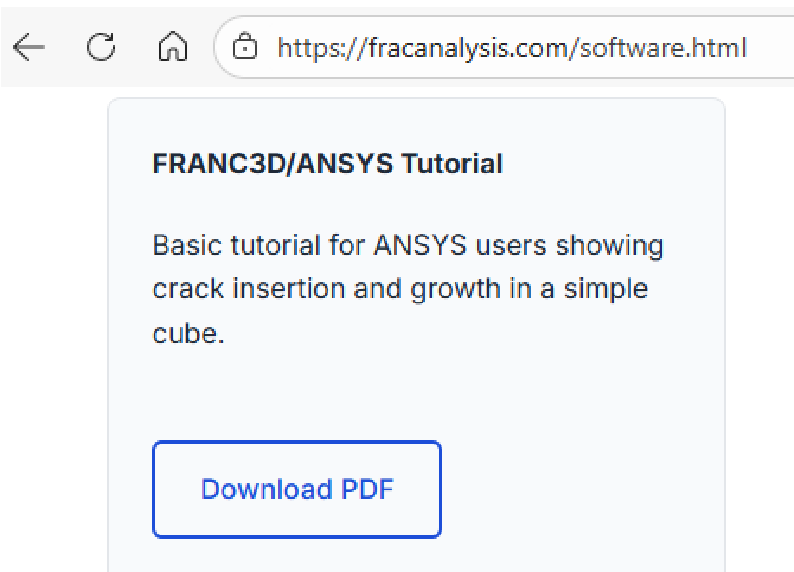
FRANC3D & ANSYS:
Crack Insertion & Growth Training

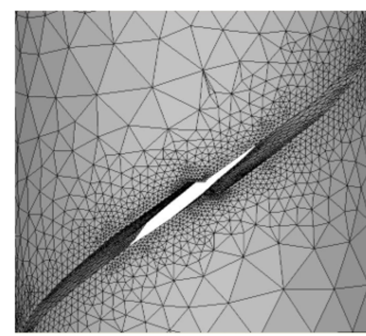


FRANC3D

ANSYS™
Tutorial

ANSYS is a registered trademark of ANSYS, Inc
www.ansys.com

Version 8.8

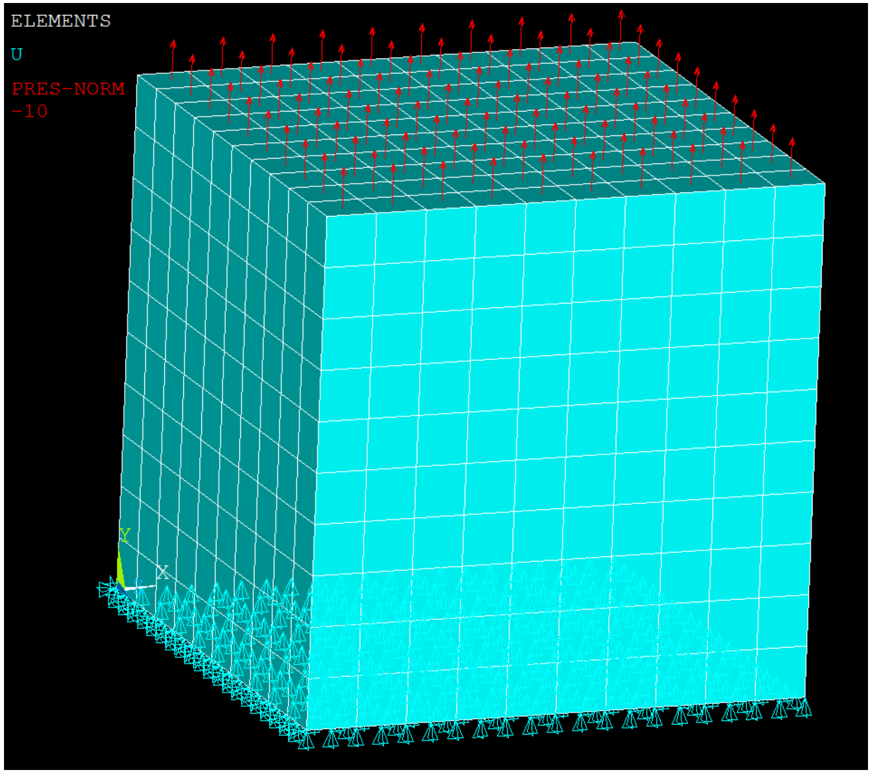


Fracture Analysis Consultants, Inc
https://www.fracanalysis.com/

Revised: Feb 2026

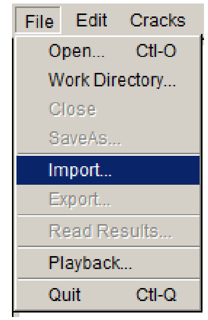
Step 1: Create the ANSYS FE Model

- 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)

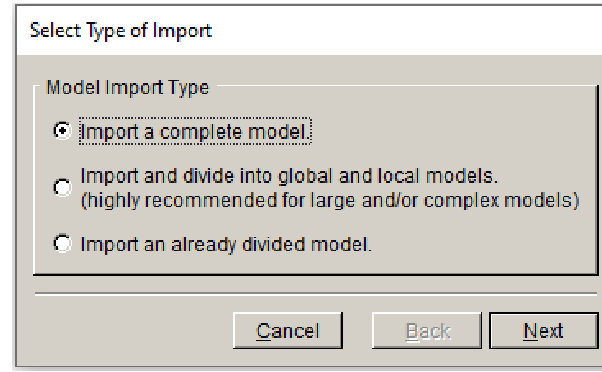


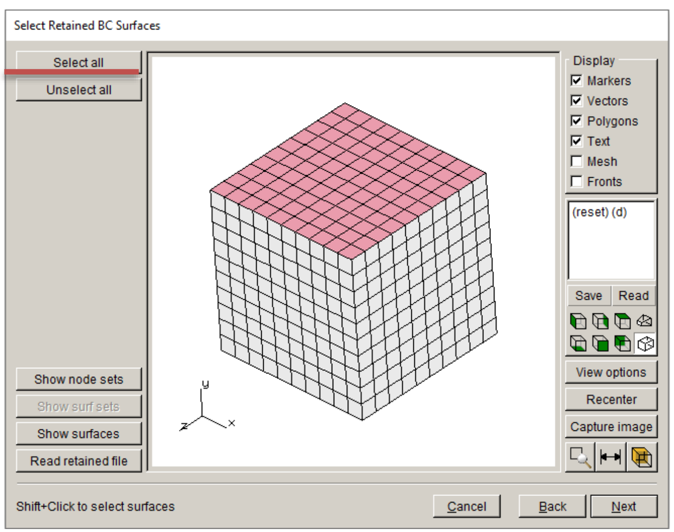
Step 2: Import into FRANC3D

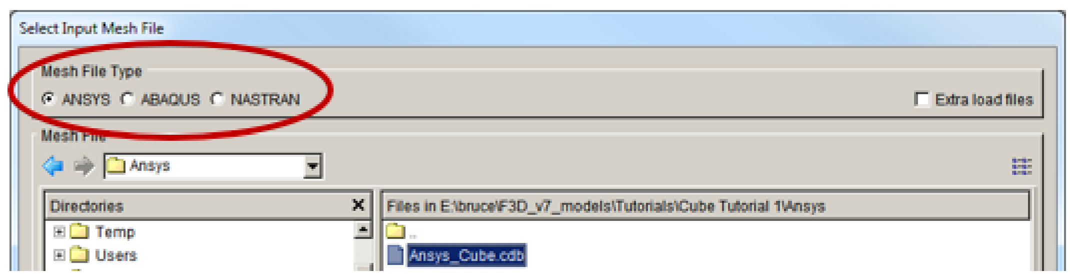
- File → Import
  - Select ANSYS mesh type
  - Retain BC surfaces
  - Set FE model units

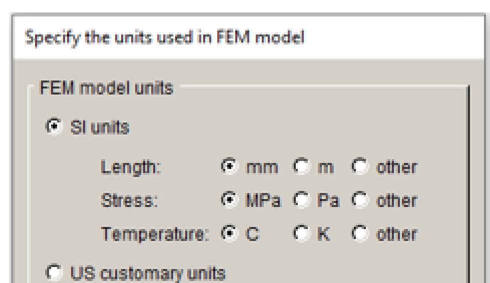


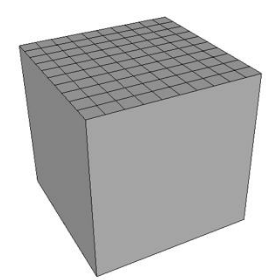
<u>import a complete model</u> should only be done for very simple models.





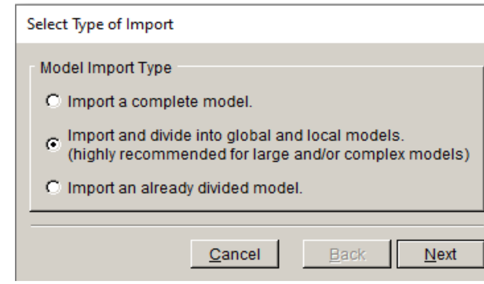


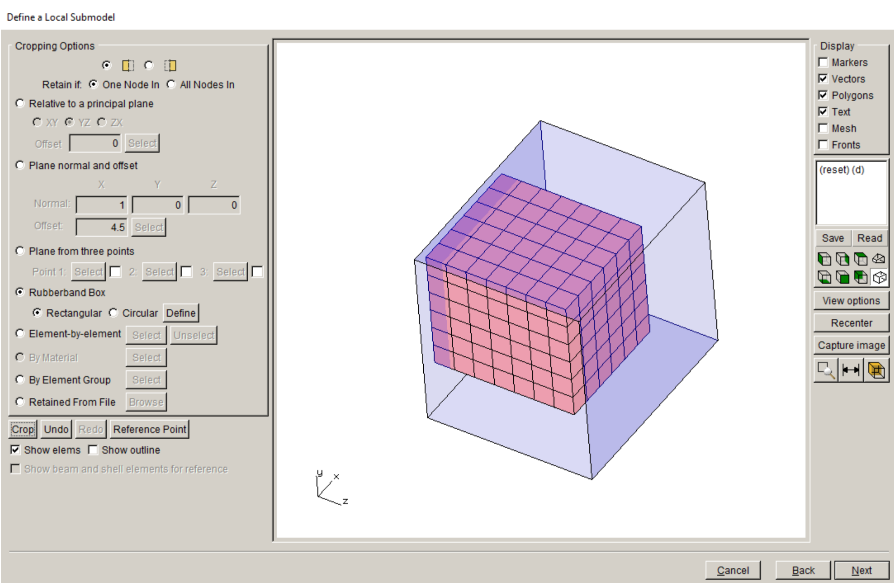


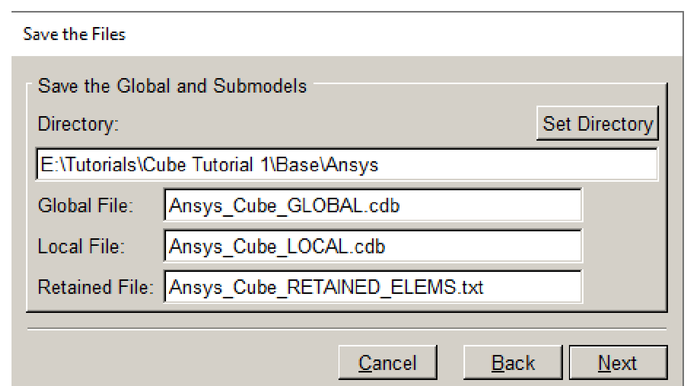


Step 3: Submodeling

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

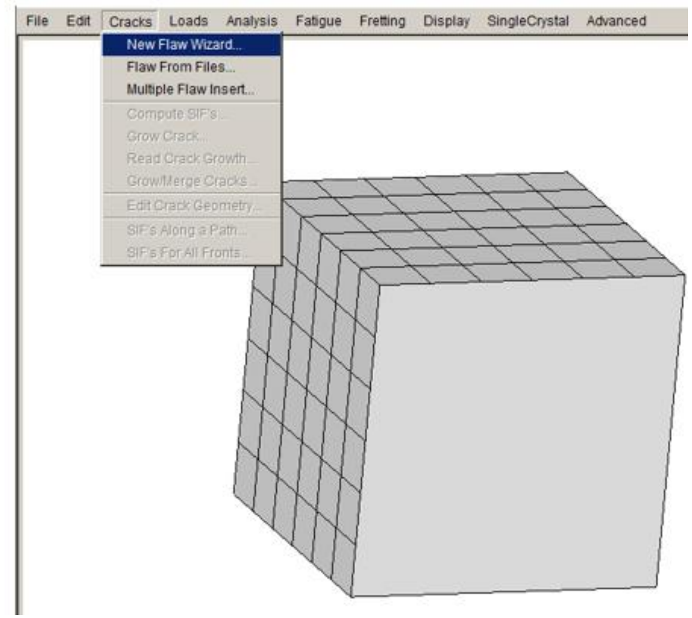


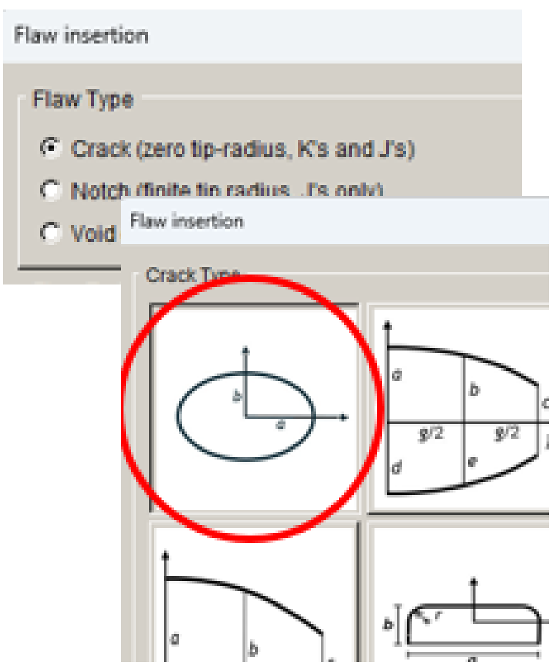


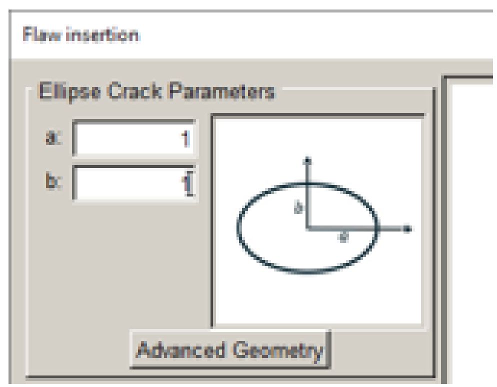


Step 4: Insert Surface Crack

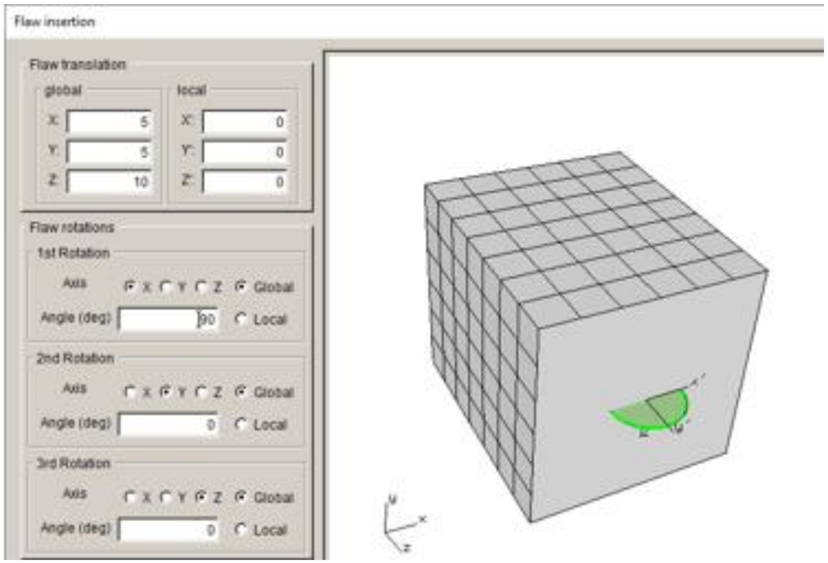
- Cracks → New Flaw Wizard
  - Elliptical crack (a=b=1 mm)
  - Location (5,5,10)
  - Automatic remeshing

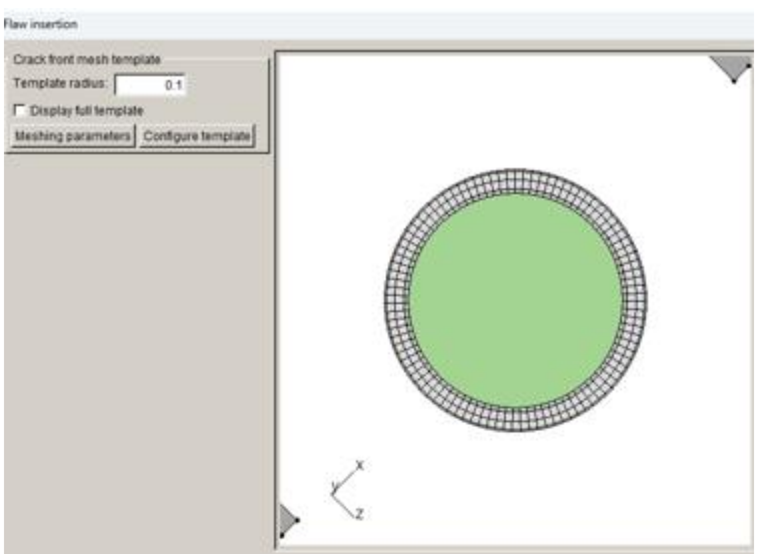


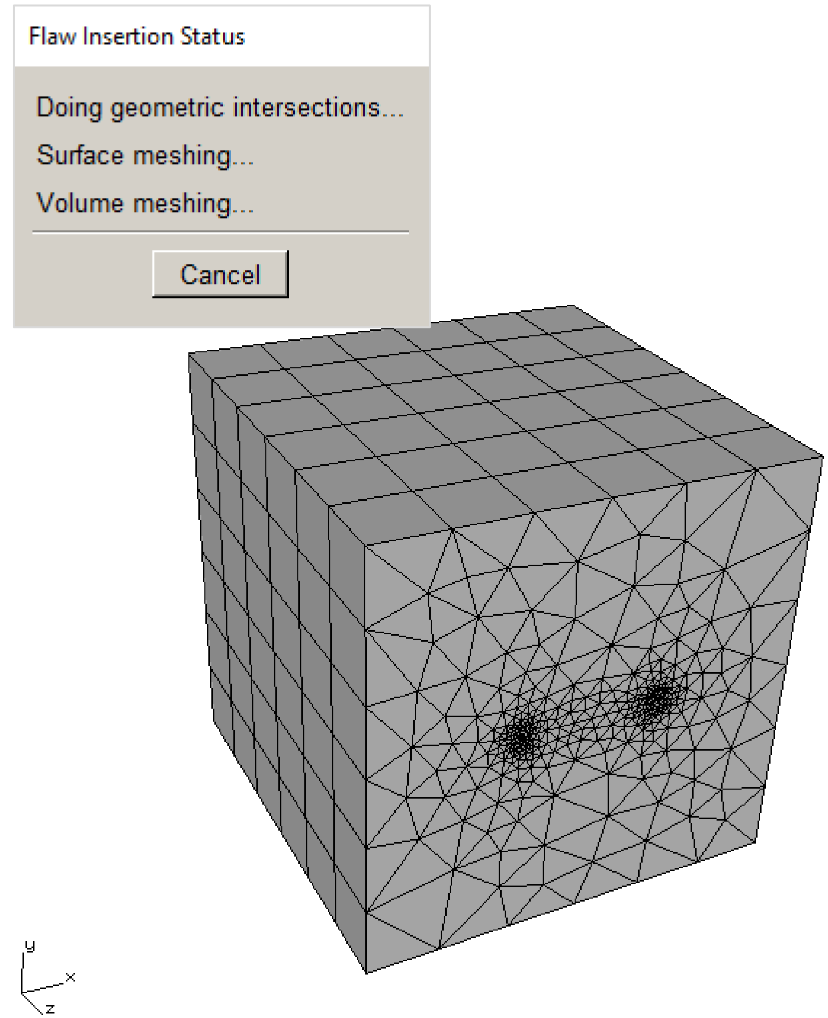




Step 4: Insert Surface Crack

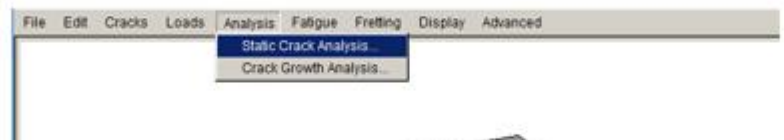


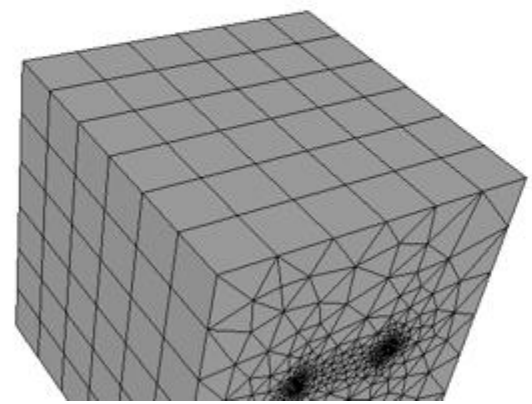


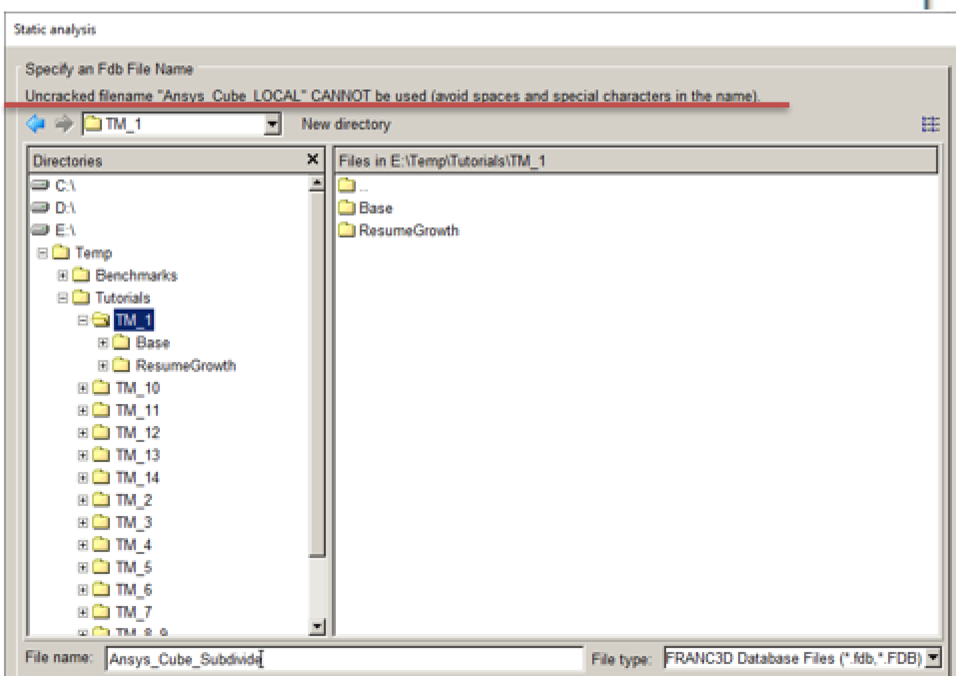


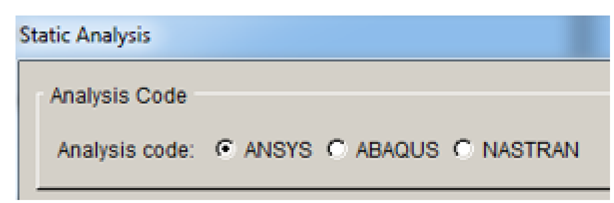
Step 5: Static Crack Analysis

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

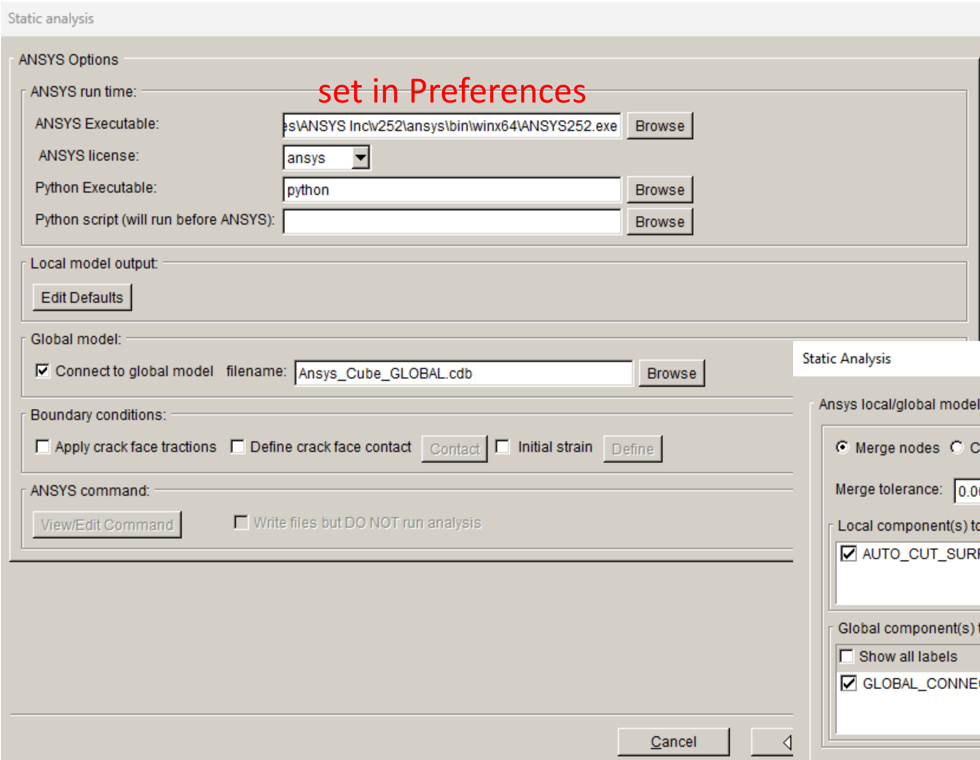




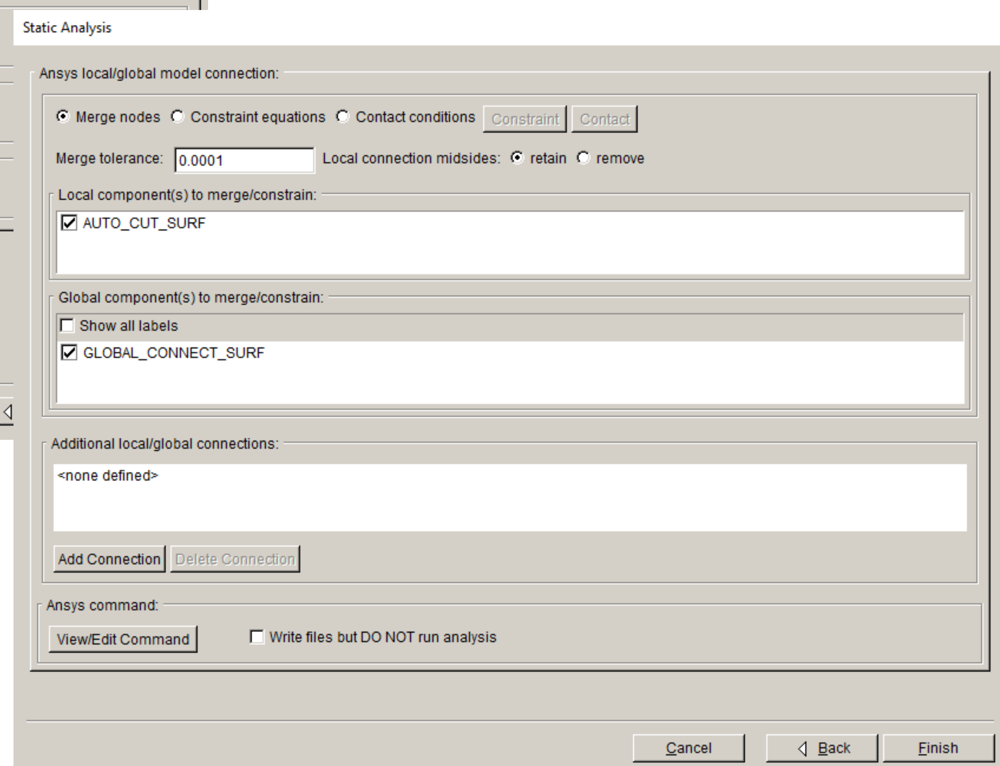




Step 5: Static Crack Analysis



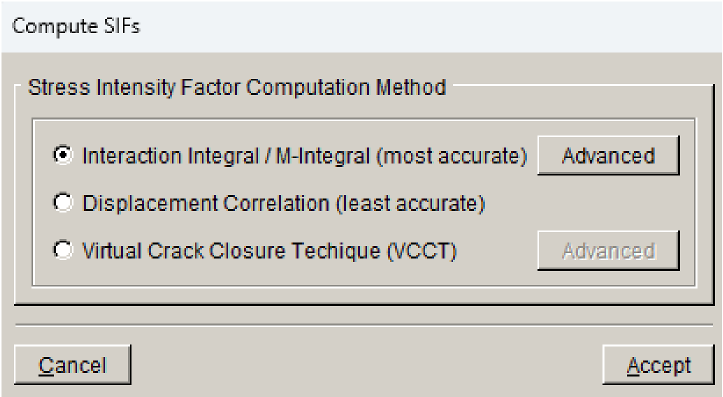
If you import the complete model, this dialog will not appear.

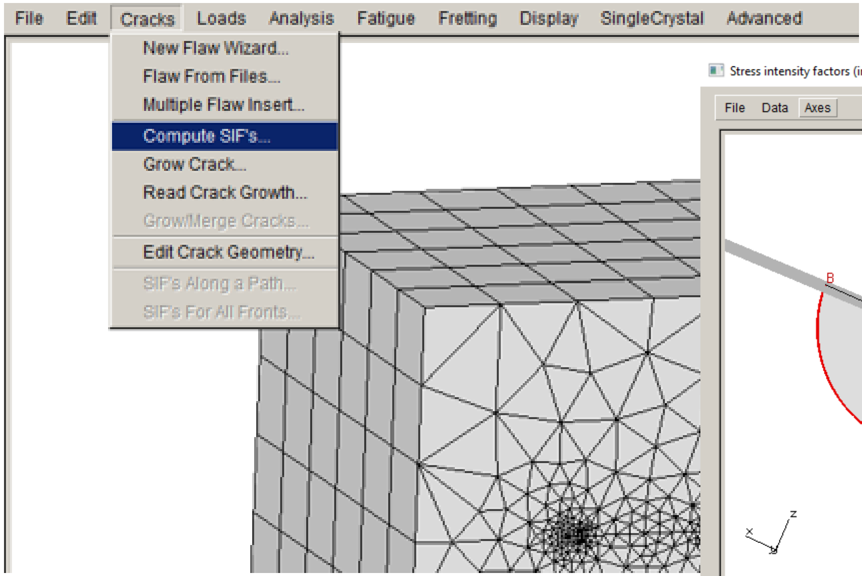


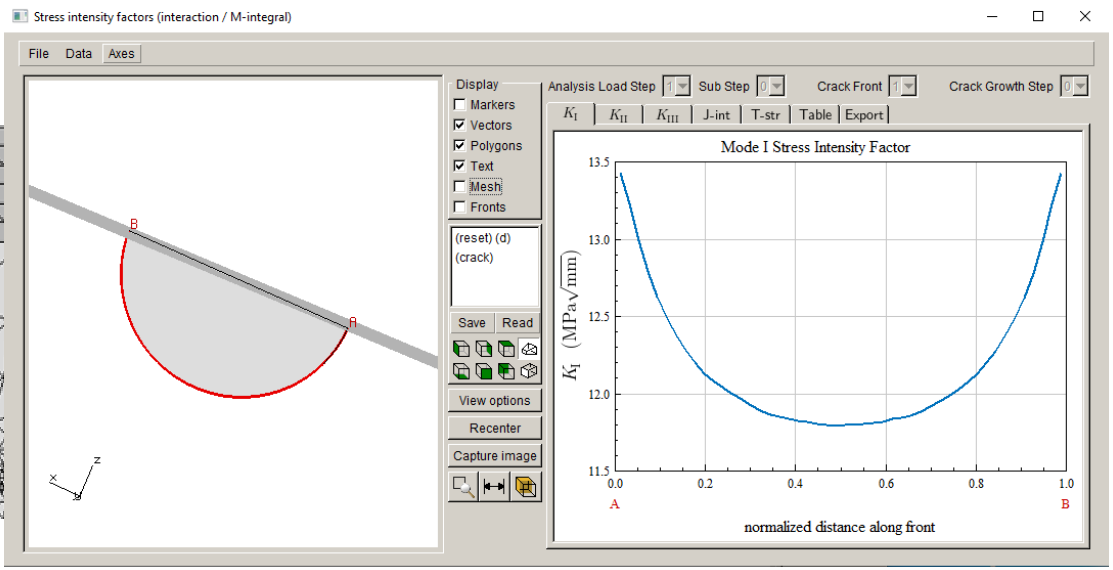
Compare ANSYS displacements for the uncracked and cracked models once the analysis has finished.

Step 6: Compute SIFs

- Cracks → Compute SIFs
  - Use M-integral
  - Plot KI, KII, KIII
  - Units: MPa√mm



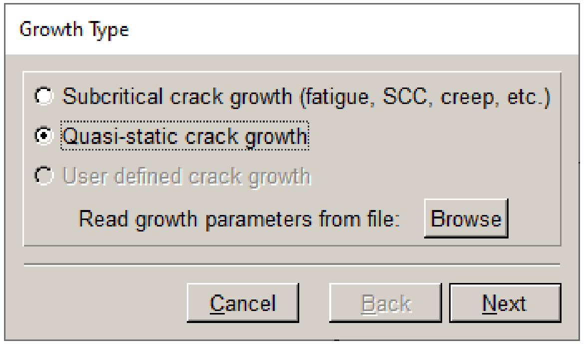


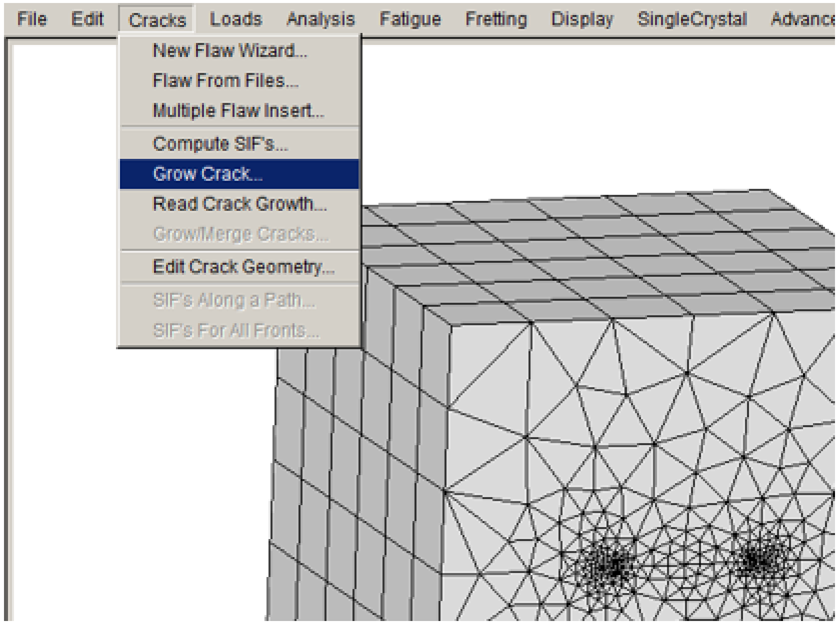


Step 7: Manual Crack Growth

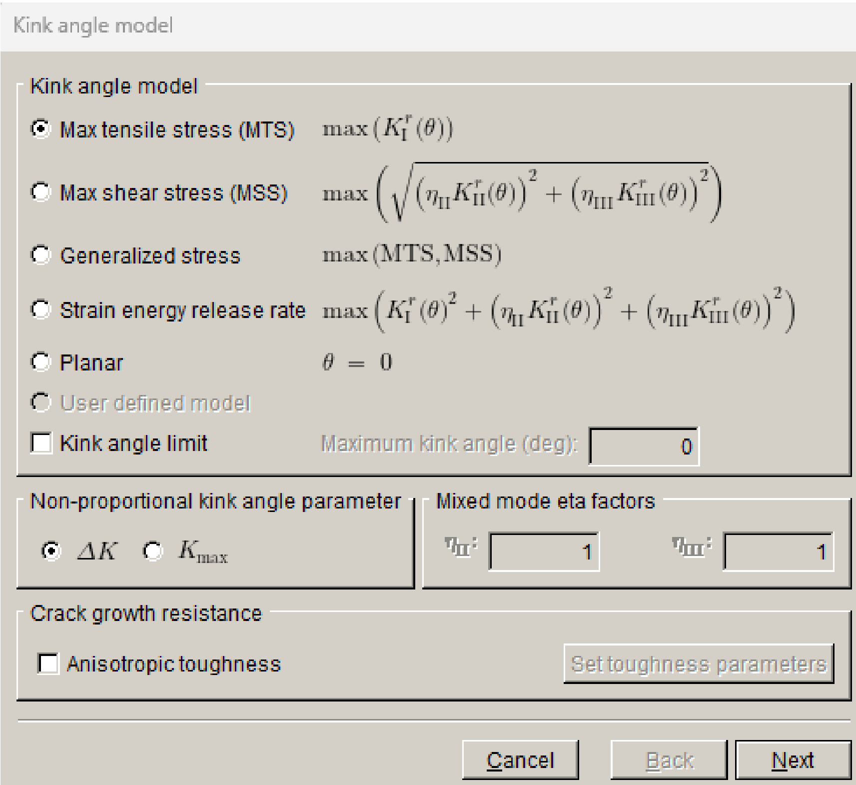
- Cracks → Grow Crack
  - Quasi-static growth
  - MTS criterion
  - Polynomial fit
  - Template radius control

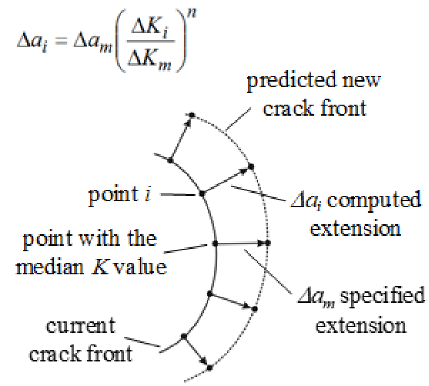
Quasi-static growth is a simplified Paris-type growth rate model.

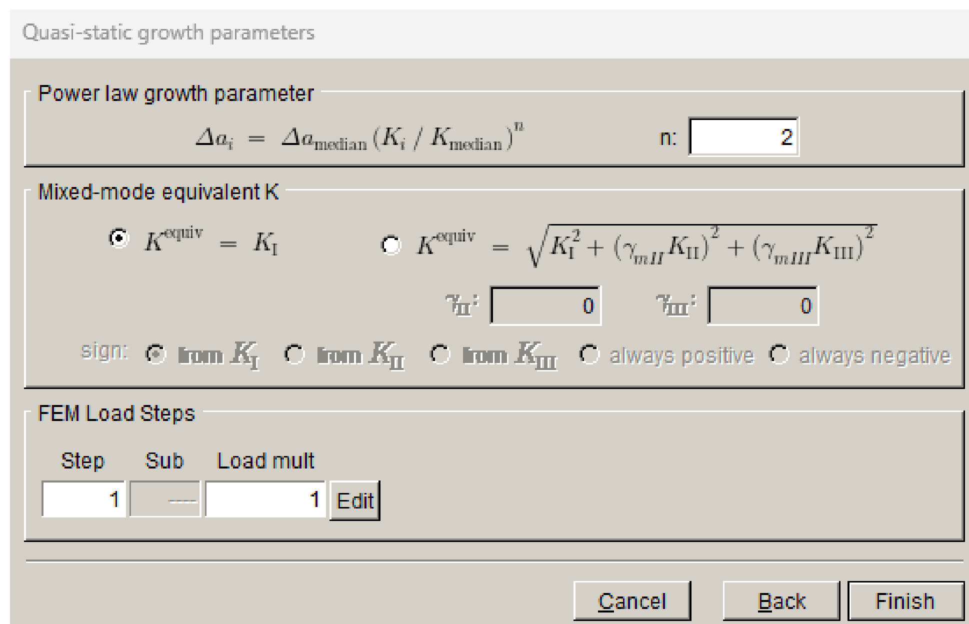




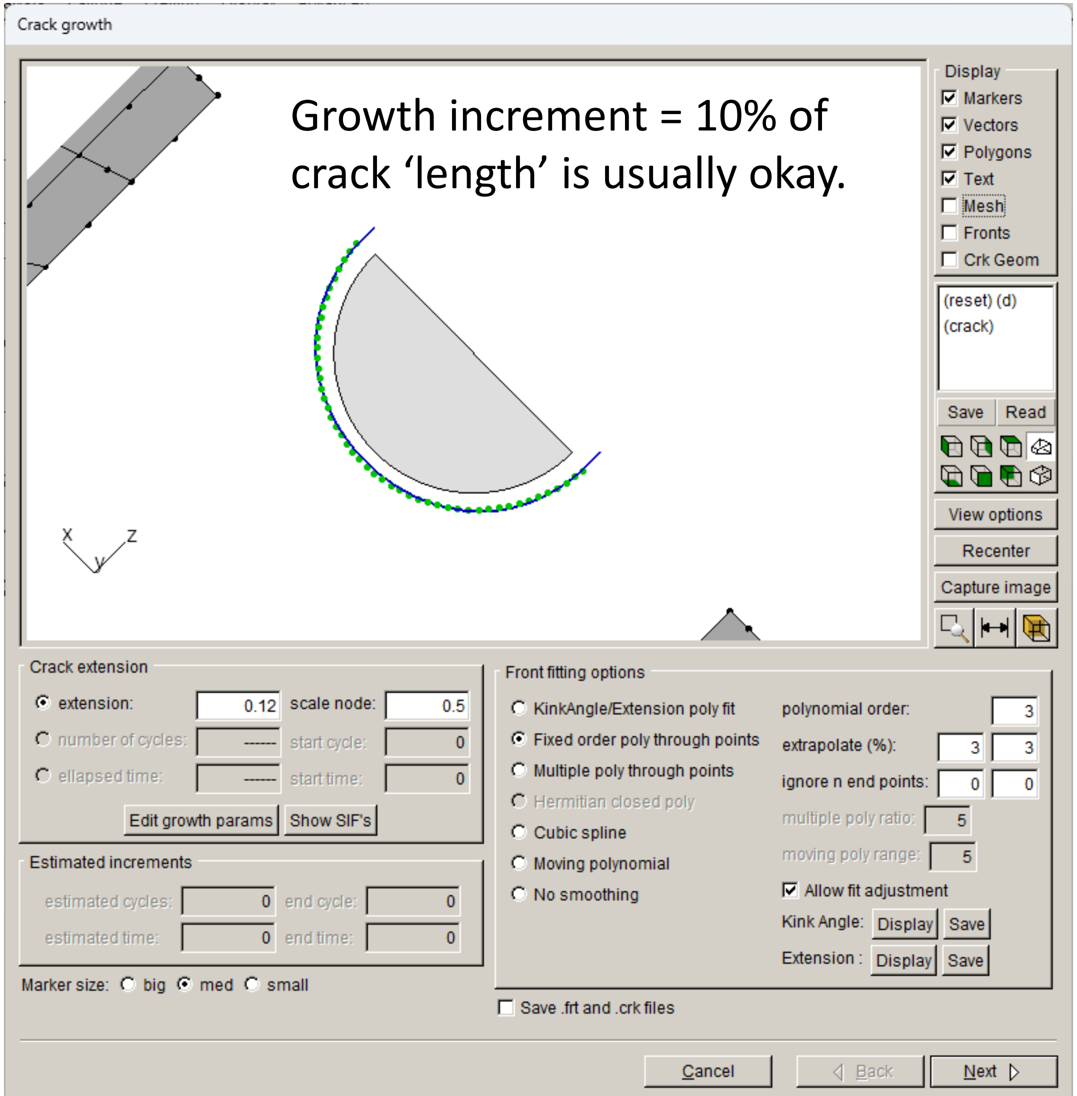
Step 7: Manual Crack Growth





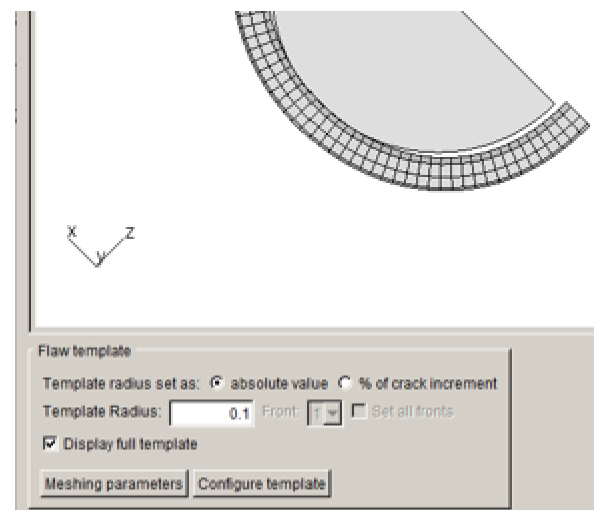


Step 7: Manual Crack Growth



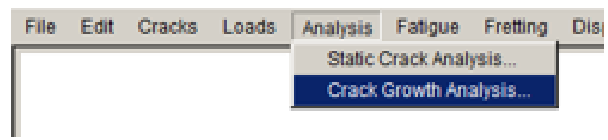
User specified extension applied at the front point where KI = median KI (if scale node = 0.5)

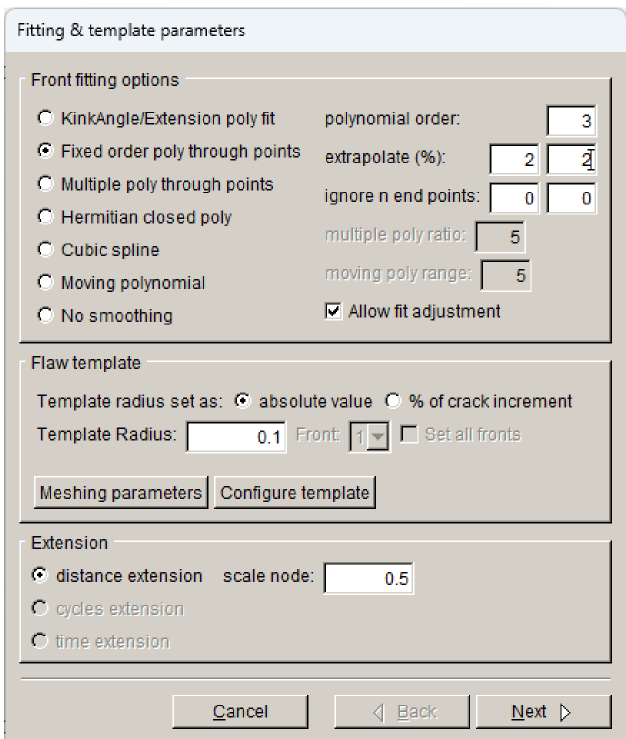
Use the simplest form of front point fitting that gives a reasonable fit (usually a single 3rd order polynomial is adequate)

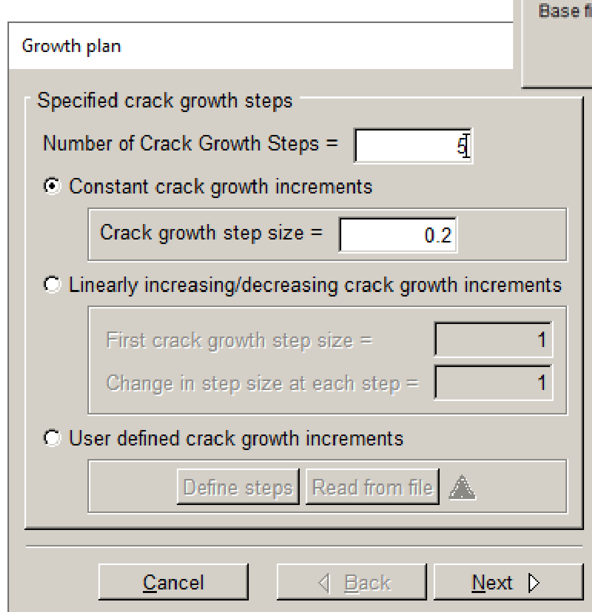


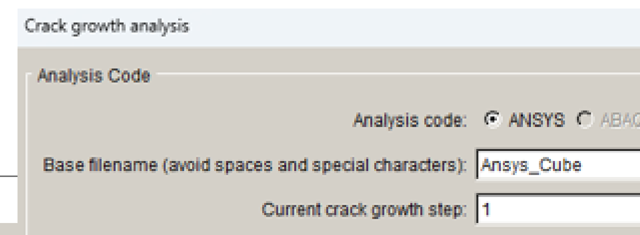
Step 8: Automatic Crack Growth

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







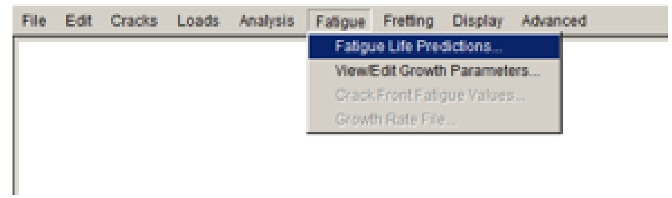


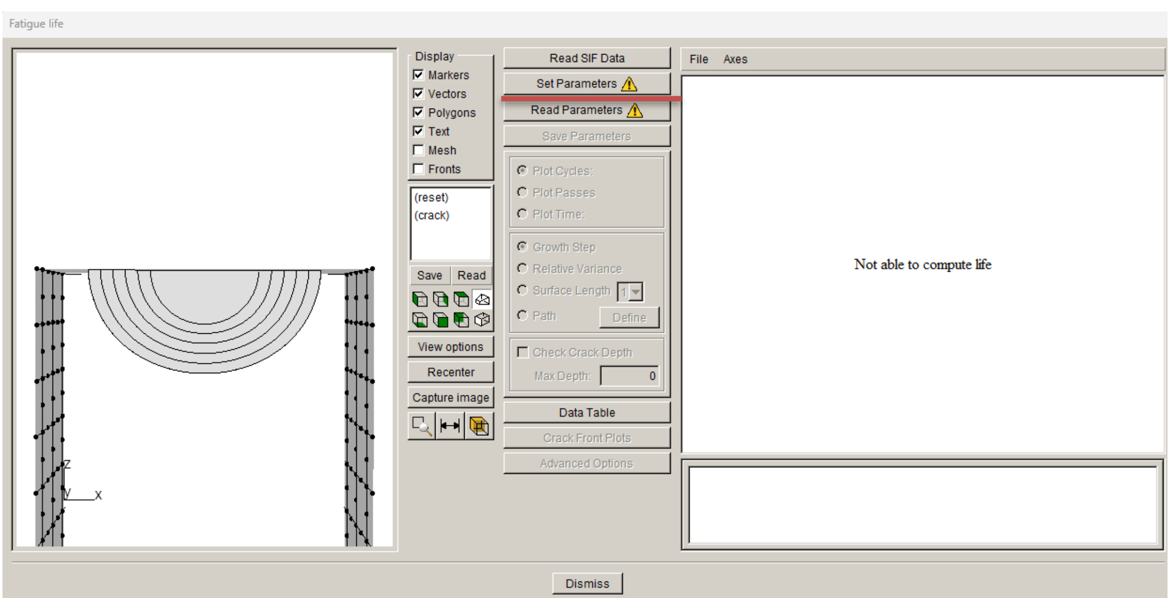
Each step of crack growth is numbered: _STEP_###.

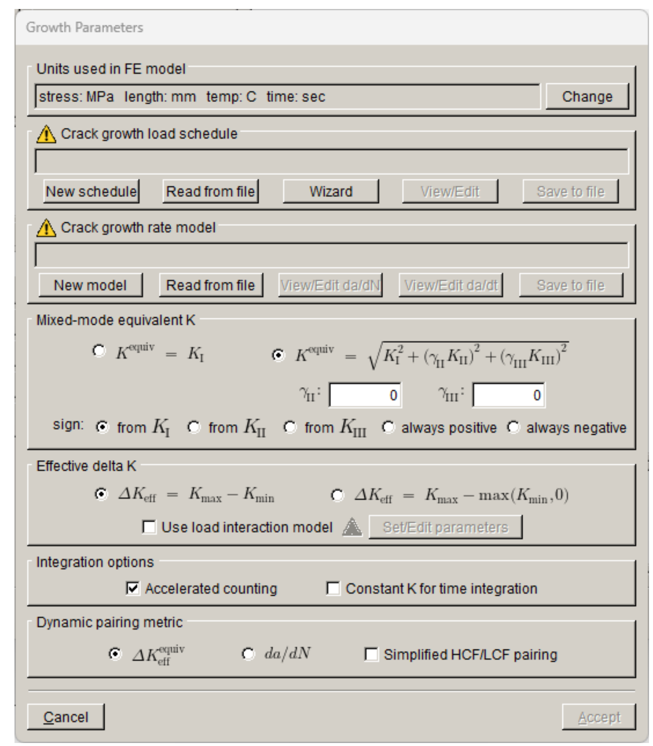
Files are saved at each step of growth so that the user can restart from any step.

Step 9: Fatigue Life Prediction

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

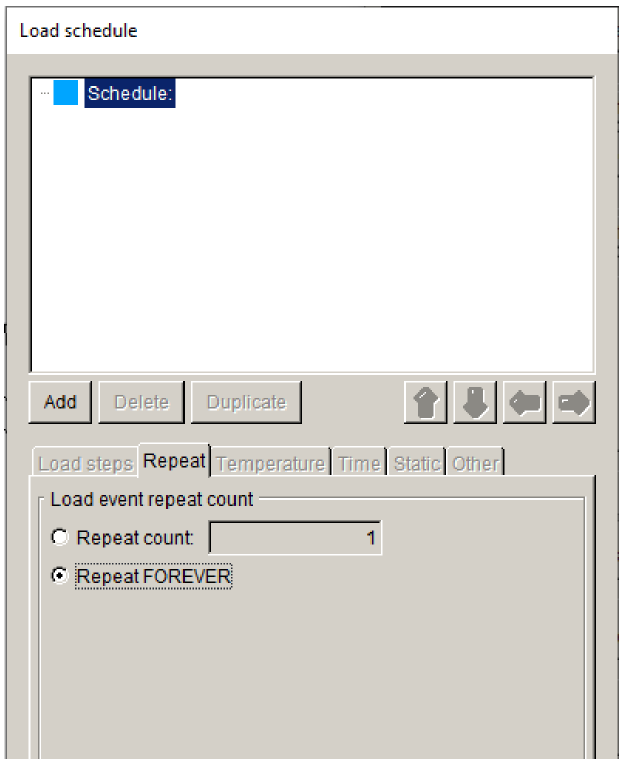


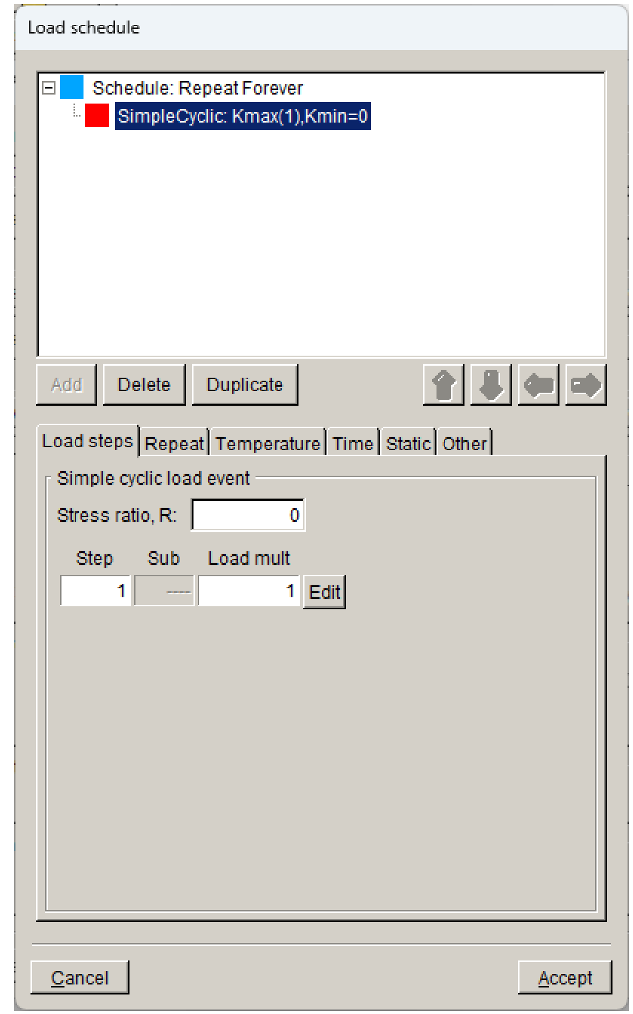




Step 9: Fatigue Life Prediction

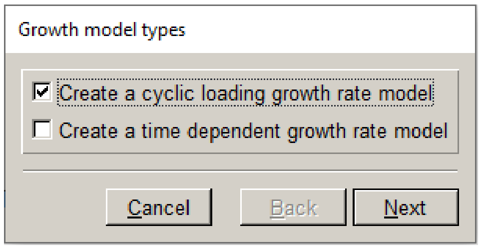
Define the load schedule:

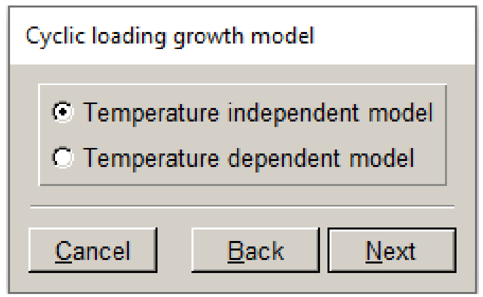


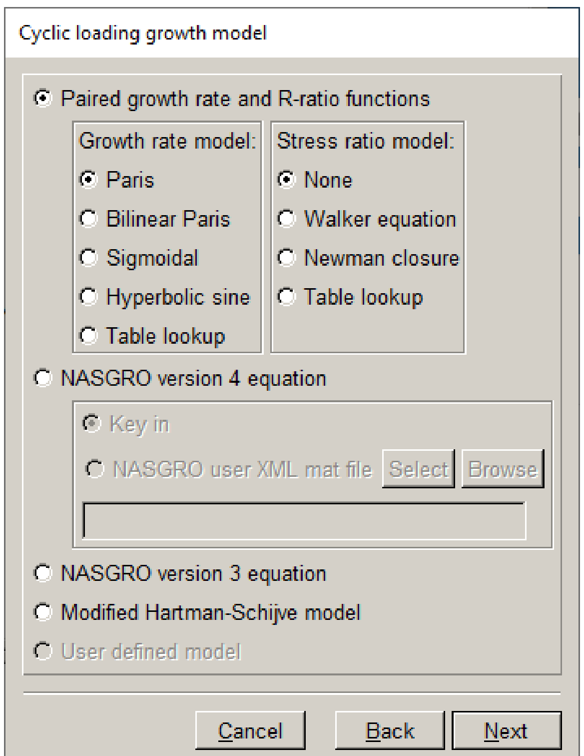


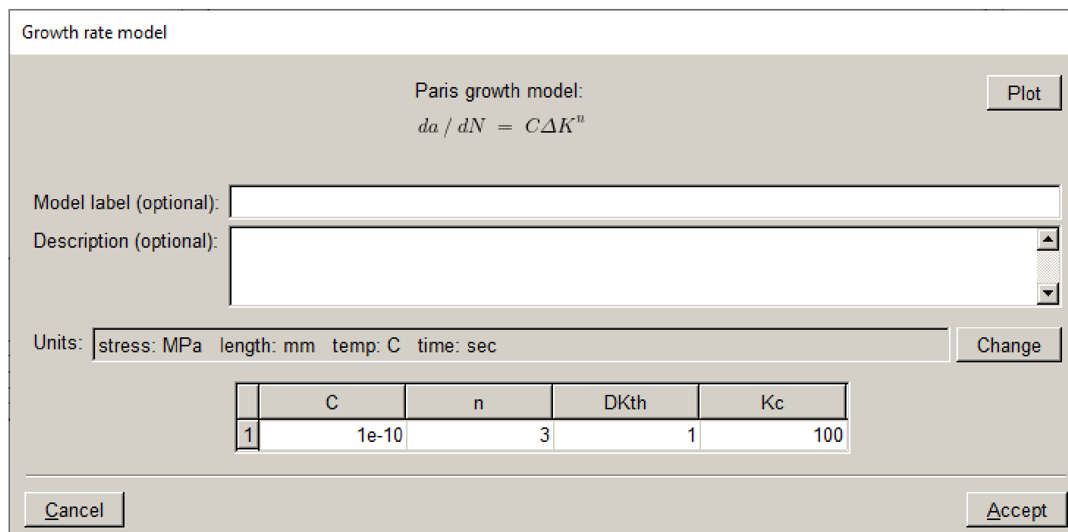
Step 9: Fatigue Life Prediction

Define the growth rate model:

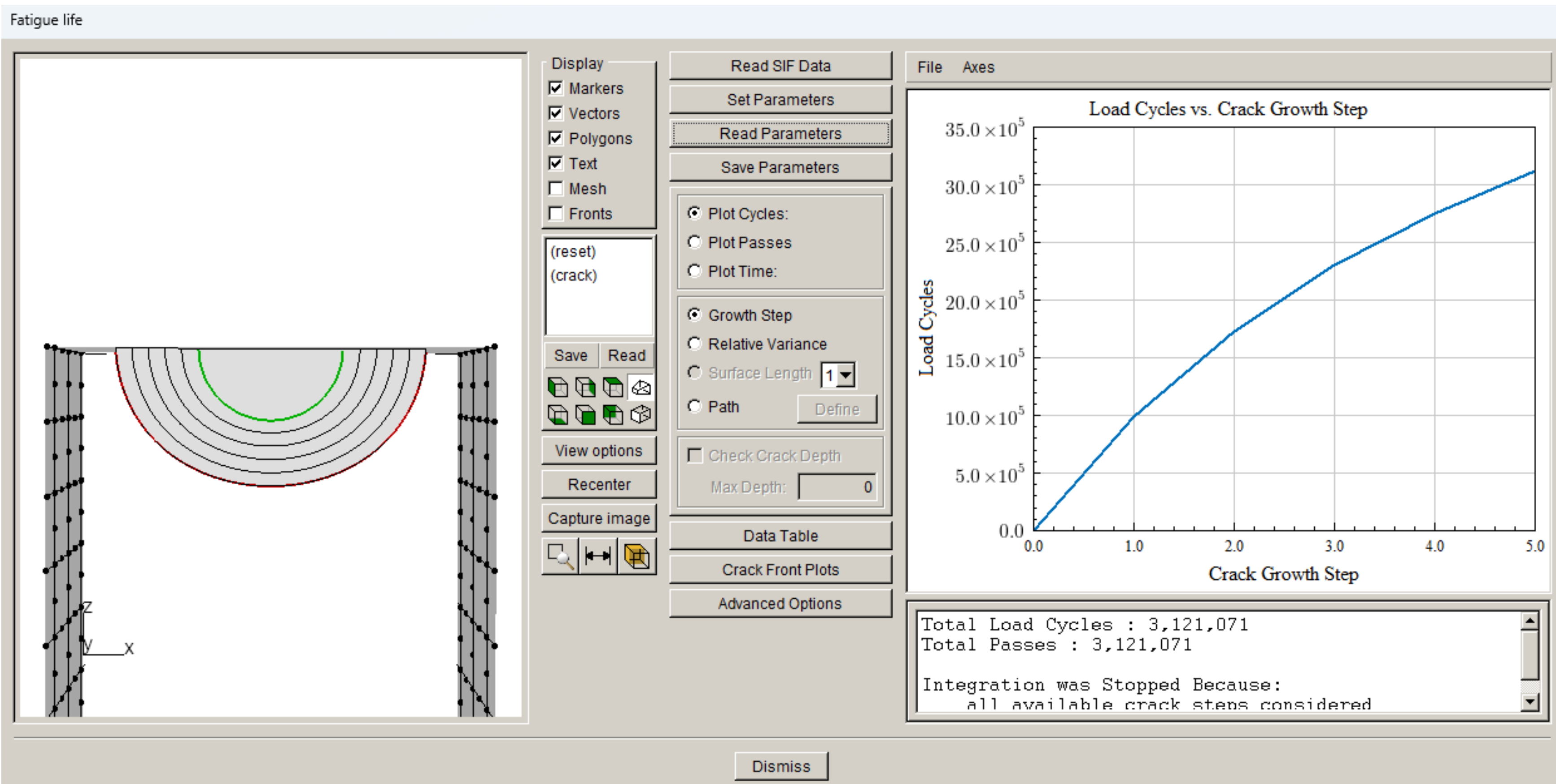








Step 9: Fatigue Life Prediction



Conclusion

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