

# FRANC3D & ANSYS: Crack Insertion & Growth Training

# FRANC3D



## FRANC3D/ANSYS Tutorial

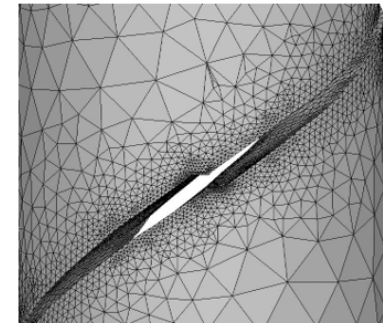
Basic tutorial for ANSYS users showing crack insertion and growth in a simple cube.

[Download PDF](#)

# ANSYS™ Tutorial

ANSYS is a registered  
trademark of ANSYS, Inc  
[www.ansys.com](http://www.ansys.com)

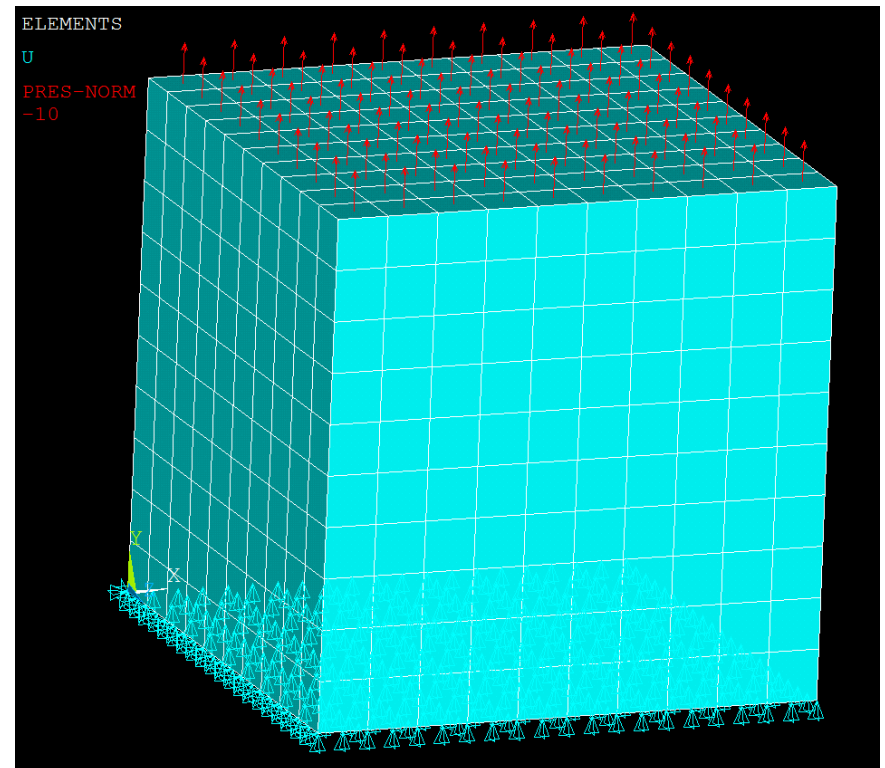
## Version 8.8



Fracture Analysis Consultants, Inc  
<https://www.fracanalysis.com/>

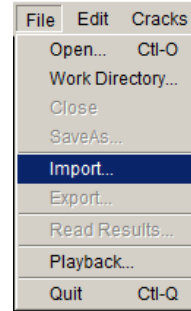
# Step 1: Create the ANSYS FE Model

- 10 x 10 x 10 mm cube
  - Quadratic brick elements
  - $E = 10000$  MPa
  - Poisson's ratio = 0.3
  - 10 MPa traction load
  - Export .cdb file; if you are using APDL, cdwrite, and if using WB, you can add APDL to WB (Append A in the F3D/ANSYS Tutorial)

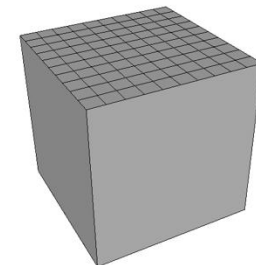
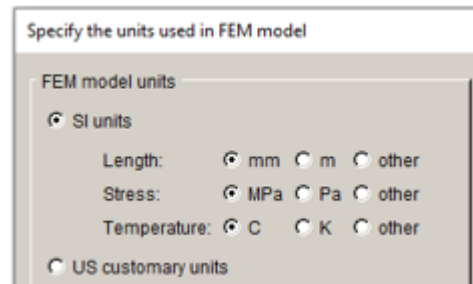
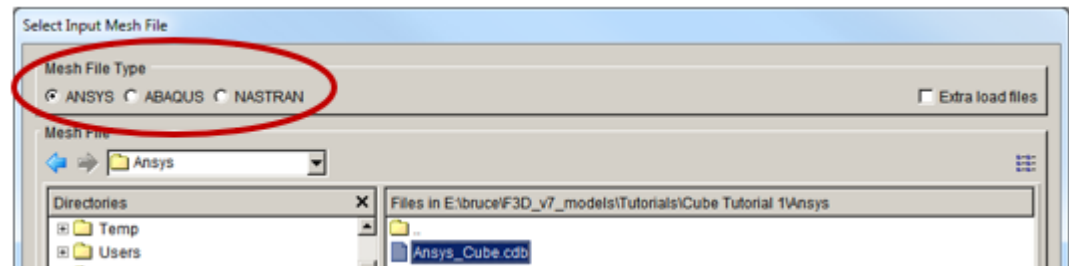
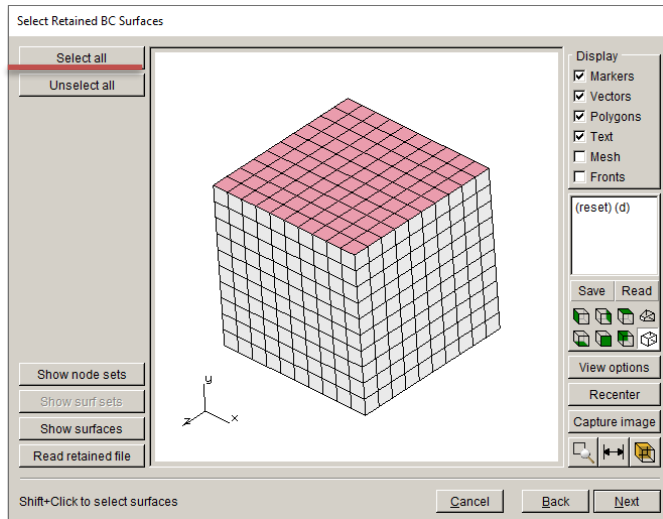
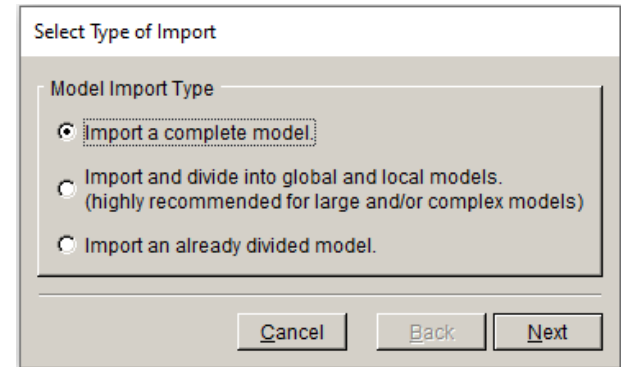


# Step 2: Import into FRANC3D

- File → Import
  - Select ANSYS mesh type
  - Retain BC surfaces
  - Set FE model units

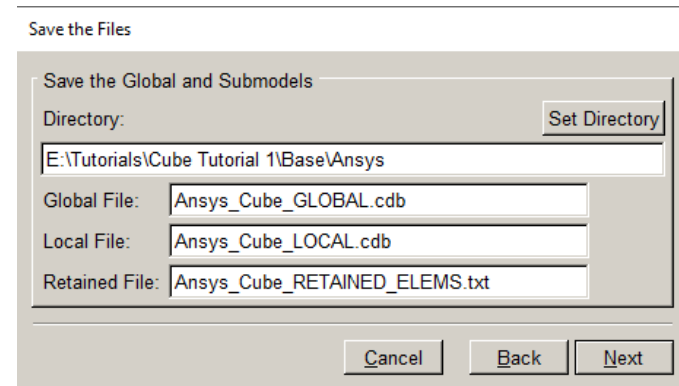
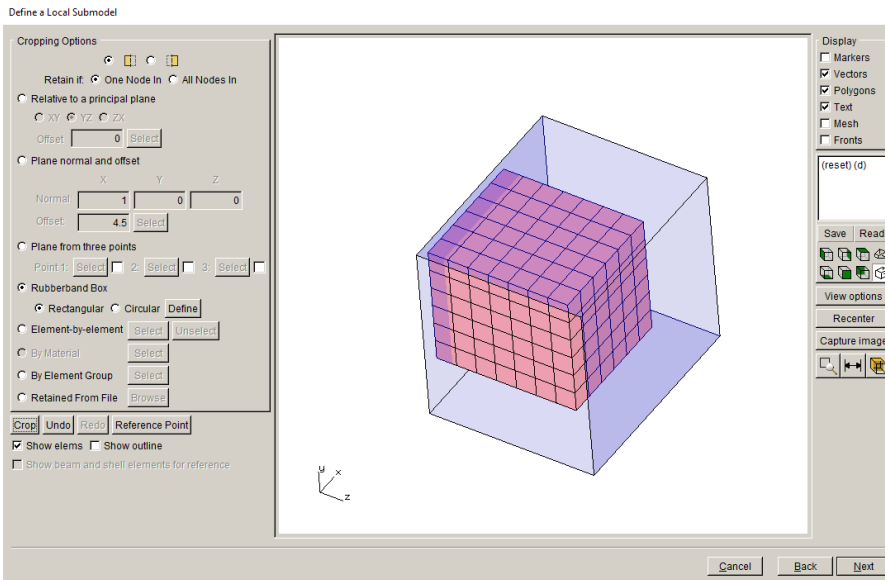
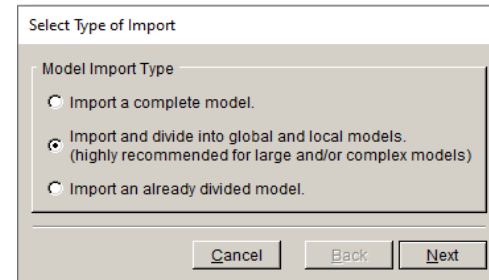


import a complete model should only be done for very simple models.



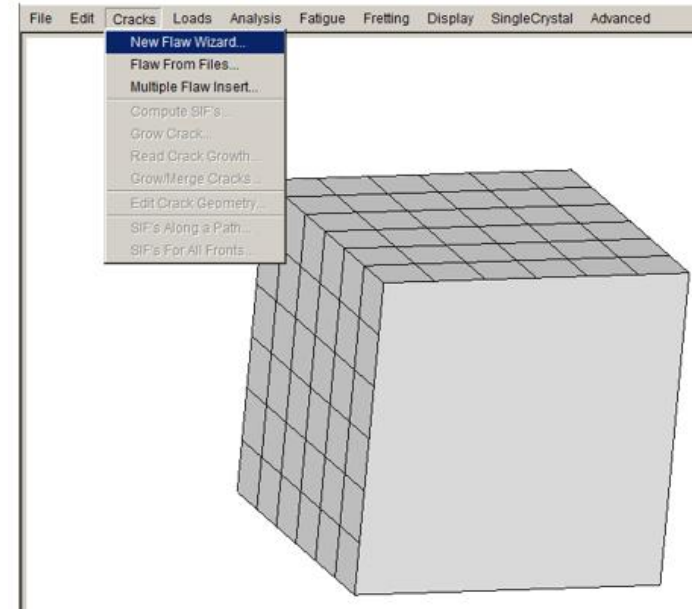
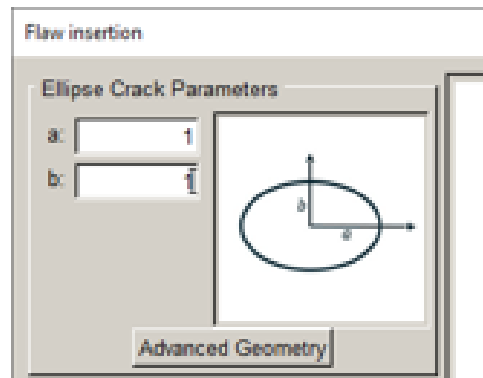
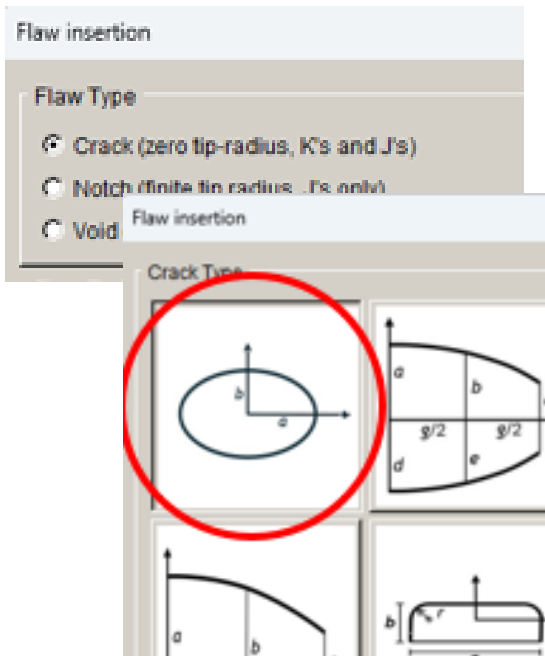
# Step 3: Submodeling

- Divide into local and global models
  - Use Rubberband Box tool
  - Crop region that will contain the crack
  - Save local/global files

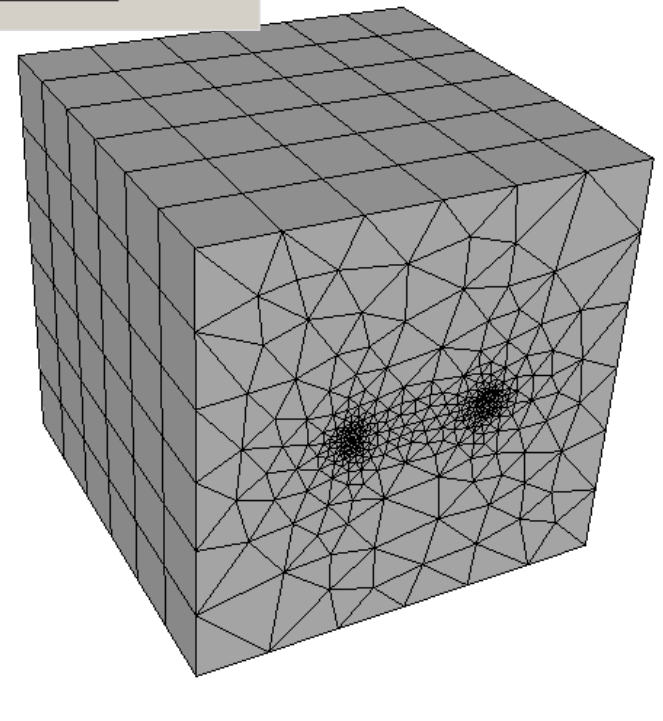
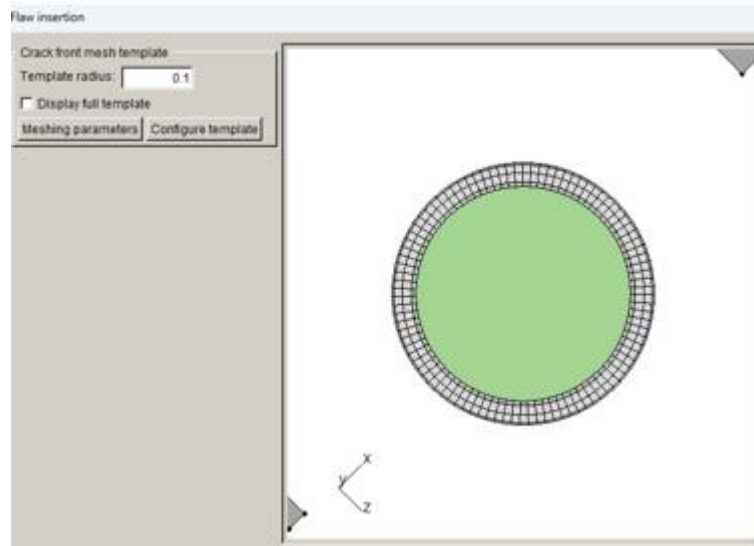
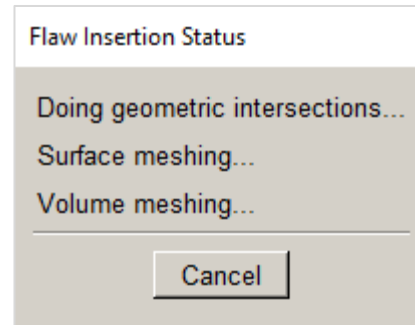
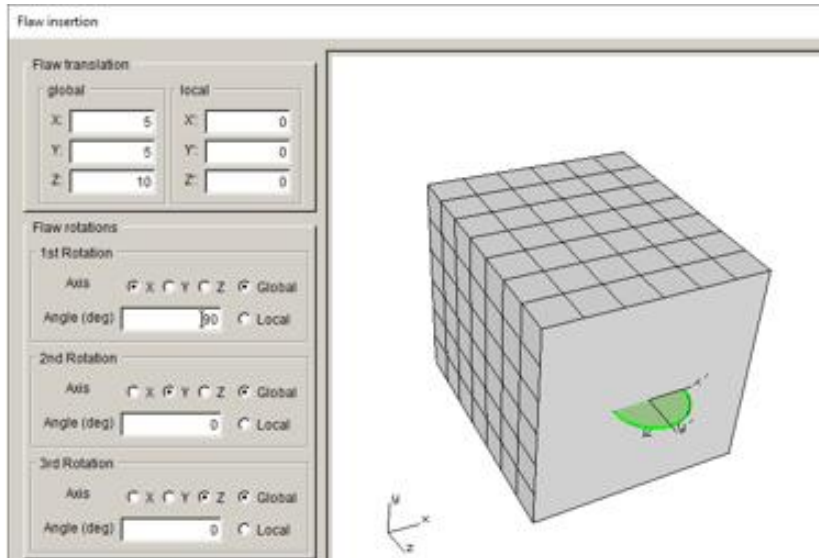


# Step 4: Insert Surface Crack

- Cracks → New Flaw Wizard
  - Elliptical crack ( $a=b=1$  mm)
  - Location (5,5,10)
  - Automatic remeshing

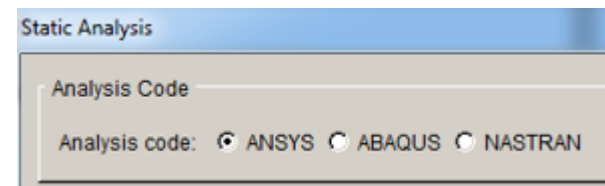
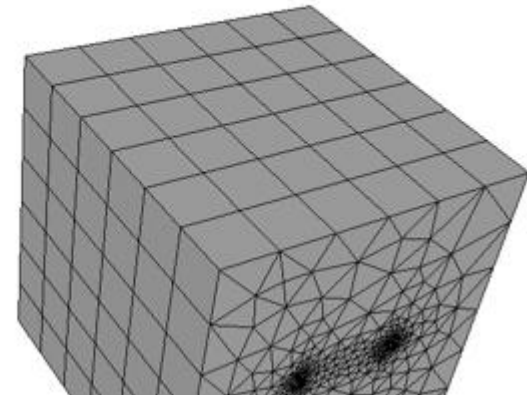
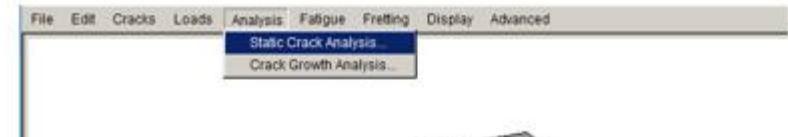
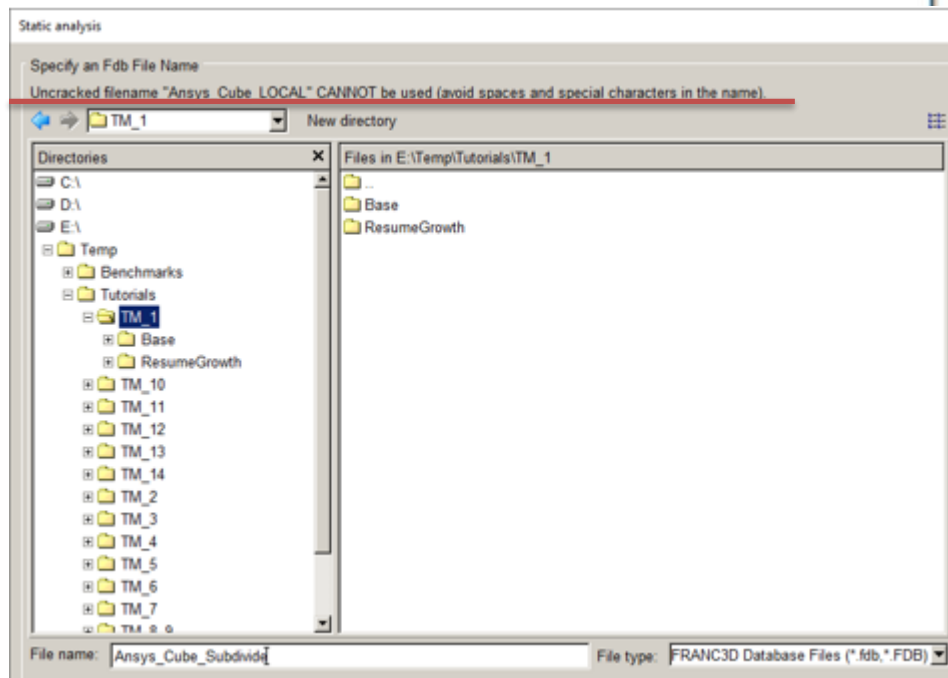


# Step 4: Insert Surface Crack



# Step 5: Static Crack Analysis

- Analysis → Static Crack Analysis
  - Using ANSYS solver
  - Verify displacements



# Step 5: Static Crack Analysis

Static analysis

ANSYS Options

ANSYS run time: **set in Preferences**

ANSYS Executable:

ANSYS license:

Python Executable:

Python script (will run before ANSYS):

Local model output:

Global model:

Connect to global model filename:

Boundary conditions:

Apply crack face tractions  Define crack face contact   Initial strain

ANSYS command:

Write files but DO NOT run analysis

If you import the complete model, this dialog will not appear.

Static Analysis

Ansys local/global model connection:

Merge nodes  Constraint equations  Contact conditions

Merge tolerance:  Local connection midsides:  retain  remove

Local component(s) to merge/constrain:

AUTO\_CUT\_SURF

Global component(s) to merge/constrain:

Show all labels

GLOBAL\_CONNECT\_SURF

Additional local/global connections:

<none defined>

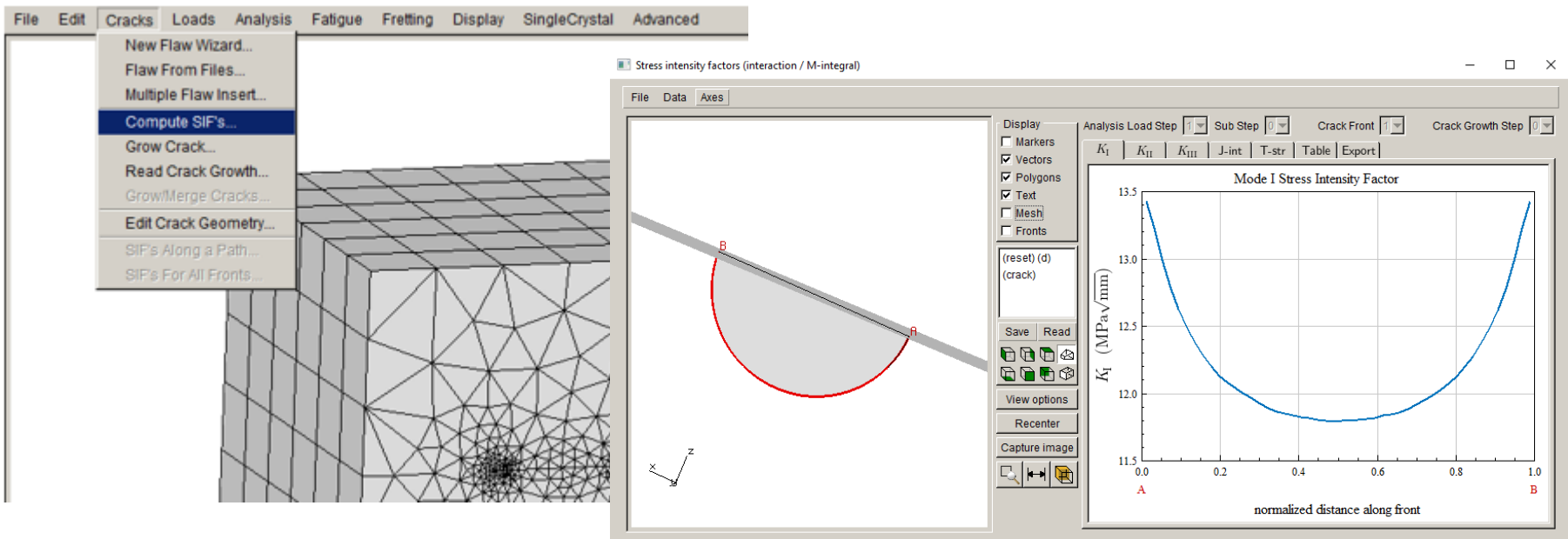
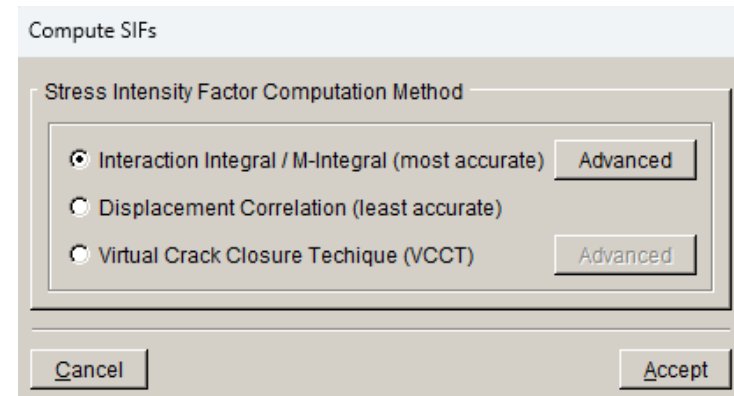
Ansys command:

Write files but DO NOT run analysis

Compare ANSYS displacements for the uncracked and cracked models once the analysis has finished.

# Step 6: Compute SIFs

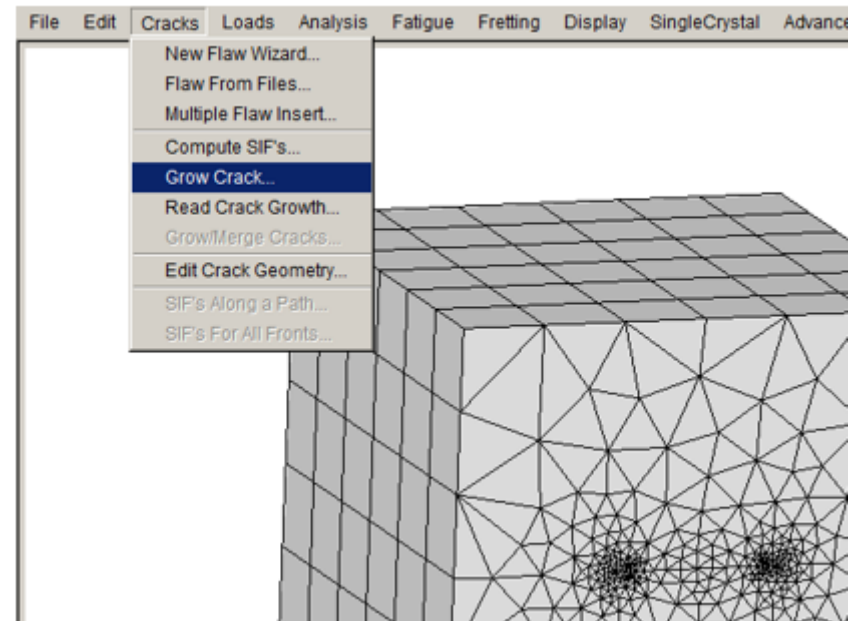
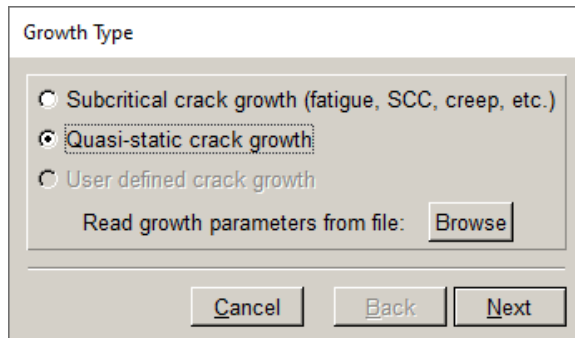
- Cracks → Compute SIFs
  - Use M-integral
  - Plot  $K_I$ ,  $K_{II}$ ,  $K_{III}$
  - Units:  $\text{MPa}\sqrt{\text{mm}}$



# Step 7: Manual Crack Growth

- Cracks → Grow Crack
  - Quasi-static growth
  - MTS criterion
  - Polynomial fit
  - Template radius control

Quasi-static growth is a simplified Paris-type growth rate model.



# Step 7: Manual Crack Growth

Kink angle model

Kink angle model

Max tensile stress (MTS)  $\max(K_I^r(\theta))$

Max shear stress (MSS)  $\max\left(\sqrt{(\eta_{II}K_{II}^r(\theta))^2 + (\eta_{III}K_{III}^r(\theta))^2}\right)$

Generalized stress  $\max(\text{MTS}, \text{MSS})$

Strain energy release rate  $\max\left(K_I^r(\theta)^2 + (\eta_{II}K_{II}^r(\theta))^2 + (\eta_{III}K_{III}^r(\theta))^2\right)$

Planar  $\theta = 0$

User defined model

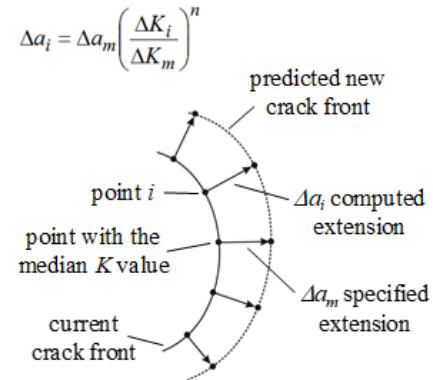
Kink angle limit Maximum kink angle (deg):

Non-proportional kink angle parameter   $\Delta K$    $K_{\max}$

Mixed mode eta factors  $\eta_{II}^r$    $\eta_{III}^r$

Crack growth resistance

Anisotropic toughness



Quasi-static growth parameters

Power law growth parameter

$$\Delta a_i = \Delta a_{\text{median}} \left( K_i / K_{\text{median}} \right)^n$$

n:

Mixed-mode equivalent K

$K^{\text{equiv}} = K_I$    $K^{\text{equiv}} = \sqrt{K_I^2 + (\gamma_{mII}K_{II})^2 + (\gamma_{mIII}K_{III})^2}$

$\gamma_{II}^r$    $\gamma_{III}^r$

sign:  from  $K_I$   from  $K_{II}$   from  $K_{III}$   always positive  always negative

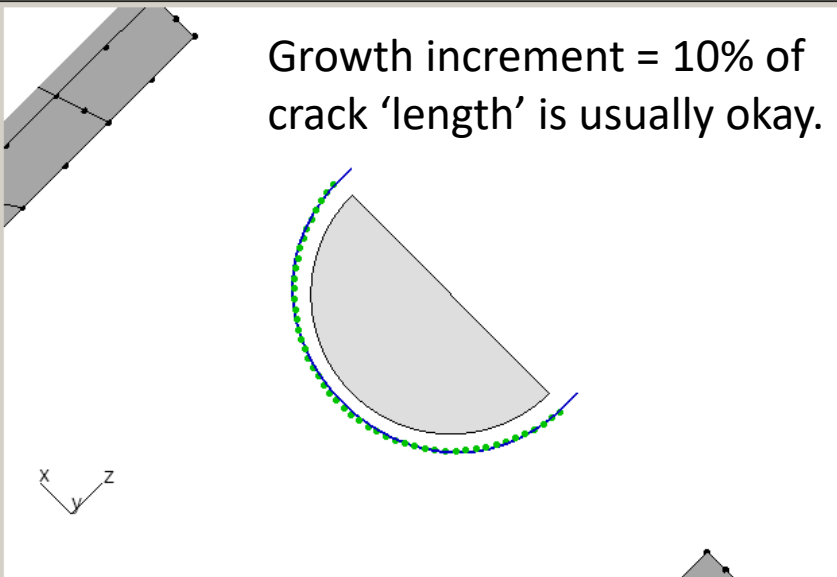
FEM Load Steps

Step	Sub	Load mult
<input type="text" value="1"/>	<input type="text" value=""/>	<input type="text" value="1"/> <input type="button" value="Edit"/>

# Step 7: Manual Crack Growth

Crack growth

Growth increment = 10% of crack 'length' is usually okay.



Crack extension

extension: 0.12 scale node: 0.5

number of cycles: ----- start cycle: 0

elapsed time: ----- start time: 0

Edit growth params Show SIF's

Estimated increments

estimated cycles: 0 end cycle: 0

estimated time: 0 end time: 0

Marker size:  big  med  small

Front fitting options

KinkAngle/Extension poly fit polynomial order: 3

Fixed order poly through points extrapolate (%): 3 3

Multiple poly through points ignore n end points: 0 0

Hermitian closed poly multiple poly ratio: 5

Cubic spline moving poly range: 5

Moving polynomial

No smoothing

Allow fit adjustment

Kink Angle: Display Save

Extension: Display Save

Save .frit and .crk files

Display

Markers

Vectors

Polygons

Text

Mesh

Fronts

Crk Geom

(reset) (d)  
(crack)

Save Read

View options

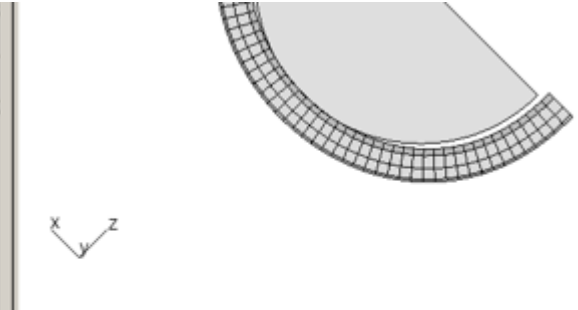
Recenter

Capture image

Cancel Back Next

User specified extension applied at the front point where  $K_I$  = median  $K_I$  (if scale node = 0.5)

Use the simplest form of front point fitting that gives a reasonable fit (usually a single 3<sup>rd</sup> order polynomial is adequate)



Flaw template

Template radius set as:  absolute value  % of crack increment

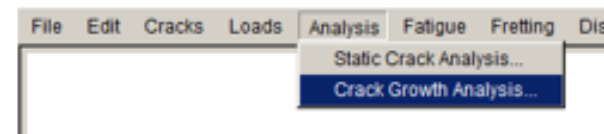
Template Radius: 0.1 Front: [dropdown]  Set all fronts

Display full template

Meshing parameters Configure template

# Step 8: Automatic Crack Growth

- Analysis → Crack Growth Analysis
  - Define growth steps
  - Constant extension



Fitting & template parameters

Front fitting options

KinkAngle/Extension poly fit    polynomial order:

Fixed order poly through points    extrapolate (%):

Multiple poly through points    ignore n end points:

Hermitian closed poly    multiple poly ratio:

Cubic spline    moving poly range:

Moving polynomial     Allow fit adjustment

No smoothing

Flaw template

Template radius set as:  absolute value     % of crack increment

Template Radius:  Front:   Set all fronts

Extension

distance extension    scale node:

cycles extension

time extension

Crack growth analysis

Analysis Code

Analysis code:  ANSYS     ABAQUS

Base filename (avoid spaces and special characters):

Current crack growth step:

Growth plan

Specified crack growth steps

Number of Crack Growth Steps =

Constant crack growth increments

Crack growth step size =

Linearly increasing/decreasing crack growth increments

First crack growth step size =

Change in step size at each step =

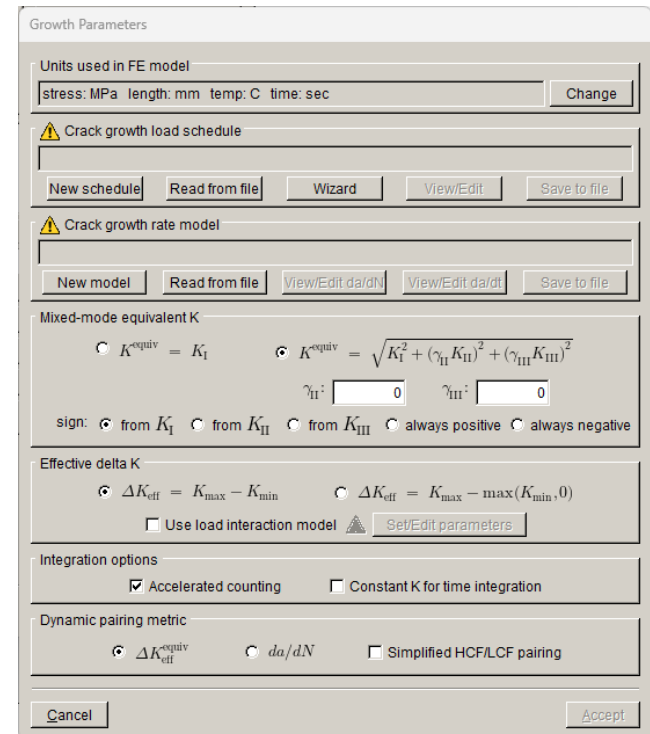
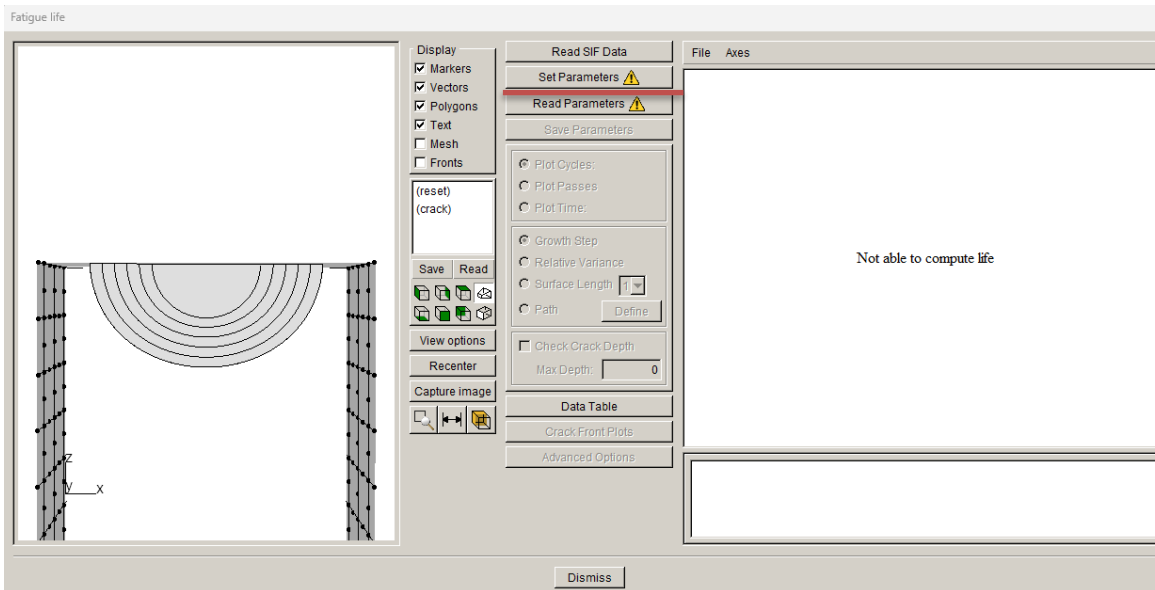
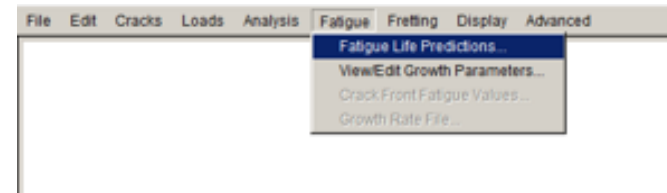
User defined crack growth increments

Each step of crack growth is numbered: `_STEP_###`.

Files are saved at each step of growth so that the user can restart from any step.

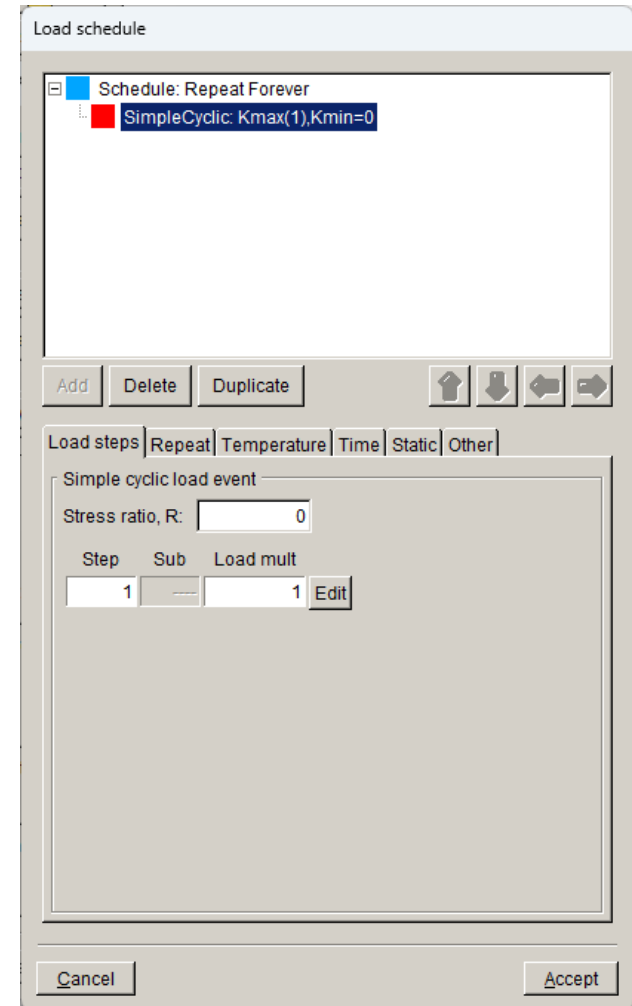
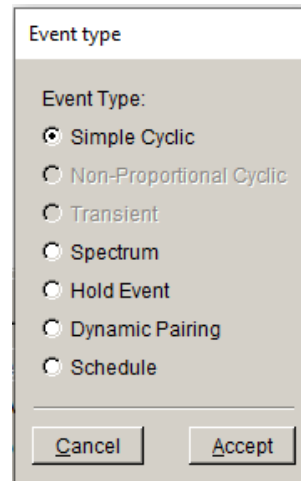
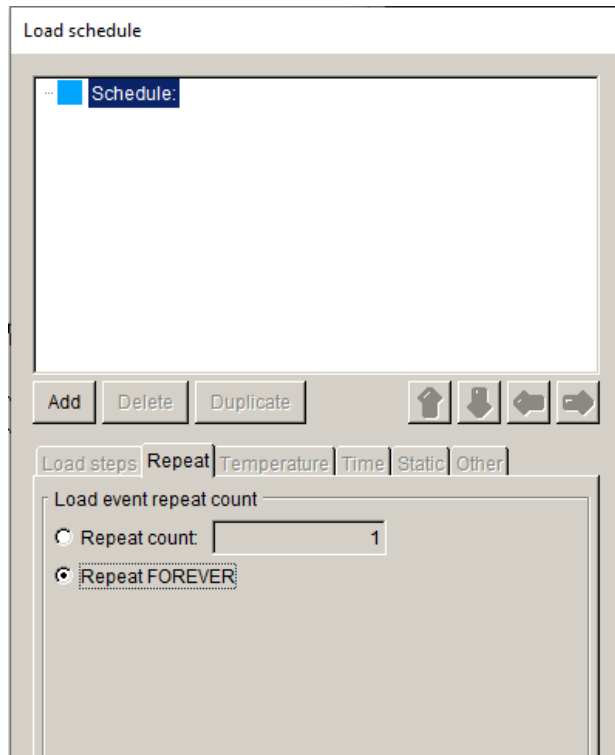
# Step 9: Fatigue Life Prediction

- Fatigue → Fatigue Life Predictions
  - Define load schedule
  - Apply Paris model
  - Plot cycles vs crack length



# Step 9: Fatigue Life Prediction

Define the load schedule:



# Step 9: Fatigue Life Prediction

Define the growth rate model:

Growth model types

Create a cyclic loading growth rate model  
 Create a time dependent growth rate model

Cancel Back Next

Cyclic loading growth model

Temperature independent model  
 Temperature dependent model

Cancel Back Next

Cyclic loading growth model

Paired growth rate and R-ratio functions

Growth rate model:  Paris  
 Bilinear Paris  
 Sigmoidal  
 Hyperbolic sine  
 Table lookup

Stress ratio model:  None  
 Walker equation  
 Newman closure  
 Table lookup

NASGRO version 4 equation

Key in  
 NASGRO user XML mat file

NASGRO version 3 equation  
 Modified Hartman-Schijve model  
 User defined model

Cancel Back Next

Growth rate model

Paris growth model:   
 $da / dN = C \Delta K^n$

Model label (optional):

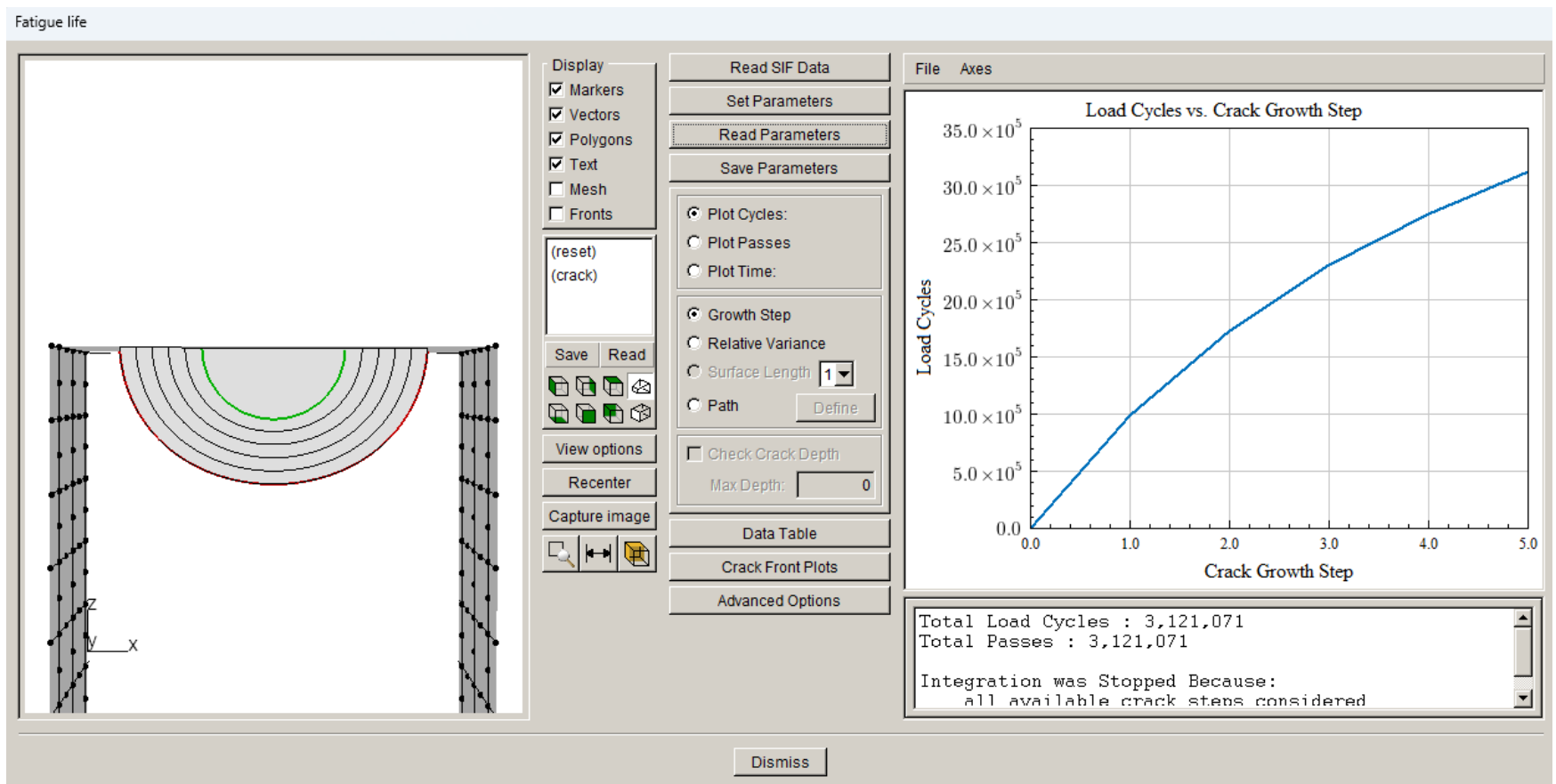
Description (optional):

Units: stress: MPa length: mm temp: C time: sec

	C	n	DKth	Kc
1	1e-10	3	1	100

Cancel

# Step 9: Fatigue Life Prediction



# Conclusion

- Integrated FRANC3D + ANSYS workflow
  - Crack insertion and ANSYS analysis
  - Stress intensity factor analysis
  - Crack propagation
  - Fatigue life evaluation