Overview Slides

October, 2003

Cornell Fracture Group
Realistic 3D crack growth simulations require:
- actual geometric models, including crack(s)
- ability to discretize and analyze the model
- ability to post-process analysis results
- and the ability to incorporate the results into design and/or assessment

A framework for simulation requires:
- an intuitive, relatively simple, user interface
- sophisticated capabilities, while hiding complex details
- automation, speed, efficiency
- testing, benchmarking, and support
Example: Life Prediction in Transmission Gears

U.S. Army OH-58 Kiowa

Allison 250-C30R Engine

Fatigue Cracks in Spiral Bevel Power Transmission Gear

U.S. Army Vehicle Technology Center
NASA/Lewis Research Center
Constrained Shape: Known Solution Method

Characteristic—Library:
look-up stress intensity factor solutions for finite number of simple shapes and boundary conditions

Benefits:
no model to build,
no meshing,
fast analysis,
simple interface.

Drawbacks:
models do not match real geometry and boundary conditions,
no multi-scale,
no multi-physics.
A fast, accurate solution to the wrong problem.

Examples: NASGRO, AFGROW
Unconstrained Shape: Computed Solution

FRANC3D:
actual part
with actual
boundary
conditions

Benefits:
realistic
solution,
multi-scale,
multi-physics
capable.

Drawbacks:
time
consuming.
FRANC3D / OSM

X Windows-based, menu driven, C code
FRANC3D / OSM
what they are and are not!

FRacture ANalysis Code - 3D
- hybrid software that combines solid modeling, mesh generation, and fracture mechanics for nucleating and propagating cracks in the model geometry.

FRANC3D is not a boundary nor a finite element analysis code, although it is capable of writing input files for a variety of in-house and commercial BEM and FEM codes.

Object Solid Modeler
- generates boundary representation (B-rep) solid model for FRANC3D from user-defined geometry points, curves and surfaces, or by converting FEM data.

OSM is not a general purpose solid modeler or CAD program.
Supported FEM / BEM Analysis Codes

**BES** - Cornell University Boundary Element System

**CPTC** - Cornell University parallel finite element code (ongoing research and development)
Overview of Simulation Process

• a B-rep solid model is the primary input to FRANC3D; this normally is generated by OSM.

• boundary conditions can be specified within FRANC3D; these include imposed loads, imposed displacements, temperatures, and crack face pressure interpolated from finite element stresses.

• material properties can be specified within FRANC3D; available materials include elastic, orthotropic, and elasto-plastic.

• the model is discretized or meshed within FRANC3D; this could include surface (for BEM) and volume meshing (ANSYS).

• analysis input files are generated by FRANC3D.

• an external program provides displacements and stresses; these should be imported into FRANC3D.
Overview of Simulation Process (continued)

- one or more flaws can be initiated in the model; the location, orientation, size and shape could be based on the stress analysis, or based on field evidence, or completely arbitrary.

- fracture parameters, such as stress-intensity factors, are computed based on the displacements from the external stress analysis (e.g. ANSYS).

- crack propagation is based on the computed fracture parameters and a “rule” to predict the new crack front – orientation and advance. The model geometry is modified automatically to grow the crack.

- a history of the stress intensity factors as a function of the flaw size can be used in conjunction with the crack growth properties of the material and load history to make a prediction of the fatigue life.
Simple Cube Example – Hands-on-training

- demonstrate ANSYS FEM to OSM conversion
- demonstrate MRP boundary conditions – transfer ANSYS stresses to crack face tractions for BEM analysis

- simple cube with 3x3x3 grid of SOLID95 elements
- kinematic constraint on base nodes
- surface traction on upper surface
- linear elastic ANSYS analysis
Simple Cube Example

Steps:

• read the ANSYS .cdb and .igs files into OSM and extract the geometry features

• discuss MRP’s generated by OSM from ANSYS .cdb file and from ANSYS nodal results saved as .dsp and .str files

• read model into FRANC3D and apply MRP boundary conditions

• add a crack and analyze using BES

• compare stress intensity factors based on both BES and ANSYS results
OSM can convert ANSYS iges and cdb files.

Iges file contains only geometry and if a solid model does not exist there is no iges information.

cdb file contains finite element mesh information along with boundary conditions and materials.
Simple Cube Example

OSM reads the .str file and saves a _res_str.mrp file that contains the mesh information from the .cdb file as well as the stress results.

We can use the ANSYS mesh facets to define FRANC3D geometry, but it is not recommended for big models.

The MRP is imported into FRANC3D and used in the Face Boundary Condition dialog box – this is then attached to the crack surface.
The BEM analysis provides the stresses and displacements for the cracked model. Stress intensity factors can be computed.

A part through crack is inserted into the FRANC3D model and the face boundary condition defined previously is attached to this surface.
• incremental process involving ‘n’ steps of crack growth in a “true” geometric model of the structure.
The essence of the problem is predicting and tracking the changing geometry as the crack grows. A topology-based data structure underlies the solid model geometry, which allows for efficient interaction and accurate representation of evolving cracks.
Geometric Model and Topology

radial edge topological database

Topological edges provide ordering based on the edge uses.
Topological entities have orientations and uses and the data is stored accordingly, such that traversals and queries are efficient.
Geometric Model and Topology

Allows for the definition of complex topological models, including the definition of complex cracks.
• main and mate crack topological surfaces share the same geometry
• the normal to the main side surface points into the crack towards the mate
• the crack tip and branch vertices mark the ends of crack fronts
• crack fronts can grow independently
Topological Operators for Cracks

Tear operators are used in sequence to create a crack.

Glue operators can be used to delete a crack.
A crack can be nucleated manually by adding the edges and faces and ‘tearing’ the entities using the **Arbitrary 3D Crack** menu option.
Crack Nucleation

Completely arbitrary initial flaws (arbitrary size, orientation, location, and shape, including non-planar flaws) can be specified in the geometrical model.

• start with a cube
• define edges on the surfaces and in the interior
• define faces from the edges
• pick the faces and create a crack
Crack Propagation

The 3D crack front is divided into a number of segments by defining geometric points along the edge.

At each point, the plane orthogonal to the local tangent on the crack front is defined and 2D theories for crack growth are invoked.

Fracture mechanics theories and crack growth models are encapsulated in easily modified modules.
Stress and displacement near the crack tip as a function of $r$ and $\theta$:

\[
\begin{align*}
\sigma_x &= \frac{K_I}{\sqrt{2\pi r}} \cos \theta \left(1 - \sin \frac{\theta}{2} \sin \frac{3\theta}{2}\right) \\
\sigma_y &= \frac{K_I}{\sqrt{2\pi r}} \cos \theta \left(1 + \sin \frac{\theta}{2} \sin \frac{3\theta}{2}\right) \\
\tau_{xy} &= \frac{K_I}{\sqrt{2\pi r}} \sin \frac{\theta}{2} \cos \theta \cos \frac{3\theta}{2} \\
u &= 2(1 + \nu) \frac{K_I}{E} \sqrt{\frac{r}{2\pi}} \cos \theta \left[1 - 2\nu + \sin^2 \frac{\theta}{2}\right] \\
v &= 2(1 + \nu) \frac{K_I}{E} \sqrt{\frac{r}{2\pi}} \sin \theta \left[2 - 2\nu - \cos^2 \frac{\theta}{2}\right]
\end{align*}
\]
Crack Propagation: Displacement Correlation

Stress intensity factors can be extracted from boundary or finite element analyses using displacement correlation:

The crack-tip displacement fields are given by:

\[
\begin{bmatrix}
u \\
v
\end{bmatrix} = 2(1 + \nu) \frac{K_I}{E} \sqrt{\frac{r}{2\pi}} \begin{bmatrix}
\cos \frac{\theta}{2} & 1 - 2\nu + \sin^2 \frac{\theta}{2} \\
\sin \frac{\theta}{2} & 2 - 2\nu + \cos^2 \frac{\theta}{2}
\end{bmatrix} + 2(1 + \nu) \frac{K_I}{E} \sqrt{\frac{r}{2\pi}} \begin{bmatrix}
\sin \frac{\theta}{2} & 2 - 2\nu + \cos^2 \frac{\theta}{2} \\
\cos \frac{\theta}{2} & 1 + 2\nu + \sin^2 \frac{\theta}{2}
\end{bmatrix}
\]

At 180 degrees, along the crack flank:

\[
\begin{bmatrix}
u \\
v
\end{bmatrix} = \frac{(1+\nu)(1-\nu)}{E} \sqrt{\frac{r}{2\pi}} \begin{bmatrix}
K_{II} \\
K_I
\end{bmatrix}
\]

Using displacements from a finite or boundary element analysis, we can solve for the K’s:

\[
\begin{bmatrix}
K_{II} \\
K_I
\end{bmatrix} = \frac{E}{(1+\nu)(1-\nu)} \sqrt{\frac{2\pi}{r}} \begin{bmatrix}
u_{FEM} \\
v_{FEM}
\end{bmatrix}
\]
Crack Propagation: Direction and Extension

Maximum Hoop Stress Criterion: \( \sigma_{\theta\theta} \) is maximum when \( \sigma_{r\theta} = 0 \)

\[
\sigma_{r\theta} = \frac{K_I}{\sqrt{2\pi r}} \sin \frac{\theta}{2} \cos^2 \frac{\theta}{2} + \frac{K_{II}}{\sqrt{2\pi r}} \left( \frac{1}{4} \cos \frac{\theta}{2} + \frac{3}{4} \cos \frac{3\theta}{2} \right)
\]

Solve for \( \theta \):

\[
\tan \frac{\theta_0}{2} = \frac{1}{4} \frac{K_I}{K_{II}} \pm \frac{1}{4} \sqrt{\left( \frac{K_I}{K_{II}} \right)^2 + 8}
\]

Crack Extension Rule:

\[
a_i = a_{max} \left[ \frac{K_I}{K_{I_{max}}} \right]^b
\]

\( a_{max} \) is user defined

\( K_{I_{max}} \) is the maximum computed \( K_I \)
Crack Propagation

At each point on the crack front, a new front point is computed.

The new front points are smoothed by doing least-squares fitting of polynomials.

Connect old and new front edges and create faces.

Curve fitting & smoothing.
Crack Propagation

If the points fall outside of the region, the crack front breaks into segments.

open boundary edge

The collection of edges must be turned into a collection of faces that are then ‘torn’.
Propagation in a more complex geometry, such as through a material interface or around a re-entrant corner requires careful tracking and manipulation of both geometry and topology.
FRANC3D has 5 levels of the model:

- all five levels are automatically updated in response to predicted crack growth

- crack growth occurs at the geometry level first, but all lower levels are constrained by the levels above.

- remeshing is localized to the crack region

- local remeshing near the crack simplifies mapping of information from the previous equilibrium state
Boundary Conditions

Model boundary conditions: rotational and linear acceleration, nodal temperatures

Surface boundary conditions: displacement and pressure, uniform or arbitrary distributions

Edge boundary conditions: displacement and force, uniform or arbitrary distributions

Point boundary conditions: displacement and force

Arbitrary distributions applied using MRPs (Mesh RePresentation)

Coupled Sets: cyclic symmetry

Contact Surfaces: master/slave
MRP boundary conditions

Linear pressure distribution on a cantilever beam

ANSYS model with 20-noded brick elements

MRP: 12 elements, 20 nodes, 20 nodal pressures

FRANC3D model with 10-noded tetrahedral elements

MRP must cover the same geometric location as the model surface where the MRP is to be applied but the mesh does not have to match.
Superposition Based BC’s

linear superposition [see Broek 1986]

The magnitudes of the tractions applied to the crack faces are equal in magnitude and opposite in direction to those holding an imaginary crack closed in the finite element analysis and are determined from the stresses from the finite element analysis.

All results -- stresses, strains, displacements, and stress-intensity factors -- for the model of interest, model (a), are the algebraic sum of the results of models (b) and (c). In the case of stress-intensity factors, \( K_a = K_b + K_c \), but since \( K_b \) is equal to zero, \( K_a = K_c \).
Meshing & Discretization

The arbitrary region mesh can still be modified by moving nodes and by adding or deleting edges.
Meshing & Discretization

- sweep bricks or wedges depending on source face mesh

The mesh around the crack front can be:
- bricks
- wedges (singular)
- tetrahedra
- wedges surrounded by bricks, then pyramids and then tetrahedra

source face mesh

internal penny-shape crack with crack front subvolume (subvolume can be swept)

subdivided edge in sweep direction

swept brick mesh
advancing front, unstructured tetrahedral mesh generation

• generate an octree “background mesh” to estimate element size.

• geometry-based advancing front - elements are generated based on ideal shapes.

• topology-based advancing front – all topologically valid elements are generated.

• backtracking to mesh any non-meshable polyhedra.

• edge and face swapping and laplace smoothing for local shape improvement.

sliver element  delete adjacent element  restart advancing front
Tetrahedral Volume Meshing

some examples

inclined borehole

coarse and refined meshes

torus

course and refined meshes

engine part
OSM Overview

Specifically designed to create B-rep models for FRANC3D:
- closed solid 3D models
- curved shells and 2D plate models

B-rep solid model for FRANC3D requires:
- watertight geometry
- exterior surface normals pointing outwards
- interior surfaces defined as ‘interior’
- plates and curved shell models are not ‘closed’

OSM capabilities:
- creates lines, arcs, spline curves
- creates triangular Bezier or bi-cubic b-spline surfaces
- planar surfaces are supported but cannot be created manually
- extrude, translate, rotate, mirror, cut & paste
- copy objects between multiple windows
- finite element model translation and conversion
- ANSYS generated IGES file conversion
OSM: FEM Conversion

- start with FE data file that contains node and element information
- extract the surface facets with their edges and nodes (vertices)
- eliminate all interior features:
  facets used by two elements are not surface facets and can be removed

identify key nodes and edges based on angle changes
OSM: FEM Conversion

- compose chains of key edges between end key nodes
- form loops from the chains
- form faces from the loops

A limitation that has been partly resolved:
- faces must be 3 or 4-sided
OSM: FEM Conversion

original solid model geometry

original mesh

extracted geometry
Models usually require that the user manually complete the process of forming a closed B-rep solid model. Problems arise if there are contact surfaces or other features where the original mesh facets do not match exactly. Problems also arise for surfaces that consist of more than four ‘geometric’ edges.
Description of Files

OSM files:

* .mod  a formatted text file - OSM restart file containing surface, edge, and vertex geometry information.

* .osm  a formatted binary file based on XDR routines for cross platform portability - OSM restart file containing surface, edge, and vertex geometry information.

FRANC3D files:

* .dat  a formatted text file - FRANC3D input file containing surface patch geometry information.

* .fys  an unformatted binary file - FRANC3D restart file containing all of the FRANC3D model information.

* .afys  a formatted text file - FRANC3D restart file containing all of the FRANC3D model information. Although this file is a formatted text file, the file should not be edited manually.
Description of Files (continued)

FRANC3D files (continued):

*.frt  a formatted text file containing crack front geometric point coordinates.

*.mrp  a formatted text file containing finite element mesh information and boundary condition data. This file is used to specify arbitrary loading conditions. NOTE: this file is generated by OSM and replaces the .conn and .fstr files. It can consist of 2D or 3D finite element information. It is currently generated from ANSYS .cdb files.

*.conn  (obsolete for Version 2.4) a text file containing finite element mesh information in FRANC3D generic format consisting of nodal coordinates and element connectivity. This file can be generated by OSM when translating FEM data from ANSYS .cdb files for example.

*.fstr  (obsolete for Version 2.4) a text file containing finite element nodal stress information. This file along with the .conn file is used by FRANC3D when applying crack face tractions based on FEM results.
Description of Files (continued)

BES files:

* .bes  a formatted text file - BES input file containing nodal coordinates, element connectivity, boundary conditions, and material properties.
coef.dat  an unformatted binary file containing the BES coefficient matrix. This file is created when running the out-of-core version of the BES solver and the file should be deleted after BES is finished.
* .besout  a formatted text file - BES output file containing computed nodal displacements and tractions.
* .con  a formatted text file - BES output file containing computed element stresses.

ANSYS files:

* .cdb  a formatted text file of the model data, including nodes, elements, etc
* .igs  an IGES formatted text file describing the model geometry
* .dsp  listing of nodal displacements
* .str  listing of nodal stress components
* .tmp  listing of nodal temperatures
* .eps  listing of nodal plastic strain
Simple Example To Illustrate Concepts

2219-T851 aluminum beam laboratory test specimen in four-point bending

\[ \Delta p = 11.54 \text{ kips, } R = 0.214 \]

- Crack Tip 1
- Crack Tip 2
- Projection onto Section A - A
- \( a_1 \)
- \( a_2 \)
- \( a_3 \)
- 0.25R" EDM flaw
- Initial notch
- Crack

Dimensions:
- 2.412"
- 3"
- 1.5"
- 12"
- 2"
- 45° angle
Initial Boundary Element Mesh

Geometric Model

Flaw Mouth

Initial Flaw

Full Beam
Predicted and Observed Crack Fronts

Predicted Transition into Corner “a”

Predicted Transition into Corner “b”

Final Fatigue Crack Front

Observed Fatigue Crack Shape
Final Predicted Crack Shape

Deformed solid

Comparison of computed and observed crack trace on the surface
## Computed SIF History and Comparison to Observations

<table>
<thead>
<tr>
<th>Event</th>
<th>Observed (cycles)</th>
<th>FRANC3D (cycles)</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>a-Corner</td>
<td>106,800</td>
<td>140,000</td>
<td>32</td>
</tr>
<tr>
<td>b-Corner</td>
<td>171,000</td>
<td>170,000</td>
<td>-0.5</td>
</tr>
<tr>
<td>Last Front</td>
<td>175,000</td>
<td>190,000</td>
<td>8.5</td>
</tr>
<tr>
<td>a₁ (in.)</td>
<td>1.26</td>
<td>1.42</td>
<td>12.7</td>
</tr>
<tr>
<td>a₂ (in.)</td>
<td>1.38</td>
<td>1.34</td>
<td>-2.8</td>
</tr>
</tbody>
</table>
Summary

• We can translate the ANSYS model from either the mesh information (.cdb file) or from the solid model geometry (.iges file).

• Based on uncracked model stress results, we can predict the location of crack initiation.

• Model boundary conditions are mapped from the uncracked model to the cracked model.

• The orientation of the maximum principal stress provides the plane for the initial crack.

• The orientation can be input into the FRANC3D library flaw dialog box or can be “user defined”.

• Subsequent crack trajectory is predicted within FRANC3D based on the maximum hoop stress criterion.

• The resulting output can be a history of SIF versus crack size, which can then be input into a life analysis tool.