.06.2011	
ZR	C tmp	
Ø	🛞 🗁 Yeni Klasör 🛛 💌	
Material Properties	Yeni Klasör Oluştur Tamam İptal	
Material Constant (C)		
Material Constant (n)	6 1	
Modulus of Elasticity (E)	7 1 🚔 Start Crack Growth	
Poisson Ratio (v)	8 1 - Analysis	
Integration Order		
Number of Processor	10 1 Plot Crack Profiles	
Use transition element	Crack_Lesize: Number of elements along crack front	

Project Name	test		
ANSYS Path	Aç	Life Calculatio	n
ANSYS Path	Konum:		
	— 🙆 i	acon.exe SDYNA120.exe	
	Kullandıklarım	Animate.exe Storepost.exe Ans.Deployment.Config.ANSYS.exe Sxpost.exe	
Cylinder and Crack Dimensions		Ans.Deployment.Unconfig.ANSYS.exe Atdsh84.exe Ans_admin.exe Atdsh.exe	
D Crack Length (c)		ansys120.exe Awish84.exe Ansys120.exe Awish84.exe	Life
Crack Depth (a)		Diat.exe mitshell.exe Cal	culation
н		DisplayW.exe	
XR	Bilgisavanm	launcher 120.exe LicAlert.exe	
YR J ZR		s971.exe	
Ø	Ağ Bağlantılanm (losya adi: ansys120.exe 💌 Aç	
		osya tūrū: Executable File(*.exe) iptal	
Material Properties Material Constant (C)		5 1 -	
Material Constant (n)		6 1 2	
Modulus of Elasticity (E)		7 1 🚎 Start Crack Growth	
Poisson Ratio (v)		8 1 Analysis	
Number of December		9 1	
Number of Processor			
		10 1 -	
Use transition element Cylinder Displacement Change Working Directory Project Name	C:Documents and Set	10 1 E Crack Lesize: Number of elements along crack front	
Use transition element Use transition element Cylinder Displacement Change Working Directory Project Name ANSYS Path ANSYS Path C.\Program Files\ANSYS Inc.\v120\a	C:\Documents and Sett lest Copy Macr nays bin kintel'anays 120 exe	10 Crack_Lesize: Number of elements along crack front Cra	• • • • • • • •
Use transition element Use transition element Cylinder Displacement Change Working Directory Project Name ANSYS Path ANSYS Path C.\Program Files\ANSYS Inc\v120\va		10 Crack_Lesize: Number of elements along crack front Crack_Lesize: Number of elements along crack front	•
Use transition element Use transition element Cylinder Displacement Change Working Directory Project Name ANSYS Path ANSYS Path C.\Program Flex\ANSYS Inc\w120\a Cylinder and Crack Dimensions D	C:\Documents and Set lest Copy Macr nys' bin'intel'ansys 120.exe	10 Crack_Lesize: Number of elements along crack front rgs/Admin/Desidop tmp Choose Axes of the Ellipse: Choose Axes of the Ellipse: Choose Axes of the Ellipse: Ellipse Select Select	•
Use transition element Cylinder Displacement Change Working Directory Project Name ANSYS Path ANSYS Path Cylinder and Crack Dimensions D Crack Length (c)	C-VDocuments and Sett Pest Copy Macr nays bin kintel anays 120 exe	10 Crack_Lesize: Number of elements along crack front rga/Admin/Desktop/tmp Choose Axes of the Ellipse: Choose Axes of the Ellipse: Choose Axes of the Ellipse: Ellipse	Life culation
Use transition element Cytinder Displacement Change Working Directory Project Name ANSYS Path ANSYS Path CNProgram Files\ANSYS Inc.v120va Cylinder and Crack Dimensions D Crack Length (c) Crack Depth (a) H		10 Crack_Lesize: Number of elements along crack front nga/Admin/Desktop/tmp Choose Axes of the Ellipse: Life Calculation X_ellipse Select Y_Select Y Select Y Starting Crack Front Number	n Life culation
Use transition element Use transition element Change Working Directory Project Name ANSYS Path ANSYS Path C.\Program Flex-VANSYS Inc.vr120va Cylinder and Crack Dimensions D Crack Length (c) Crack Depth (a) H XR	ChDocuments and Sett I est Copy Macr sys'bin'intel'anays 120 exe	10 Crack_Lesize: Number of elements along crack front ngs/Admin/Desktop/tmp Choose Axes of the Ellipse: Life Calculation x_ellipse y_ellipse Select y Blocks Select y Starting Crack Growth Increment Blocks Crack_Lesize Increment No Number of Steps Ca(max.) Crack_Lesize Crack_Lesize	n Life culation
Use transition element Cylinder Displacement Change Working Directory Project Name ANSYS Path ANSYS Path CvProgram Files VANSYS Inclv/120va Cylinder and Crack Dimensions D Crack Length (c) Crack Length (c) H XR YR		10 Crack_Lesize: Number of elements along crack front ngs \Admin \Desidep \tmp Choose Axes of the Ellipse; Life Calculation x_ellipse y_ellipse Select y Blocks Select y Number of Crack Growth Increment Blocks Call Starting Crack Front Number Image Date 1 Image Date 2 Image Date	n Life culation
Use transition element Cylinder Displacement Change Working Directory Project Name ANSYS Path ANSYS Path CvProgram Files\ANSYS Inc\v120\u Cylinder and Crack Dimensions D Crack Length (c) Crack Depth (a) H XR YR ZR Ø		10 Crack_Lesize: Number of elements along crack front rgaVddnin\Desktop\tmp	n Life culation
Use transition element Cylinder Displacement Change Working Directory Project Name ANSYS Path ANSYS Path C:\Program Files\ANSYS Inc\v120\a Cylinder and Crack Dimensions D Crack Length (c) Crack Depth (a) H XR YR ZR Ø		10	Life culation
Use transition element Cylinder Displacement Change Working Directory Project Name ANSYS Path ANSYS Path CvProgram FleevANSYS Inclusions D Crack Length (c) Crack Length (c) Crack Length (c) H XR YR ZR Ø Material Properties Material Concenter (C)		10 Crack_Lesize: Number of elements along crack front ngp/Admin/Desktop/tmp Choose Axes of the Ellipse: y_ellipse y_ellipse Select y Blocks Select Number of Crack Growth Increment Blocks Crack_Lesize 1 1 2 1 3 1 4 1 5 1	n Life culation
Use transition element Cylinder Displacement Change Working Directory Project Name ANSYS Path ANSYS Path Cylinder and Crack Dimensions D Crack Length (c) Crack Length (c) H XR YR ZR Ø Material Properties Material Constant (C) Material Constant (n)		10 Crack_Lesize: Number of elements along crack front nge \Admin \Desidep \tmp Choose Axes of the Ellipse; Stating Crack Growth Increment Blocks Select v Number of Crack Growth Increment Blocks Select v Increment No Number of Steps 1 1 2 1 3 1 4 1 5 1 6 1	Life culation
Use transition element Cylinder Displacement Change Working Directory Project Name ANSYS Path C.VProgram Files\ANSYS Inc.v120va Cylinder and Crack Dimensions D Crack Length (c) Crack Length (c) Crack Depth (a) H XR Ø Material Properties Material Constant (n) Modulus of Elassicity (E)		10 Crack_Lesize: Number of elements along crack front nga/Vchain/Desktop/tap Choose Axes of the Ellipse: x_ellipse y_ellipse Starting Crack Growth Increment Blocks Starting Crack Front Number 1 1 1 1 2 1 3 1 4 1 5 1 6 1 7 1	Life culation
Use transition element		10	n Life culation
Use transition element Cytinder Displacement Change Working Directory Project Name ANSYS Path ANSYS Path CNProgram FleeVANSYS IncW120Va Cylinder and Crack Dimensions D Crack Length (c) Crack Depth (a) H XR YR ZR Ø Material Constant (C) Material Constant (n) Modulus of Elasticity (E) Poisson Ratio (v) Integration Order		10	n Life culation
Use transition element Cylinder Displacement Change Working Directory Project Name ANSYS Path ANSYS Path Cylinder and Crack Dimensions D Crack Length (c) Crack Length (c) Crack Depth (a) H XR YR ZR Ø Material Properties Material Constant (C) Material Constant (n) Modulus of Elasticity (E) Poisson Ratio (v) Integration Order Number of Processor		10 Crack_Lesize: Number of elements along crack front nge V.drint \Desidep \trip Choose Axes of the Ellipse: x_ellipse y_ellipse Starting Crack Growth Increment Blocks Starting Crack Front Number 1 1 2 1 3 1 4 1 5 1 4 1 5 1 4 1 5 1 6 1 7 1 9 1 9 1 10 1	Life culation
Use transition element Cylinder Displacement Change Working Directory Project Name ANSYS Path ANSYS Path C:\Program Files\ANSYS Inc\v120\us Cylinder and Crack Dimensions D Crack Length (c) Crack Length (c) Crack Length (c) Crack Depth (a) H XR YR ZR Ø Material Properties Material Constant (C) Material Constant (n) Modulus of Elasticity (E) Poisson Ratio (v) Integration Order Number of Processor Use transition element		10	Life culation

When we select "ANSYS Path", "Copy Macro" button is enabled. We click "Copy Macro" button and copy cylinder displacement macro into the working directory. At the same time, when we click "Copy Macro", "Choose x_ellipse" is enabled.

Characterize Disasters 1			
Change Working Directory C:\Docu	ments and Settings \Admin \Desktop \tmp		
Project Name			
ANSYS Path		Choose Axes of the Ellipse;	Life Calculation
ANSYS Path	Copy Macro	x ellinse v ellinse	
		Select V Select V	
C:\Program Files\ANSYS Inc\v120\ansys\bin\intel\	ansys120.exe		
Cylinder and Crack Dimensions	Blocks		
Crack Length (c)	Number of Crack Growth Increment E	Blocks	Life
Crack Denth (a)	Starting Crack Front Number	1 =	Calculation
	-	,	
YP	Increment No Number of Steps /	Δa(max.) Crack_Lesize	
YR	- 1 1 🔁 0.	80	
ZR	2 1		
Ø	3 1 🚽		
	4		
Material Properties			
Material Constant (C)			
Material Constant (n)	6 1 🔆		
Modulus of Elasticity (E)	7 1 😤	Start Crack Growt	h
Poisson Ratio (v)	8 1 📩	Analysis	
Integration Order	9 1		
Number of Processor	10	Plot Crack Profile	s

In this example, we select X and Y axes for ellipse fitting.

Change Working Directory C:\Documents and Se	sttings\Admin\Desktop\tmp				
Project Name test					
ANSYS Path			Choose Axes of the	Ellipse;	Life Calculation
ANSYS Path Copy Mac	cro		y allinea	v ellinea	
			X V	Y V	
C:\Program Files\ANSYS Inc\v120\ansys\bin\intel\ansys120.exe	8		,		
Cylinder and Crack Dimensions	Blocks				
	Number of Crack Gro	wth Increment Blocks 1			Life
Crack Denth (a)	Starting Crack Front	Number 1	-		Calculation
н			_		
XR	Increment No Numb	er of Steps ∆a(max.)	Crack_Lesize		
YR	1 1				
ZR	2 1				
Ø	3 1				
	4 1				
Material Properties	5 1				
	6				
Material Constant (II)				1	
Madulus of Electricity (E)	/ 1			Start Crack Growth Analysis	
Modulus of Elasticity (E)					
Modulus of Elasticity (E) Poisson Ratio (V) Integration Order	8 1				
Modulus of Elasticity (E) Poisson Ratio (v) Integration Order	8 1 9 1			Plot Crack Profiler	

We write cylinder and initial crack dimensions.

Change Working Directo	ry C:\Documents :	and Settings Vidmin Desktop Ymp	
ANSYS Path	Jees	Choose Aves of the Ellipse:	
Anororum			
ANSYS Path	Co	py Macro x_ellipse y_ellipse	
C:\Program Files\ANSYS In	c\v120\ansys\bin\intel\ansys1	20.exe	
Cylinder and Crack Dimen	sions	Blocks	
D	1.0	Number of Crack Growth Increment Blocks 1	
Crack Length (c)	0.5		ion
Crack Depth (a)	0.1	Starting Crack Front Number 1 🚔	
н	5.0		
XR	-0.5	Increment No Number of Steps Za(max.) Crack_Lesize	
YR	0.0		
ZR	5.0	2 1 🚔	
ø	0.0	3 1 🚎	
		4	
Material Properties			
Material Constant (C)			
Material Constant (n)		6 1 🚊	
Modulus of Elasticity (E)		7 1 📩 Start Crack Growth	
Poisson Ratio (v)		8 Analysis	
Integration Order			
Number of Processor		Plot Crack Profiles	
		10 1 🚎	

XR: Crack center coordinate.YR: Crack center coordinate.ZR: Crack center coordinate.Ø: Angle with X axes.

Write material properties.

Cylinder Displacement				_
Change Working Director	C:\Documents and	ettings\Admin\Desktop\tmp		
Project Name	test			
ANSYS Path			Choose Axes of the Ellipse;	Life Calculation
ANSYS Path	Сору М	cro	x_ellipse y_ellipse	
C:\Program Files\ANSYS Inc\	.v120\ansys\bin\intel\ansys120.	e		
Cylinder and Crack Dimensi	ons	Blocks		
D	1.0	Number of Crack Growth Increment Blocks	*	
Crack Length (c)	0.5		Life	
Crack Depth (a)	0.1	Starting Crack Front Number 1	÷	Calculatori
н	5.0	Increment No. Number of Steps Aa(max)) Crack Leeize	
XR	-0.5			
YR	0.0			
ZR	5.0	2 1 🔆		
ø	0.0	3 1 🔆		
		4 1 😴		
Material Properties	7.1D-10	5 1		
Material Constant (C)	3.000			
Maderial Constant (n)	0			1
Modulus of Elasticity (E)	3.67	7 1 🛨	Start Crack Growth Analysis	
Poisson Ratio (v)	10.3	8 1 😴		
integration Order	10	9 1 😴		
Number of Processor	8	10 1	Plot Crack Profiles	
☑ Use transition element		Crack Lesize:	Number of elements along crack front	

Change Working Directo	C:\Documents an	d Settings\Admin\Desktop	o'tmp				
Project Name	test						
ANSYS Path					Choose Axes of the	Ellipse;	Life Calculation
ANSYS Path	Сору	Macro			x ellipse	v ellipse	
C) Browner Flori ANSYS In					X	Y	
UC: (Program Files (AINS TS Inc	C V 120 Vansys voin Vintei Vansys 120	J.exe					
Cylinder and Crack Dimen	sions	Blocks					
D	1.0	DIOCING					
Crack Length (c)	0.5	Number of Cra	ck Growth Increme	ent Blocks 5	±		Life
Crack Depth (a)	0.1	Starting Crack	Front Number	1	*		Calculation
н	5.0	Increment No.	Number of Steps	As(max.)	Crack Lesize		
XR	-0.5	increment No	Number of Steps				
YR	0.0		5 🗉	10.01	180		
ZR	5.0	2	5 🛨	0.02	80		
ø	0.0	3	5 📫	0.03	80		
		4	5 🗧	0.04	80		
Material Properties	710.10						
Material Constant (C)	7.10-10	5	2 🗉	10.05	180		
Material Constant (n)	3.0D0	6	1 🚊				
Modulus of Elasticity (E)	3.e7	7	1 -			Start Crack Growth	
Poisson Ratio (v)	0.3	8	1 =			Analysis	
Integration Order	18	9	, 2		,		
		5	1	1			

We click "Start Crack Growth Analysis" and crack growth analysis starts. Frac3D Solver

Change Working Directory	C:\Docum	nts and Settings Vdmin\Desktop\tmp	
ANSYS Path	,	Choose Axes of the Ellipse;	Life Calculation
ANSYS Path		Copy Macro x_ellipse y_ellipse	
C:\Program Files\ANSYS Inc\\	r120\ansys\bin\intel\ar	sys120.exe	
Cylinder and Crack Dimensio	ons	Blocks	
D	1.0	Number of Crack Growth Increment Blocks 5	
Crack Length (c)	0.5		Life
Crack Depth (a)	0.1	C:\WINDOWS\system32\cmd.exe	
н	5.0		
XR	-0.5	G:\PFogPam Files\HMSYS Inc\VI2U\ansys\bin\intel/cd "G:\Doc min\Desktop\tmp"	uments and Settings\Ha
YR	0.0	C:\Documents and Settings\Admin\Desktop\tmp>copy test.elis PC.elis 3d.maturon	_3d_front1.matprop 3DC
ZR	5.0	1 dosya kopyaland?.	
Ø	0.0	C:\Documents and Settings\Admin\Desktop\tmp>copy test_Macr 3DCPC_Macro_Parameter.inp 1 dosya kopyaland?.	o_Parameter_front1.inp
Material Properties		C:\Documents and Settings\Admin\Desktop\tmp>"C:\Program Fi	les\ANSYS Inc\v120\ans
Material Constant (C)	7.1D-10	\3DCPC_macro.out"	ispiacement.inp 0.
Material Constant (n)	3.0D0		
Modulus of Elasticity (E)	3.e7		
Poisson Ratio (v)	0.3		
Integration Order	18		•
Number of Processor	8	10 1	Diles
The second second			

Change Working Director	y C:\Docume	nts and Settings\Admin\Desktop\tmp	
Project Name	test		
ANSYS Path		Choose Axes of the Ellipse;	Life Calculation
ANSYS Path	1	Copy Macro	
C:\Program Files\ANSYS Inc	\v120\ansys\bin\intel\an	sys 120.exe	
Cylinder and Crack Dimens	ions	Blocks	
D	1.0	Number of Crack Growth Increment Blocks 5	
Crack Length (c)	0.5		Life
Crack Depth (a)	0.1	C:\WINDOWS\system32\cmd.exe	
н	5.0		
XR	-0.5	Wall Clock Time (Seconds)= 3.76 Starting Fracture Data Pre-processing	
YR	0.0	Wall Clock Time (Seconds)= 4.37	
ZR	5.0	Hall Clock Time (Seconds)= 5.09	
ø	0.0	Start Adjusting Data Structure	
		ADJUSTMENT OF DATA STRUCTURE IS FINISHED Wall Clock Time (Seconds)= 5.09	
Material Properties		Adjusting Data Structure Done	
Material Constant (C)	7.1D-10	Wall Clock Time (Seconds)= 5.09 Start for Searching Non-Zero Terms	_
Material Constant (n)	3.0D0	SEARCHING FOR NONZERO TERMS IN STRUCTURE IS DONE	
	3.e7	Wall Clock Time (Seconds)= 10.42 Searching Non-Zero Terms Done	
Modulus of Elasticity (E)		Wall Clock Time (Seconds)= 10.46	
Modulus of Elasticity (E) Poisson Ratio (v)	0.3	Stant Element Matnix Calculation & Occomblu	
Modulus of Elasticity (E) Poisson Ratio (v) Integration Order	0.3	Start Element Matrix Calculation & Assembly	-

After solution, click "Plot Crack Profiles" button.

charge moning process		and sense of second second					
Project Name	test						
ANSYS Path					Choose Axes of the	Ellipse;	Life Calculation
ANSYS Path	1 0	oov Macro			a allows		
					X_empse	y_empse	
C:\Program Files\ANSYS Inc\	v 120'aneys \bin \intel`aneys	120.exe			1.9	1 . 2	
Cylinder and Crack Dimensi	ons	Blocks					
D	11.0	Number of Cra	ck Growth Increm	nent Blocks 5	축		116
Crack Length (c)	0.5						Calculation
Crack Depth (a)	0.1	Starting Crack	Front Number	11	1		
н	5.0	Increment No	Number of Steps	s Δa(max.)	Crack_Lesize		
XR	-0.5	1	5 -1	0.01	80		
YR	0.0	2	[E]	0.02	lon		
ZR	5.0		12 11	10.02	100		
0	lora	3	5 当	10.03	80		
Meterial Descention		4	5 1	0.04	80		
Material Constant (C)	7.1D-10	5	2 🛨	0.05	80		
Material Constant (n)	3.000	6	月日				
Modulus of Elasticity (E)	3.e7	7				Start Crack Growth	
Poisson Ratio (v)	0.3	8	1 -			Analysis	
Integration Order	18			·	· · · · · ·	1	
Number of Processor	8		1 2			Plot Crack Profiles	
		10	1 =	1			
🔽 Use transition element			Cra	ck_Lesize:	Number of element	nts along crack front	

гаск_рготне	es								
rowse	ellipse_final.inp Select "step.inp" file		1.2		· · · ·	Title			_,
	Plot Crack Pro	Aç Konum:	: 🗀 tmp		▼ ⇐ €] • 🗗 🎦			
<-Y Values		En Son	ellipse_final.inp step.inp						-
		Kullandıklarım							
		Masaüstü							
		Belgelerim							
		Bilgisayanm							
		S							
		Ag baglantilarim	Dosya adı: jei Dosya türü: el	ipse_final.inp ipse_final(*.inp)		Aç ipta			
			0.2						-
otal profiles :	00		0.0					· · · · · · ·	
			0.0	0.2	0.4	0.6	0.8	1.0	1

Select step.inp file from the working directory.




You can see symmetry of the crack profiles.





EXAMPLE.6. Plate Crack Insertion and Fracture Analysis

We generate finite element model without crack using ANSYS[™] Dimensions of plate are 2W=50 mm, 2H=50mm, t=5 mm. Plate is subjected to uniform tension loading.



After we get finite $ANSYS^{TM}$ model without crack, we start to insert crack into the plate. In this example, a/t and a/c ratios are equal to 0.2. So, crack length (2c) is 5 mm. and also crack depth (a) is 1 mm.

2W= 50 mm. 2H= 50 mm. t= 5 mm. 2c=5 mm. a= 1 mm.



First, we select "Crack Insertion and Fracture Analysis" button.

Select "working directory" and *.node extension file, comes from $ANSYS^{TM}$.

prking Directory-File Format Crack Insertion GEO File RUN File Fracture Analysis Fracture Info Post Processing Visualization	Control Panel
Vorking Directory C:Documents and Settings/Admin/Deaktop/1 Node File Inode	Change Working Directory Working Directory Crack Insertion GEO File Run File Fracture Analysis Fracture Info Post Processing Visualization

In this tab, we perform "Crack Insertion".

- Select *.node2 extension file using "Browse" button.
- Give coordinates of crack center.
- Chunk radius: Chunk is a volume that contains crack.
- Give Crack Length, Crack Depth, Number of nodes along crack front and angles.
- Click "ChunkSeperator" button and create chunk with crack.

le Help		
king Directory-File Format Crack Inset	an GEO File RUN File Fracture Analysis Fracture Info Post Processing Vaualization	
	leaved and shall	Control Panel
le Name without extension	1 Browse	tetvi ce vi nese
oordinates of Crack Center	x y z 25 25 5 0 0 1 Normal dat	
rection Cosines of Crack's T Akis		*
enerate Chunk		
hunk Radius	5.5	
rack Length	9	
rack Depth		Working Directory
umber of nodes along crack front	70 ChunkSeperator	Crack Insertion
ingle of rotation about the free surface	ormal (Gamma) 0	GEO File
ingle of rotation about the crack face x	oois (Theta) 0	Run File
		Fracture Analysis
		Fracture Info
	atasia (^y	Visualization
(14		
1		\sim
. Y		
ť		
, ⊢,		

Here, we write "Element edge length". Click;

- 1)Run Tetgen
- 2)Unify Chunk Original Meshes
- 3)Generate Quadratic Elements Midside Nodes.

Now, cracked finite element is ready to fracture analysis.



In "RUN File" tab, we select material properties.

Help			
ing Directory File Format Crack Insets	on GEO File RUN File Frad	ture Analysia Fracture Info Post Processing Vaualization	
-	and the restant of the rest of the rest		Control Panel
Run File Name without ".run"	1		
Analysis Type	C Linear C Norlinea	r 🗆 Thermal Stress 🗭 Fracture 🔽 FGM	1 14 1 1 10000000 -07
Solver Type	C Frontal @ PCG Toler	rance-> 1e3	0.113384621+08 0.2000 0.2000 0
Material Properties	Linux - Junit Steeler		CTTT TERMAL LACEDING
Number of different material in the m	vodel 1		
Ex Ey Ez	Gxy Gyz Gxz	z Nuxy Nuyz Nuxz Behaviour	
3e7 127 1287	115304 115304 115304	0.3 C Orthotropic Basto Plastic	
		C Orthotropic C Basto Plastic	
1 1 1		Basto Plastic	
FGM Properties(8	E(x)=E0 exp(B x) Nu(x)=Nu0 exp	p(n,x),Alpha(x)=Alpha0.exp(w.x))	
	B	4 W	
T Is Material1 FC	5M2		
			Working Directory
T la Material2 FC	SM? [Working Directory
Is Material2 FC	3M2		Working Directory Crack Insertion
I'' la Material2 FC I''' la Material3 FC	SM2 [] SM2 []		Varking Directory Crack Insertion GEO File
Is Material2 F0 Is Material2 F0 Is Material3 F0 Do You Have Other Tied DDF S	5M7 [] 5M7 []		Working Directory Crack Insertion GEO File
In Material2 FC In Material2 FC In Material3 FC Do You Have Other Tied DOF S	5M7 [[5M7 [[ets]	CTE Values (XXCYY/22) and Traf	Working Directory Crack Insertion GEO File Run File
Is Material2 FC Is Material3 FC Do You Have Other Tied DOF S How many DOF do you have?	5M7 [[5M7 [[ets	CTE Values (00CYY22) and Traf	Verking Directory Crack Insertion GEO File Run File Fracture Analysis
Is Material 2FG Is Material 2FG Is Material 2FG Do You Have Other Tied DOF S How many DOF do you have? How many nodes	5M7	CTE Values (XXXY/22) and Traf	Working Directory Crack Insention GEO File Run File Fracture Analysis
Is Material2 FC Is Material2 FC Is Material3 FC Do You Have Other Tied DDF S How many DDF do you have? How many nodes	5M7	CTE Values (XXXY/2Z) and Tref [1112 [Temp Depender Material Property Initial Temperature	Verking Directory Crack Insertion GED File Run File Fracture Analysis Fracture Info
Is Material 2 FC Is Material 2 FC Is Material 3 FC Do You Have Other Tied DOF 5 How many DOF do you have? How many nodes Node number and direction	347 547 572	CTE Values (XXXY22) and Tref T112 CTE Values (XXXY22) and Tref T Temp Depender Material Property Initial Temperature CT E of Material 1 is temperature dep.	Vlorking Directory Crack Insertion GEO File Run File Fracture Analysis Fracture Info Post Processing
Is Material 2 FC Is Material 2 FC Is Material 3 FC Do You Have Other Tied DDF S How many DDF do you have? How many nodes Node number and direction	5M2	CTE Values (XXXYY.22) and Tref 112 CTE Values (XXXYY.22) and Tref 112 Timer Dependent Antonia Property Initial Temperature FT E of Mannial 1 is temperature dep Timer Prese	Violking Directory Crack Insention GED File Run File Fracture Analysis Fracture Info Post Processing
Is Material 2 FG Is Material 2 FG Is Material 3 FG Do You Have Other Tied DOF S How many DOF do you have? How many nodes Node number and direction	5M7	CTE Values (XXCYY22) and Tref 1172 Temp Depender Material Property Initial Temperature T E of Material 1 is temperature dep Temp Points Ex.Ey.Ez.Gx.Gy.Gz 1167367367	Vlorking Directory Crack Insertion GEO File Run File Fracture Analysis Fracture Info Post Processing Visualization
Is Material 2 FC Is Material 2 FC Is Material 3 FC Oo You Have Other Tied DOF S How many DOF do you have? How many modes Node number and direction Have Body Force Loading	547 547 eta	CTE Values (XXYY.22) and Tref TTT2 CTE values (XXYY.22) and Tref TTT2 TT Emp Dependent Property Initial Temperature dep. Temp: Points Ex.Ey,Ez,Gx,Gy,Gz Temp: Points KEx.Ey,Ez,Gx,Gy,Gz Temp: Points Ctaterial T is temperature dep.	Vloking Directory Crack Insention GEO File Run File Fracture Analysis Fracture Info Post Processing Visuelization
Is Material 2 FG Is Material 2 FG Is Material 3 FG Do You Have Other Tied DOF S How many DOF do you have? How many nodes Node number and direction Have Body Force Loading Type of Body Force Loading	SM7	CTE Values (XXCYY.ZZ) and Tref TTrep Depended Material Property Instal Temperature E E of Material 1 is temperature dep. Temp. Points. Exc.Fy.Ez.GX.Gy.Gz 107101307 Two of Material 1 is temperature dep. Tamp. Temp. Two of Material Te temperature dep.	Vorking Directory Crack Insertion GED File Pour File Fracture Analysis Fracture Info Post Processing Visualization
Is Material 2 FC Is Material 2 FC Is Material 3 FC Do You Have Other Ted DOF S How many DOF do you have? How many nodes Node number and direction Have Body Force Loading Type of Body Type of Body Type of	SMO	CTE Values (00CYY.22) and Tref T112 T Temp Depender Material Property Initial Temperature fep. Temp Points Ex.By Ez.Gx Gy.Gz 107107107 Two of Material T is temperature dep. Temp. Points Nuc. 1010103	Working Directory Crack Insenson GEO File Run File Fracture Analysis Fracture Info Post Processing Visualization
Is Naterial 2 FC Is Material 2 FC Is Material 3 FC Is Material 3 FC How many DOF do you have? How many nodes Node number and direction Have Body Force Loading Type of Body Force Loading Type of Body Force Loading The Acceleration Internation	SM2 SM2 F x F y F z Gravity C Centrilugal	CTE Values (XXYY.22) and Tref TTT2 CTE values (XXYY.22) and Tref TTT2 Temp Dependent Material Property Initial Temperature dep Temp. Points Ex.Ey, Ez.Gx, Gy, Gz 1010103 Temp. Points Nux.Nuy Nuz Temp. Points I CTE temperature depended	Working Directory Crack Insenson GEO File Run File Fracture Analysis Fracture Info Post Processing Visualization
Is Material 2 FG Is Material 2 FG Is Material 3 FG Is Material 3 FG Is wanted to be a set of the set	SM7 SM7 ets F x F y F z Gravity C Centrilugal seting [111]	CTE Values (XXYY/22) and Tref T172 CTE remp Depender Material Property Initial Temperature F E of Material 1 is temperature dep Temp Points Ex.Ey.Ez.Qx.Qy.Gz T0120103 Tmon. Points Nax.Nay.Naz T0120103 Tim Number of Importative points Tim Number of Importative points	Working Directory Crack Insertion GED File Pun File Fracture Analysis Fracture Info Post Processing Visualization
Is Material 2 FC Is Material 2 FC Is Material 3 FC Is Material 3 FC How many DOF do you have? How many nodes Node number and direction Type of Body Force Loading Type of Body Type of Body Type of Body Type of Body Type of Body	SM7 SM7 ets F x F y F z Gravity C Centrilugal	CTE Values (XXXY/22) and Tref TTT2 CTE Temp Dependent Material Property Initial Temperature E. 64 Material 1 is temperature dep. Temp. Points Ex.Ey, Ez.Gx, Gy, Gg Temp. Points Kuc.Nay Nuc Temperature depended The Number of temperature points Temperature and corresponding	Working Directory Crack Insention GEO File Run File Fracture Analysis Fracture Info Post Processing Visualization
Is Material 2 FG Is Material 2 FG Is Material 3 FG Is Material 3 FG Is Material 3 FG Is with the second	SM7 SM7 Fits Fity Fits Gravity C Centrifugal Inding Fits	CTE Values (XXCYY.ZZ) and Tref T112 Terre Depended Material Property Instal Temperature Property F E of Material 1 is temperature dep. Temp. Points ExcEy.Ez.GX.Gy.Gz 101701307 Two of Material 1 is temperature dep. Temp. Points Nuc.Nuc.Nuc. TCE Valuesprature depended The Number of temperature points. Temperature and corresponding CTE Values(x; z points) for point.	Vorking Directory Crack Insertion GED File Plue File Fracture Analysis Fracture Info Post Processing Visualization
Is Material 2 FC Is Material 2 FC Is Material 3 FC Is Material 3 FC Is Material 3 FC Is way 200F do you have? How many 200F do you have? How many acides Node number and direction The Acceleration The Acceleration Ingut Linecton Costenes of Grav. La Direction (min Clobal Coord () Second on Julis Second on Jul	SM7 SM7 Es F x F y F z Gravity C Centrilugal adding [111	CTE Values (XXCYY/Z2) and Tref	Vorking Directory Crack Insertion GED File Paul File Fracture Analysis Fracture Info Post Processing Visualization
Is Material 2 FG Is Material 2 FG Is Material 3 FG Is Material 3 FG Is Material 3 FG Is with a second s	SM7	CTE Values (0XYY,22) and Tref TTT2	Working Directory Crack Insertion GED File Paun File Fracture Analysis Fracture Info Post Processing Visualization
Is Material 2 FC Is Material 2 FC Is Material 3 FC Is Material 3 FC Do You Have Other Tied DOP S How many DOP do you have? How many nodes Node number and direction Type of Body Force Loading Type of Body Force Loading Type of Body Force Loading Type of Body Force Loading Type of Body Force Loading Type of Body Force Loading Type of Body Force F	SM7 SM7 es F x F y F 2 Gravity C Centrifugal sading [11]	CTE Values (XXXY/22) and Tref CTE Values (XXXY/22) and Tref Inter Dependent Material Property Instal Temperature CE of Material 1 is temperature dep. Temp. Points Ex.Ey.Ez.GX.Gy.Gz CTE Values Paints CTE Values Paints CTE Values Paints CTE Values Caresporting CTE Values Caresporting CTE Values CTE	Vorking Directory Crack Insertion GEO File Run File Fracture Analysis Fracture Info Post Processing Visualization Generate RUN File
Is Material 2 FG Is Material 2 FG Is Material 3 FG Is M	SM2 SM2 F x F y F z Gravity C Centrifugal seding F 117	CTE Values (XXCYY/22) and Tref TTT2 CTE Values (XXCYY/22) and Tref TTT2 Temp Depender Material Property Initial Temperature dep Temp Points Ex.Ey.Ez.Cx.Cy.Cg 107101307 No of Material T is temperature dep Temp Points Ex.Ey.Ez.Cx.Cy.Cg 107101307 The Number of temperature points Temperature and corresponding CTE Values(x,y z points) for point 1 Integration Drafe 24 V Use Transition Element" Generalized Plane Strain Problem Input Number of Incorrents. Max. Number of	Working Directory Crack Insertion GED File Run File Fracture Analysis Fracture Info Post Processing Visualization Generate RUN File

Fracture Analysis. "Frac3D Solver".

le Help					
king Directory-File Format	Crack Insetton GEO File RUN File Frac	ture Analysis Fracture Info Post Proce	ming Vaualization		
	RUN File Name GEO File Name I Thermal Analysis? TEM File Name Number of Processors to be Used	[1 [1 [2	Browse Browse Browse RUN		
	Do you accept License Agreement?		•	Review Error File	Working Directory Crack Insertion GEO File
	FCPAS Solver Fracture and Crack Propagation And LICENSETERMS	Nysia System	*		Run File Fracture Analysis Fracture Info Post Processing Visualization
	² Compute and PRO (Picel), given Permission is hereby granted, free of perion obtaining a copy of this software, subconstituents and the associated docu the "Software", for the nights to use, copy and publish resultifion the Software subs ³ Usage of the Software is permissible and UNDER THE CONDITION THAT THE S INCLUDER APPROPRIATE SUBCONS.	charge to any it all mentation files , menge software. cted to the following conditions: freely available OFTWARE. TUTUENTS, REMAIN UNDER COPYRIGI	а		 ↔
	OF THE RESPECTIVE AUTHOR(S) AND DISTRIBUTED TO ANY OTHER THRD "Usage of the Software for private use, res and education 15 PERMISSIBLE	PARTIES PARTIES warch, publication			

Fracture Info and SIFs graphic.

Help		
ng Directory-File Format Crack Insection GEO File RUN File Fracture Analysis Fracture Info Pos	t Processing Vaualization	Control Panel
t this section, you can view contents of output files and plot stress intensity factors along crack from		
Open File : C:Documents and Settings/Admin/Desktopl.111.ork	Plot SIFs :	
	al I	
	<u> </u>	
A 12 M 12 M 12 M 10 USED FOR ENROLED COVER IF ELEMENTS		
CRACK NO: 1 RACK TIP NODES: 572 49535 5880 72363 5881 71630 5882 74790 5883 74747		Vilorking Directory
5884 64844 5885 64868 5886 74634 5887 77583 5888 74560 5889 68551 5890 53084 5891 53067 5892 64663 5893 74365		Crack Insertion
5894 74272 5895 59519 5896 70804 5897 54028 5898 61634 5899 61651 5900 66219 5901 68296 5902 49009 5933 58289 5404 74727 5405 5586 5405 5407 54170 5407 4659 5408 500 500		GEO File Run File
5509 63480 5910 73911 5911 71107 5912 70338 5913 73796 5914 73865 5915 73833 5916 73836 5917 73793 5918 50108		Fracture Analysis
5919 73736 5920 73716 5921 55859 5922 73611 5923 45891 5924 47181 5925 46599 5926 64825 5927 73385 5928 47757 5926 5434 5430 7396 5931 73926 5927 57385 5928 47757		Fracture Info
5334 57513 5935 67608 5936 71290 5937 54296 5938 64418 5939 53178 5940 44951 5941 41271 5942 43708 5943 45972		Visualization
5944 53543 5945 61150 5946 52468 5947 47067 5948 44765 5949 51283 5950 51610 5951 41465 5952 48324 5953 66819		
3354 4,302 3353 4,224 3355 4,9494 3357 4,9493 3356 49331 5959 45024 5950 56204 5961 48792 5962 59543 5963 42623 5964 55531 5965 62265 5966 4888 5967 51922 5968 41288		
5969 56159 5970 41322 5971 57525 5972 49882 5973 48353 5974 46379 5975 42177 5976 56420 5977 42833 5978 65191		
5979 62269 5980 57492 5981 46920 5982 54619 5983 49905 5884 41167 5985 58493 5986 58340 5887 46519 5988 56635		
2853 41/24 2570 2516/ 2571 4/2/5 2572 40/33 2573 4393 5994 58535 5995 51693 5996 51788 5997 45555 5998 45007 5999 51258 6000 50121 6001 49738 6002 65953 6003 52066		
6004 44149 6005 50250 6006 55373 6007 44436 6008 44096 6009 47572 6010 62216 6011 57892 6012 54763 6013 47898		







References

[1] ANSYS 12.0 Academic Research Advanced Version, Canonsburg, PA, U.S.A.

Contributions/Applications By:

C. Kurtiş M. Uslu G. Atalı İ. Y. Sülü H. Pekel İ. Kacar A. R. Zaloğlu E. Nart A.O. Ayhan H. F. Nied

Contact: Dr. Ali O. Ayhan, aoayhan@yildiz.edu.tr