FRANC3D / OSM

Overview Slides

October, 2003

Cornell Fracture Group

INTRODUCTION

http://ppe.navail.navy.mil

Realistic 3D crack growth simulations require:

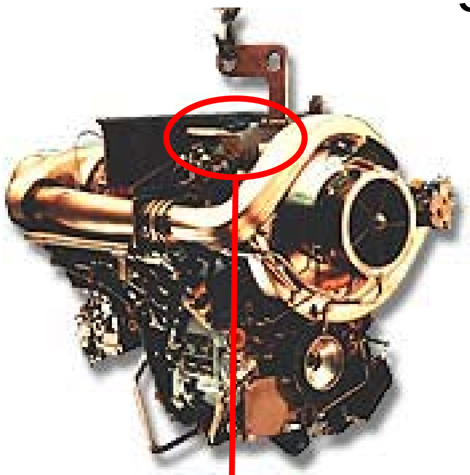
- actual geometric models, including crack(s)
- ability to discretize and analyze the model
- ability to post-process analysis results
- and the ability to incorporate the results into design and/or assessment

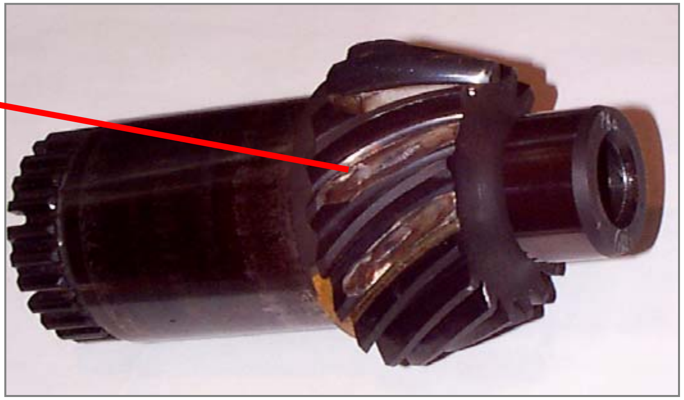
A framework for simulation requires:

- 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

Allison 250-C30R Engine





Fatigue Cracks in Spiral Bevel Power Transmission Gear

U.S. Army Vehicle Technology Center

NASA/Lewis Research Center

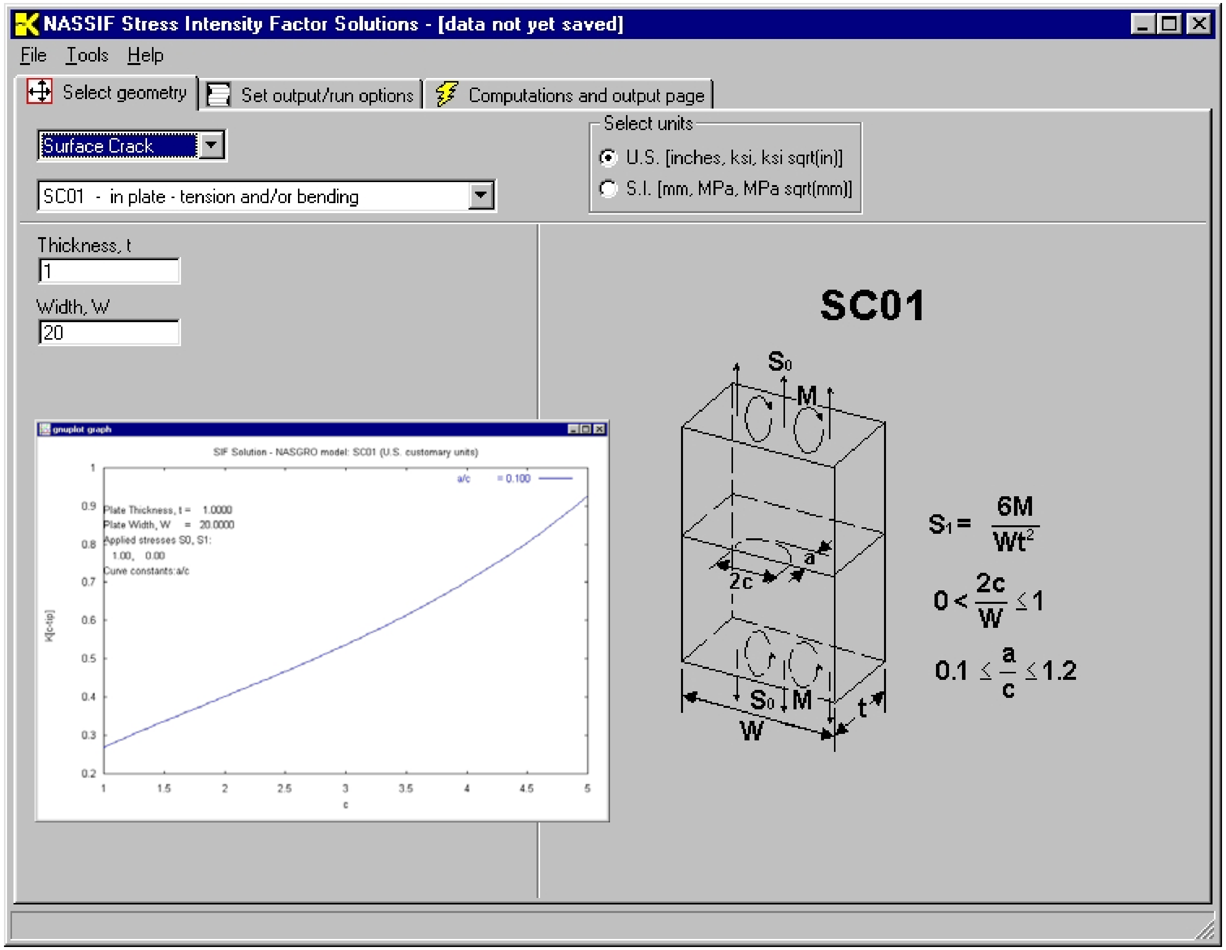
Constrained Shape: Known Solution Method

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

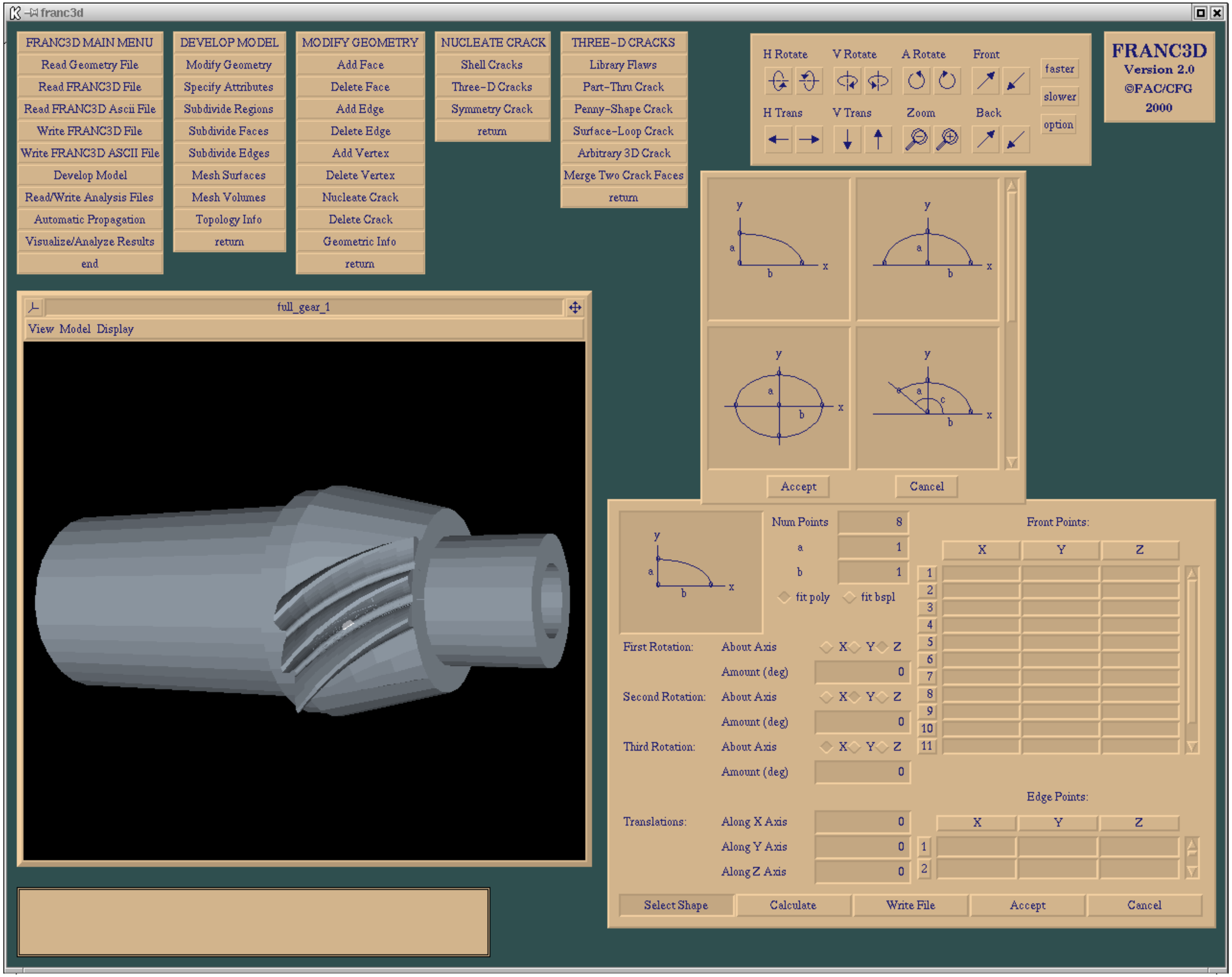


Unconstrained Shape: Computed Solution

FRANC3D:
actual part with actual boundary conditions

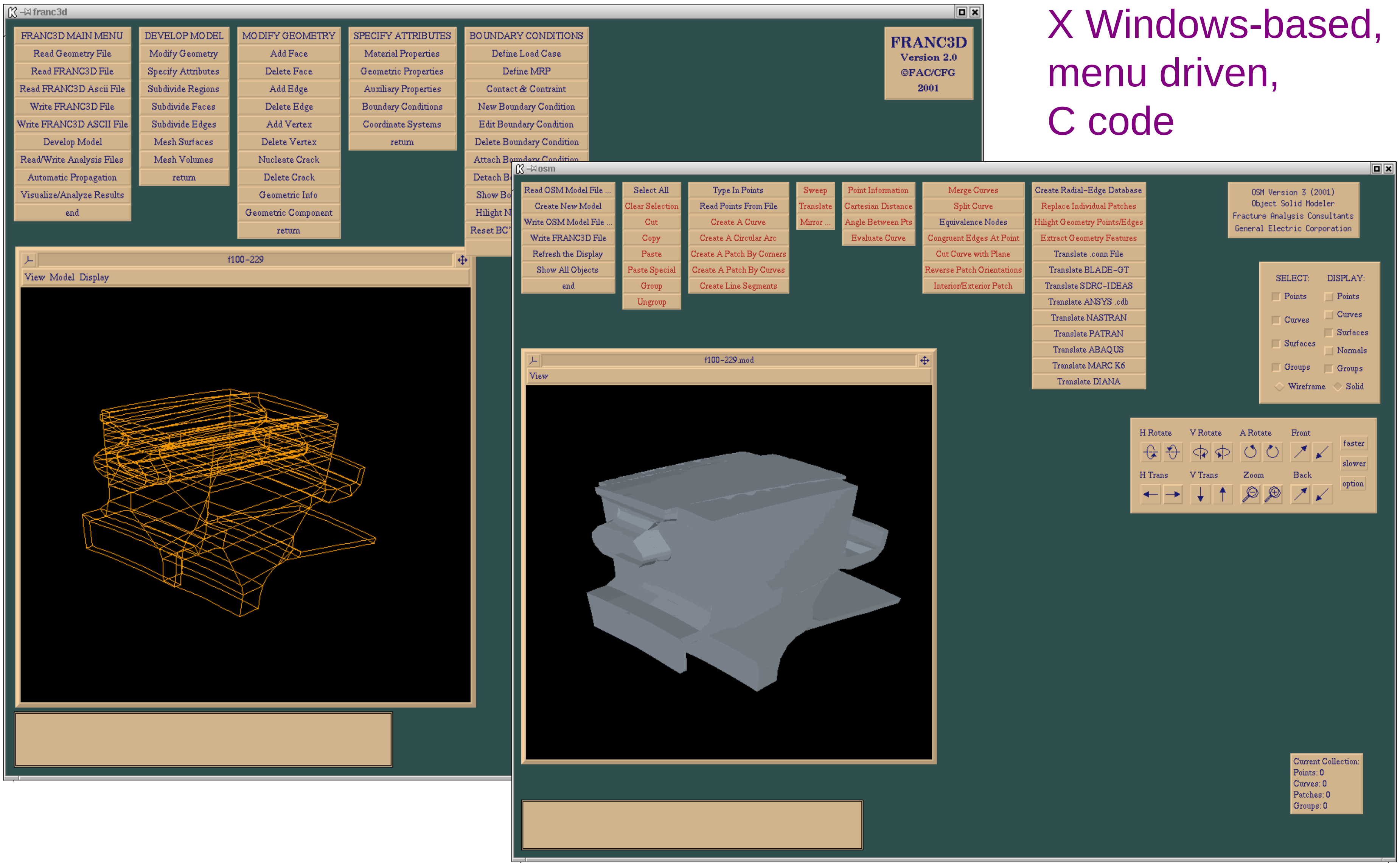
Benefits:
realistic solution, multi-scale, multi-physics capable.

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)

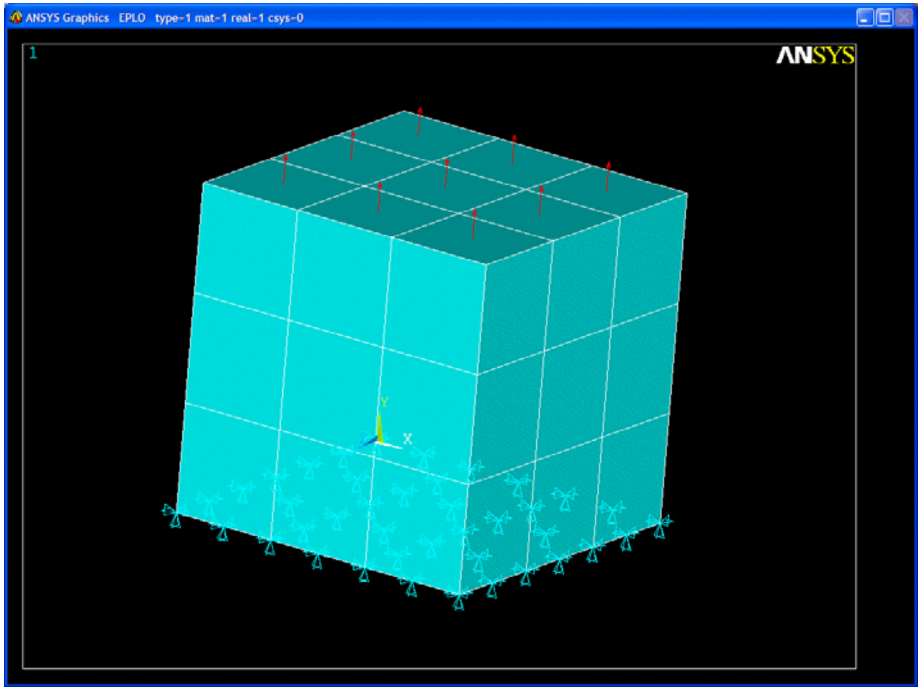
- one or more flaws can be initiated in the model; the location, orientation, size and shape could be based on the stress analysis, or based on field evidence, or completely arbitrary

- 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.

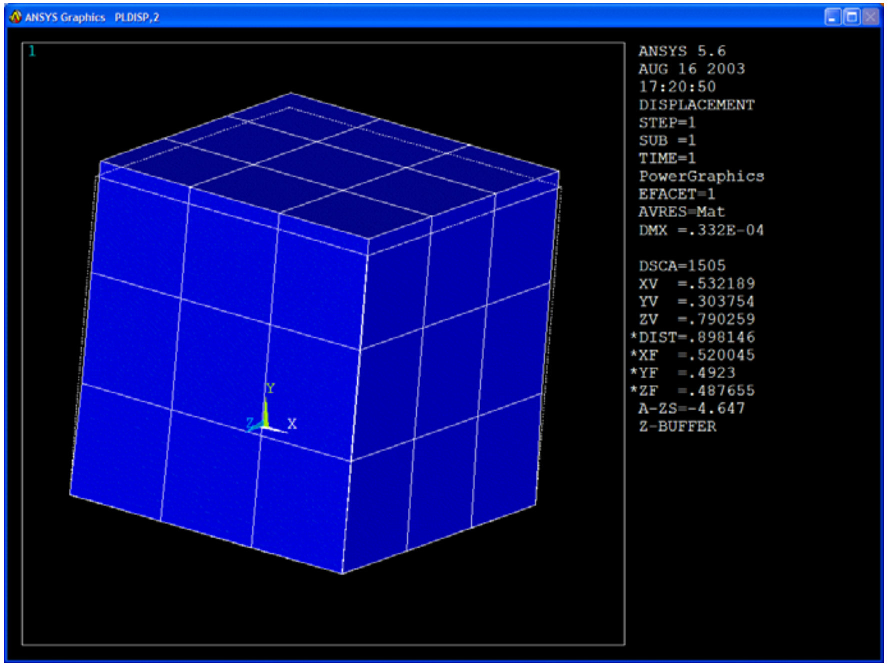
- a history of the stress intensity factors as a function of the flaw size can be used in conjunction with the crack growth properties of the material and load history to make a prediction of the fatigue life.

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

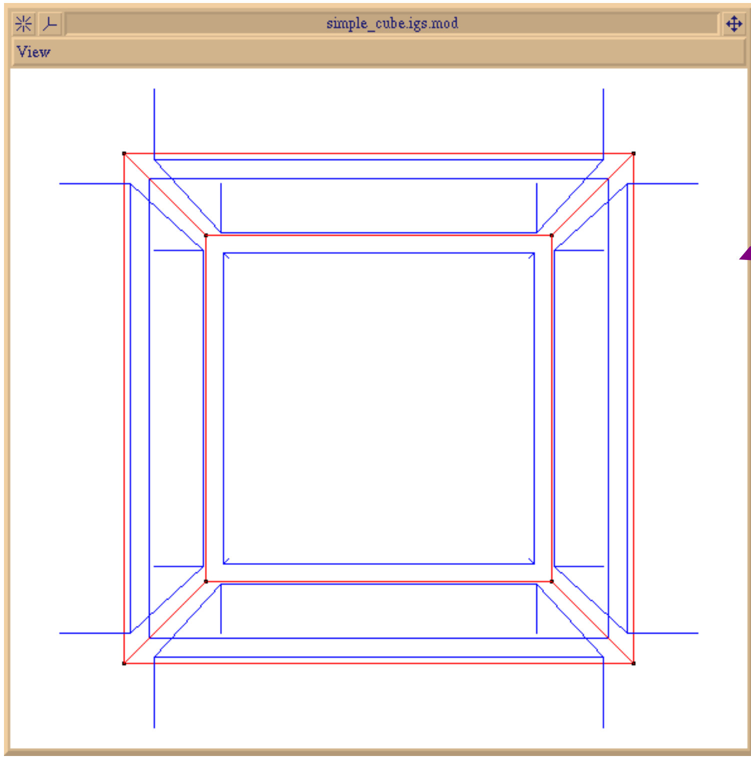
- discuss MRP's generated by OSM from ANSYS .cdb file and from ANSYS nodal results saved as .dsp and .str files

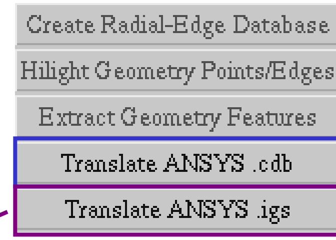
- read model into FRANC3D and apply MRP boundary conditions

- add a crack and analyze using BES

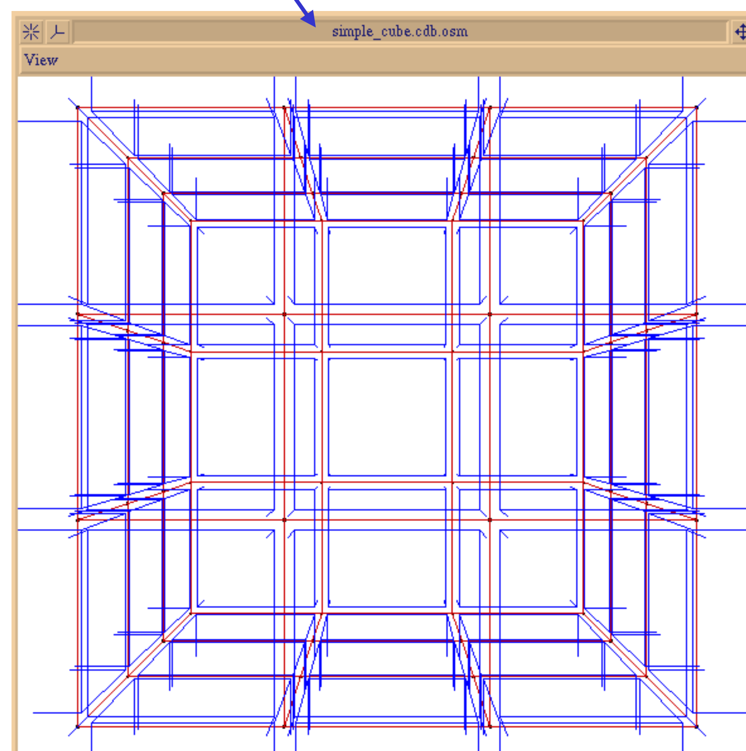
- compare stress intensity factors based on both BES and ANSYS results

Simple Cube Example





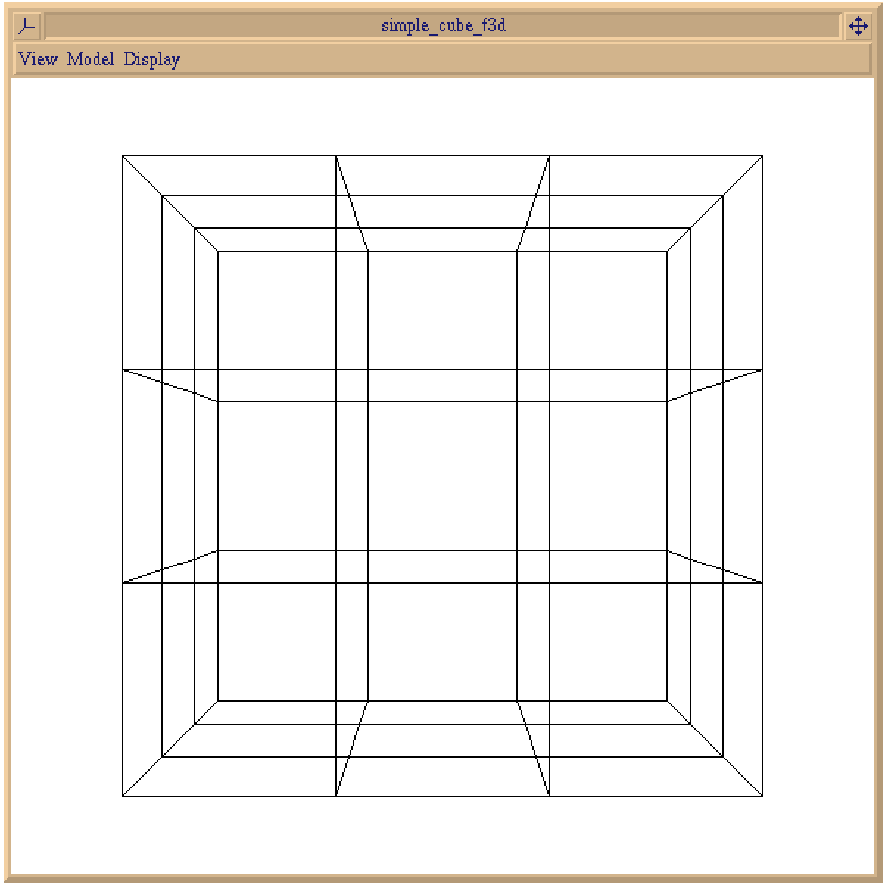
OSM can convert ANSYS iges and cdb files



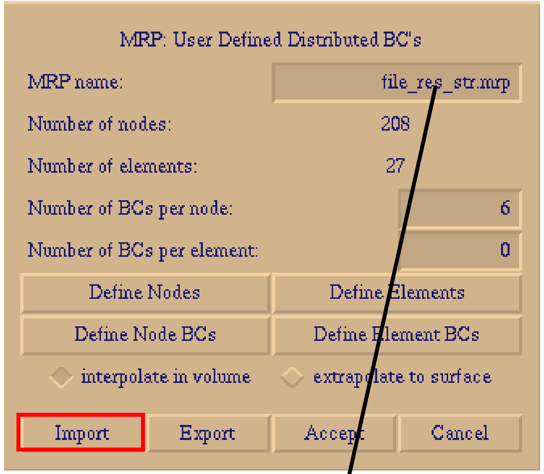
iges file contains only geometry and if a solid model does not exist there is no iges information

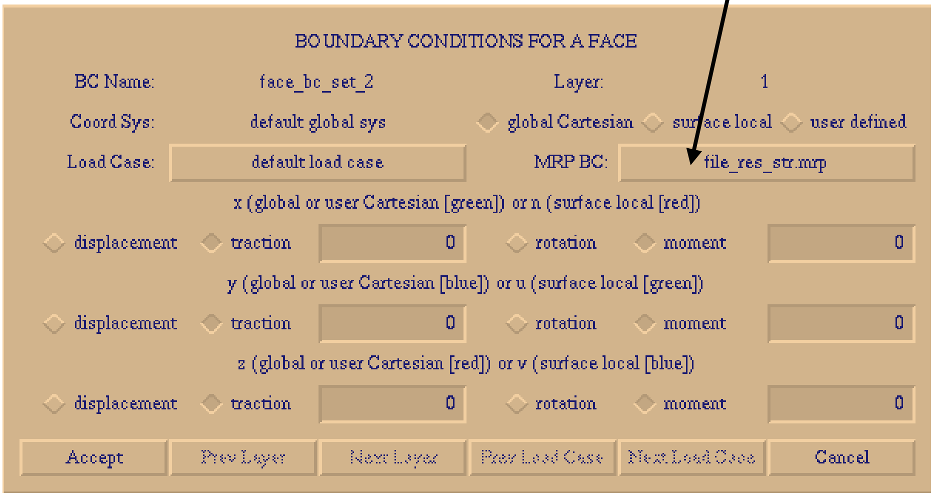
cdb file contains finite element mesh information along with boundary conditions and materials

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

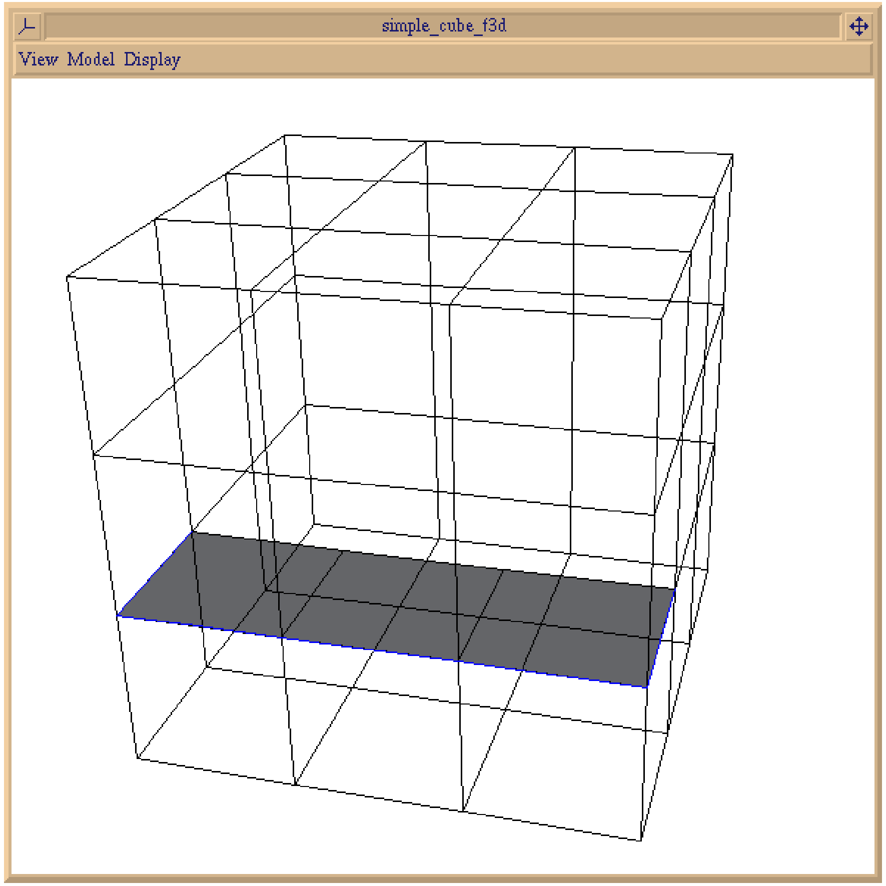




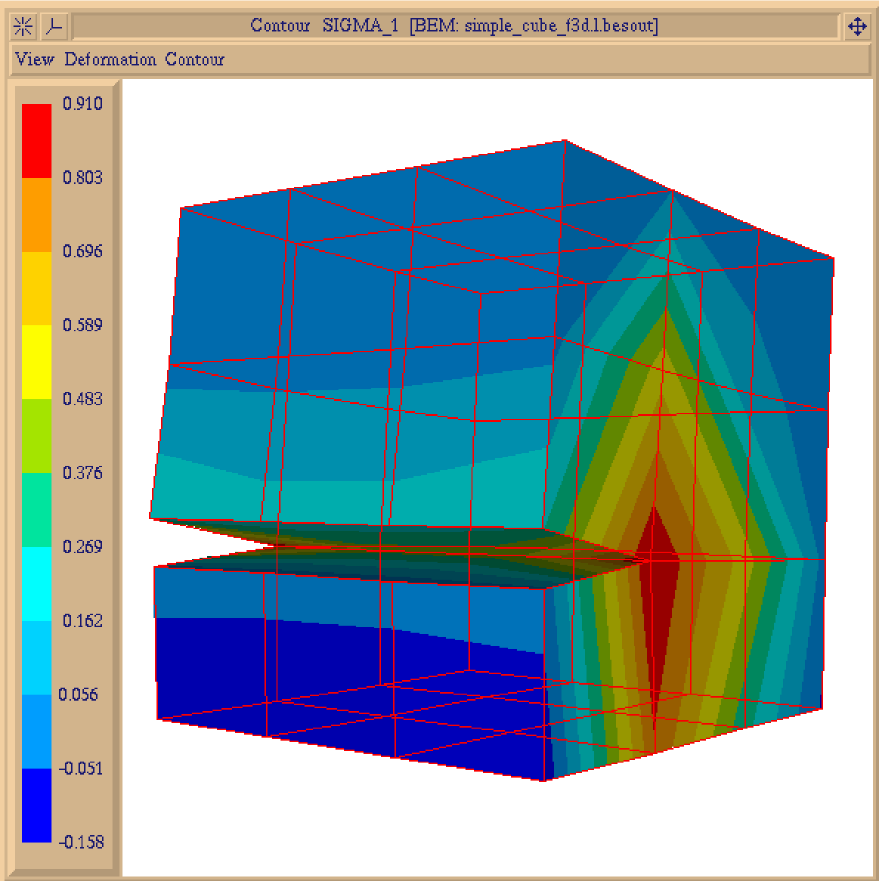
We can use the ANSYS mesh facets to define FRANC3D geometry, but it is not recommended for big models.

The MRP is Imported into FRANC3D and used in the Face Boundary Condition dialog box – this is then attached to the crack surface.

Simple Cube Example

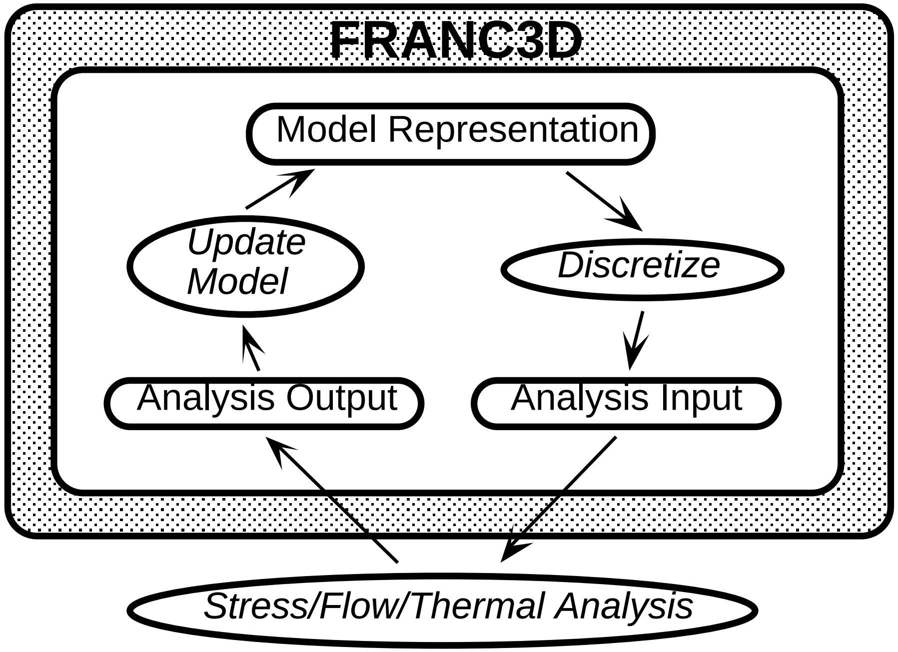


The BEM analysis provides the stresses and displacements for the cracked model. Stress intensity factors can be computed.

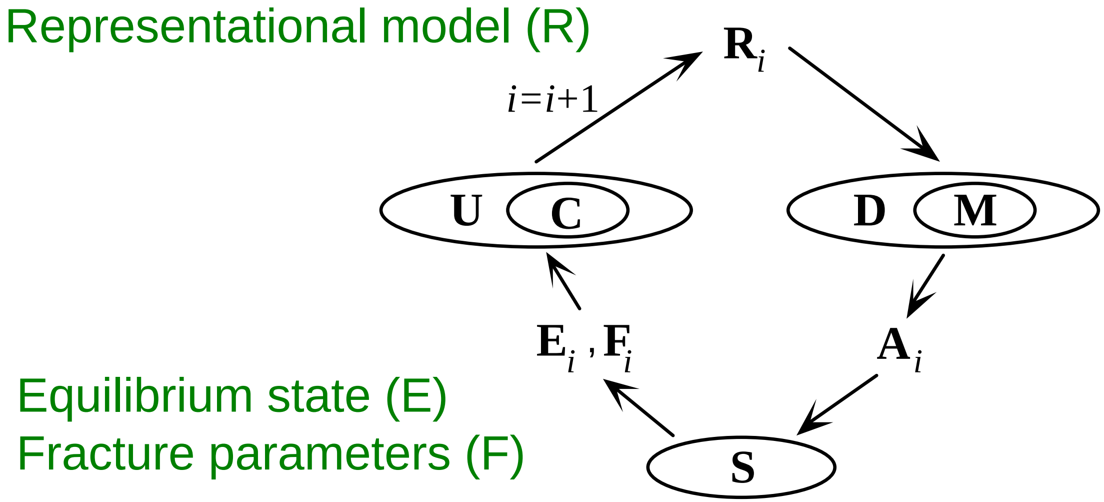


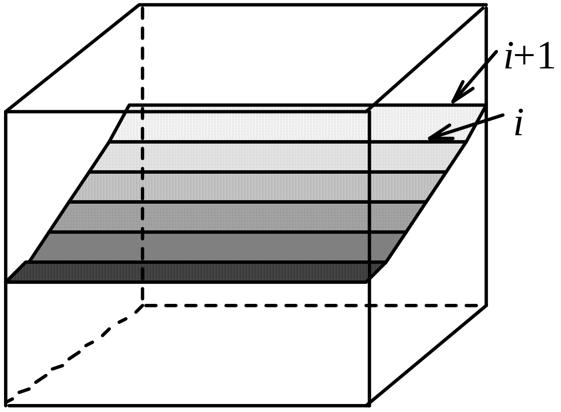
A part through crack is inserted into the FRANC3D model and the face boundary condition defined previously is attached to this surface.

FRANC3D Concepts



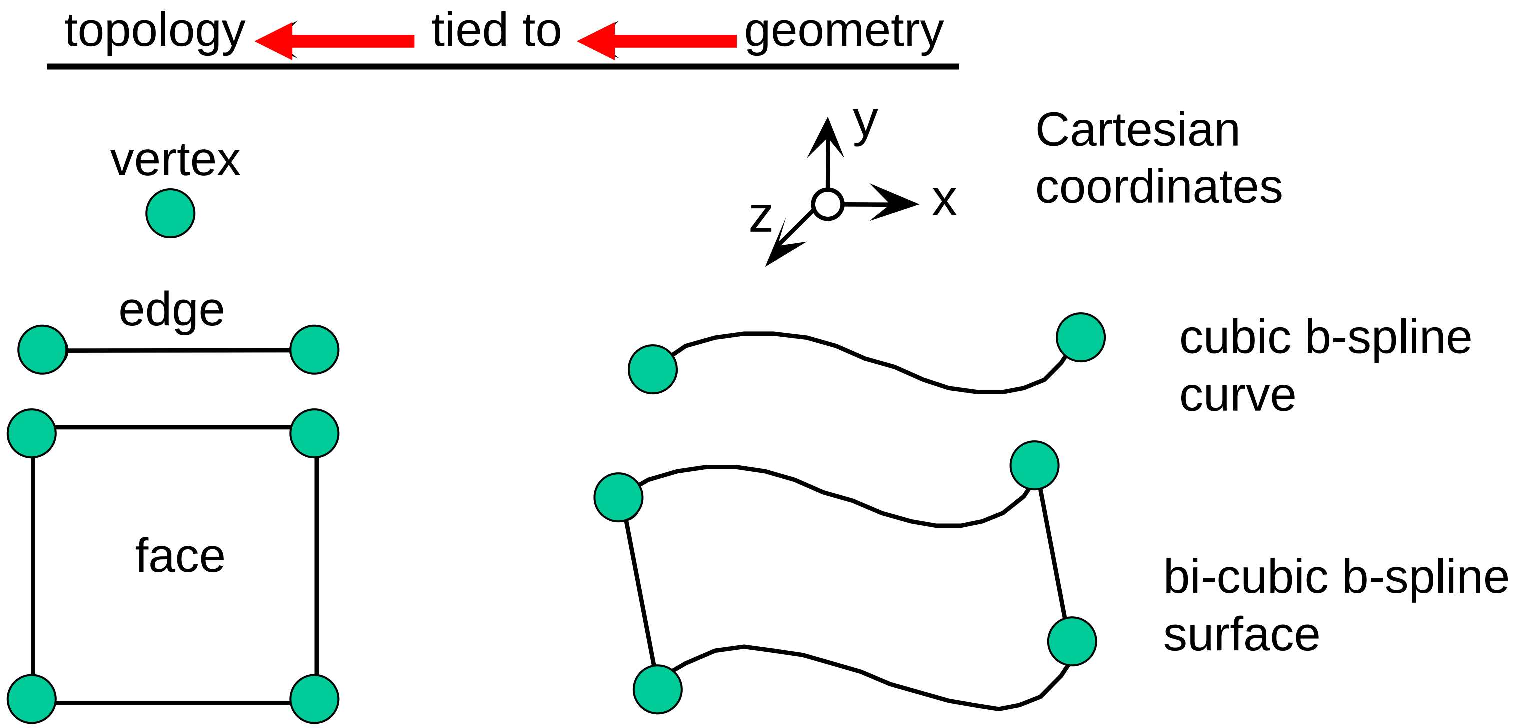
• incremental process involving ‘n’ steps of crack growth in a “true” geometric model of the structure.





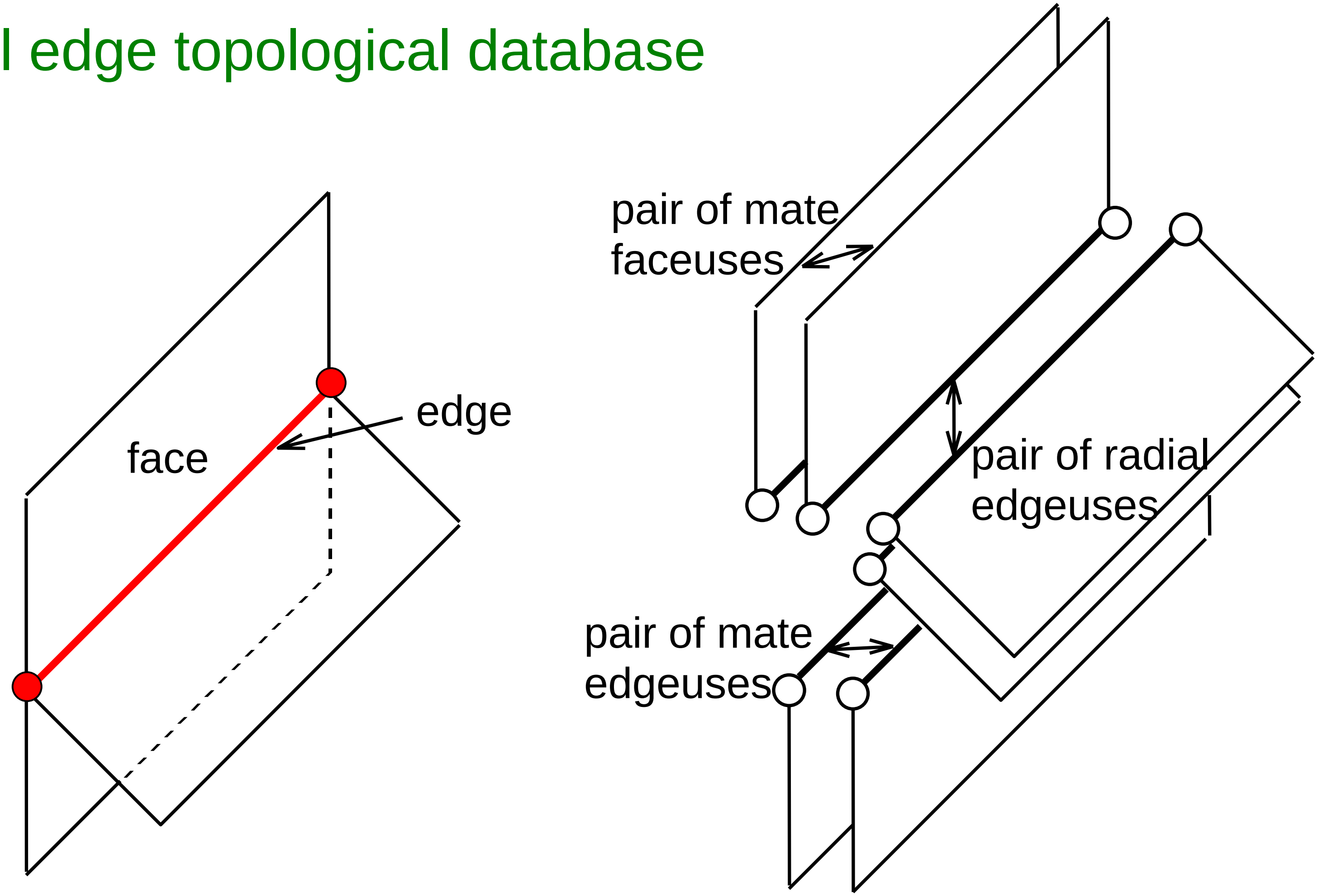
Geometric Model and Topology

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

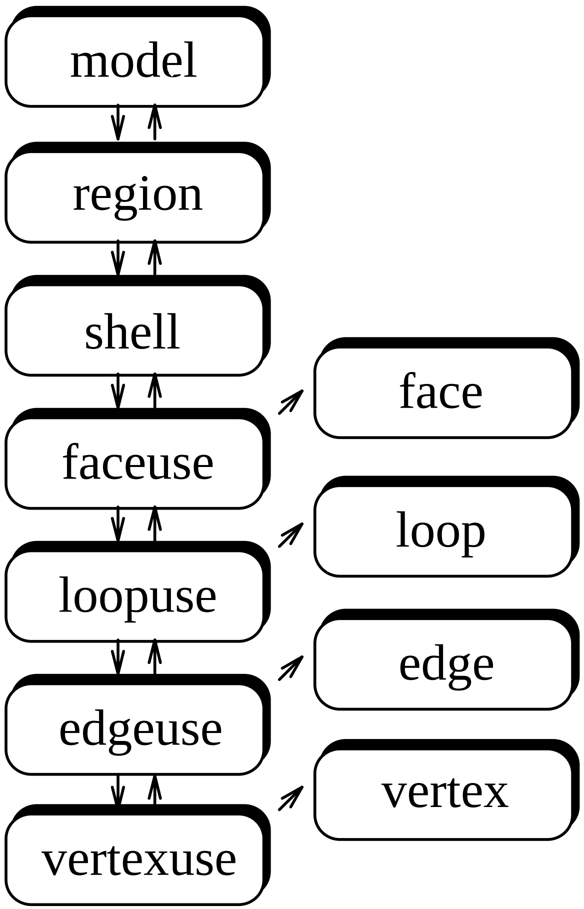
radial edge topological database

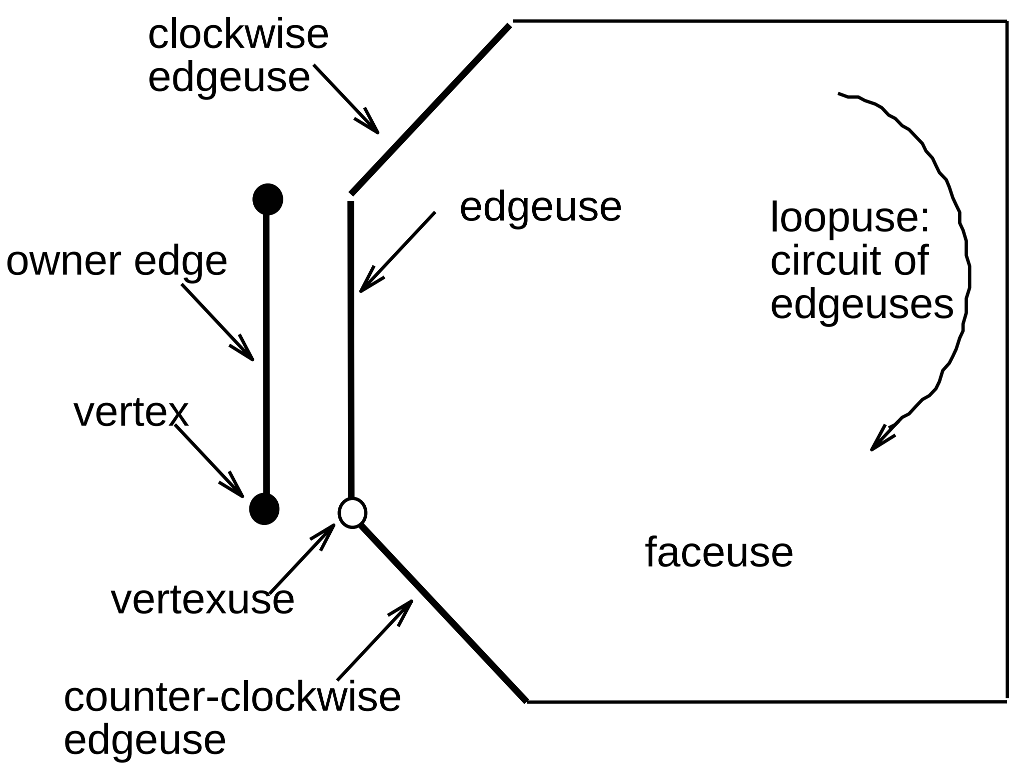


Topological edges provide ordering based on the edge uses.

Geometric Model and Topology

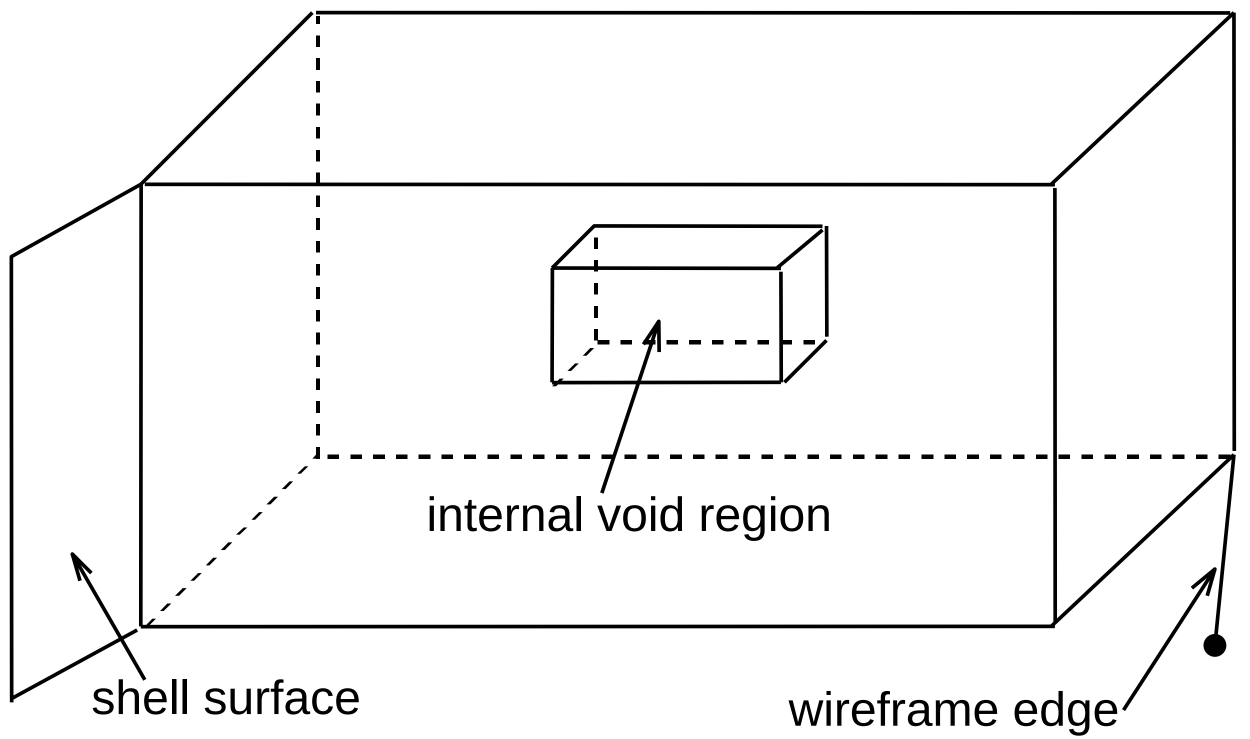
Topological entities have orientations and uses and the data is stored accordingly, such that traversals and queries are efficient.





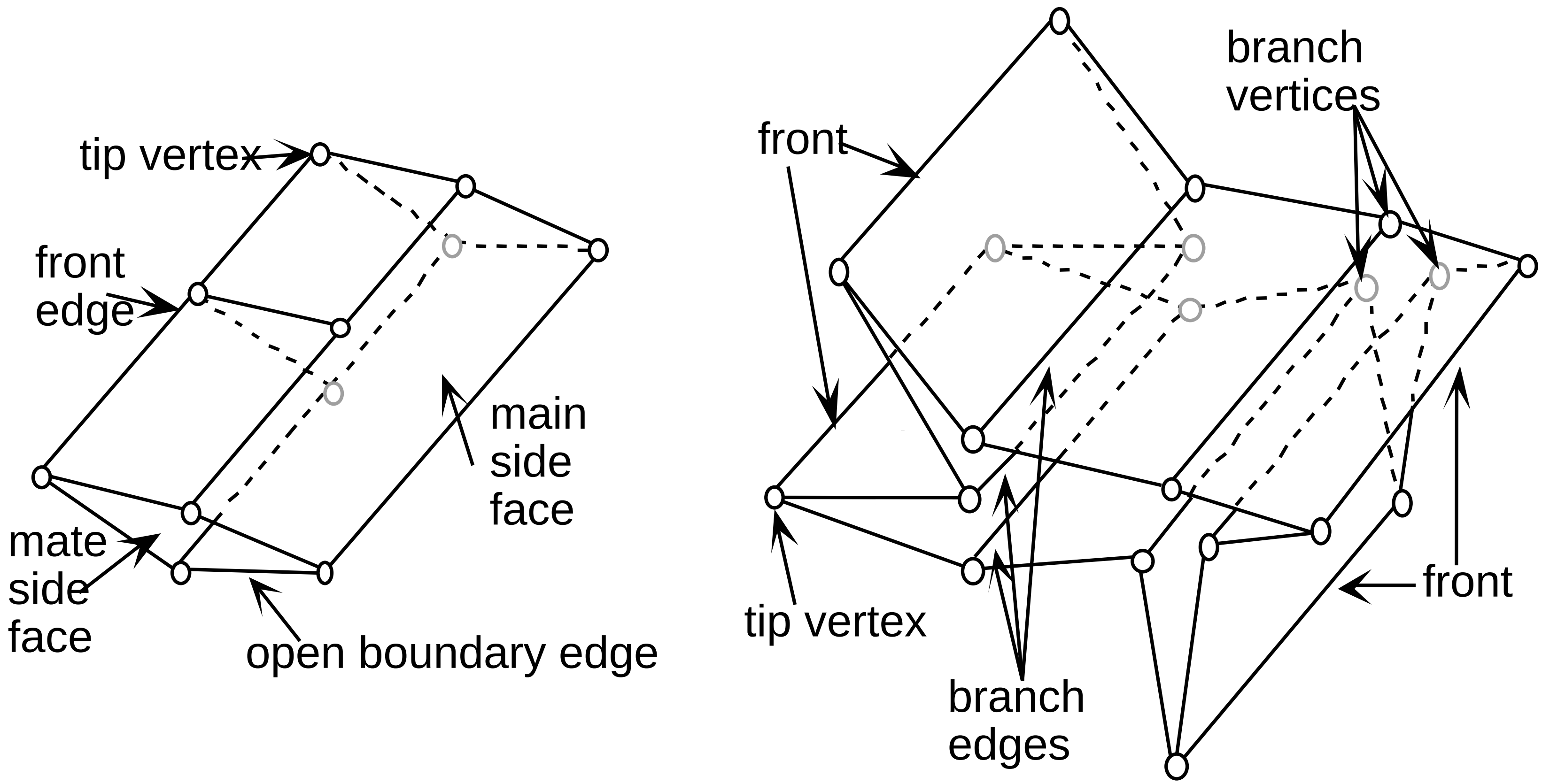
Geometric Model and Topology

Allows for the definition of complex topological models,



including the definition of complex cracks.

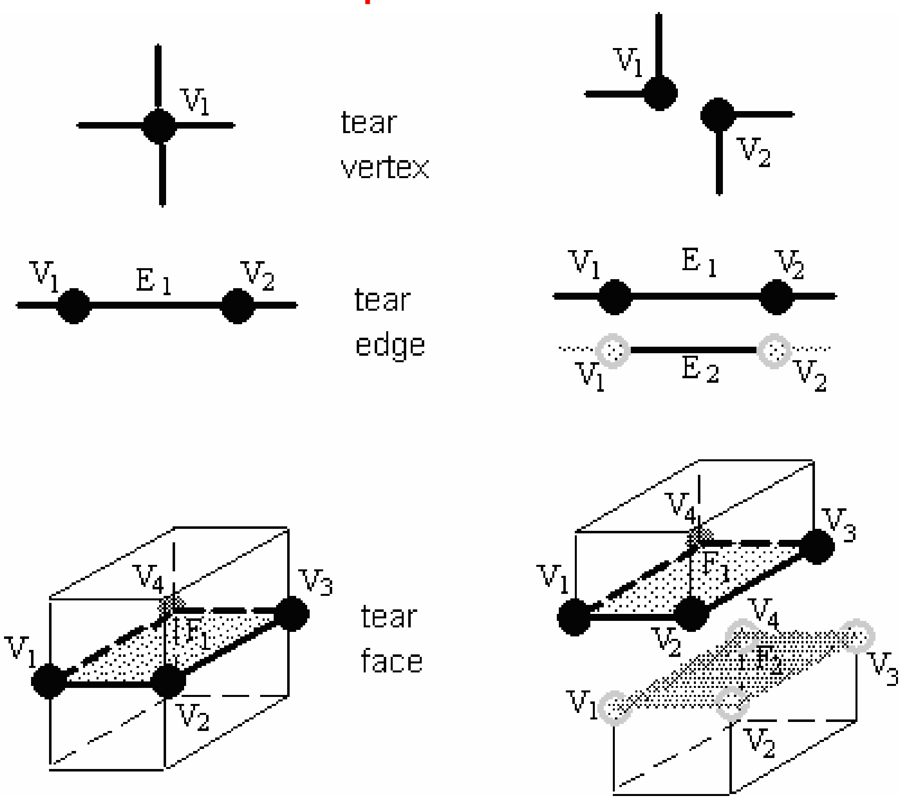
Geometric Model and Topology



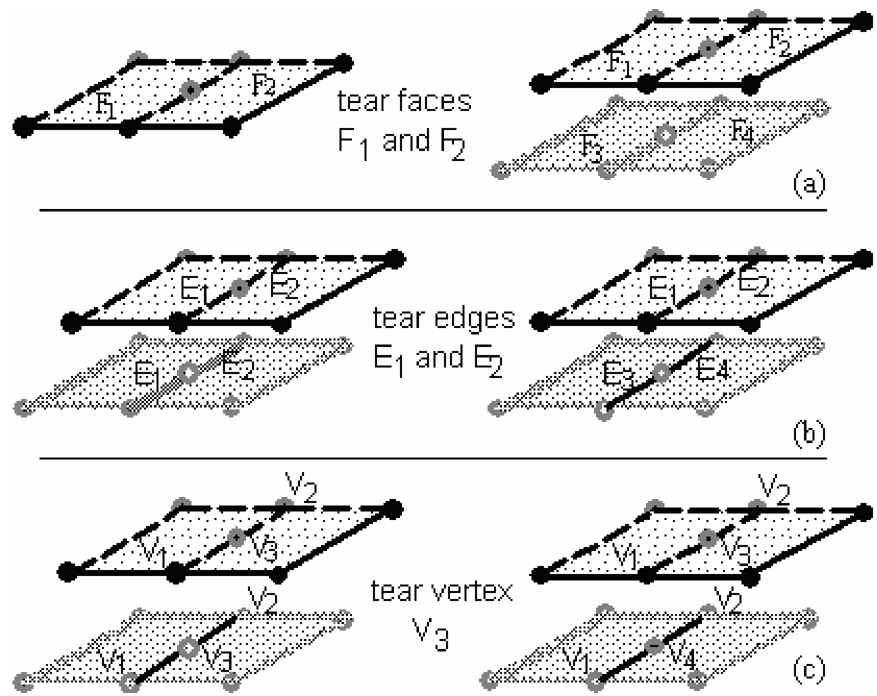
- main and mate crack topological surfaces share the same geometry
- the normal to the main side surface points into the crack towards the mate
- the crack tip and branch vertices mark the ends of crack fronts
- crack fronts can grow independently

Topological Operators for Cracks

tear operators

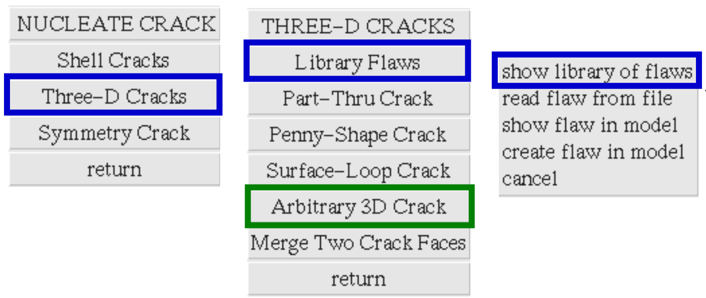


tear operators are used in sequence to create a crack

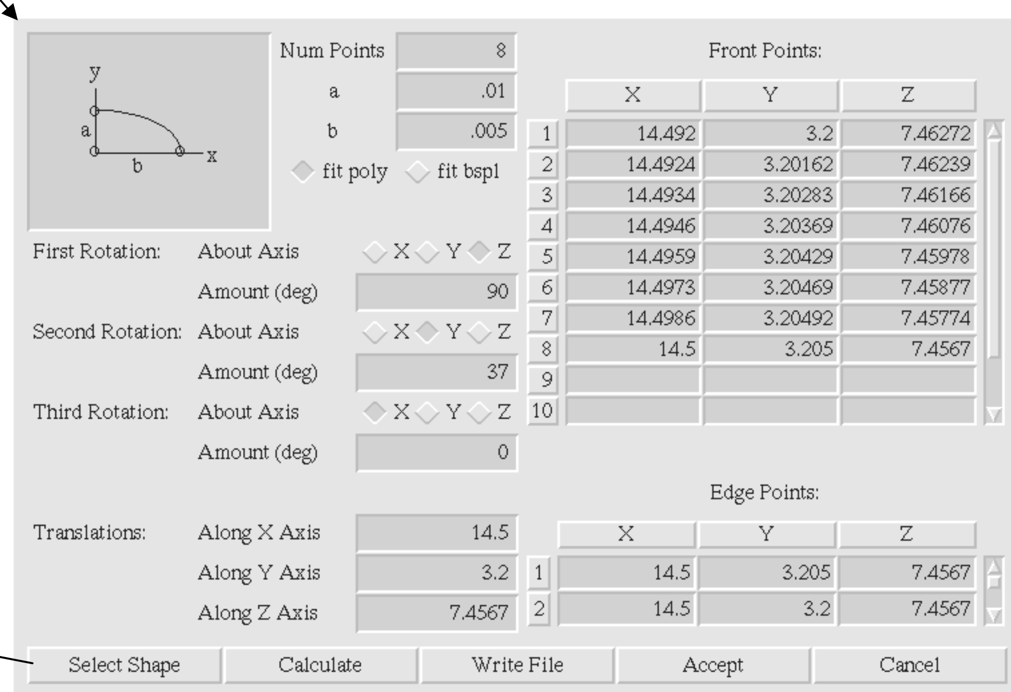


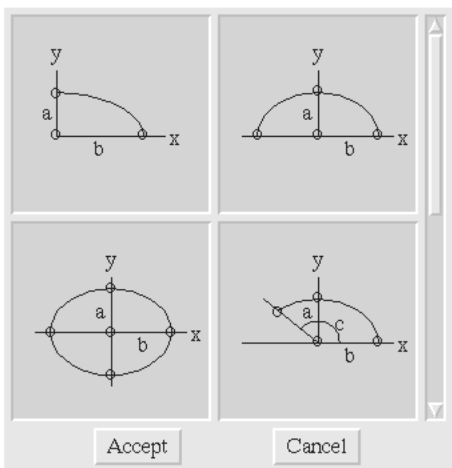
glue operators can be used to delete a crack

Crack Nucleation



a library of common flaw shapes



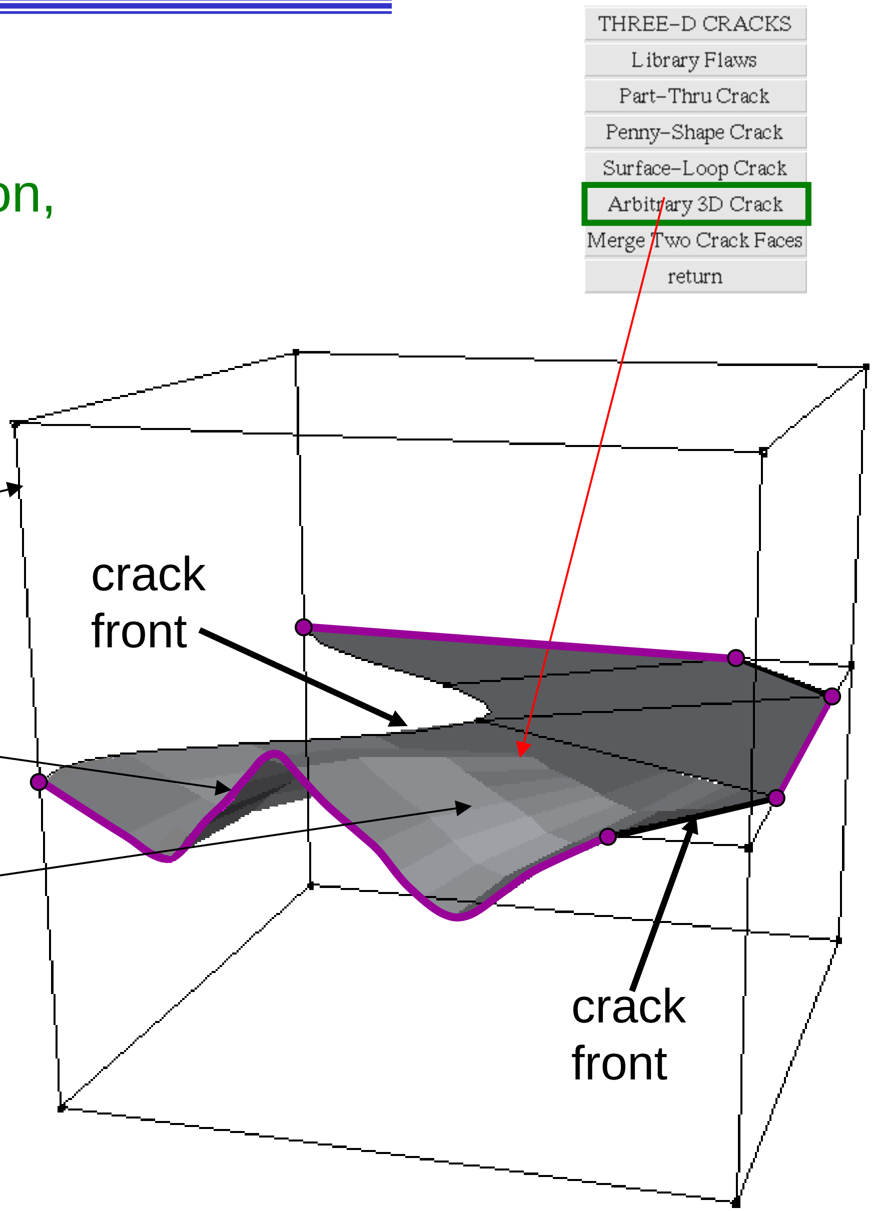


A crack can be nucleated manually by adding the edges and faces and ‘tearing’ the entities using the Arbitrary 3D Crack menu option.

Crack Nucleation

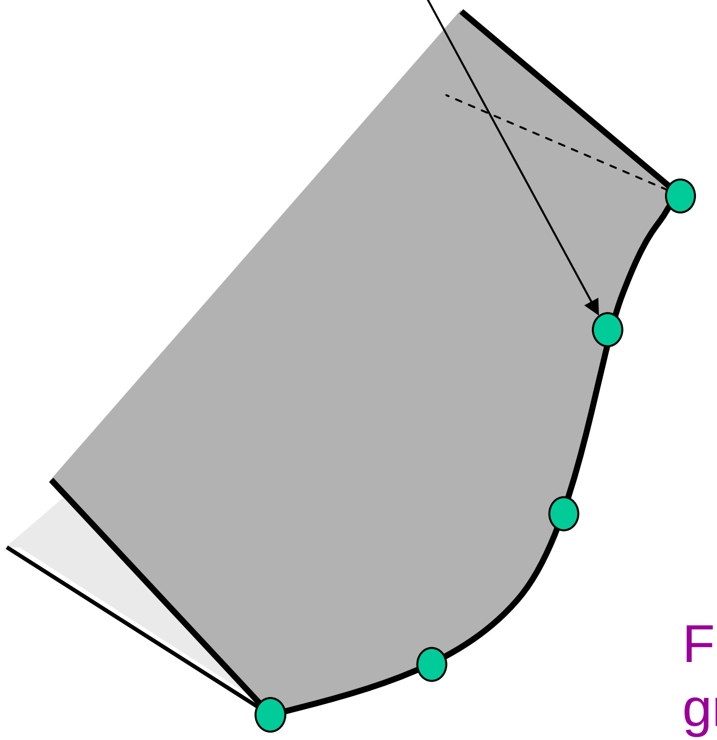
Completely arbitrary initial flaws (arbitrary size, orientation, location, and shape, including non-planar flaws) can be specified in the geometrical model.

- 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

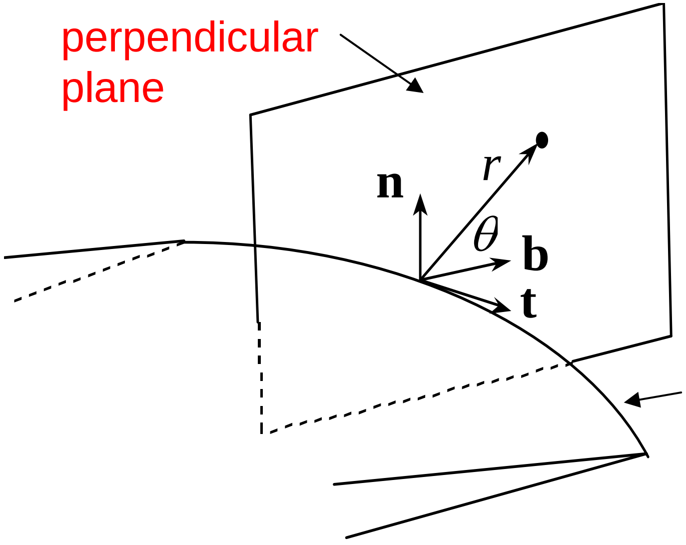


Crack Propagation

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

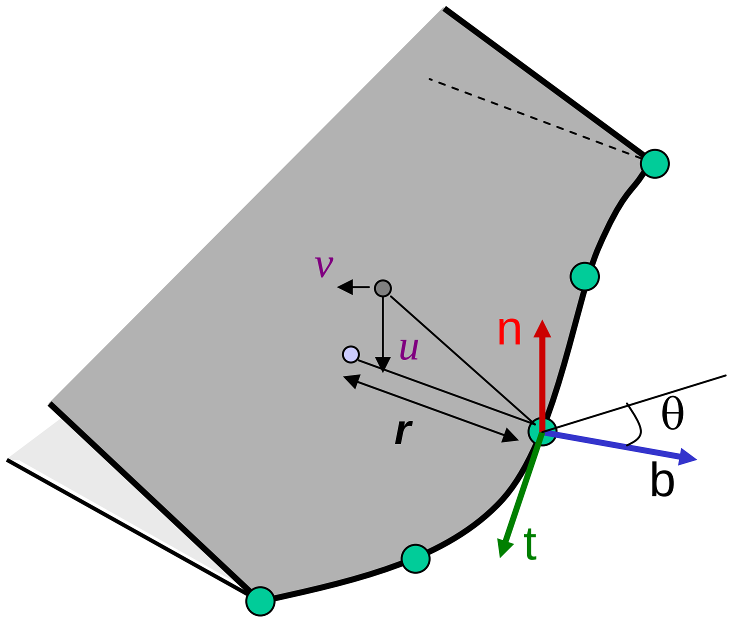


At each point, the plane orthogonal to the local tangent on the crack front is defined and 2D theories for crack growth are invoked.



Fracture mechanics theories and crack growth models are encapsulated in easily modified modules.

Crack Propagation: Displacement Correlation

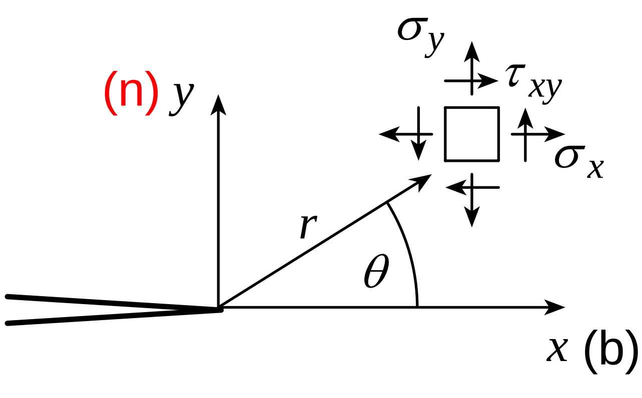


Stress and displacement near the crack tip as a function of r and θ:

$$\sigma_x = \frac{K_I}{\sqrt{2\pi r}} \cos\frac{\theta}{2}\left(1 - \sin\frac{\theta}{2}\sin\frac{3\theta}{2}\right)$$

$$\sigma_y = \frac{K_I}{\sqrt{2\pi r}} \cos\frac{\theta}{2}\left(1 + \sin\frac{\theta}{2}\sin\frac{3\theta}{2}\right)$$

$$\tau_{xy} = \frac{K_I}{\sqrt{2\pi r}} \sin\frac{\theta}{2}\cos\frac{\theta}{2}\cos\frac{3\theta}{2}$$



$$u = 2(1+\nu)\frac{K_I}{E}\sqrt{\frac{r}{2\pi}}\cos\frac{\theta}{2}\left[1 - 2\nu + \sin^2\frac{\theta}{2}\right]$$

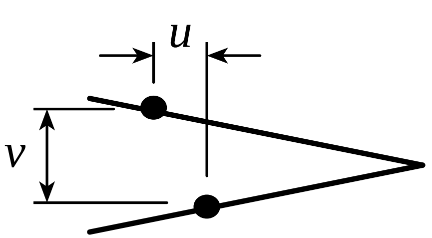
$$v = 2(1+\nu)\frac{K_I}{E}\sqrt{\frac{r}{2\pi}}\sin\frac{\theta}{2}\left[2 - 2\nu - \cos^2\frac{\theta}{2}\right]$$

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:

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



At 180 degrees, along the crack flank:

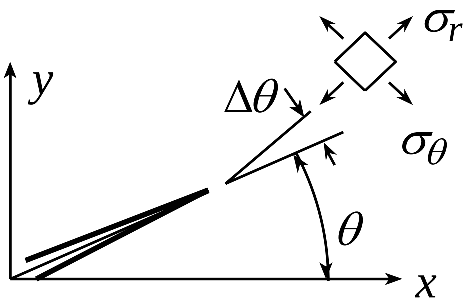
$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \frac{(1+\nu)(1-\nu)}{E}\sqrt{\frac{r}{2\pi}}\begin{Bmatrix} K_{II} \\ K_I \end{Bmatrix}$$

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

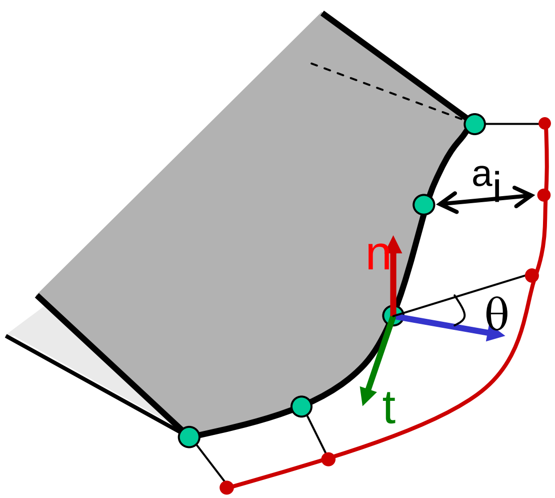
Maximum Hoop Stress Criterion: $\sigma_{\theta\theta}$ is maximum when $\sigma_{r\theta} = 0$



crack-tip "kinks" in the direction where $\sigma_{r\theta} = 0$

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

Solve for θ: $$\tan\frac{\theta_0}{2} = \frac{1}{4}\frac{K_I}{K_{II}} \pm \frac{1}{4}\sqrt{\left(\frac{K_I}{K_{II}}\right)^2 + 8}$$



Crack Extension Rule:

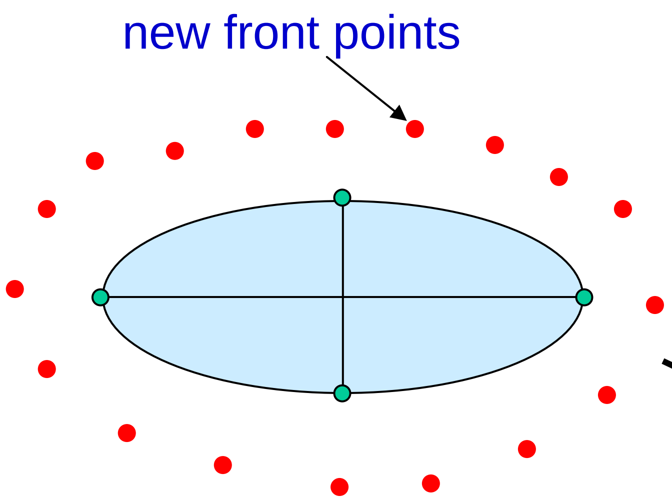
$$a_i = a_{\max}\left[\frac{K_I}{K_{I\max}}\right]^b$$

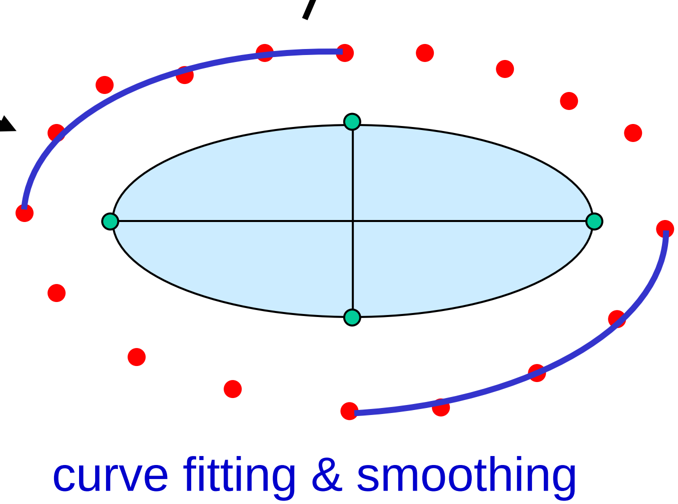
$a_{\max}$ is user defined

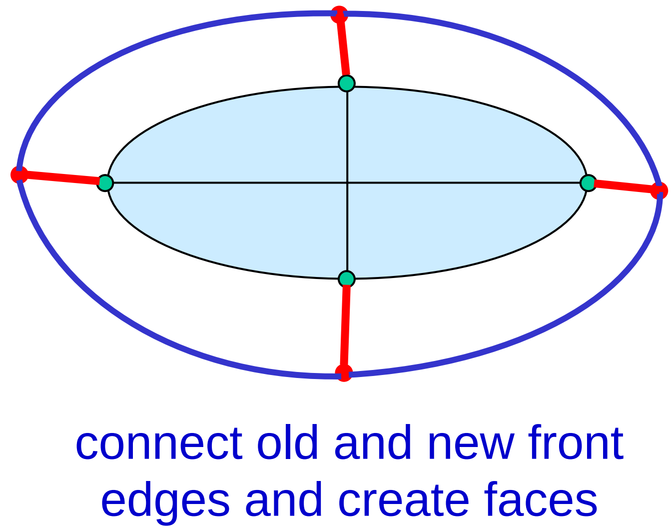
$K_{I\max}$ is the maximum computed K_I

Crack Propagation

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

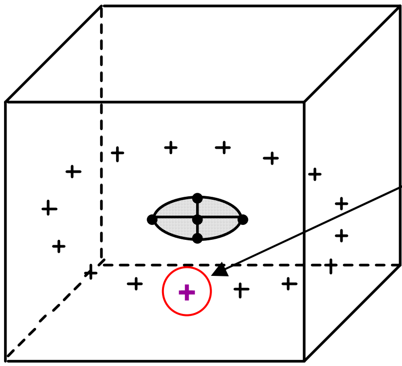




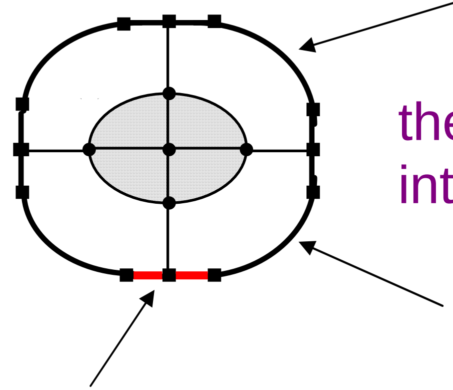


The new front points are smoothed by doing least-squares fitting of polynomials.

Crack Propagation

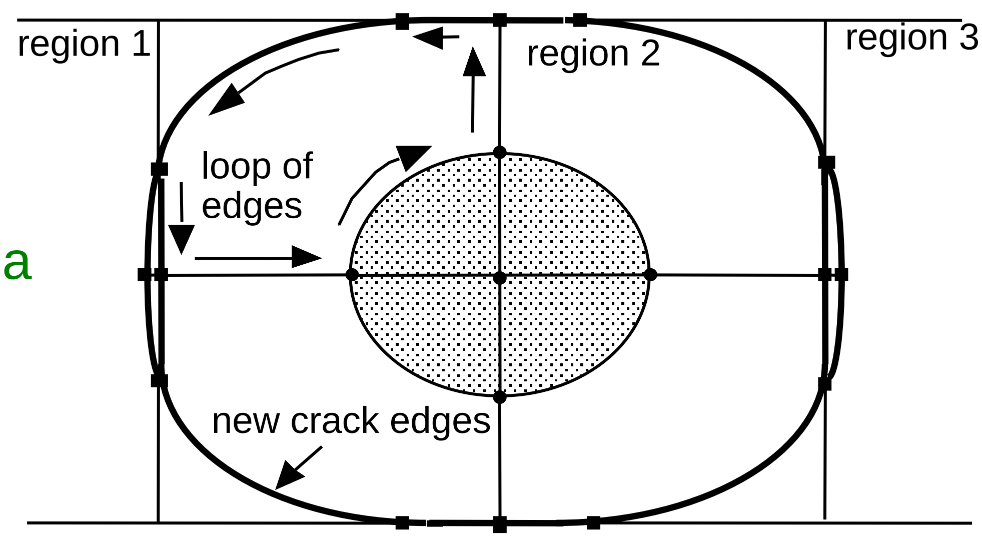


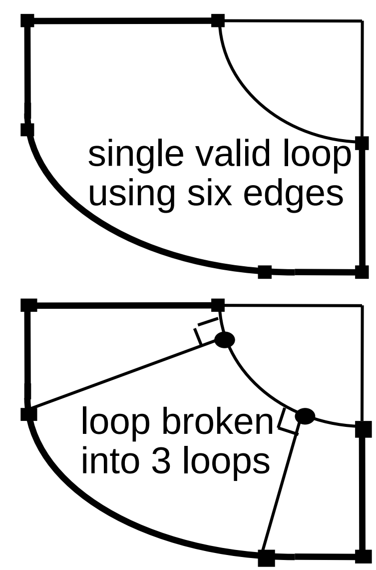
If the points fall outside of the region,



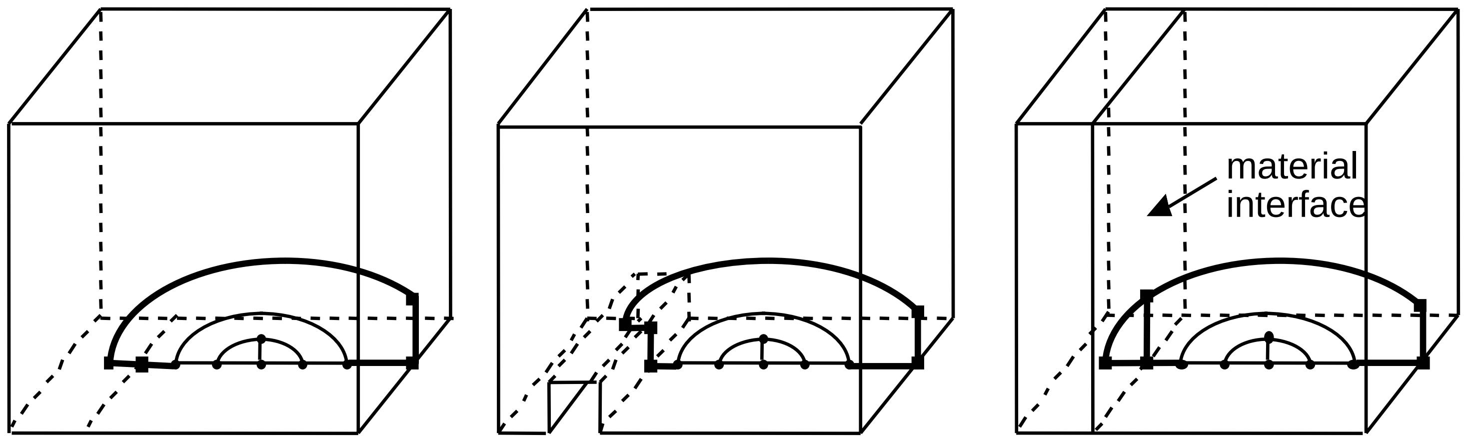
the crack front breaks into segments.

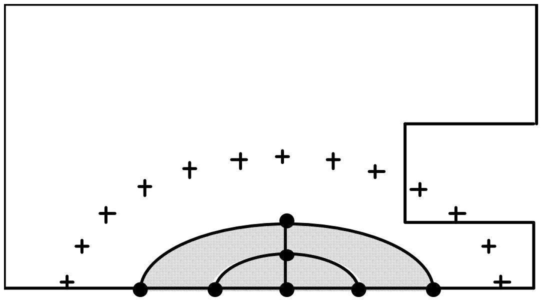
The collection of edges must be turned into a collection of faces that are then ‘torn’.



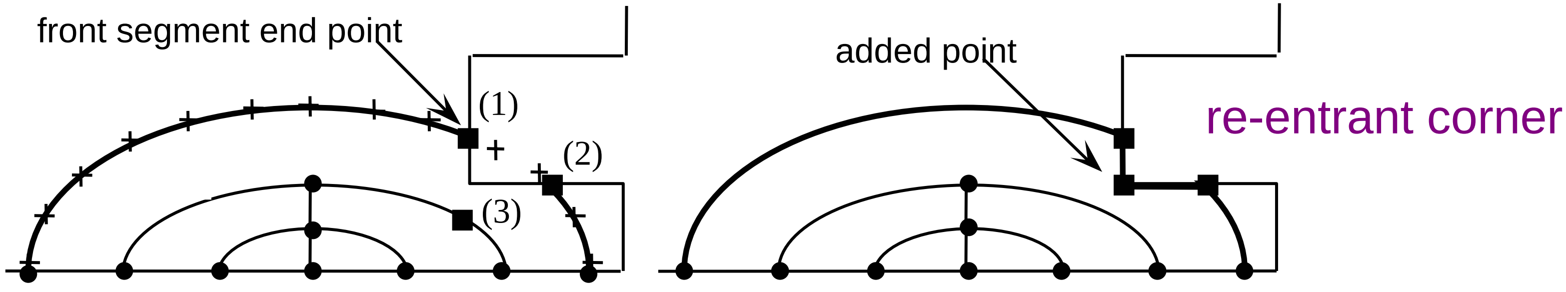


Crack Propagation





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.



Crack Propagation

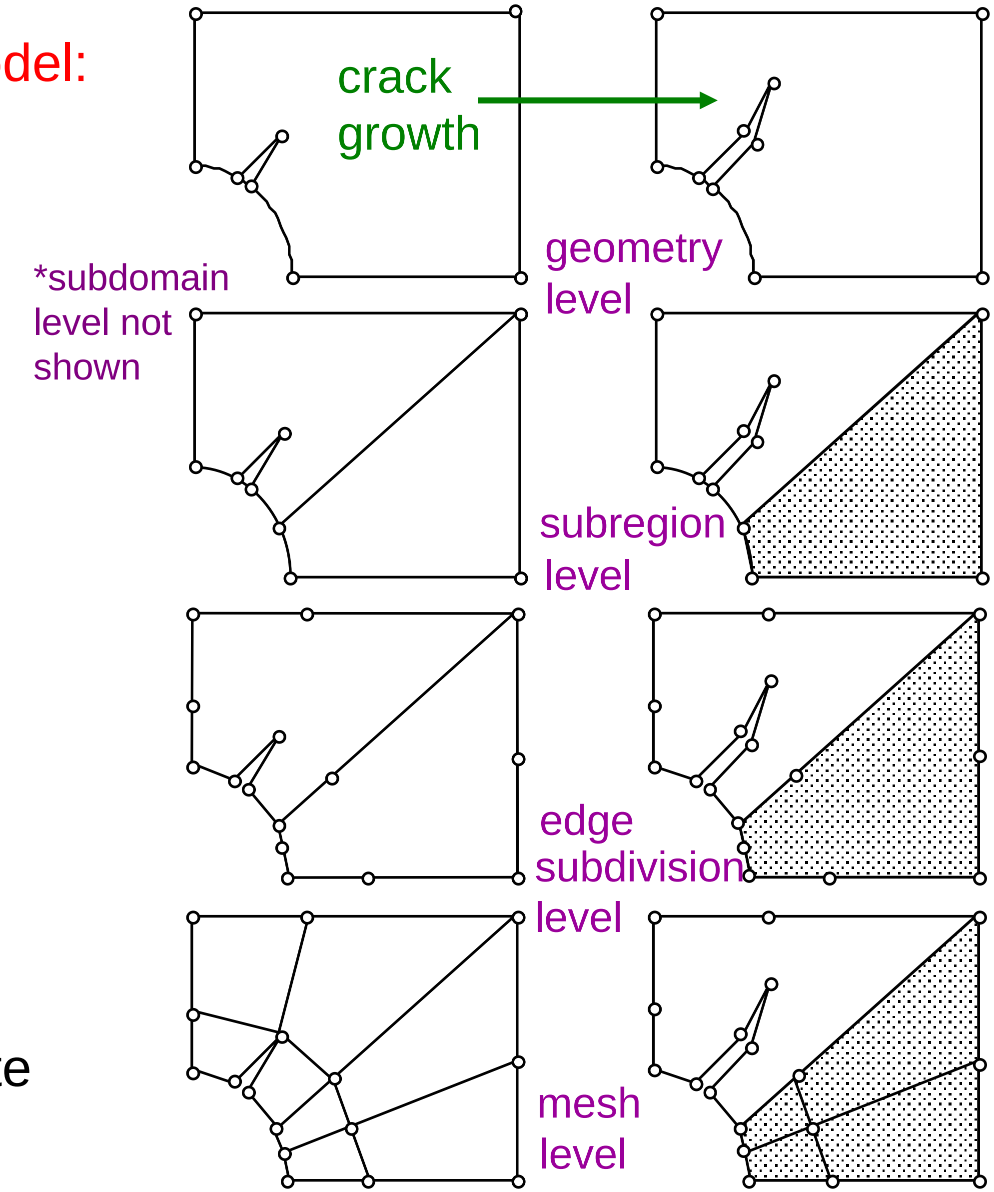
FRANC3D has 5 levels of the model:

- 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

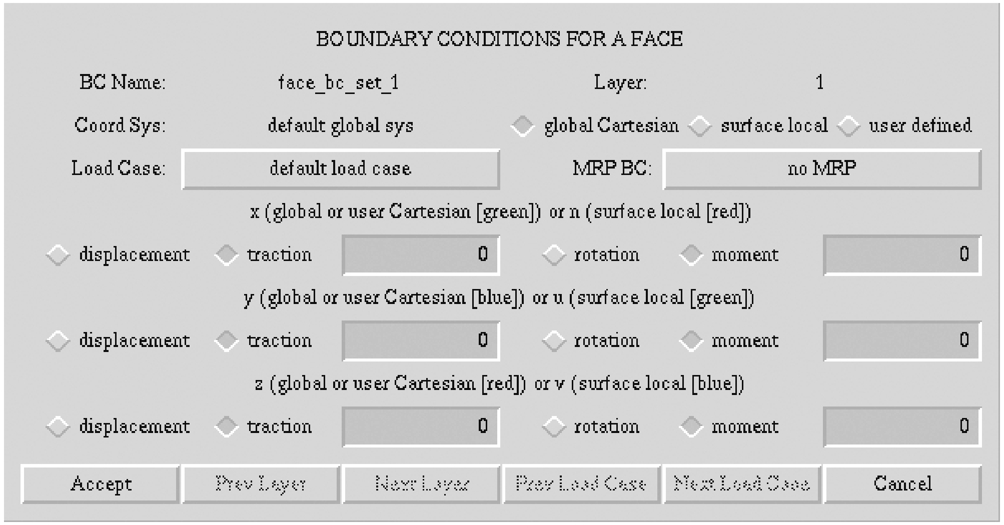
Edge boundary conditions: displacement and force uniform or arbitrary distributions

Point boundary conditions: displacement and force

Arbitrary distributions applied using MRPs (Mesh RePresentation)

Coupled Sets: cyclic symmetry

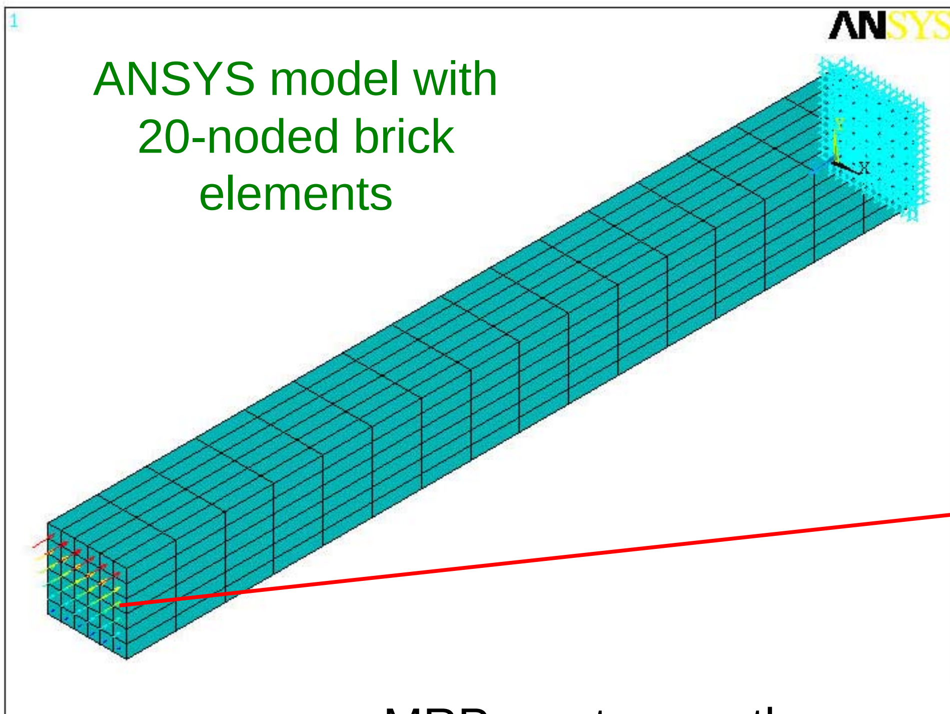
Contact Surfaces: master/slave

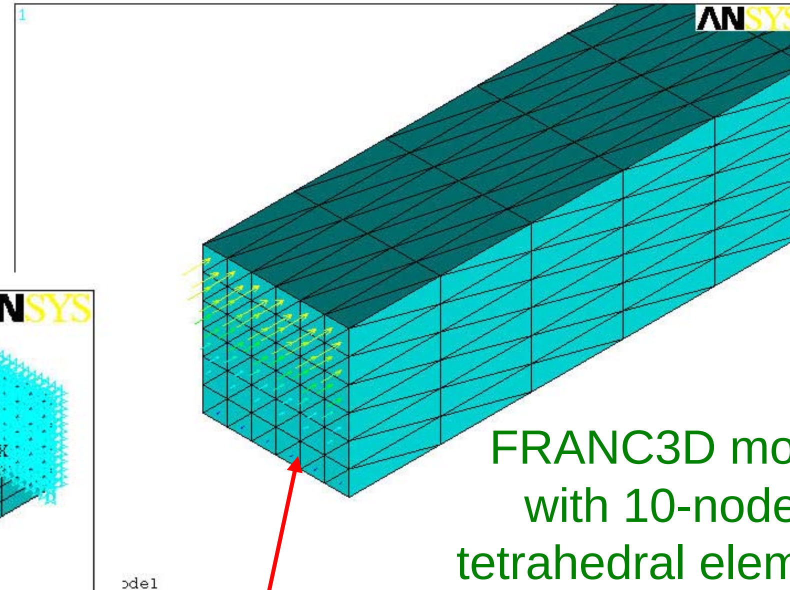


MRP boundary conditions

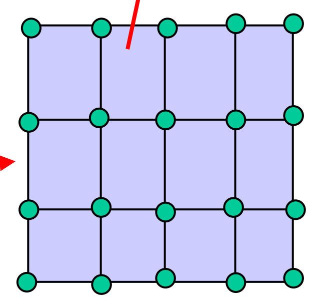
linear pressure distribution
on a cantilever beam

ANSYS model with
20-noded brick
elements





FRANC3D model
with 10-noded
tetrahedral elements

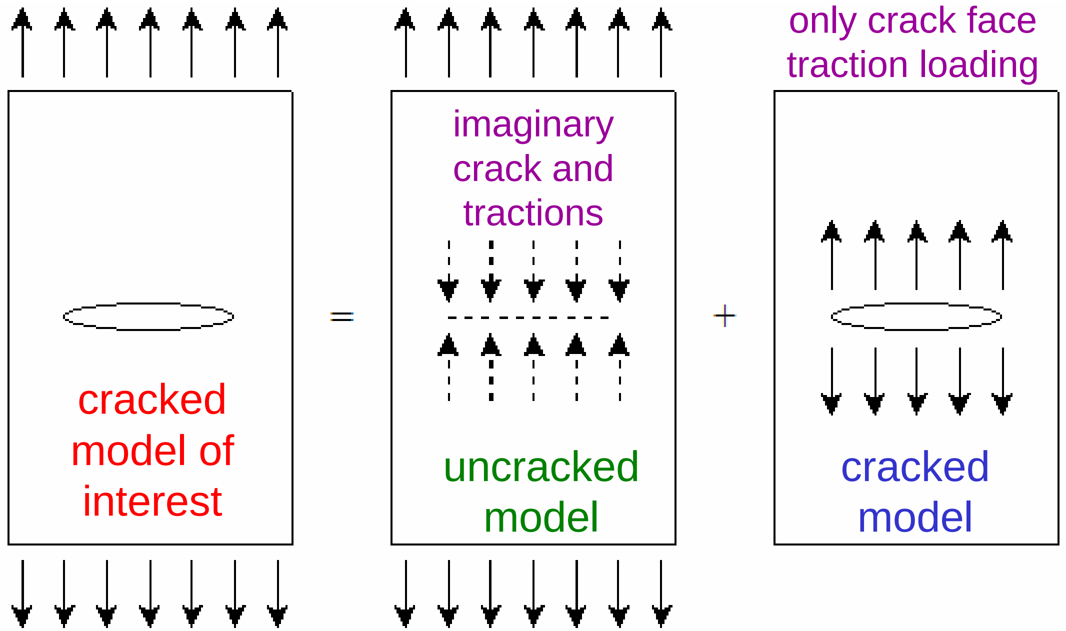


MRP:
12 elements
20 nodes
20 nodal pressures

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]



finite element analyses of an uncracked model (might include complicated loading and stress)

+

boundary element analysis with traction on the crack faces only and constraint where the FEM model has fixed displacement

=

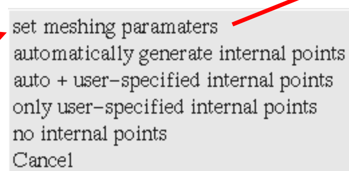
cracked model of interest

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.

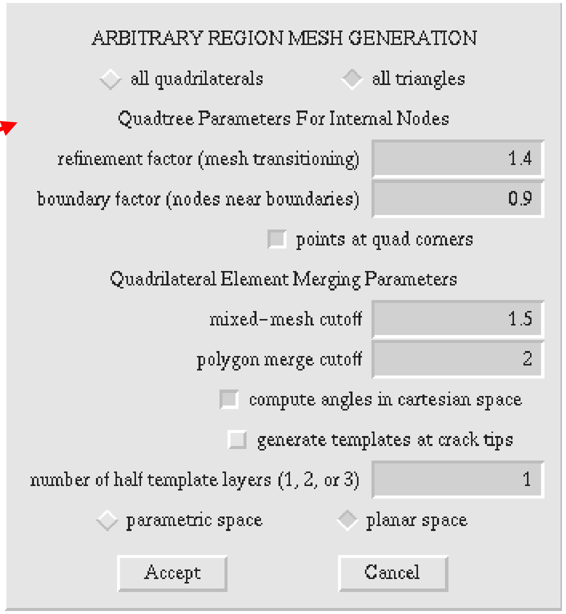
All results -- stresses, strains, displacements, and stress-intensity factors -- for the model of interest, model (a), are the algebraic sum of the results of models (b) and (c). In the case of stress-intensity factors, Ka = Kb + Kc, but since Kb is equal to zero, Ka = Kc.

Meshing & Discretization

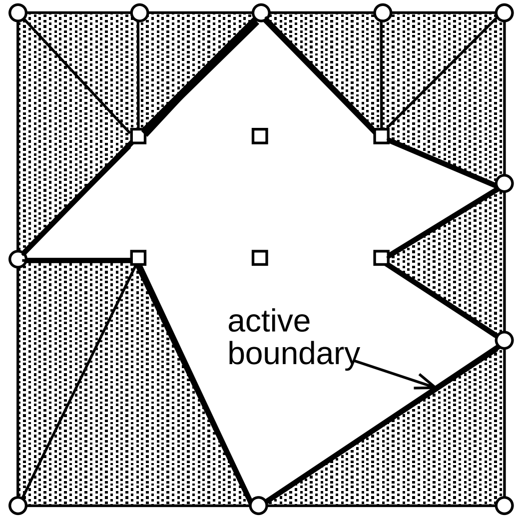
standard mapped meshing algorithms



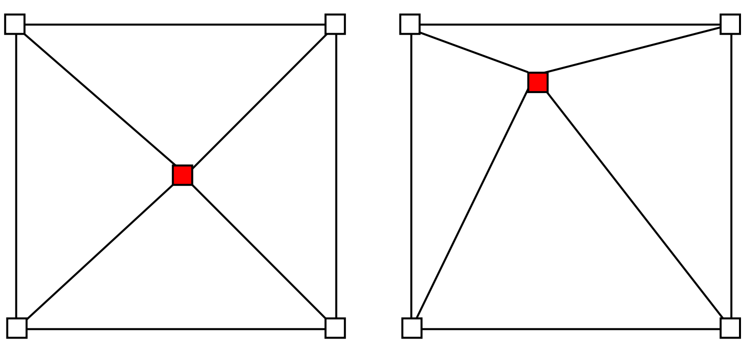
parameters control mesh density



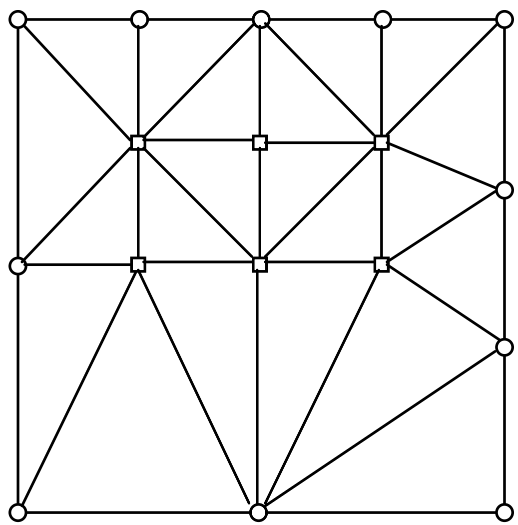
advancing front algorithm



control mesh manually by moving points



final triangular mesh

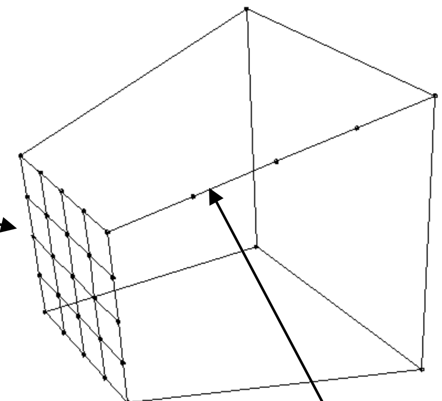


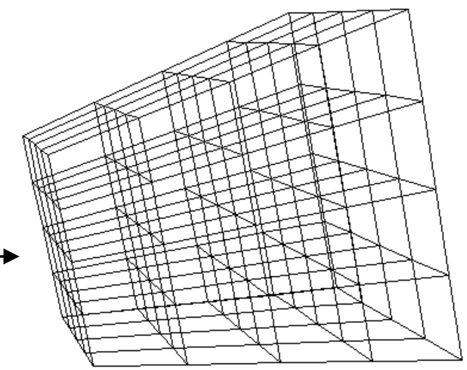
The arbitrary region mesh can still be modified by moving nodes and by adding or deleting edges.

Meshing & Discretization

- sweep bricks or wedges depending on source face mesh

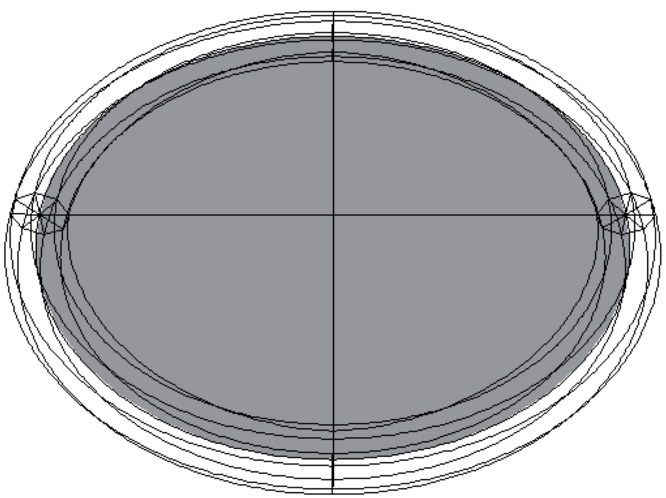
source face mesh





swept brick mesh

subdivided edge in sweep direction



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

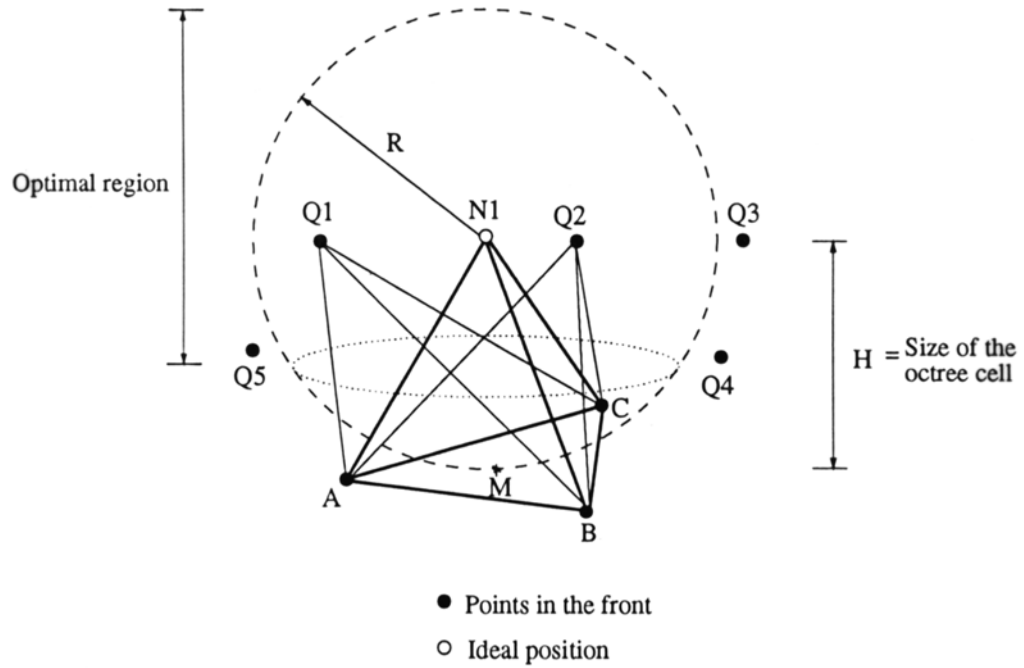
The mesh around the crack front can be:

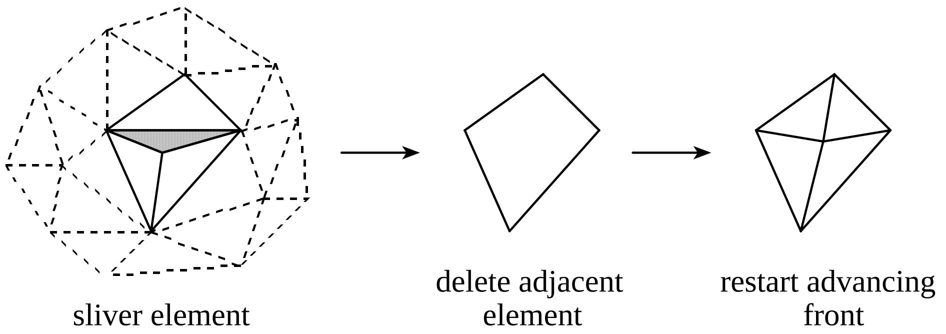
- bricks
- wedges (singular)
- tetrahedra
- wedges surrounded by bricks, then pyramids and then tetrahedra

Tetrahedral Volume Meshing

advancing front, unstructured tetrahedral mesh generation

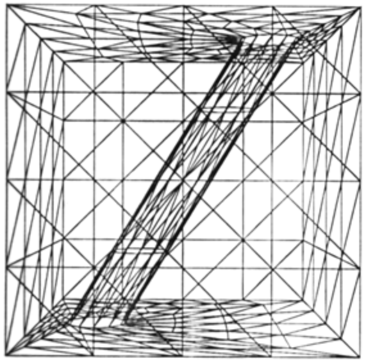
- generate an octree “background mesh” to estimate element size.
- geometry-based advancing front - elements are generated based on ideal shapes.
- topology-based advancing front – all topologically valid elements are generated.
- backtracking to mesh any non-meshable polyhedra.
- edge and face swapping and laplace smoothing for local shape improvement.



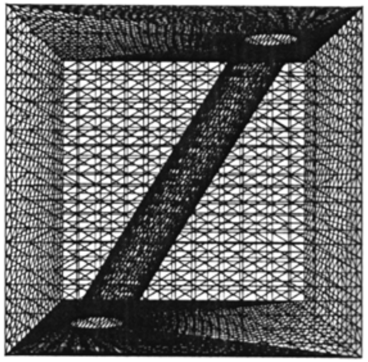


Tetrahedral Volume Meshing

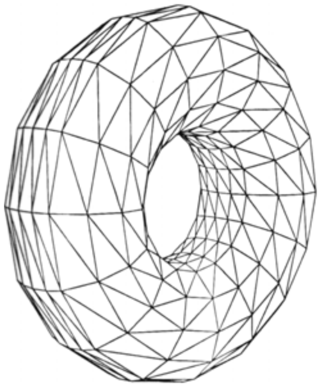
some examples



coarse
and
refined
meshes

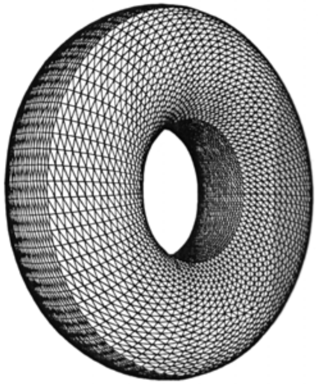


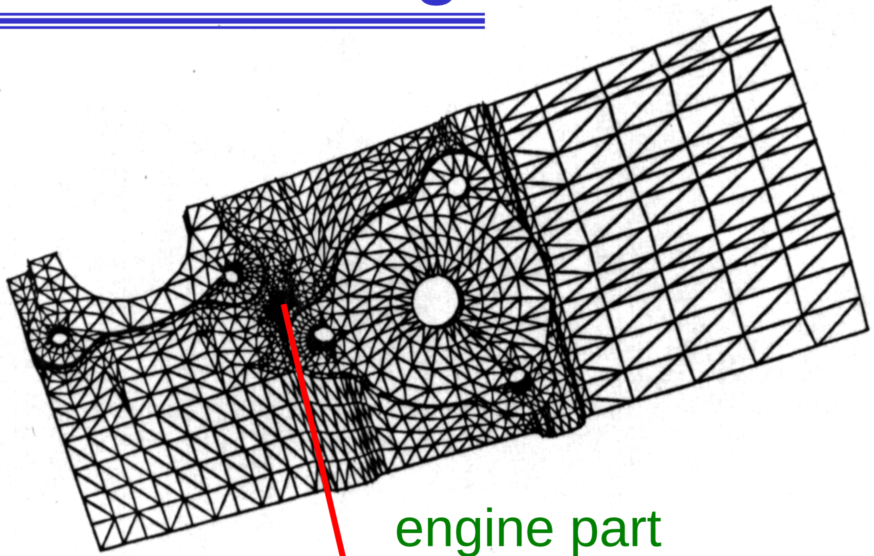
inclined borehole



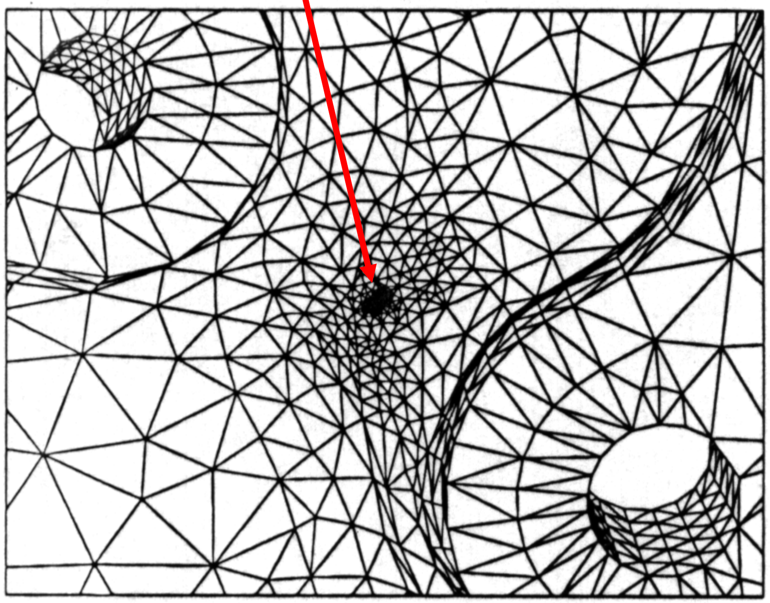
coarse
and
refined
meshes

torus





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’

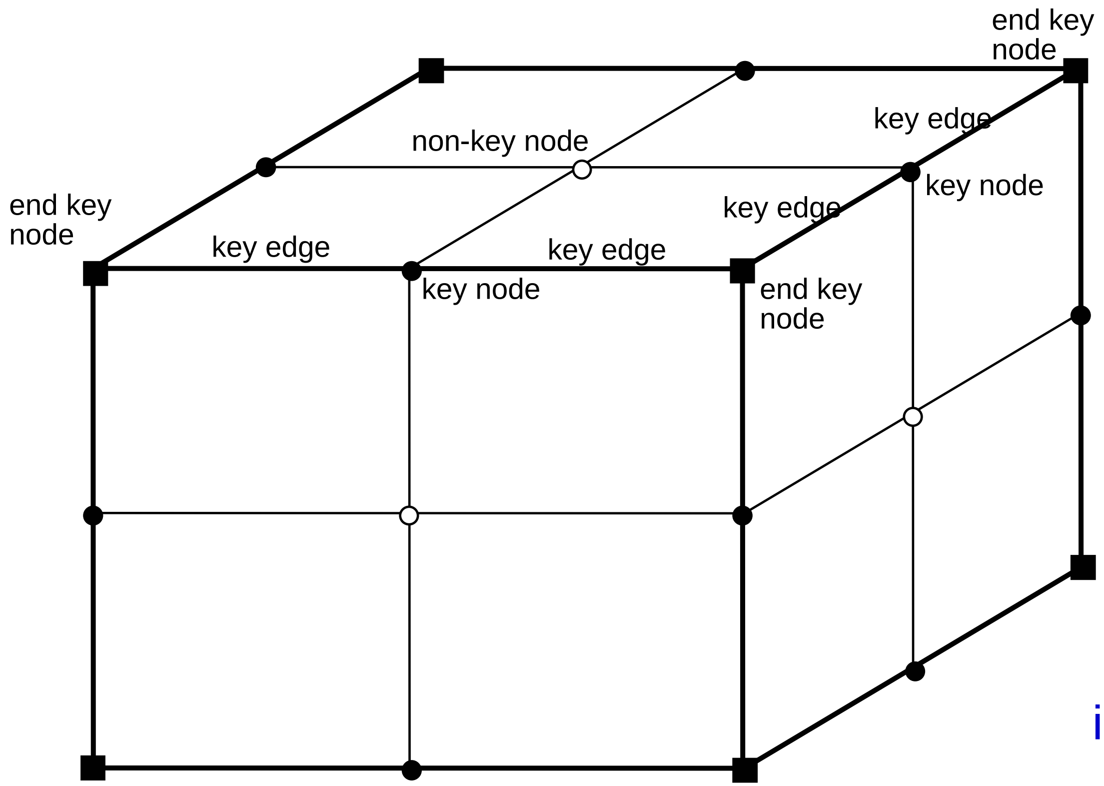
OSM capabilities:

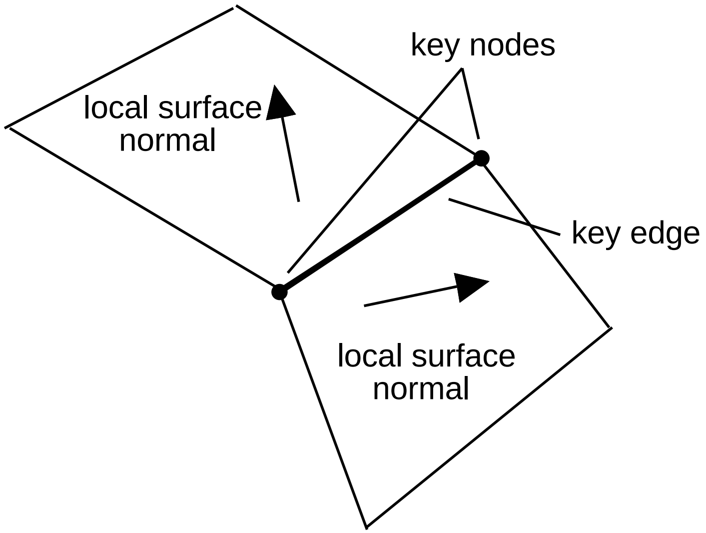
- creates lines, arcs, spline curves
- creates triangular Bezier or bi-cubic b-spline surfaces
- planar surfaces are supported but cannot be created manually
- extrude, translate, rotate, mirror, cut & paste
- copy objects between multiple windows
- finite element model translation and conversion
- ANSYS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

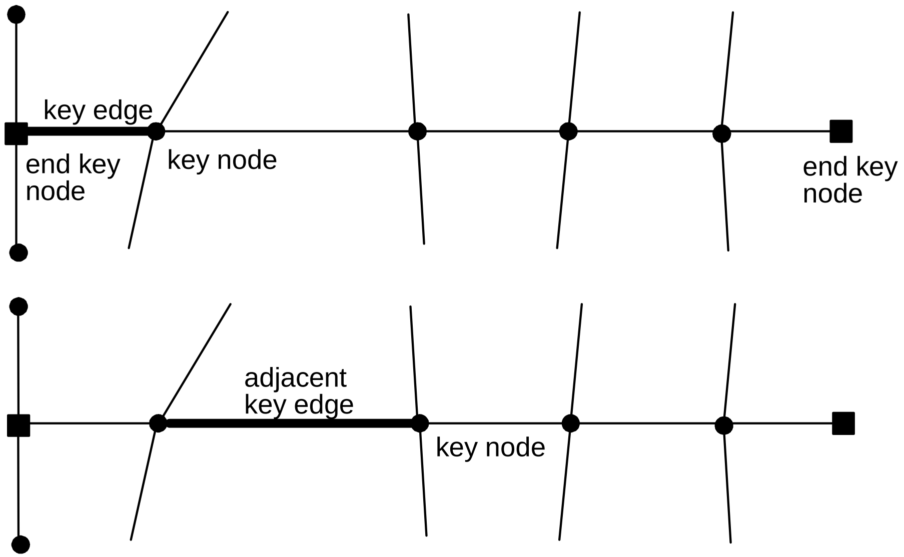
  facets used by two elements are not surface facets and can be removed



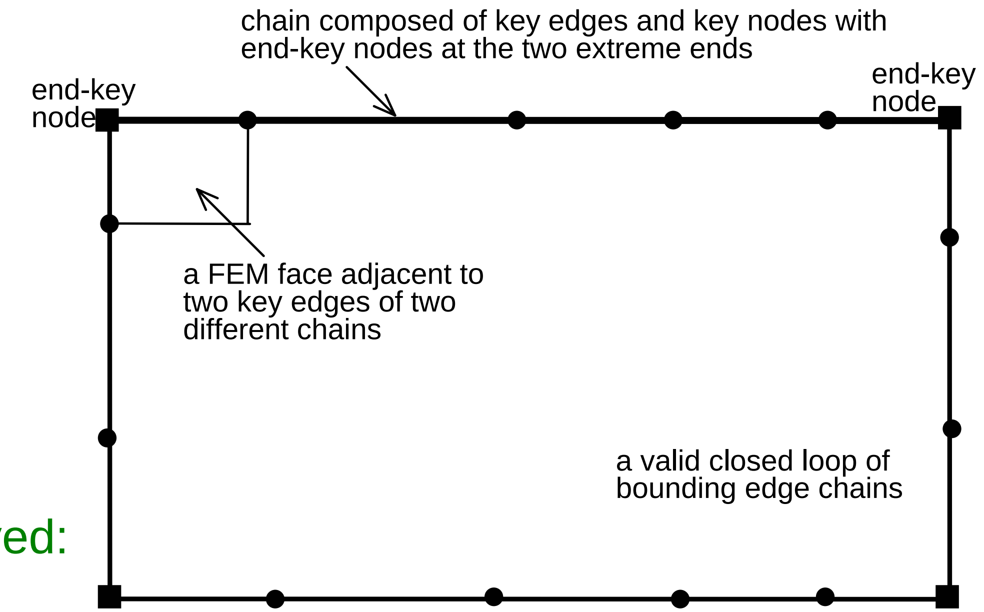


identify key nodes and edges based on angle changes

OSM: FEM Conversion



- compose chains of key edges between end key nodes

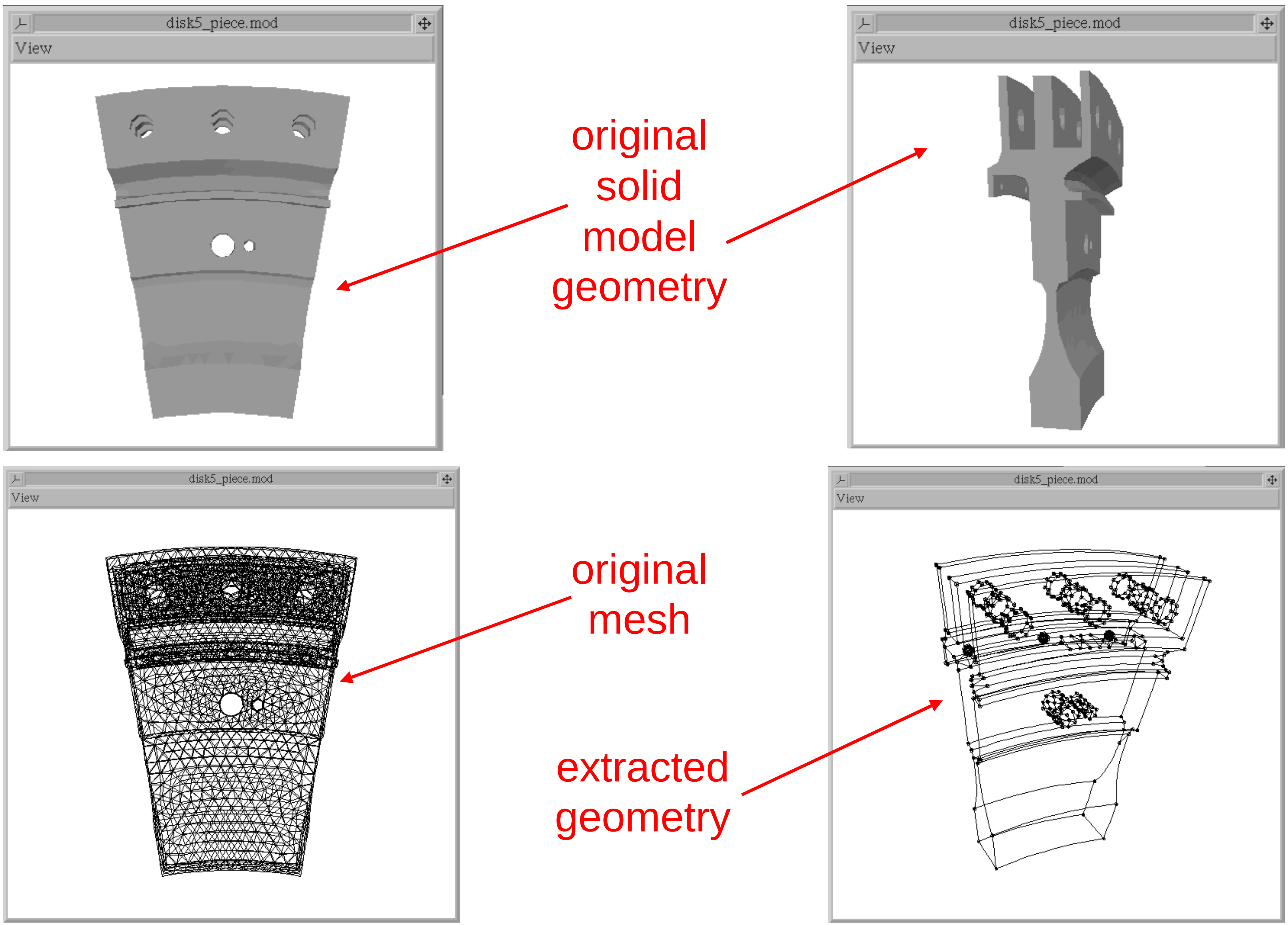


- form loops from the chains

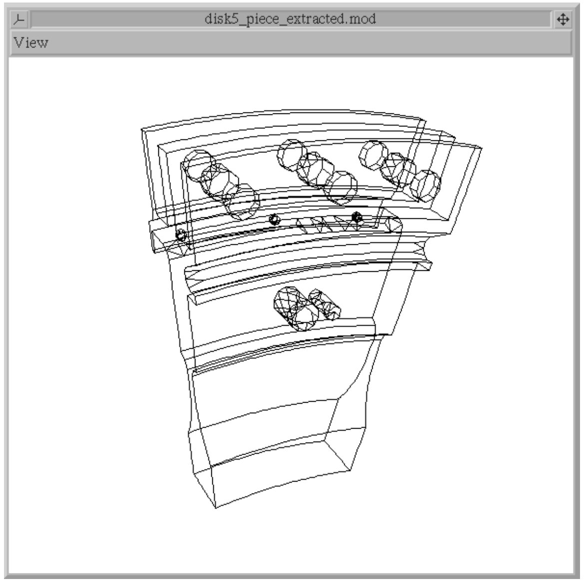
- form faces from the loops

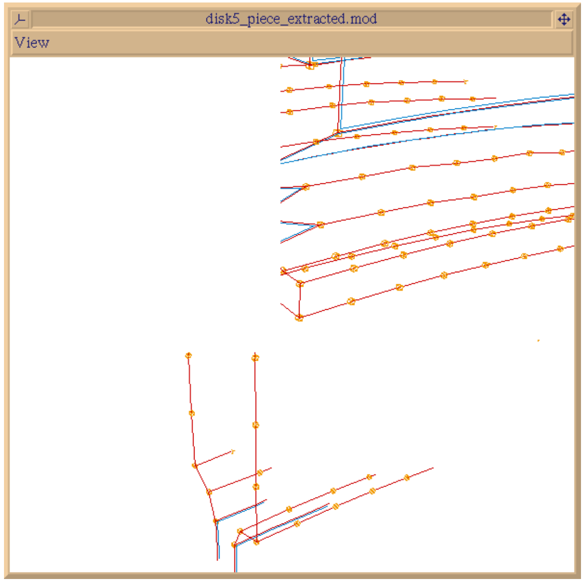
A limitation that has been partly resolved:
- faces must be 3 or 4-sided

OSM: FEM Conversion

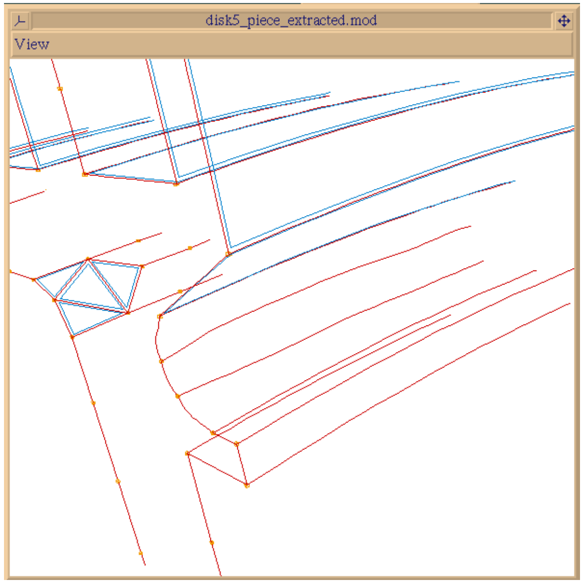


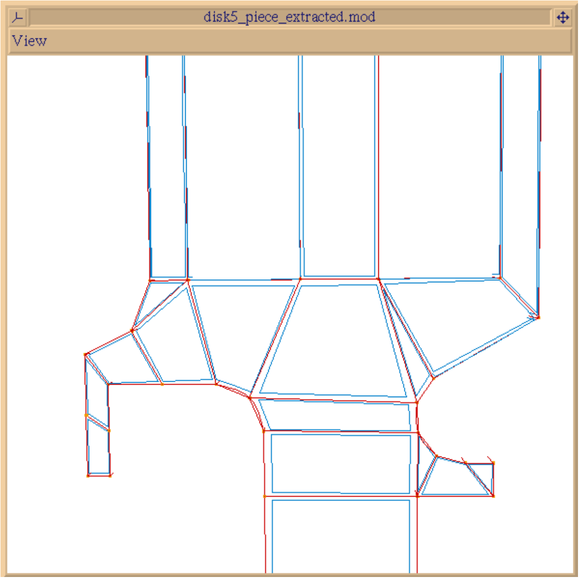
OSM: FEM Conversion





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:

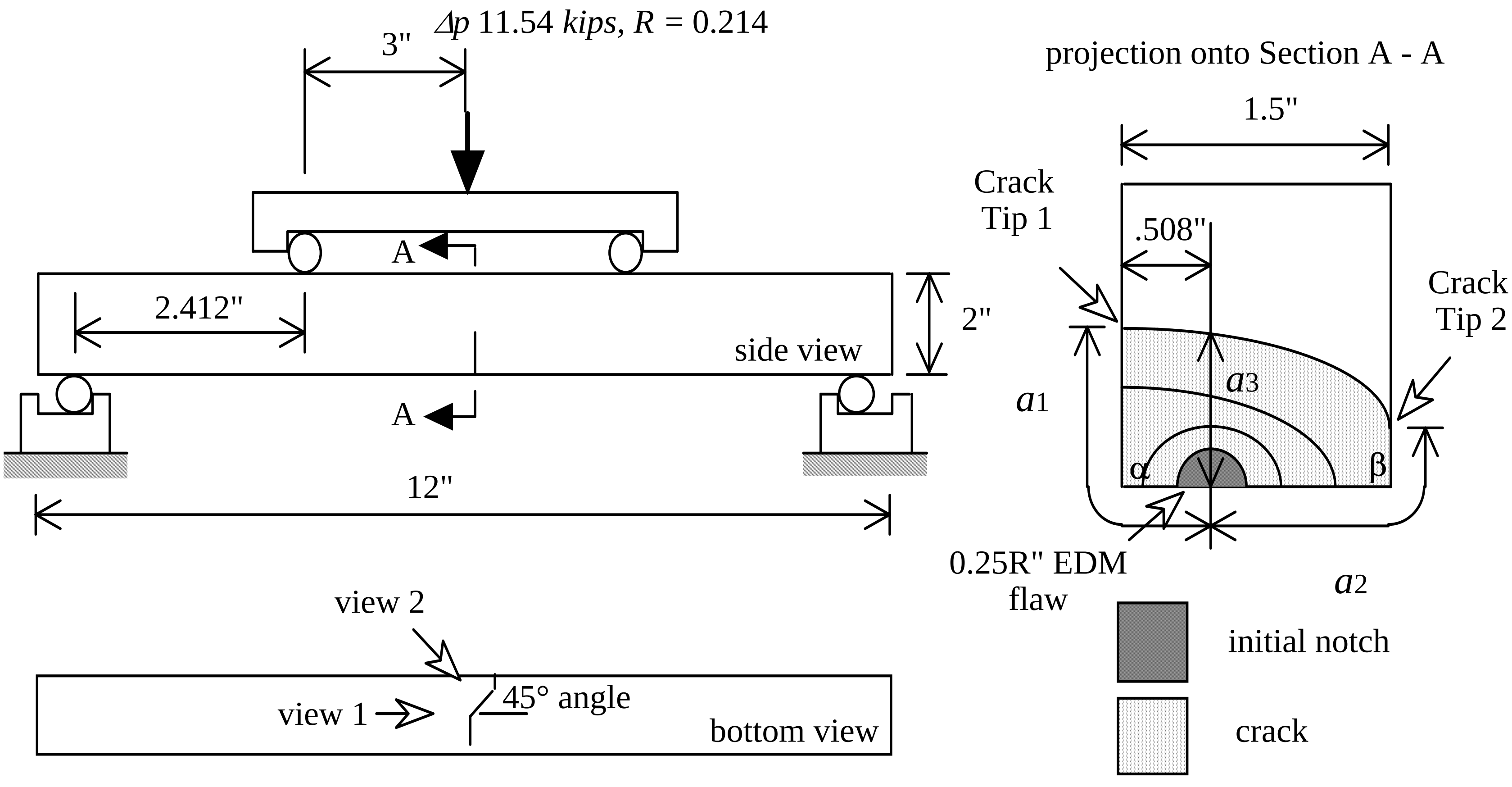
*.bes	a formatted text file - BES input file containing nodal coordinates, element connectivity, boundary conditions, and material properties.
coef.dat	an unformatted binary file containing the BES coefficient matrix. This file is created when running the out-of-core version of the BES solver and the 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:

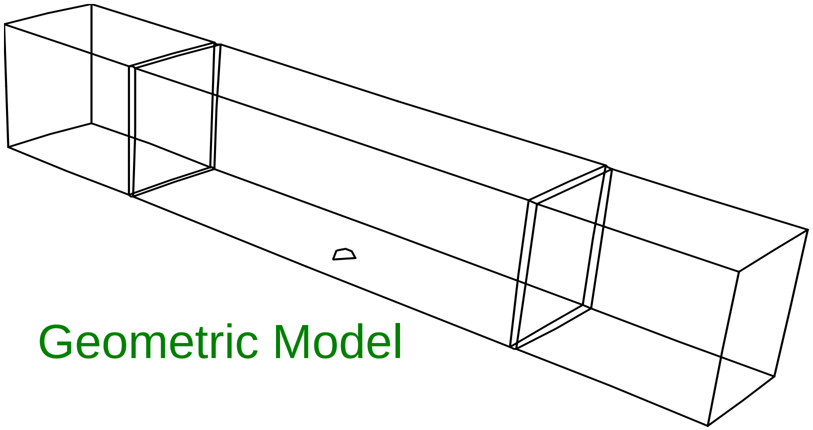
*.cdb	a formatted text file of the model data, including nodes, elements, etc
*.igs	an IGES formatted text file describing the model geometry
*.dsp	listing of nodal displacements
*.str	listing of nodal stress components
*.tmp	listing of nodal temperatures
*.eps	listing of nodal plastic strain

Simple Example To Illustrate Concepts

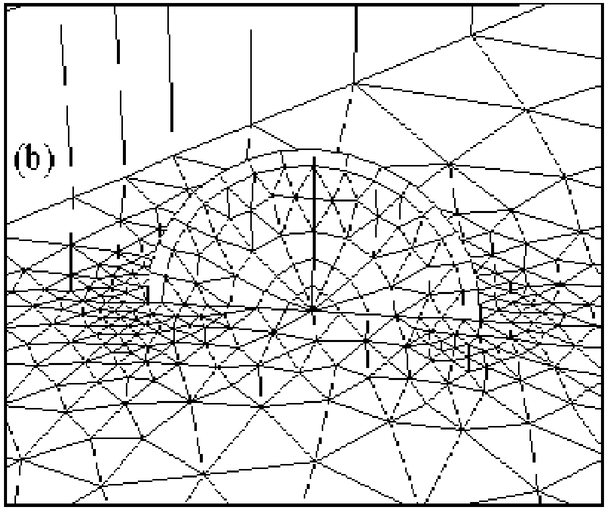
2219-T851 aluminum beam laboratory test specimen in four-point bending



Initial Boundary Element Mesh

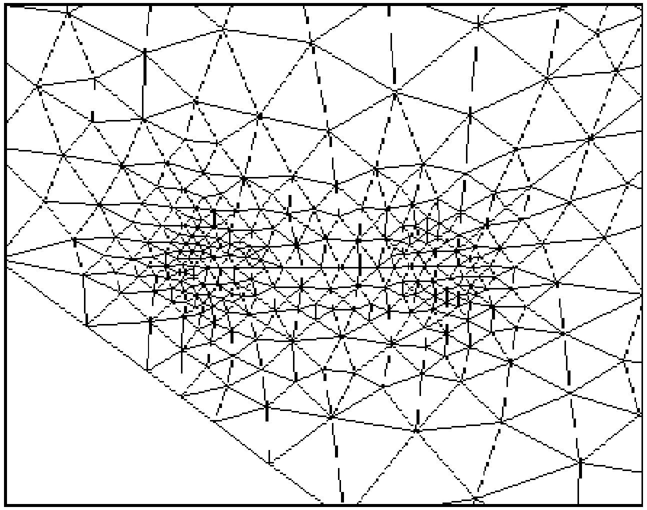


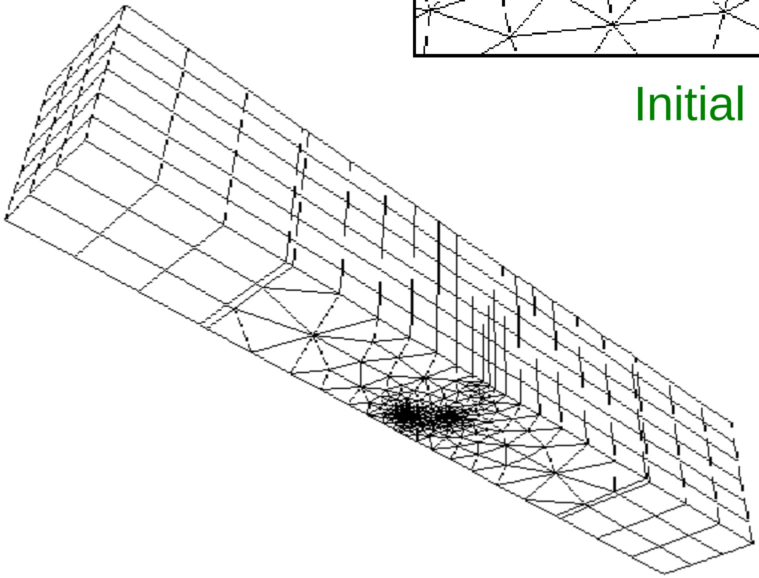
Geometric Model



Initial Flaw

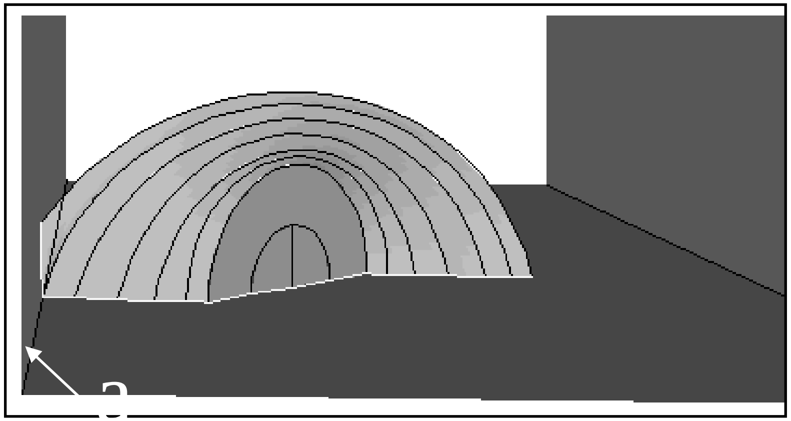
Flaw Mouth





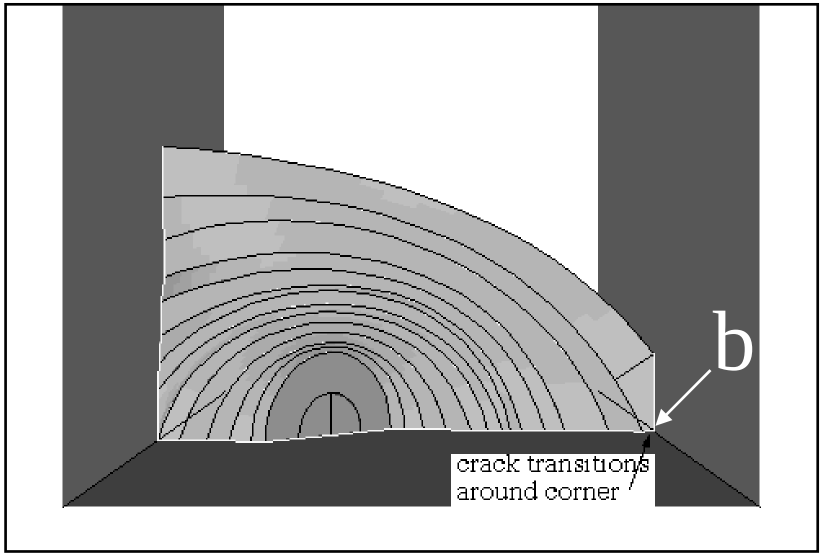
Full Beam

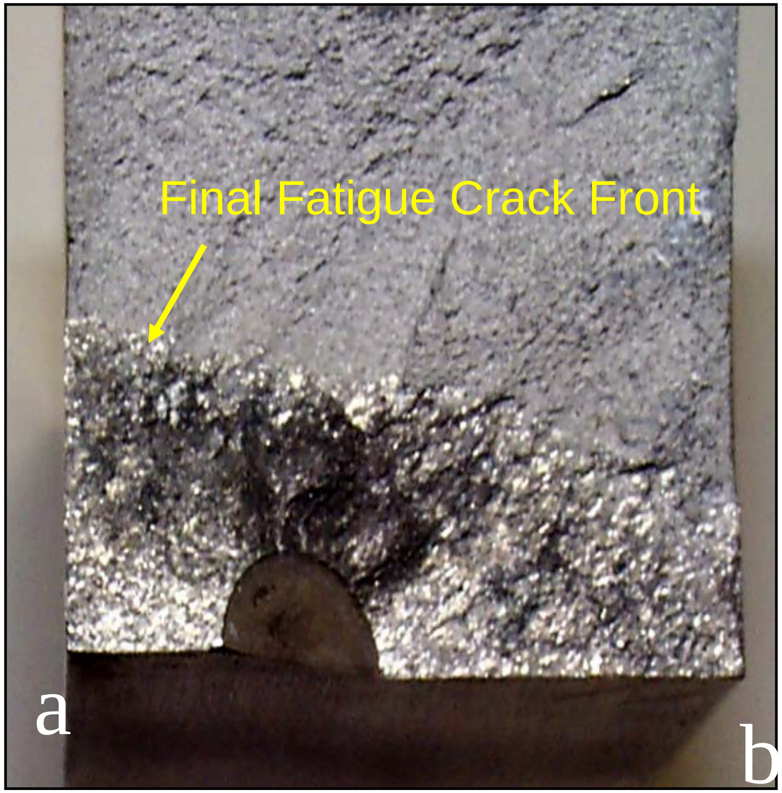
Predicted and Observed Crack Fronts



Predicted Transition into Corner "a"

Predicted Transition into Corner "b"

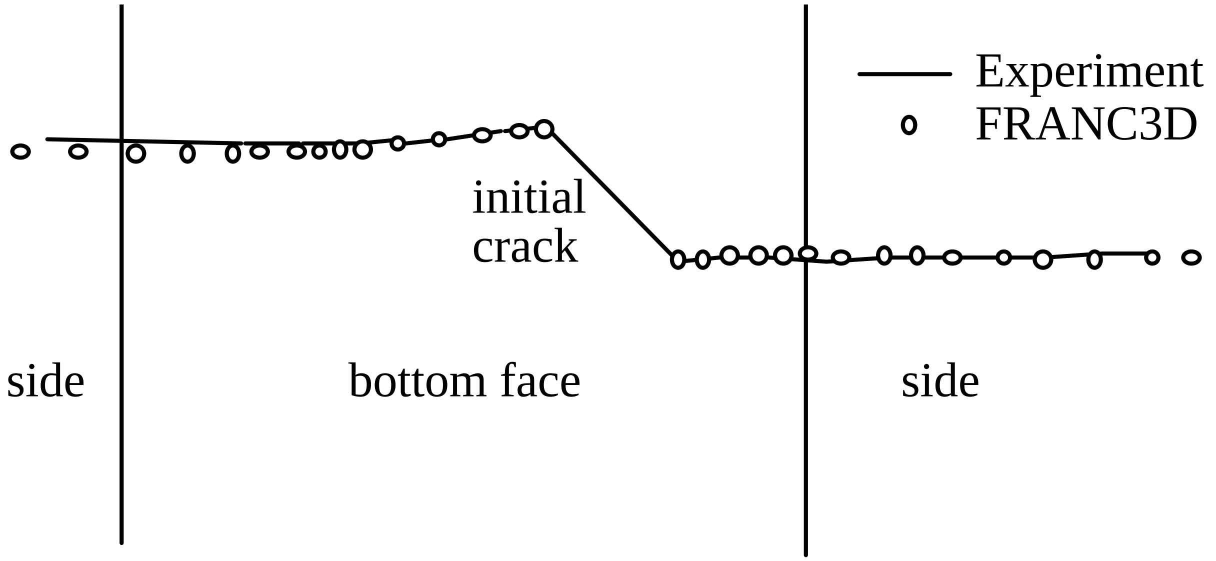




Observed Fatigue Crack Shape

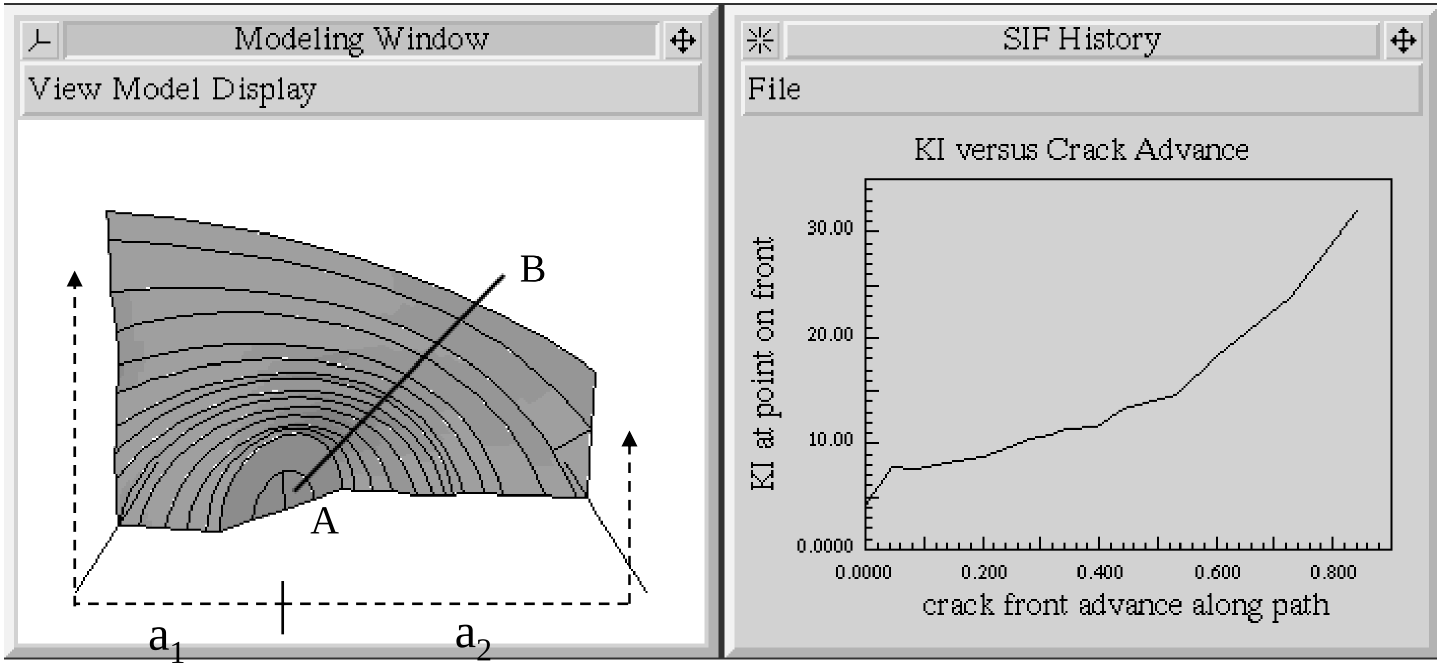
Final Predicted Crack Shape

Deformed solid



Comparison of computed and observed crack trace on the surface

Computed SIF History and Comparison to Observations



Event	**Observed**	FRANC3D	% Difference
a-Corner (cycles)	**106,800**	140,000	32
b-Corner (cycles)	**171,000**	170,000	-0.5
Last Front (cycles)	**175,000**	190,000	8.5
a_1 (in.)	**1.26**	1.42	12.7
a_2 (in.)	**1.38**	1.34	-2.8

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.