FRANC3D Training Workshop:
Part IV

February - 2016

Drs. Bruce Carter, Paul “Wash” Wawrzynek, Tony Ingraffea, and Omar Ibrahim
Objectives

• General introduction to FRANC3D:
  - capabilities and limitations
• Present theories and approaches to computational fracture mechanics built into software
• Hands-on sessions give participants time to use software
• Opportunity for participants to ask questions
Workshop Agenda

- Part I: Introduction to Fracture Mechanics Analysis
- Part II: Introduction to FRANC3D
- Part III: FRANC3D FE model import - demo & hands-on
- **Part IV: Crack Insertion Process** – demo & hands-on
- Part V: Crack Growth and Fatigue Life – Theory, Rules & Models
- Part VI: FRANC3D crack growth, SIF history & fatigue life - demo & hands-on
- Part VII: FRANC3D Session Log, Playback, Command Line & Python
- Part VIII: Known issues & what to do if something goes wrong
- Part IX: Capabilities coming soon
Crack Insertion
FRANC3D Crack Types

- Elliptical Crack
- Through-the-thickness
  - One crack front
  - Two crack fronts
- Long-shallow surface crack shape
  (use this instead of long narrow ellipses)
- Elliptical crack shape with two fronts
- User-defined crack
Crack Insertion Wizard (Elliptical Flaw)

- Set crack size/shape parameters.
- Set crack position and orientation.
- Set crack-front template mesh parameters.

Part IV
Crack Insertion Wizard – Library Shapes
User-defined crack allows for an arbitrary (planar) shape; enter (or read from a file) a series of points that define the exterior vertices of a polygon.

Crack front vertices flagged as “1”.

Part IV
Surface Mesh after Crack Insertion

Model surface mesh

Crack surface mesh
Crack-Front Element Types

- quarter-point singular wedge crack-front elements
- tetrahedral elements used for most of volume mesh
- two or more “rings” of brick elements
- pyramids provide compatibility between bricks and tetrahedra

*FRANC3D default is 8 wedge elements around the front and 2 rings of bricks.*
Crack Insertion – into uncracked FE mesh

Start with importing a 3D finite element volume mesh:

Brick, wedge, pyramid, and tetrahedral elements of first and second order can be imported. FRANC3D reads ANSYS (.cdb), ABAQUS (.inp) and NASTRAN (.bdf or .nas) files.
Uncracked FE Mesh – create surface geometry

- model surface geometry approximated from faceted-surface mesh
- remeshed surface mesh lies on curved surface

1) compute weighted average normals at all FE nodes
2) define 1 or 2 triangular Bezier patches for each FE facet
3) identify “topological” edges and group together patches that form logical faces

Bezier patches

Topological edges & logical faces

FE facets on cut surfaces retained for compatibility

Part IV
Crack Insertion – define flaw geometry

- flaws can be zero volume (cracks) or finite volume (voids)
- FRANC3D has tools to define and place a flaw interactively

- flaw surfaces defined as Bezier patches
- crack front edges can be curves (cubic splines) or straight lines

*Initial flaws can be non-planar, but there is no built-in user-interface for such a capability; FE surface meshes can be turned into cracks.*
Crack Insertion – crack-front template mesh

- templates used to place well-shaped elements at/near crack fronts
- templates are combination of brick and quarter-point wedge elements

- where templates intersect model surfaces, element topology and geometry is modified to conform to the surface geometry
Crack Insertion – crack-front template mesh

If the template intersects the model surface at shallow angles, the elements are highly distorted and do not provide accurate SIFs.

Template intersection with model can be turned off, by turning on Simple Template Intersections Only

Template does not extend to model surface
Crack Insertion – intersection and trimming

- surface/surface intersections computed for all model and flaw patches
- patches are trimmed and combined into composite objects

Trimmed patches divided into triangular sub-patches to keep model “water-tight”.

Note that these images show geometry, not a surface mesh.
Crack Insertion – surface remeshing

- triangular surface meshes generated for all “logical” model surfaces
- meshes conform to mesh on retained cut surfaces

Template mesh extracted as it is already meshed

retained cut-surface meshes

Part IV
Crack Insertion – pyramids and tetrahedra

- pyramid elements generated for compatibility:
  - between quadrilateral facets on template or retained cut-surface facets and triangular faces of tetrahedra in the volume mesh

- advancing front meshing algorithm* used to generate tetrahedral volume mesh
- algorithm respects geometrically coincident but distinct nodes on opposite sides of crack faces

Crack Insertion – meshing options

- volume meshing can be done using:
  - FRANC3D
  - ANSYS
  - ABAQUS CAE

- FRANC3D generated volume elements smoothed to improve element quality; ANSYS and ABAQUS generated meshes imported as created

- Do Local Surface Refinement - refines any surface mesh near the crack

- Do Coarsen Crack Mouth Mesh - produces a less refined mesh on the crack surface

Part IV
Crack Insertion – sub-model limitation

Difficult to mesh thin volume that has large quadrilateral patches retained as there is no room for well-shaped pyramid and tetrahedral elements.
Demo: Crack Insertion
FRANC3D Tutorial 1 – Crack Insertion

Step 1: Define crack geometry, location and orientation
Step 2: Remesh
Step 3: Run static analysis

Select “Save to File and add flaw” to save a .crk file for re-use.
FRANC3D Tutorial 1 – Crack Insertion
Remeshing happens automatically after selecting **Finish** in the crack insertion wizard (or **Accept** if the .crk file is saved).
Demo: Static Crack Analysis
Do not overwrite initial uncracked FE files.
ANSYS executable and license should be pre-set in the FRANC3D Edit - Preferences.

Write files but Do Not run if you need to run analysis on a different computer.
Demo: Compute SIFs
Step 1: Re-Open FRANC3D restart 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 the FRANC3D restart file; it 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 (/.pch file for NASTRAN).
Step 2: Select Compute SIFs

- From FRANC3D menu, select Cracks - Compute SIFs.
- \textit{M-Integral} is the default
- \textit{Thermal} and crack face traction terms will be checked if needed
- \textbf{Finish} to display SIFs Plot dialog

Plot SIF modes.
Export SIF data.
Switch between load steps, crack fronts, and steps of growth.
Hands On: Crack Insertion, Static Analysis and SIF Computation