FRANC3D Training Workshop: Part 11

Miscellaneous Topics

March - 2026

Drs. Paul "Wash" Wawrzynek, Bruce Carter,
Tony Ingraffea and Omar Ibrahim

Fracture Analysis
Consultants, Inc.

PROCESS Optimization CORPORATION

Workshop Agenda

- Part 1: Introduction to Fatigue and Damage Tolerance
- Part 2: Introduction to Fracture Mechanics Analysis
- Part 3: Introduction to FRANC3D
- Part 4: FRANC3D User Interface
- Part 5: Finite Element (FE) Model Import
- Part 6: Crack Insertion
- Part 7: Static Crack Analysis & SIF Computation
- Part 8: SIFs from FE Analysis
- Part 9: Crack Growth
- Part 10: SIF History & Fatigue Life
- Part 11: Miscellaneous Topics

Miscellaneous Topics

- FRANC3D Files and Restarts
- Reading Analysis Results
- Python Interface
- Crack Face Traction
- Merging Cracks
- Display Menu Options
- Advanced Menu Options
- Some Limitations and Potential Issues
- Elasto-Plastic Material

FRANC3D Files and Restarts

See Section 3 of the User's Guide.

Restart from FRANC3D .fdb file:

- From FRANC3D menu, select **File - Open**.
- Choose *.fdb** file and select **Accept**.
- FRANC3D automatically reads the results file** along with the initial uncracked FE*** and the cracked FE model files

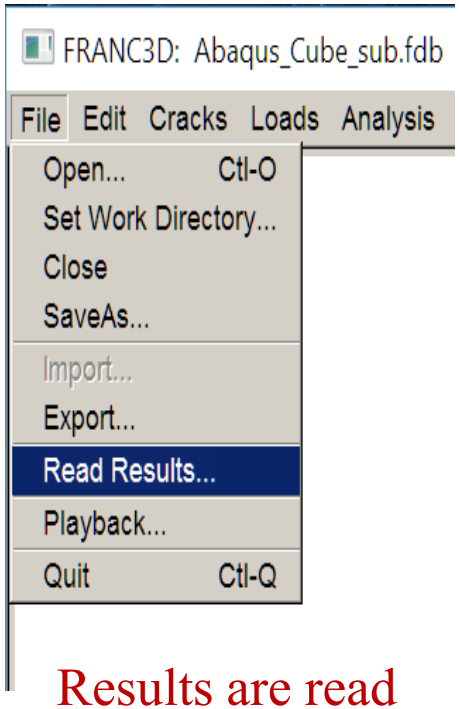
*The .fdb file is a plain text file with references to other files, the flaw geometry, mesh template and crack growth parameters, and the crack growth history.

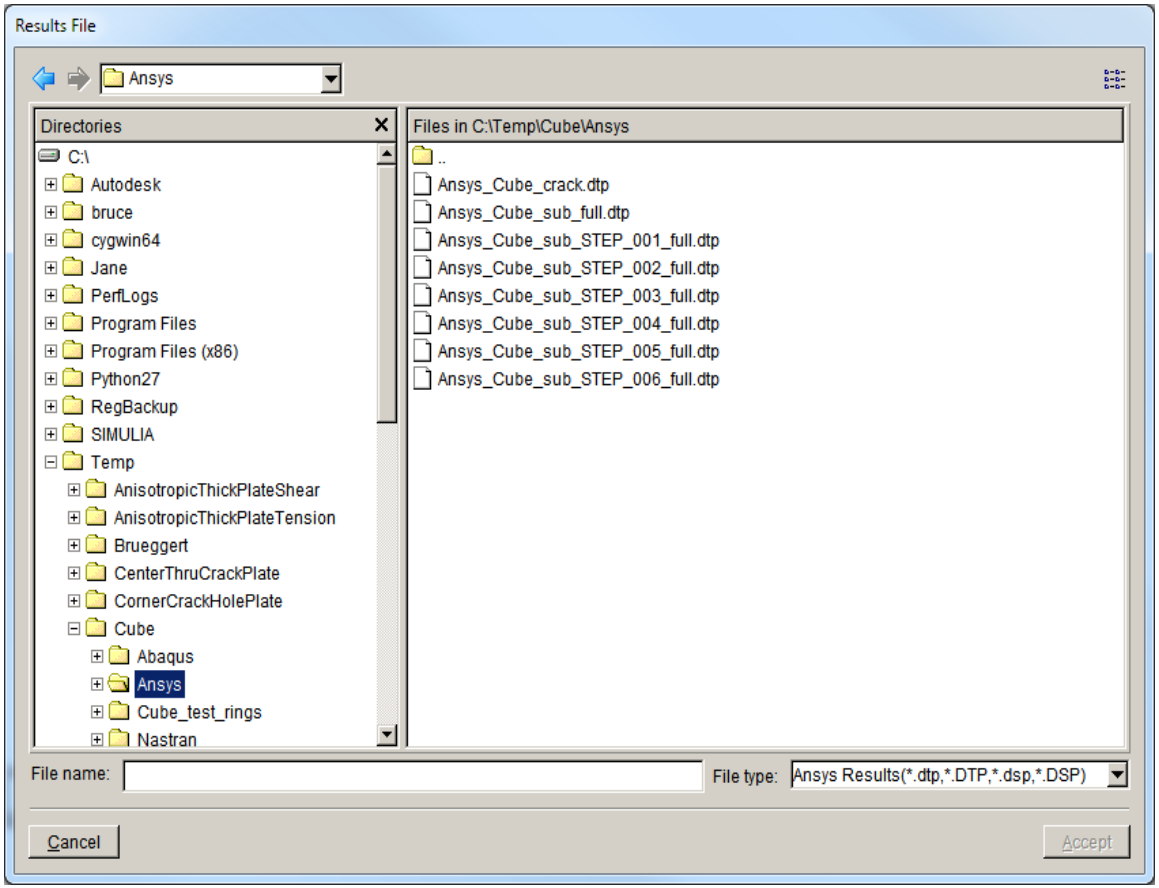
**The analysis provides nodal displacement and possibly nodal temperature and crack face contact pressure results. These will be saved in the .dtp file for ANSYS and ABAQUS; the .pch file for NASTRAN has only displacements.

***Cracks, whether initial or subsequent steps of growth, are always inserted into the original uncracked (local) FE model.

Reading Analysis Results

(if the analysis is run on a different computer)





Results are read automatically by FRANC3D when opening the .fdb file, if the results file exists.

FRANC3D PyF3D

See Commands and Python Reference

FRANC3D Python Programming Interface

- FRANC3D has a programming interface that is an extension to the Python programming language.
- Python is an open source, object oriented, scripting language, which is popular in engineering and scientific computing communities (*e.g.*, it is used with ABAQUS).
- The Python interface allows one to automate repetitive and possibly error prone tasks.
- It also provides a possible strategy for coupling FRANC3D with other computational applications.
- Use Fcl2Py executable to convert session logs to Python.

A simple PyF3D Program

```
import PyF3D

f3d = PyF3D.F3DApp()

f3d.OpenMeshModel(
   model_type="ANSYS",
   file_name='cube_orig.cdb',
   extra_files='cube_orig.s01','cube_orig.s02',
   retained_nodes_file='cube_orig_RETAINED.txt')

f3d.InsertFileFlaw(
   file_name='test.crk',
   radius=0.008)

f3d.RunAnalysis(
   model_type="ANSYS",
   file_name='test.fdb',
   flags["NO_CFACE_CNTCT"],
   connection_type="MERGE",
   command='"ANSYS252.exe" -b -p ansys -i "test.macro" -o "test.out"')

f3d.ComputeSif(
   do_therm_terms=True,
   ref_temp=71.6)
```

The session log can be converted to Python, and modified to insert a list of cracks:

```
# lists of crack size parameters to

a_sizes = [0.0160, 0.0320, 0.0480,0.0640, 0.0787, 0.2362, 0.3937]
b_sizes = [0.0160, 0.0787, 0.2362, 0.3937]

# loop through the crack size matrix

for a in a_sizes:
   for b in b_sizes:
      # create flaw, insert it, and run analysis……
```

Crack Face Traction

See Section 12 of the User's Guide

Crack Face Tractions

- Crack face traction loading can be used to include:
  - Simple pressure on the crack faces
  - Residual or initial stress condition
  - Replace full boundary conditions using the principle of linear superposition
- Crack face traction applied in FE code as nodal forces; includes normal and shear components.

Linear Superposition

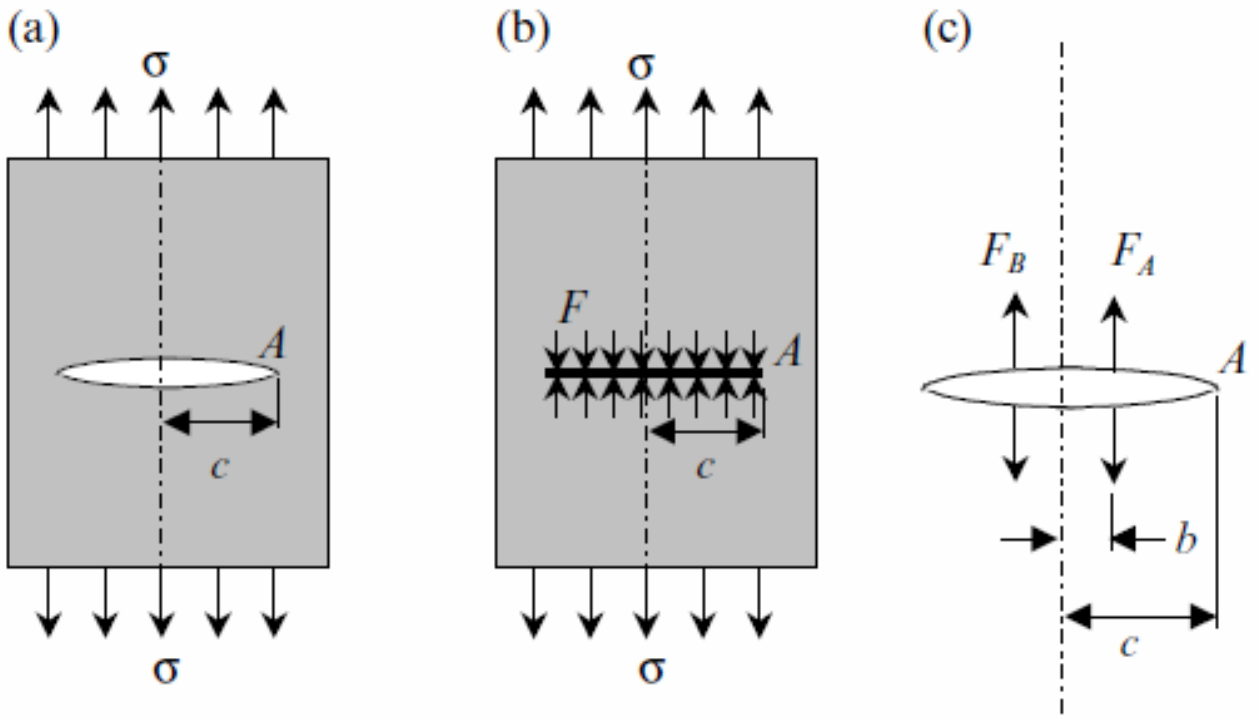


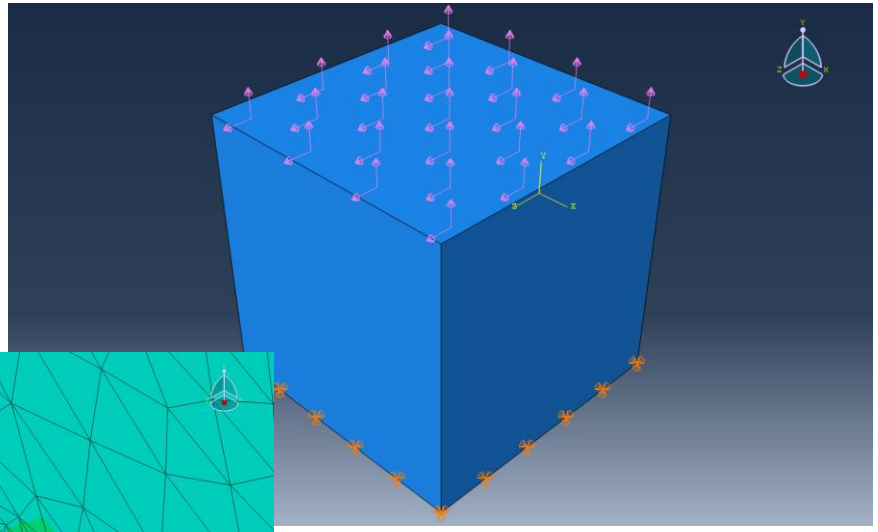
Fig 2.5.2 (a) Internal crack in a solid loaded with an external stress σ. (b) Crack closed by the application of a distribution of surface tractions F. (c) Internal crack loaded with surface tractions F_A and F_B.

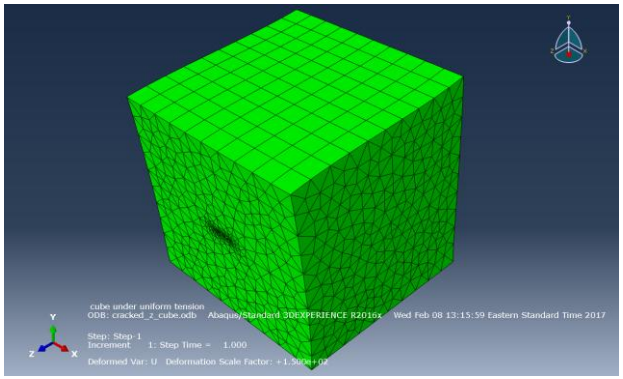
http://www.springer.com/978-0-387-68187-0

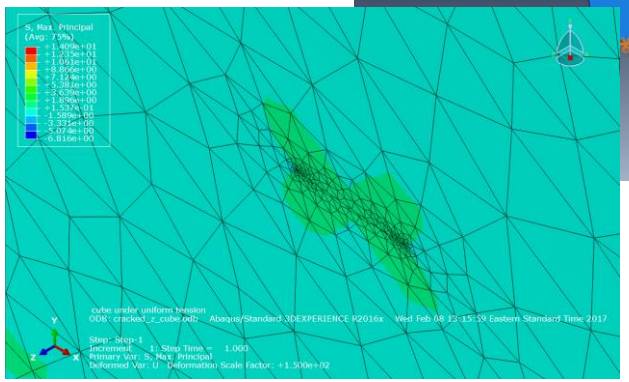
Introduction to Contact Mechanics
Fischer-Cripps, A.C.
2007, XXII, 226 p., Hardcover
ISBN: 978-0-387-68187-0

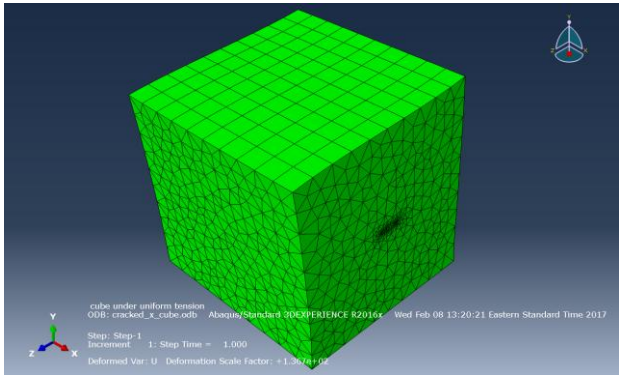
Crack Face Tractions

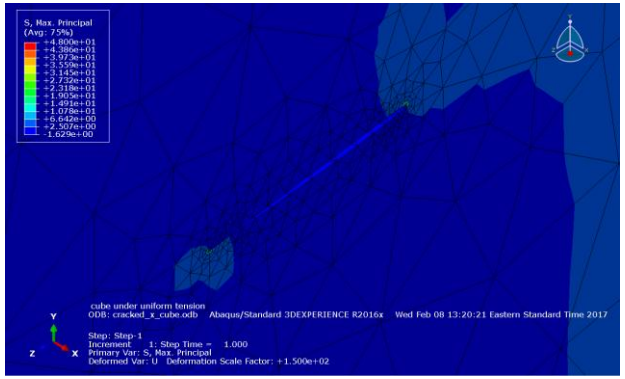
Cube model with normal tension and shear load.





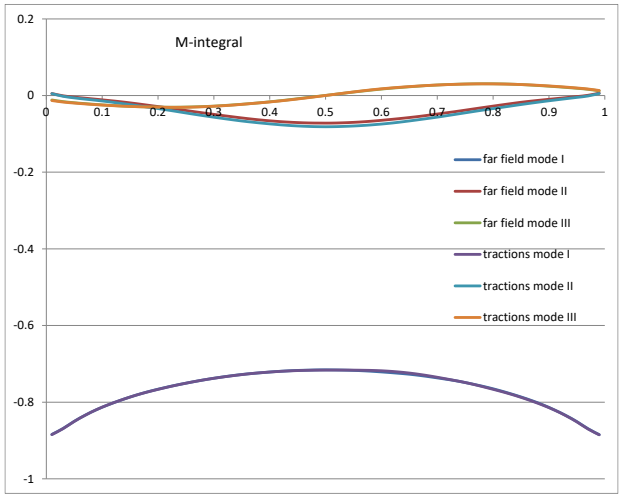


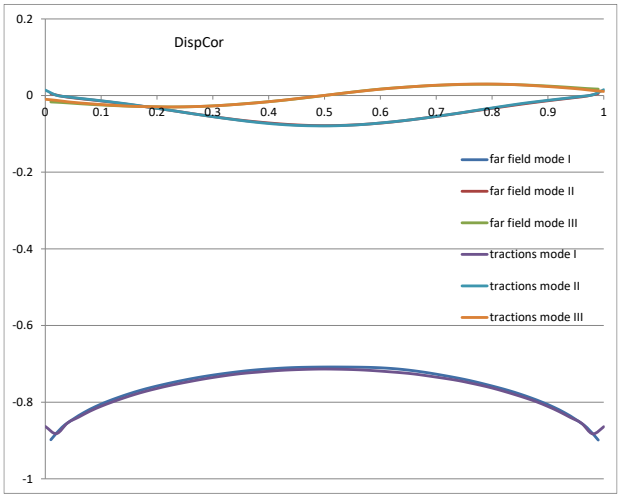




Crack on +z face and on +x face.

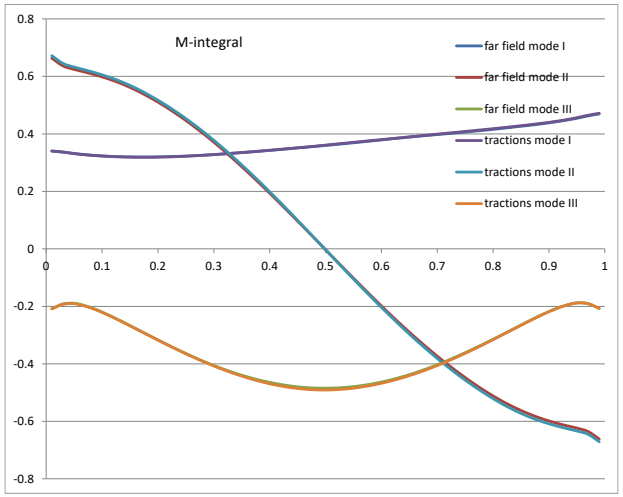
Crack Face Tractions

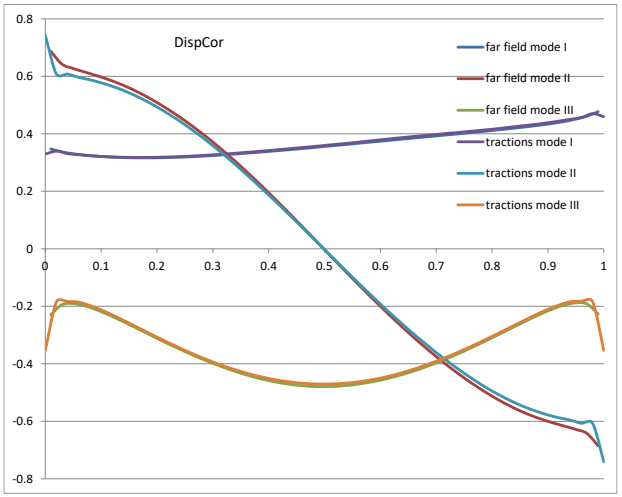




Crack on +z face

M-integral (crack face traction term included) versus displacement correlation SIFs.





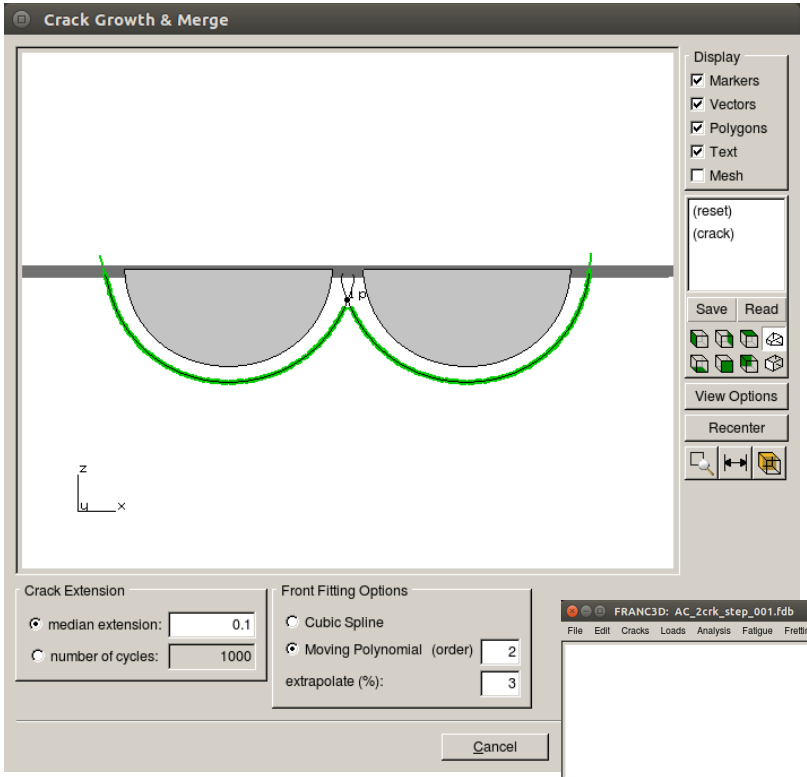
Crack on +x face

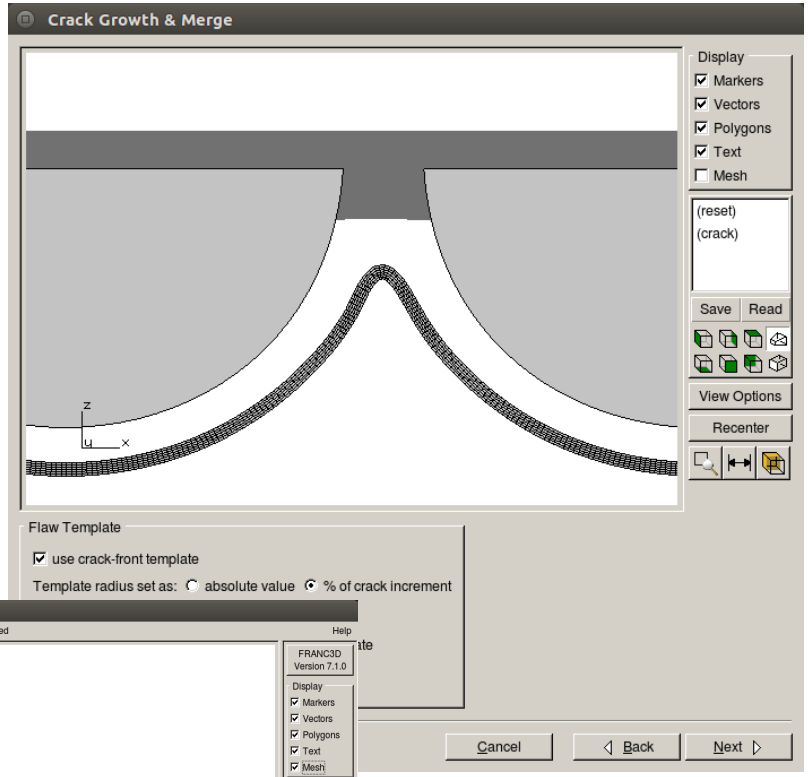
ABAQUS and ANSYS give essentially identical results.

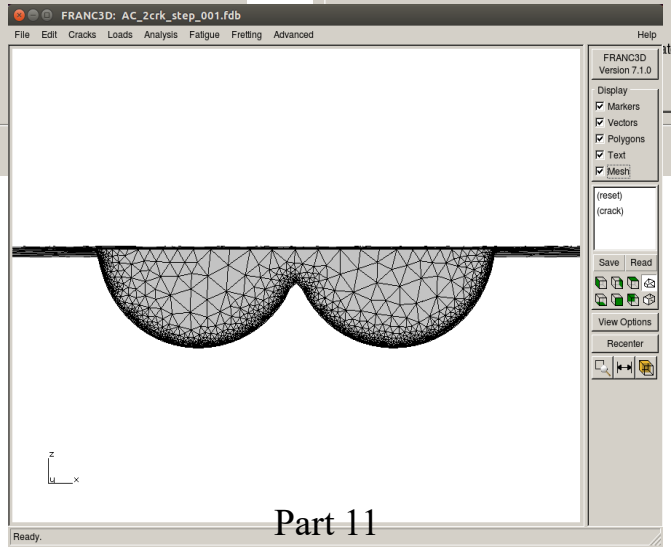
Merging Cracks

See Section 8.4 of the User's Guide

Crack Front Growth and Merge

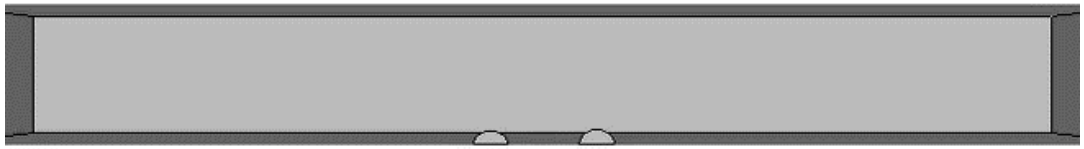


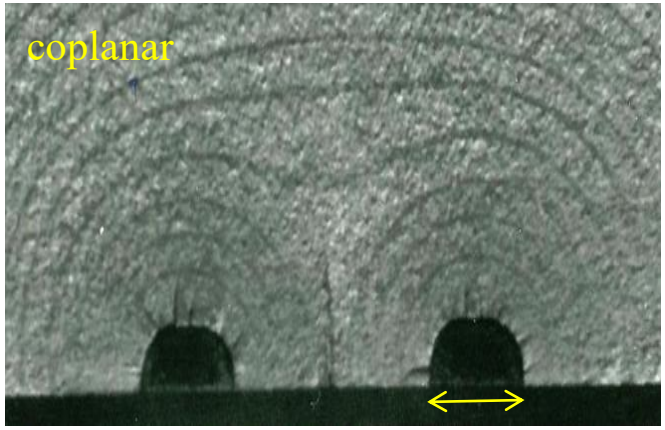




Cracks need to be mostly co-planar

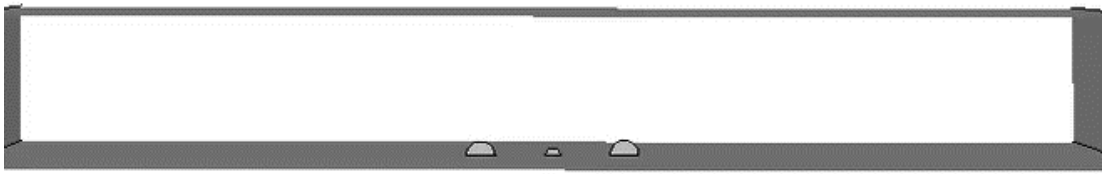
Crack Front Growth and Merge

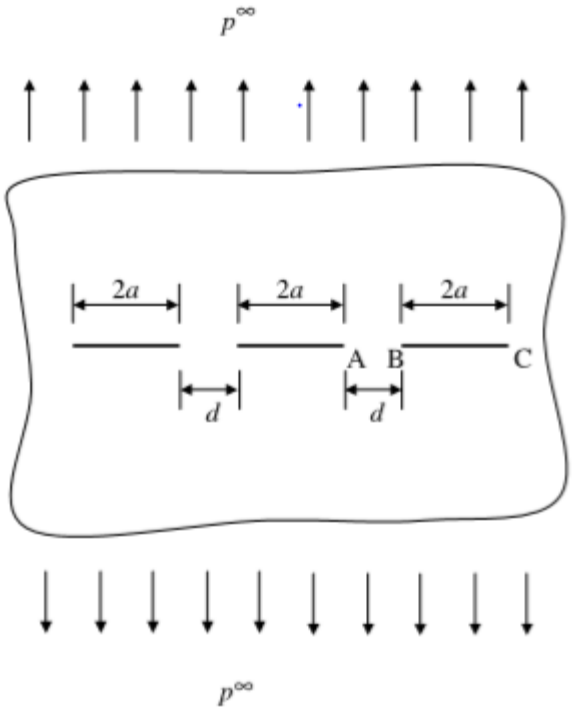




Two co-planar cracks (Manu, 1980)

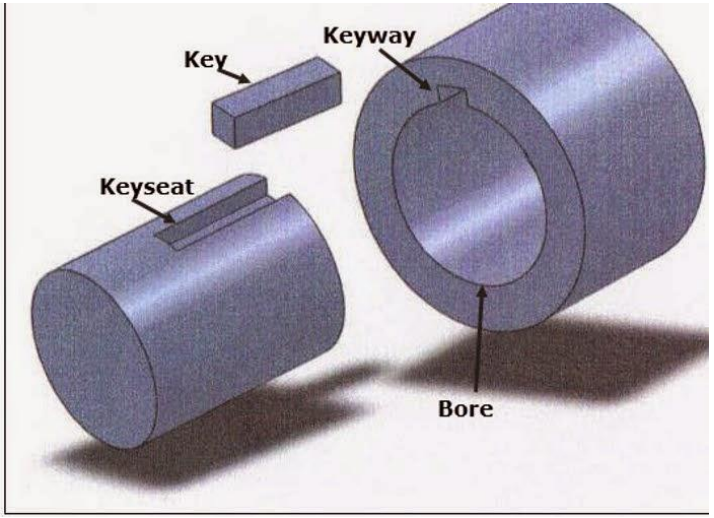
Crack Front Growth and Merge





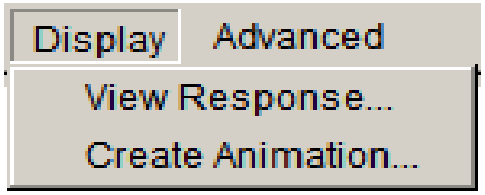
Three co-planar cracks (e.g. Li *et al*, 2003)

Crack Front Growth and Merge



Multiple cracks in a keyway

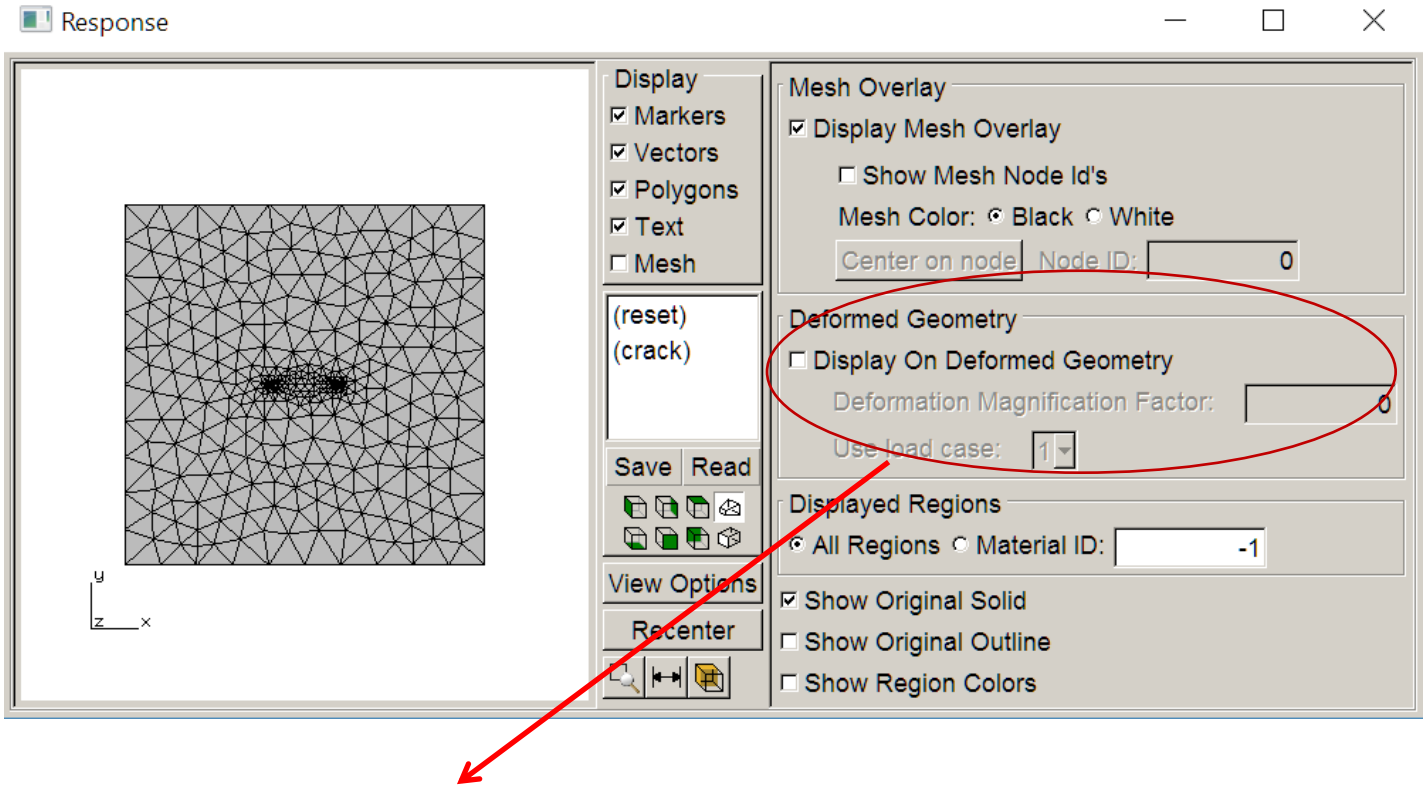
Display Menu Options



See Section 11 of the Reference

View Response

Allows one to view the deformed shape of the cracked mesh portion

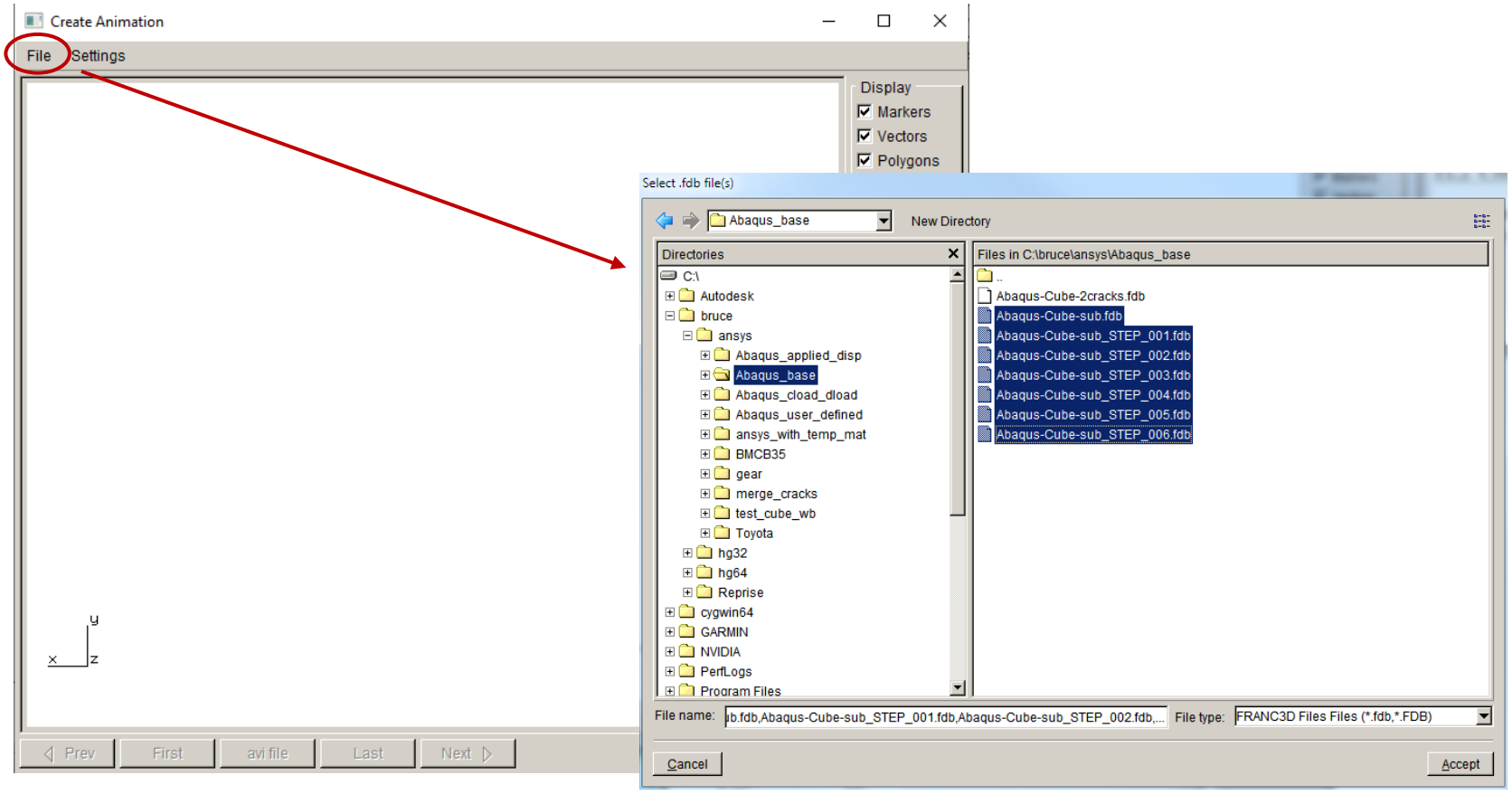


The deformed shape can be displayed by turning on the Display on Deformed Geometry option and entering a non-zero magnification factor

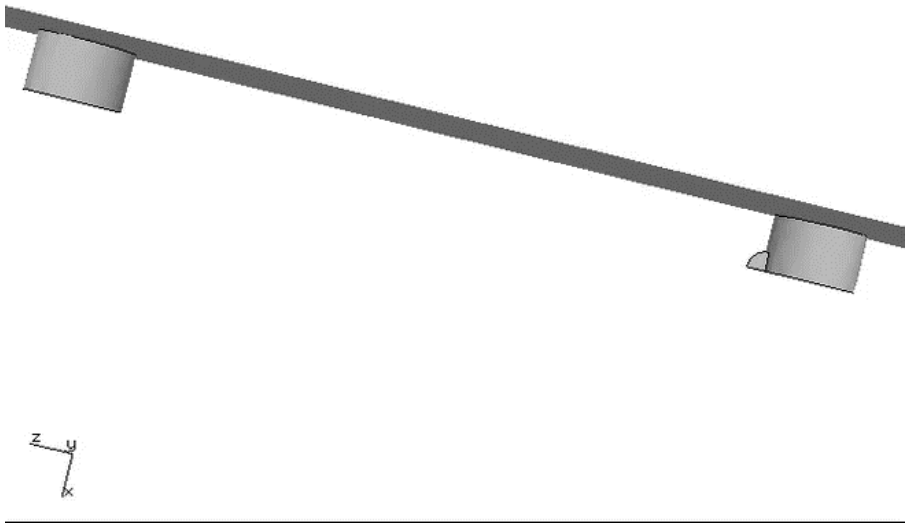
Create Animation

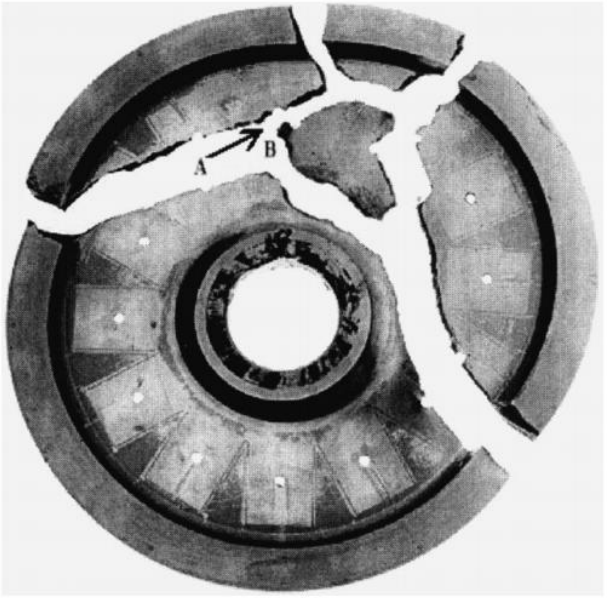
- Creates an animation of the crack growth sequence and save to an .avi file.
- Allows one to select a set of FRANC3D restart (*.fdb*) files to create an animation of the crack growth
- User must orient the model into a camera position that shows all the crack growth steps before clicking the **avi file** button

Create Animation

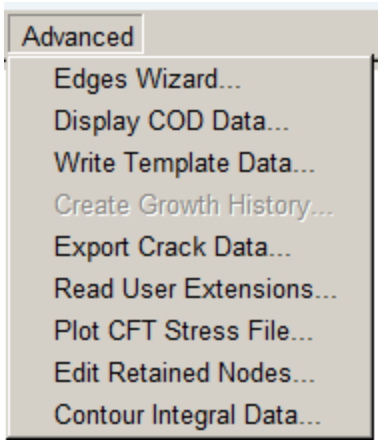


Animation of Crack Growth from a Hole in a Wheel





Advanced Menu Options



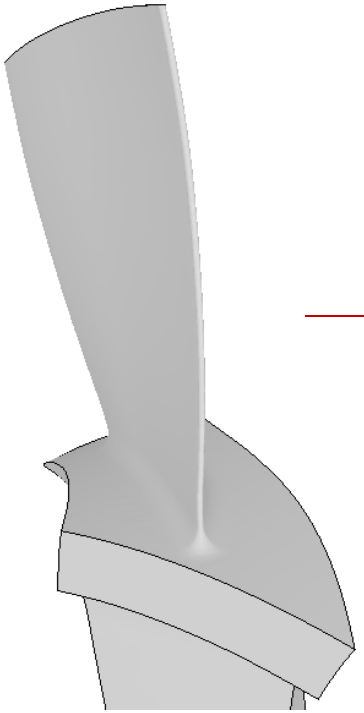
See Section 14 of the Reference

Kink Angle Edges

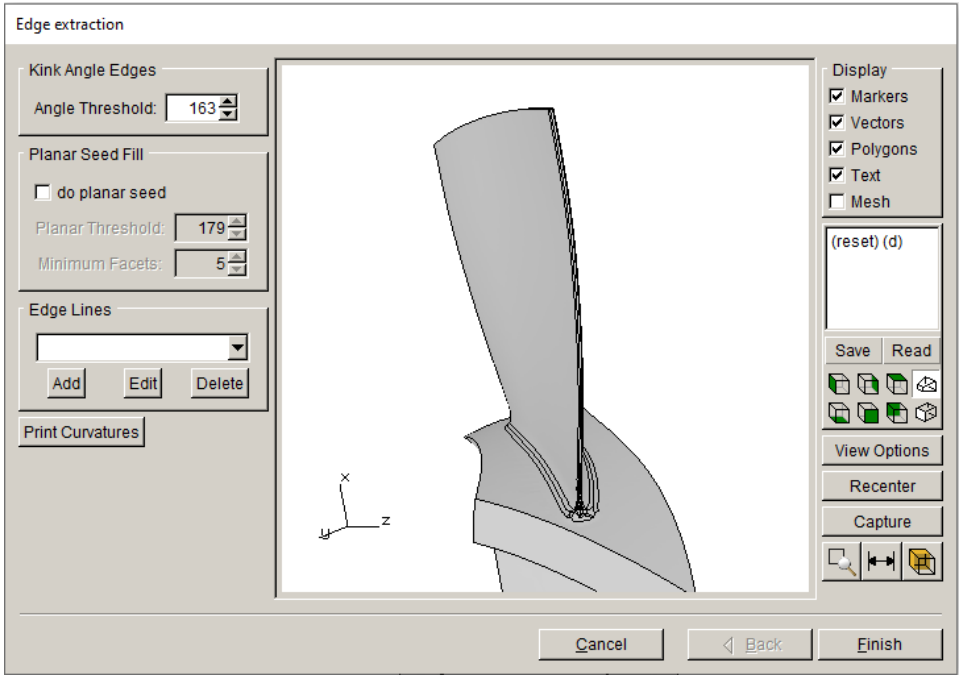
- Needed for models with no clearly defined geometry boundaries
- Allows one to control the surface geometry/topology, which might be required for crack insertion and remeshing in some models

Kink Angle Edges

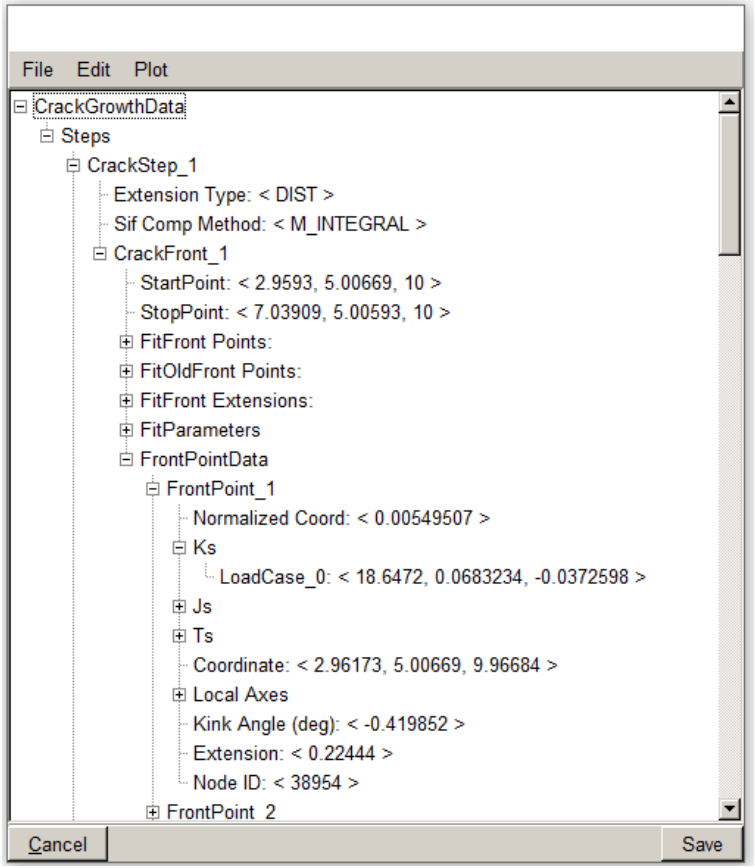
Before using edge extraction dialog



After using edge extraction dialog



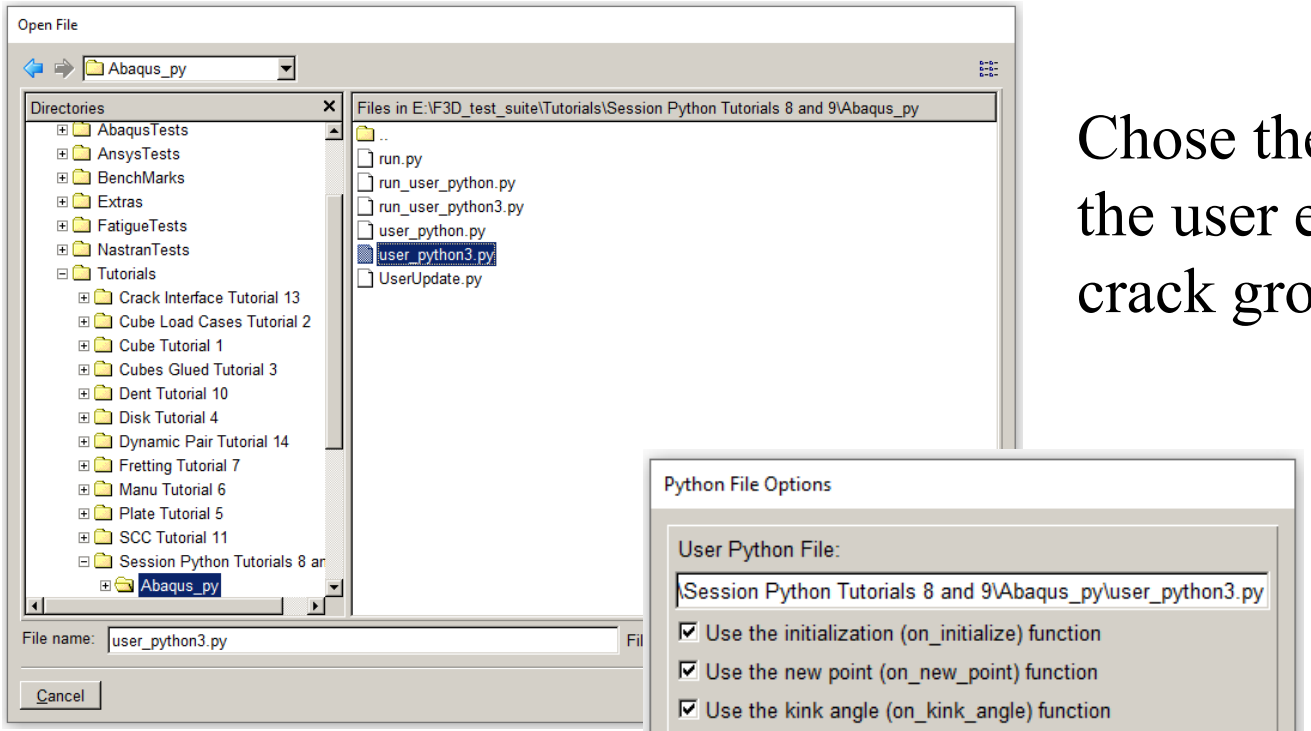
Create Growth History



Displays the crack growth data. Allows one to save this data to a .fcg file that can be used with the fatigue dialog.
Crack growth data can be combined.

See Tutorial #15

Read User Extensions (Python)



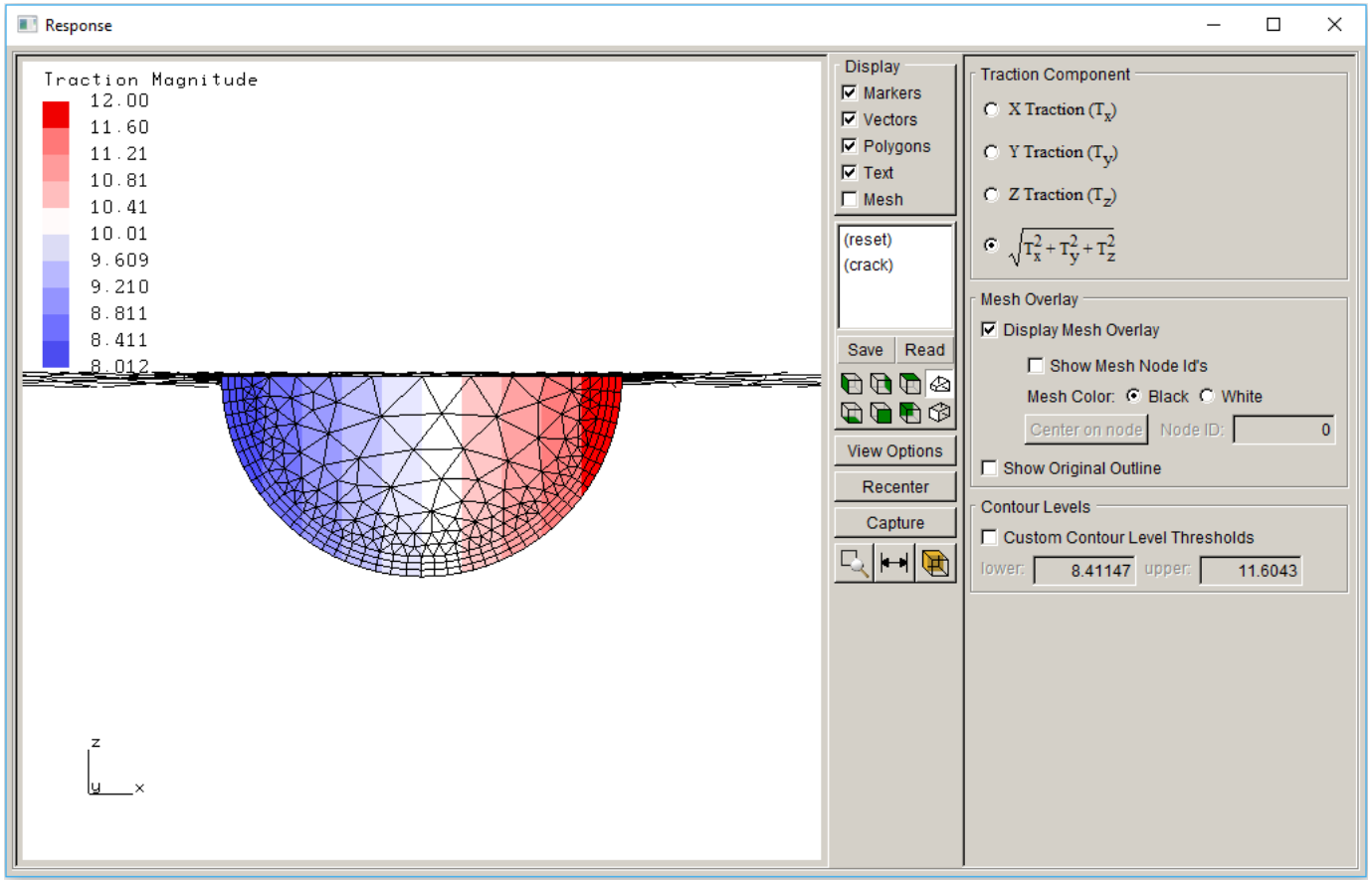
Chose the .py file with the user extensions for crack growth.

Select the functions that will be used from those defined in the .py file.

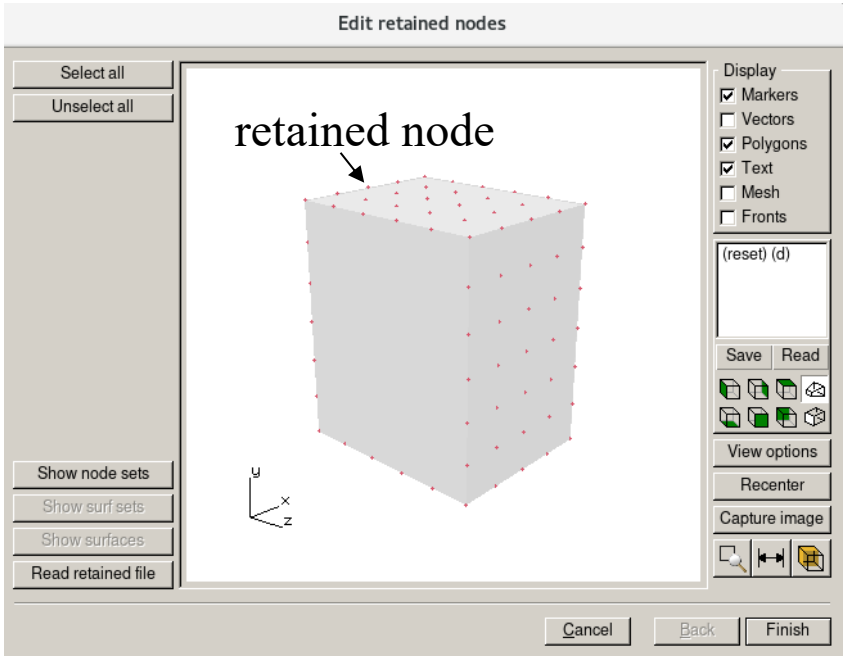
See the Commands and Python Reference.

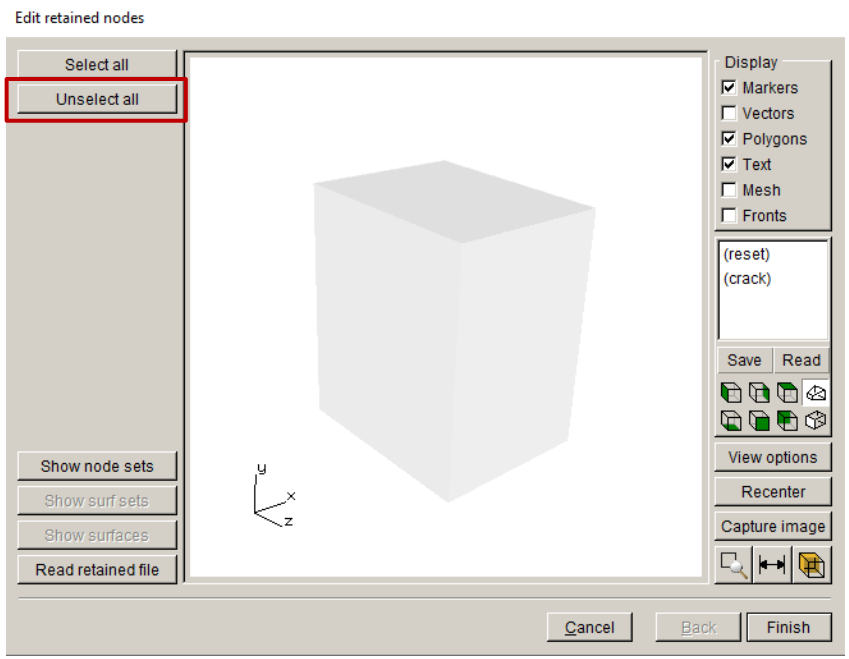
Plot CFT (section 14.9 of the Reference)

Contour crack face tractions based on mesh-based stress input



Edit Retained Nodes (section 5.6 of the User's Guide)





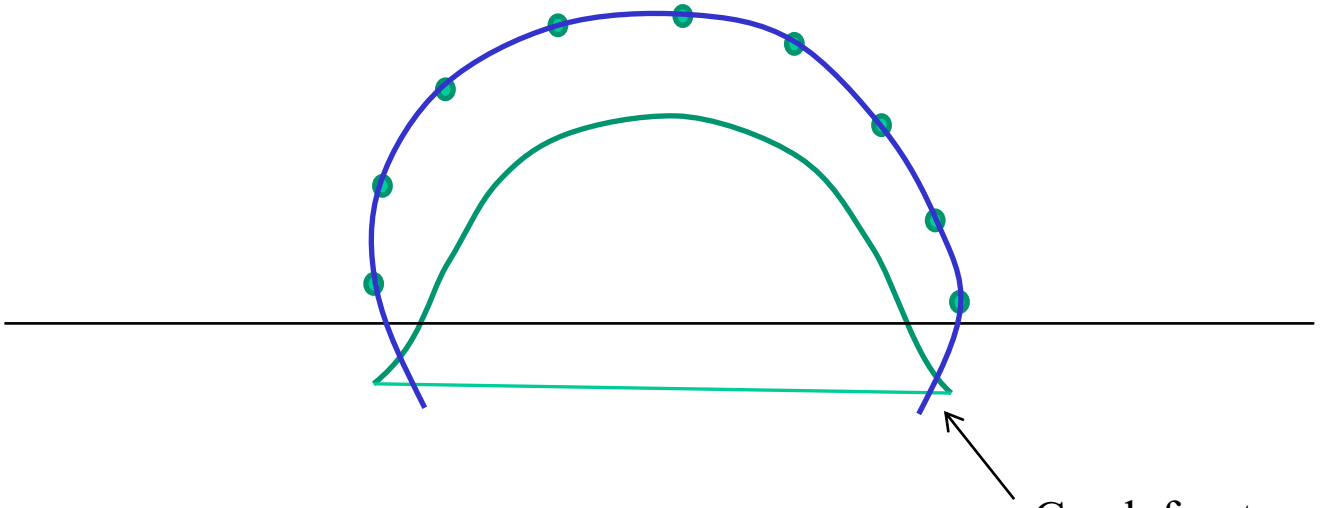
Allows one to change the retained facets if crack growth reaches a retained surface and further growth is desired.

Some Limitations and Potential Issues

See Section 8, 9 and 17 of the User's Guide

Triangulating New Crack Growth Geometry

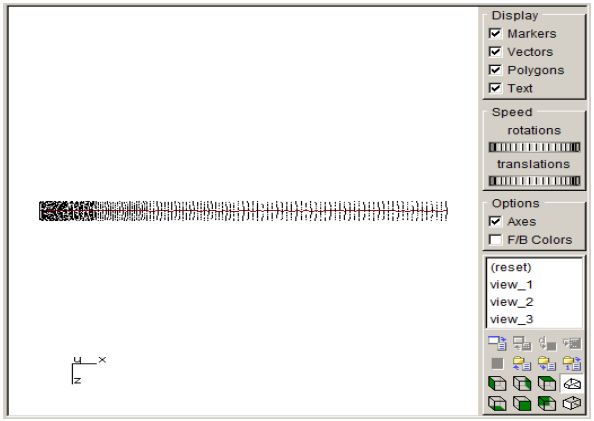
Users must make sure that crack fronts do not overlap when fitting and extrapolating.

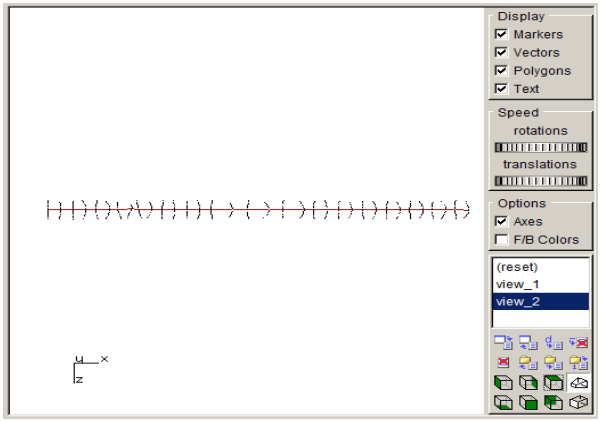


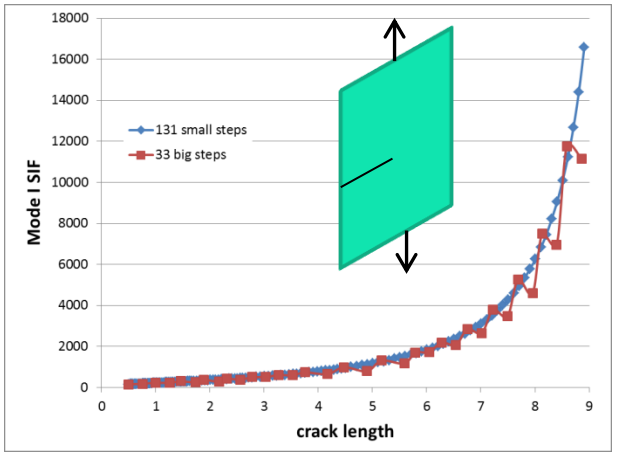
Crack front geometry overlap

```
Intersection of new and old front segments
New front point 7 intersects old front
Intersection of new and old front segments
clearing crack front ids
Failed triangulating new crack growth surface
Failed to propagate crack front 0
Failed triangulating new crack growth surface
Failed to propagate crack front 1
```

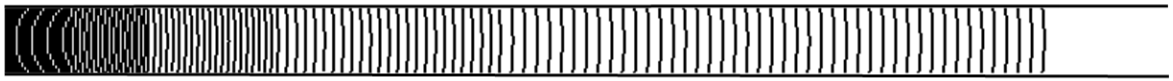
Crack Front Oscillations



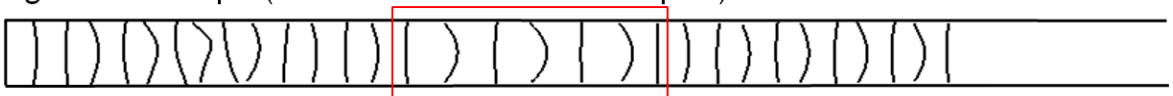


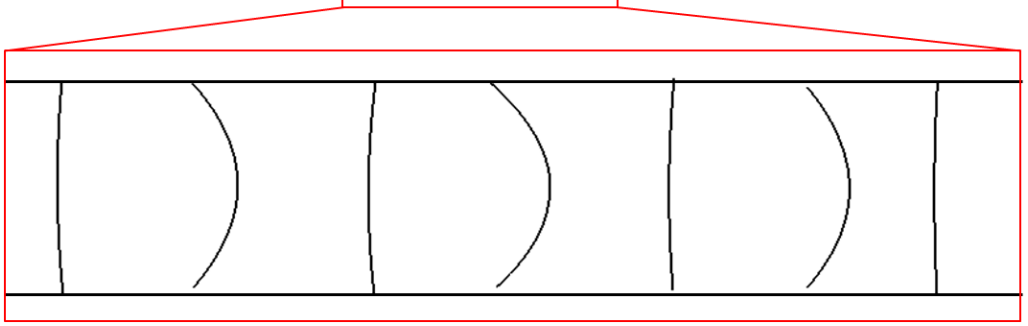


small crack steps (stable crack-front shapes)



large crack steps (unstable crack-front shapes)





If you see this type of growth, you should take smaller steps or simplify the crack front fitting, *i.e.* use a lower order polynomial.

What to do when something goes wrong during crack insertion or growth

If the program crashes before you see the "Flaw Insertion Status" window:

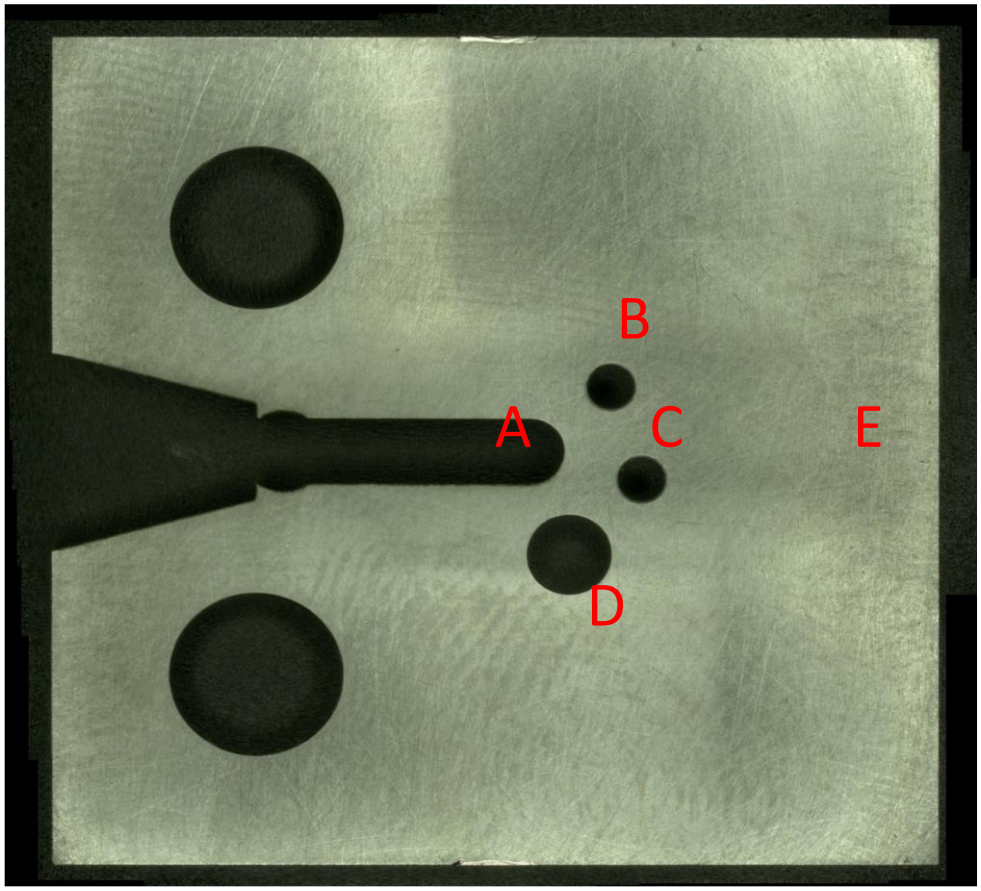
- Use the "Advanced -> Flaw to File Wizard" option to create a .crk file that describes the flaw you are trying to insert.
- Send the .crk file along with the mesh file (.inp or .cdb) to us.

If the program crashes during flaw insertion or the program reports that it cannot insert the flaw:

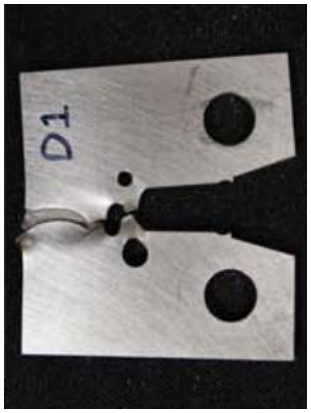
- Check to make sure that no part of the flaw or crack-front template is in the retained (cut surface) portion of the sub-model.
- Check the Command/Terminal window for messages.
- Look for a file called "debug.tst" in your working directory and send it to us.

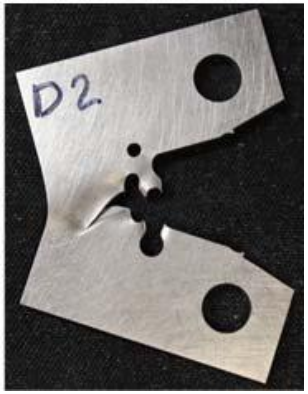
Elasto-Plastic Material

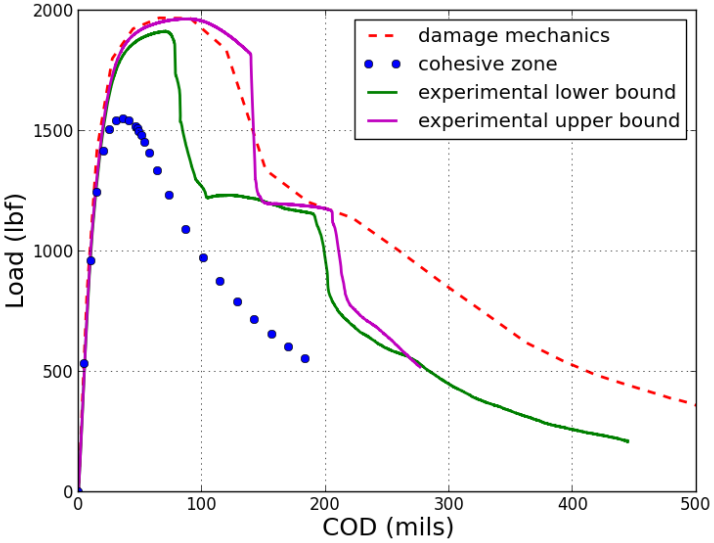
Example Simulation



33mm

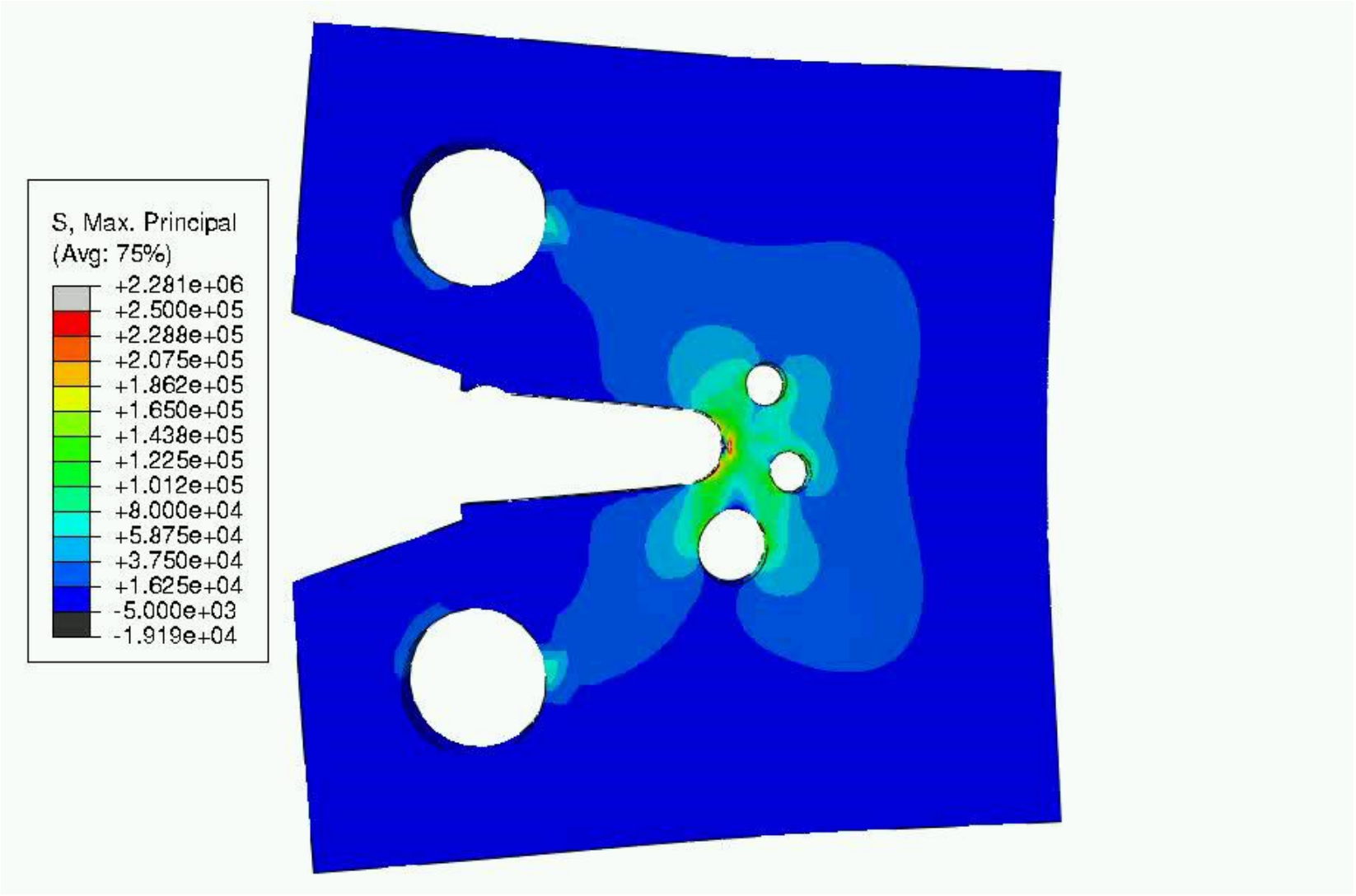




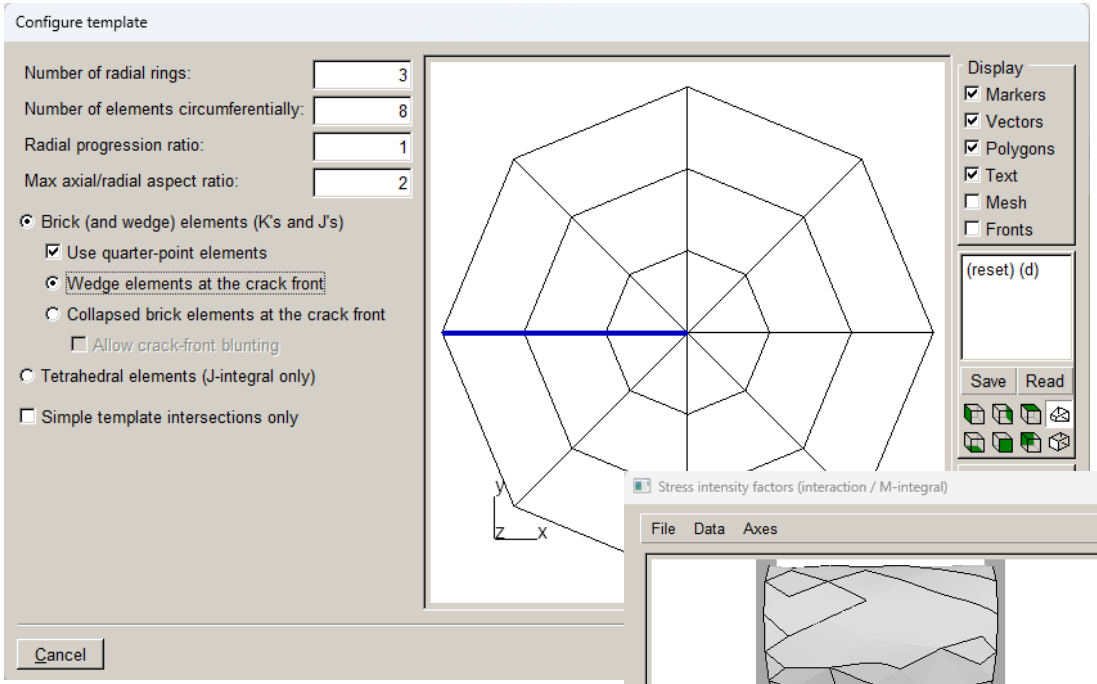


Predict fracture path and load vs COD

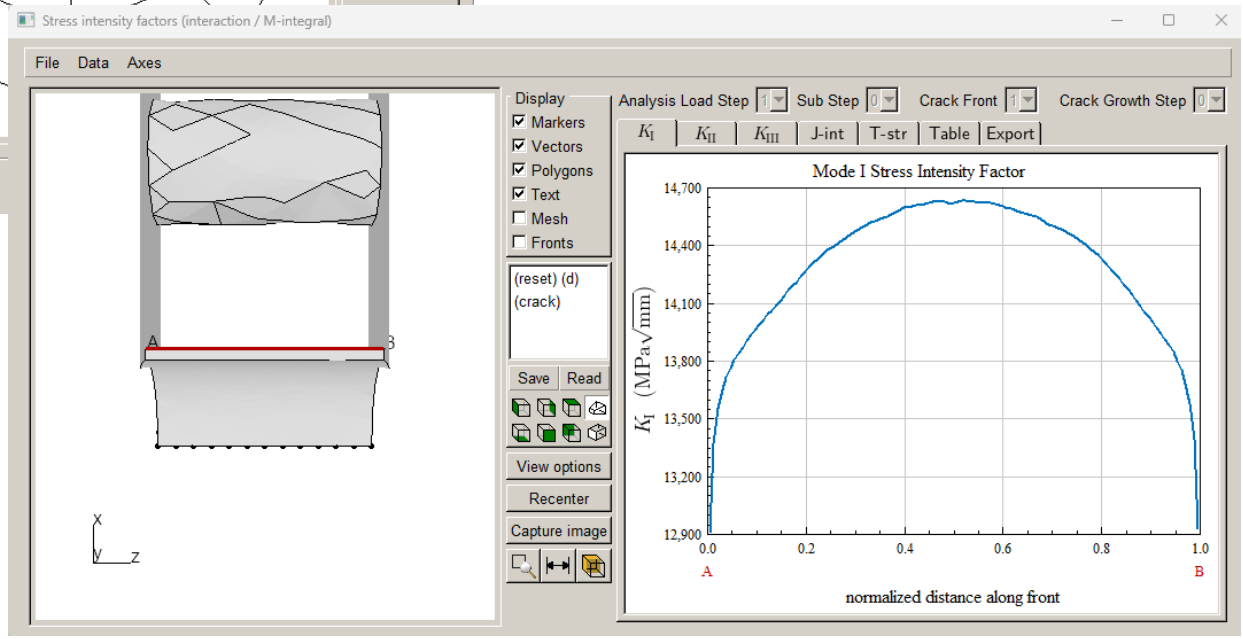
Example Simulation: LEFM



Linear Elastic – Singular Wedges

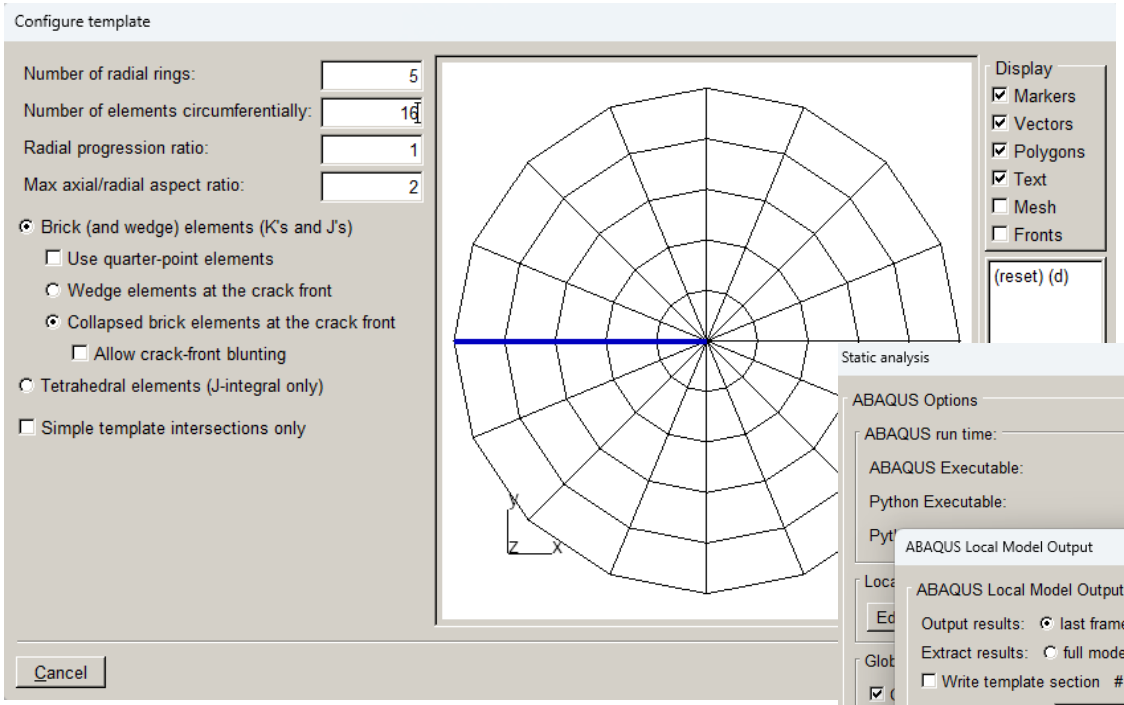


*Material, Name=Cube
*ELASTIC,
TYPE=ISOTROPIC
1.0e+04, 3.0e-01,

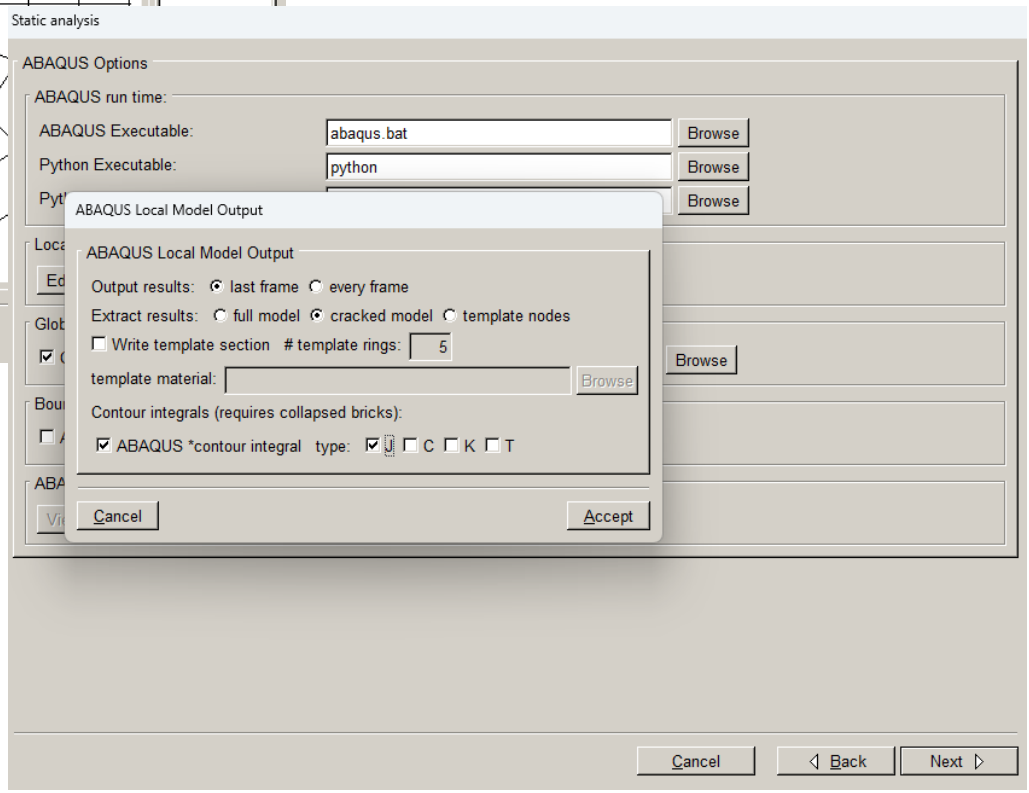


LEFM SIFs computed with M-integral

Elasto-Plastic – Collapsed Bricks with MidSides

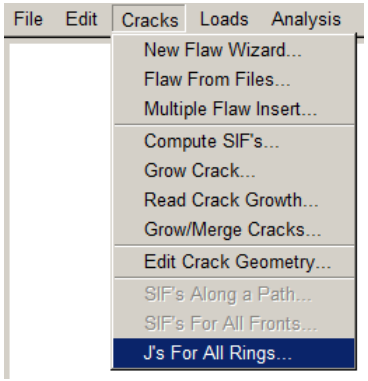


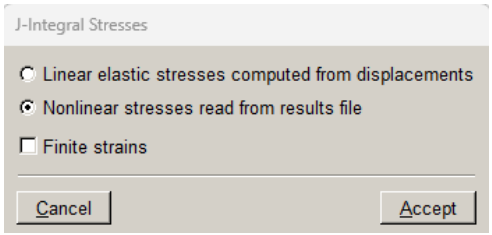
Turn on ABAQUS J-integral computation



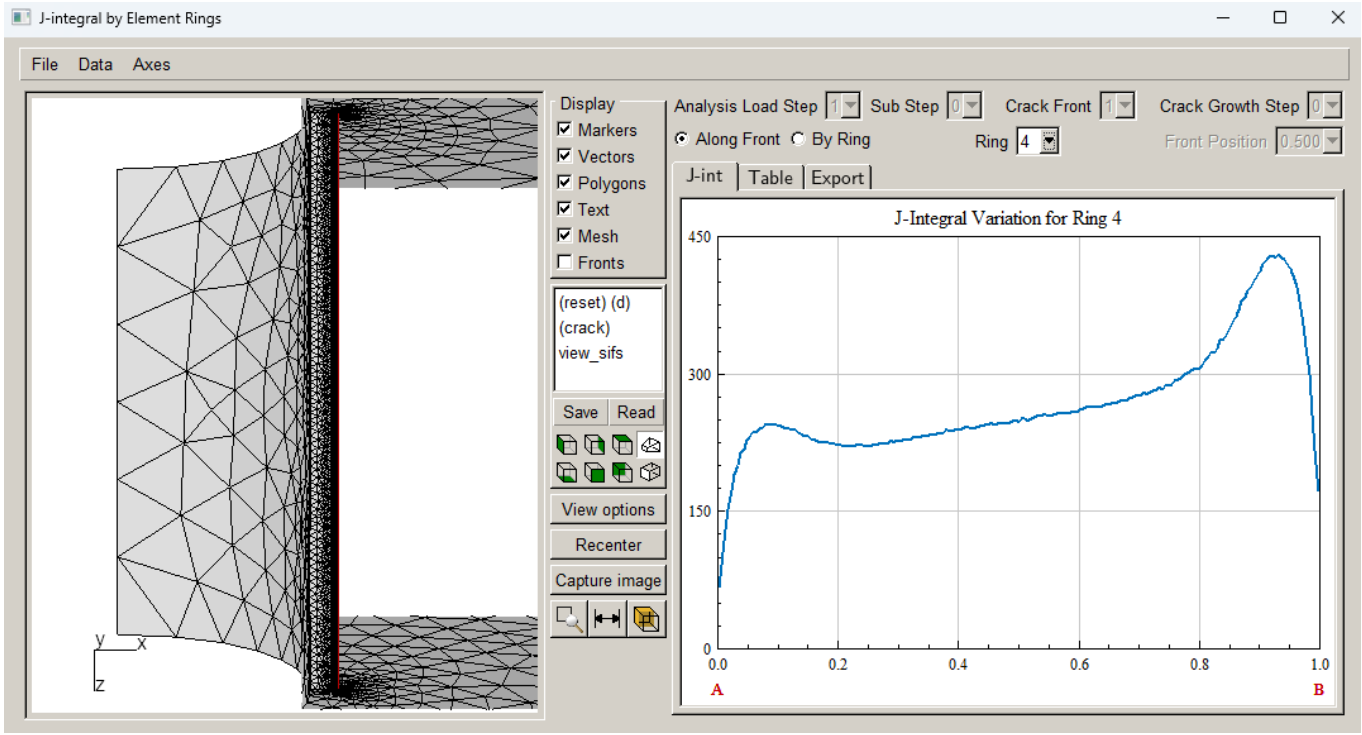

```
*Material, Name=Cube
*ELASTIC,
TYPE=ISOTROPIC
1.0e+04, 3.0e-01
*PLASTIC
6.6e+01, 0.0e+00
6.78e+01, 2.623e-03
6.85e+01, 1.3623e-02
6.99e+01, 9.3623e-02
7.05e+01, 1.33623e-01
7.10e+01, 1.73623e-01
7.11e+01, 1.93623e-01
```

FRANC3D Elasto-Plastic J-Integral

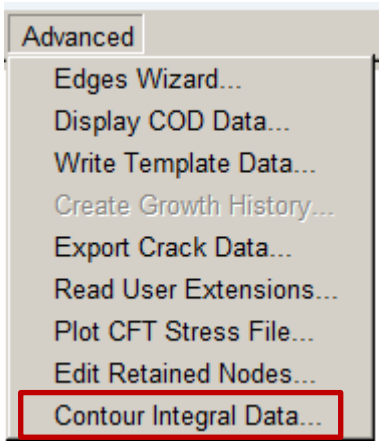




FRANC3D computes J-integral at mid-side nodes

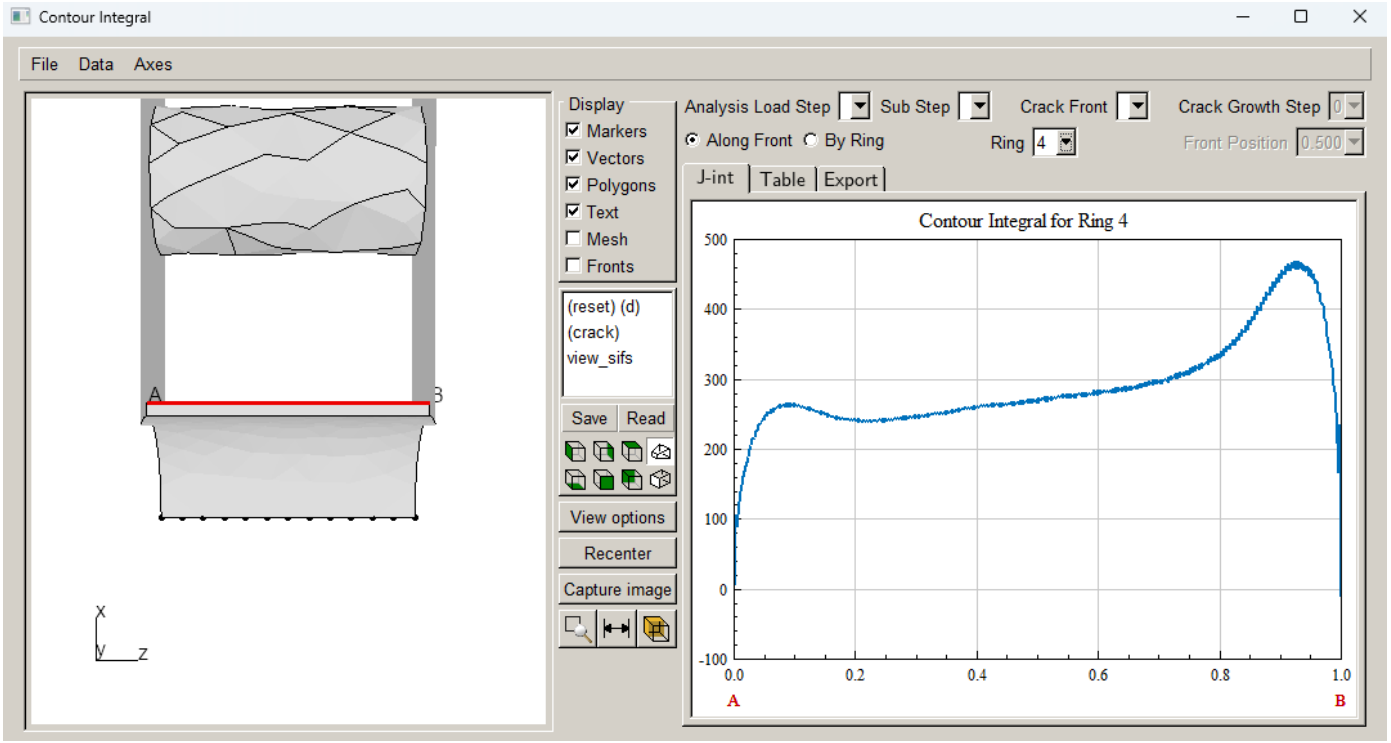


FRANC3D Elasto-Plastic J-Integral



ABAQUS computes J-integral at both mid-side and corner nodes

reads the ABAQUS *.dat* file to get the contour integral data



End Part 11