FRANC3D Version 7

Version 7.4 is the newest version — released July 2019.

FRANC3D:
- extracts a portion of a 3D FE model,
- inserts one or more cracks,
- re-meshes cracked portion,
- executes FE analysis program,
- computes fracture parameters,
- extends cracks,
- displays SIF history, and
- computes fatigue cycles.

Version 7.4 has a number of enhancements along with improved support for the commercial finite element (FE) analysis programs: ANSYS\(^1\), ABAQUS\(^2\) and NASTRAN\(^3\).

There are changes to the crack face tractions, along with preliminary capabilities for computing elasto-plastic J-integral from ABAQUS results, and preliminary capabilities for computing crack growth under combined static and dynamic loading in ANSYS.

Features include:
- Simple graphical user interface
- Import and export of ANSYS\(^1\), ABAQUS\(^2\), NASTRAN\(^3\) 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 elasto-plastic materials
- Advancing-front tetrahedral meshing with pyramid transition elements, and the ability to use ANSYS\(^1\) or ABAQUS\(^2\) volume meshing
- M-integral stress intensity factor (SIF) calculation allows for 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 to define relative crack extension
- Mapping and/or transfer of initial conditions and boundary conditions
- Multiple cracks, multiple crack fronts, and multiple load steps and substeps
- Command (batch) and Python interfaces
- Executables (64 bit) for Microsoft Windows\textsuperscript{TM} and various Linux distributions

\(^1\) ANSYS is a registered trademark of Ansys, Inc
\(^2\) ABAQUS is a registered trademark of Dassault Systemes Simulia Corp,
\(^3\) NASTRAN is a registered trademark of NASA
Wizard panels aid crack insertion — leading the user through the steps. A library of parameterized crack shapes is supplemented with user-defined crack options.

Rotating minidisk in ANSYS

Deformed shape at 40 steps of crack growth

SIF history along path through fronts

Local portion will be cracked

Retained mesh faces

Deformed shape at 10x magnification

at 50x magnification

Contact Information

For technical information:
Fracture Analysis Consultants
121 Eastern Heights Drive
Ithaca, NY 14850
Phone: 607-257-4970
Email: support@fracanalysis.com

For sales information:
Process Optimization Corp
42726 Settlers Ridge
Murrieta, California 92562
Phone: 951-461-0368
Email: info@processopt.com