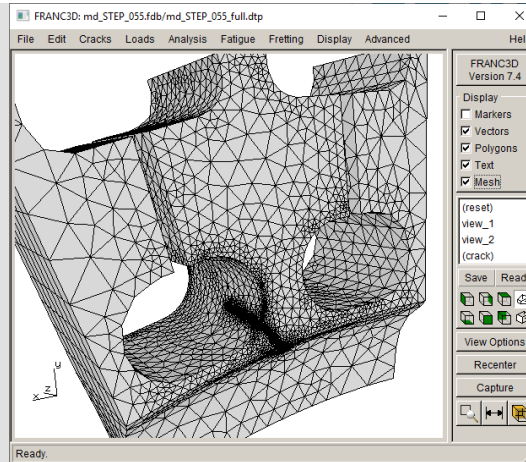


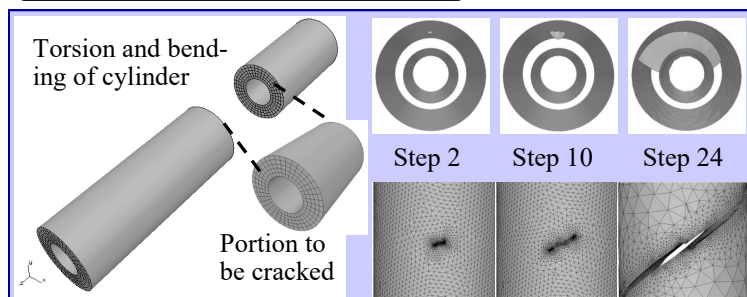
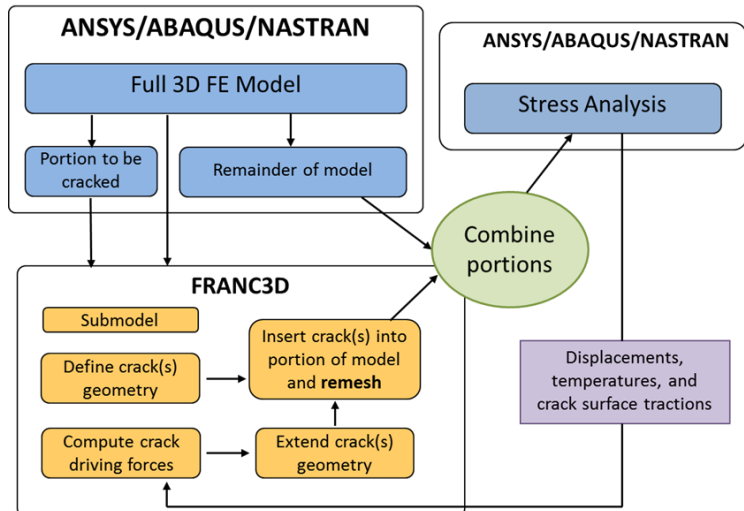
FRANC3D Version 9

FRANC3D:

- extracts portion of a 3D FE model,
- inserts one or more cracks,
- re-meshes around the cracks,
- runs a FE analysis program,
- computes fracture parameters,
- propagates the cracks,
- displays the SIF history, and
- computes fatigue cycles.



Version 9 supports the following finite element analysis programs: ANSYS¹, ABAQUS² and NASTRAN³. The fatigue life module supports combined low cycle (LC) and high cycle fatigue (HCF) loading. The crack insertion and meshing library supports symmetry surface cracks, embedded bi-material cracks, and cracks that cross bi-material interfaces. The current Python interface is based on Python 3.12, but has been tested with older versions of Python 3.



¹ ANSYS is a registered trademark of Ansys, Inc

² ABAQUS is a registered trademark of Dassault Systèmes Simulia Corp

³ NASTRAN is a registered trademark of NASA

Fracture Analysis Consultants, Inc

Fracture simulation and software development

121 Eastern Heights Dr
Ithaca, New York 14850
support@fracanalysis.com

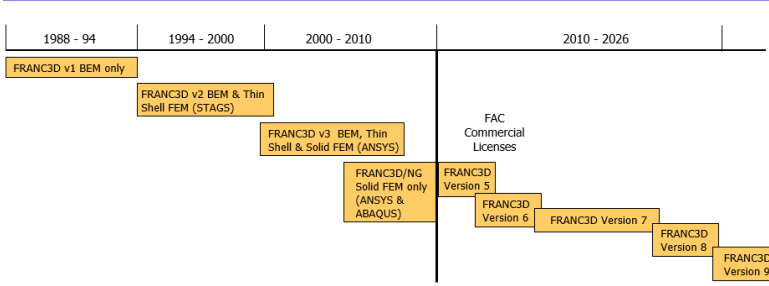
www.fracanalysis.com



Features include:

- A simple graphical user interface
- Importing and exporting ANSYS¹, ABAQUS², NASTRAN³ ASCII file formats
- Wizards and dialogs to aid crack insertion, crack growth, and analysis
- Finite-volume voids and zero-volume cracks
- Crack front template meshes include singular wedge elements or collapsed brick elements for elastoplastic materials
- Advancing-front tetrahedral meshing with pyramid transition elements, with optional ANSYS¹ or ABAQUS² volume meshing
- M-integral stress intensity factor (SIF) calculation supports anisotropic materials and accounts for temperature changes and crack face tractions or contact pressures
- User-controlled crack growth rules for kink angle and increment
- Fatigue models for relative crack extension
- Mapping and/or transferring initial and boundary conditions
- Multiple cracks, crack fronts, load steps, and sub-steps
- Command (batch) and Python interfaces
- 64-bit executables for Microsoft Windows™ and various Linux distributions

FRANC3D Development History

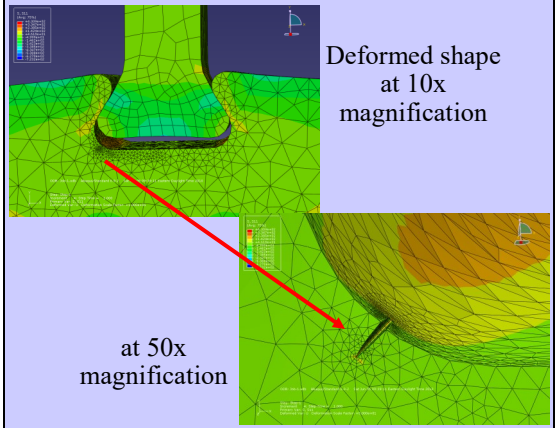
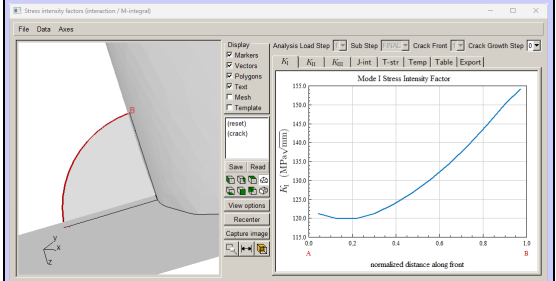
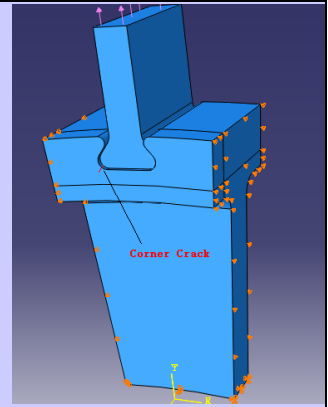


Wizard panels aid crack insertion by guiding the user through each step. A library of parameterized crack shapes is supplemented by user-defined crack (or void) options.

The wizard panels include:

- Flaw Type:** Crack (zero tip-radius, K's and J's), Notch (finite tip radius, J's only), Void (cavity).
- Crack Type:** A grid of 12 parameterized crack shapes with diagrams and meshing options.
- Flaw insertion:**
 - Ellipse Crack Parameters:** Inputs for 'a' (0.05) and 'b' (0.01).
 - Advanced Geometry:** A diagram showing the crack geometry.
 - Flaw rotations:** Controls for 1st, 2nd, and 3rd rotations (Axis and Angle).

Blade & disk assembly in ABAQUS with contact between the two parts



Rotating minidisk in ANSYS

The ANSYS simulation shows:

- Local portion will be cracked:** A callout box pointing to a specific region of the minidisk.
- Retained mesh faces:** A diagram showing the mesh structure after crack growth.
- Deformed shape at 40 steps of crack growth:** A 3D model showing the crack path and mesh deformation.
- SIF history along path through fronts:** A plot showing the variation of stress intensity factor along the crack path.

Contact Information

For technical information:
Fracture Analysis Consultants
 121 Eastern Heights Drive
 Ithaca, NY 14850
 support@fracanalysis.com

For sales information:
Process Optimization Corp
 42726 Settlers Ridge
 Murrieta, California 92562
 Phone: 951-288-8902
 info@processopt.com