

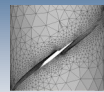
FRANC3D Version 6

Fracture
Analysis
Consultants,
Inc

Fracture simulation and
software development

121 Eastern Heights Dr
Ithaca, New York 14850
Phone: 607-257-4970

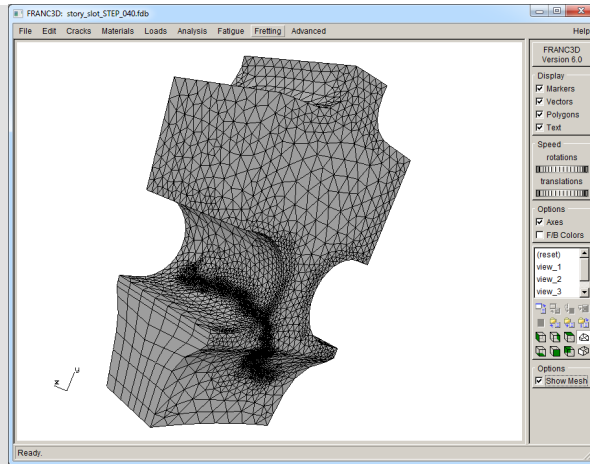
www.fracanalysis.com



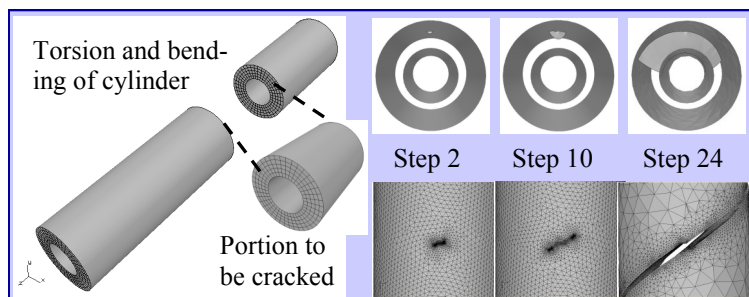
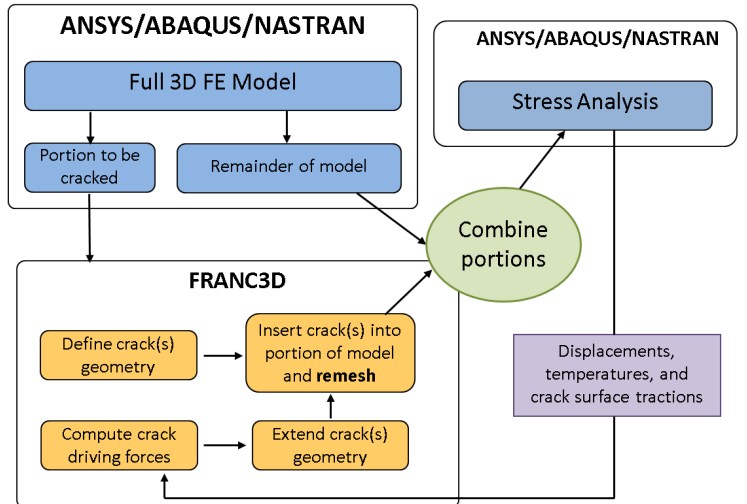
FAC

FRANC3D Version 6 is the newest 3D fracture analysis software from FAC.

FRANC3D development started at Cornell University in the late 1980's, evolving into a program that has been used worldwide in academia and industry for analyzing crack growth in complex 3D structures.



Version 6 has a number of significant enhancements over Version 5 and has improved support for the commercial finite element analysis programs: ANSYS¹, ABAQUS² and NASTRAN³. Typically, a user extracts a small portion of a complex 3D finite element model. FRANC3D reads this small portion, inserts one or more cracks, re-meshes, executes the analysis program, computes fracture parameters, extends the cracks, and computes a stress intensity factor history and fatigue life.



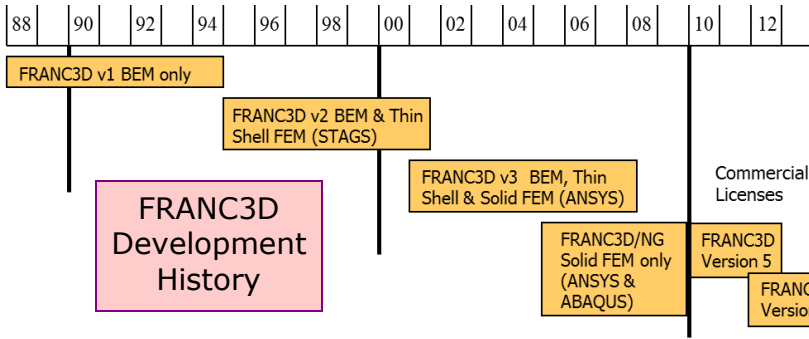
Features include:

- Simple graphical user interface with a window to display the 3D model
- Import and export of finite element file formats, including ANSYS¹, ABAQUS², NASTRAN³
- Wizards and dialogs to aid crack insertion, crack growth, and analysis
- Finite volume voids and zero volume cracks
- Crack front templates include singular wedge or collapsed brick elements for elasto-plastic material behavior
- Advancing-front tetrahedral meshing with pyramid transition elements, plus the ability to use ANSYS¹ or ABAQUS² 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 available to define relative crack extension
- Mapping or transfer of initial conditions and boundary conditions
- Multiple cracks with multiple crack fronts and multiple load steps/cases
- Executables (32 and 64 bit) for Microsoft Windows™ (XP thru 7) and Linux

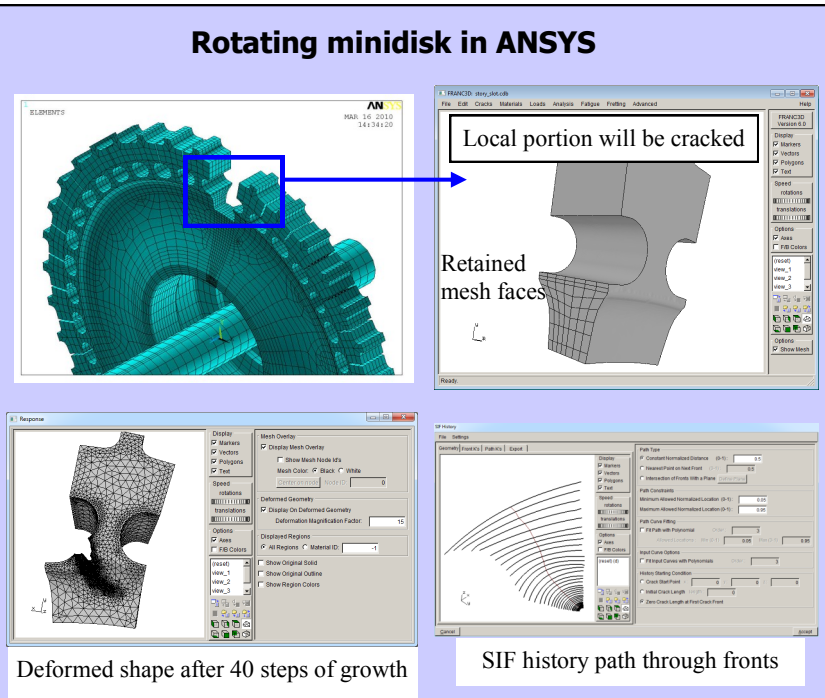
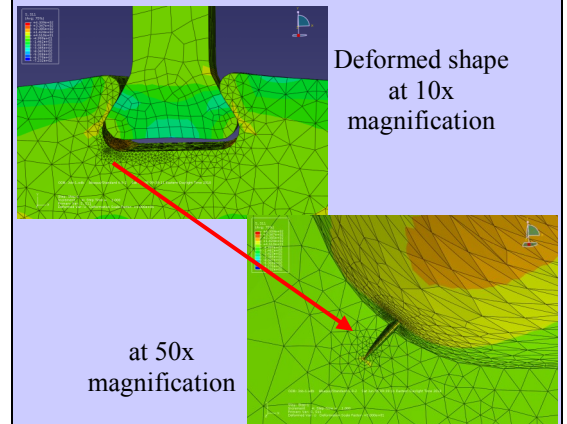
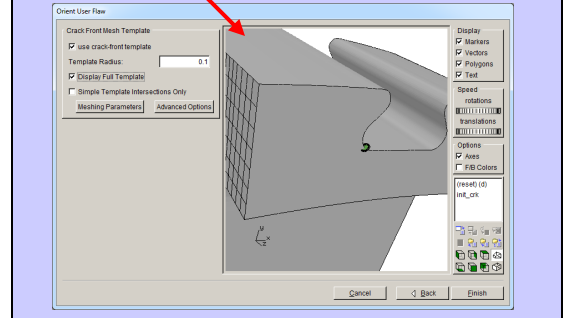
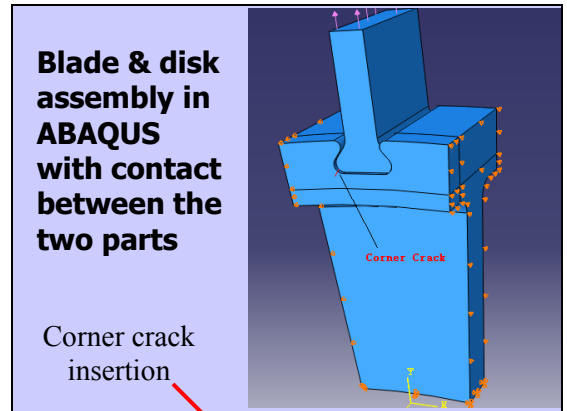
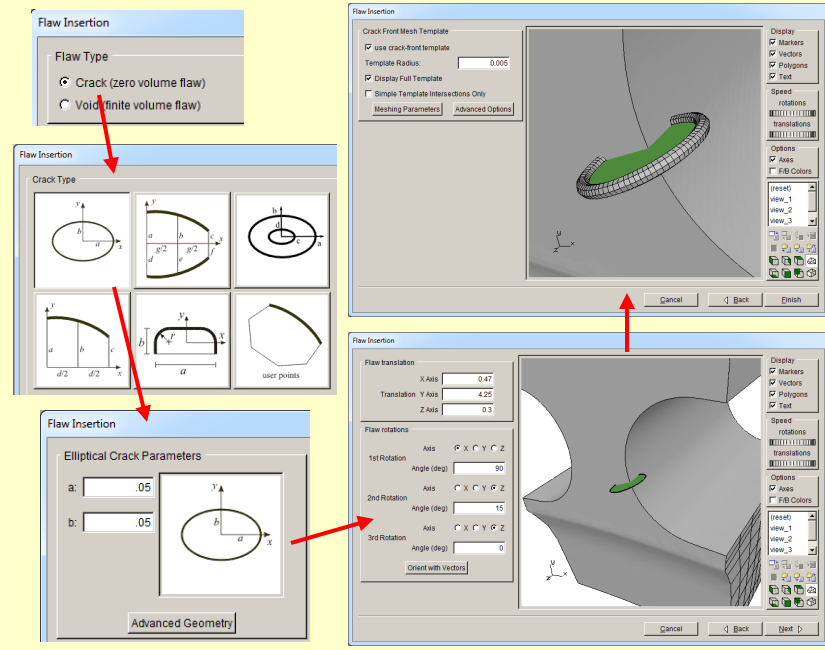
¹ ANSYS is a registered trademark of Ansys, Inc

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 a user-defined crack option.



Contact Information

For general and technical information:

Fracture Analysis Consultants
 121 Eastern Heights Dr
 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