

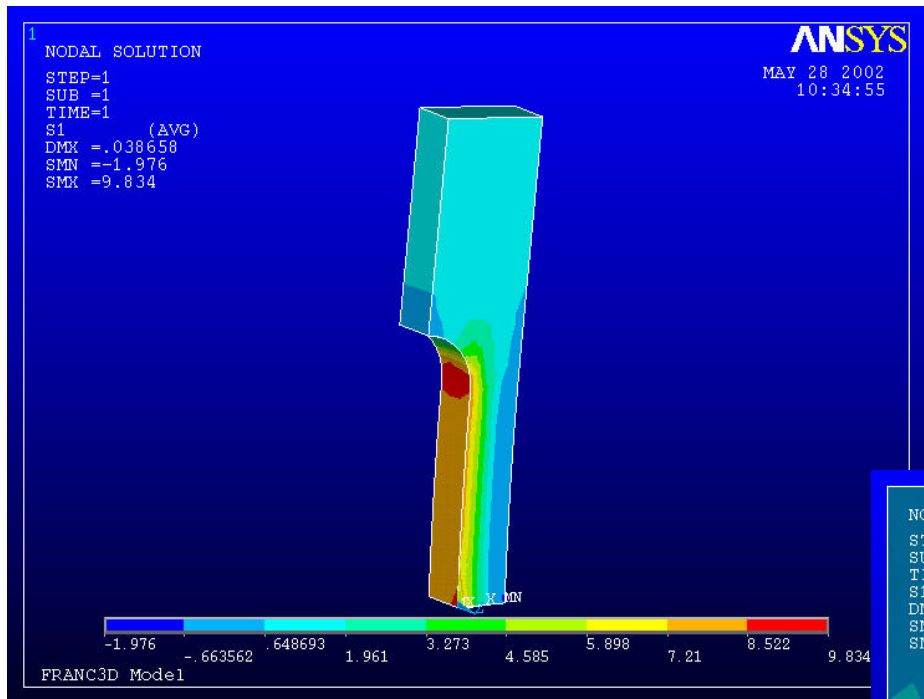
FRANC3D / OSM

Tutorial Slides

October, 2003

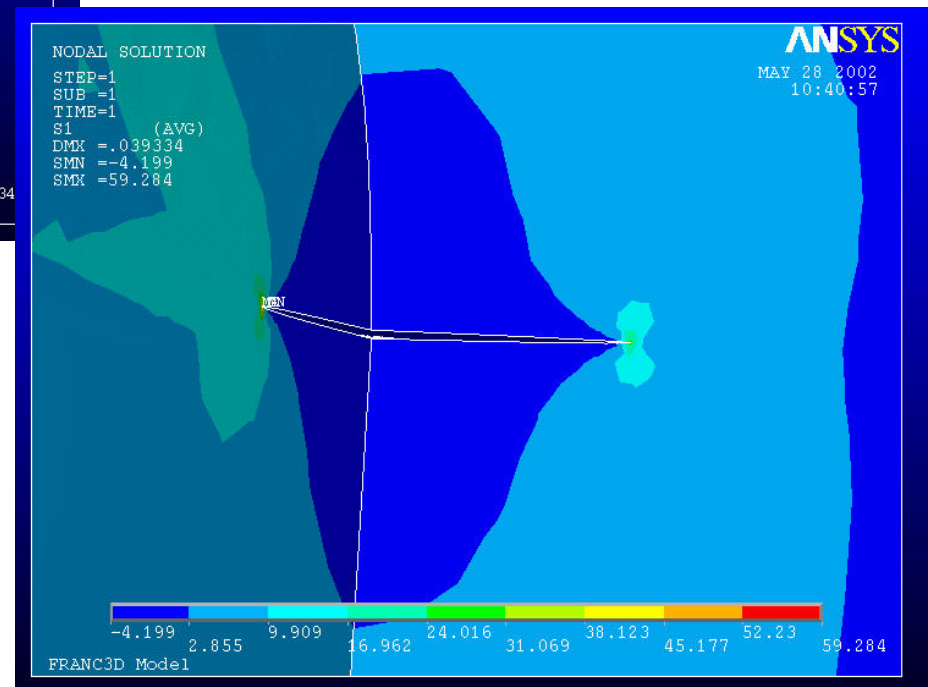
Cornell Fracture Group

Tutorial Example – Hands-on-training

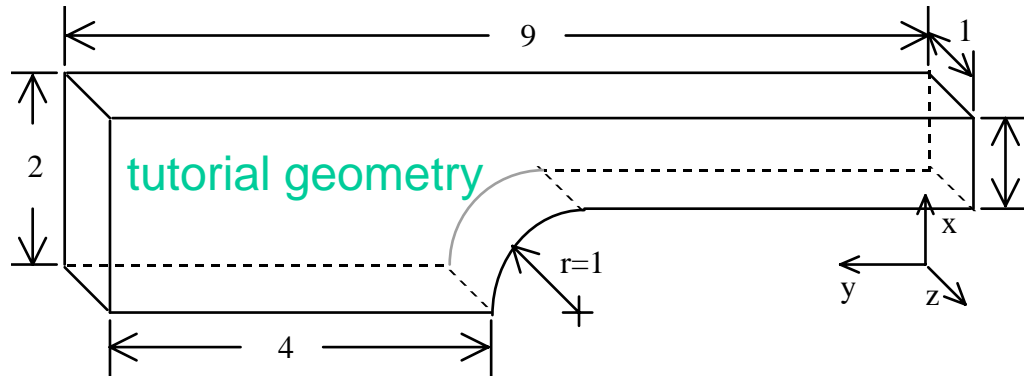


- learn how to use OSM to build simple models
- learn how to do uncracked stress analysis using FRANC3D
- learn how to nucleate a crack in FRANC3D

- learn how to analyze and propagate a crack using FRANC3D
- learn how to do FEM and BEM analyses
- compare advantages and disadvantages of BEM and FEM



FRANC3D / OSM 3D Tutorial



Steps:

- build model using OSM (or use ANSYS and then use OSM to convert it)
- read model into FRANC3D and analyze without crack using BES or ANSYS
- add a crack and analyze again using BES or ANSYS
- grow the crack for several steps
- compare stress intensity factors using BES versus ANSYS
- create stress intensity factor histories and predict fatigue life

FRANC3D / OSM 3D Tutorial

Advanced Steps:

- try automatic propagation analysis using BES
(this will be possible with ANSYS in the future)

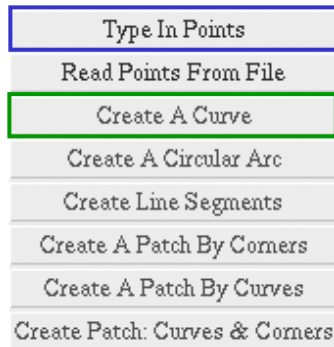
```
franc3d -b -f tutorial.fys -c crack_growth_model > junk &
```

- read the ANSYS cdb file created by FRANC3D into OSM and extract the geometry features or create the model in ANSYS and use OSM to convert
- if results are available from ANSYS discuss MRP's generated by OSM for FRANC3D and how MRP's are attached and used

FRANC3D / OSM 3D Tutorial



- display an empty modeling window



- generate a set of key points in the modeling window

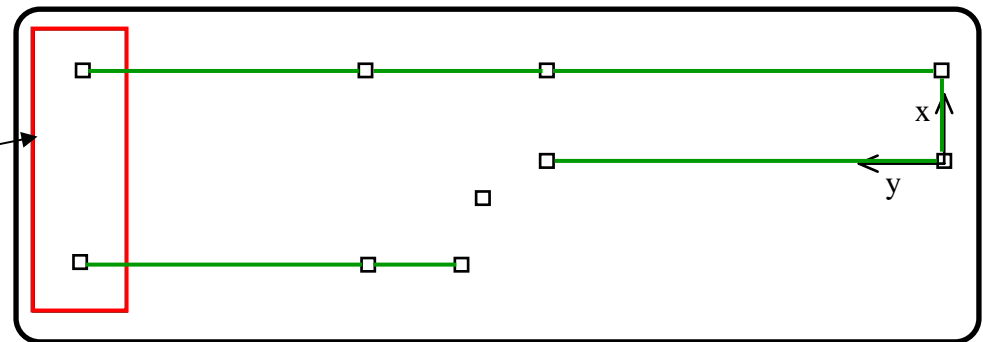
- change the view in the modeling window and **Create A Curve** for all the straight line edges



point coordinates

x	y	z
0	0	0
1	0	0
0	4	0
1	4	0
-0.2929	4.7071	0
-1	5	0
-1	6	0
1	6	0
-1	9	0
1	9	0

drag box



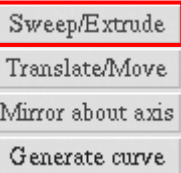
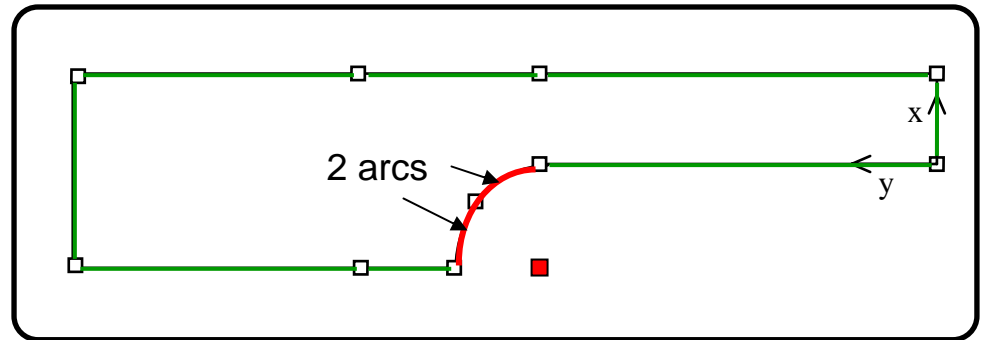
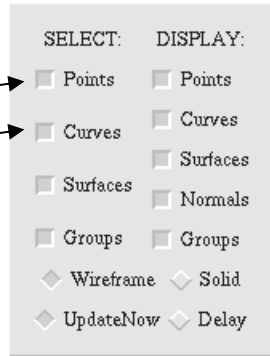
drag the mouse to select 2 points or click on the points while holding the shift key

FRANC3D / OSM 3D Tutorial

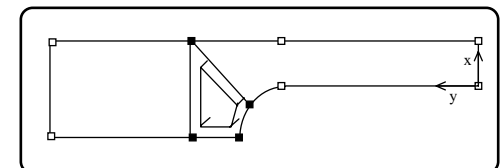
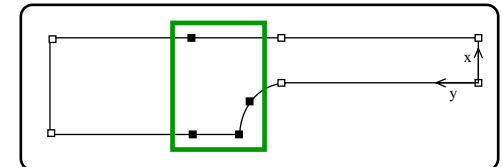
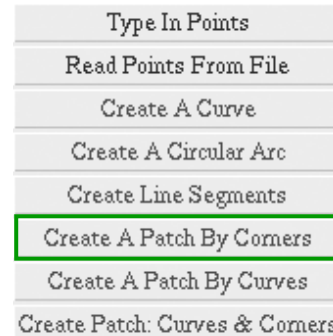
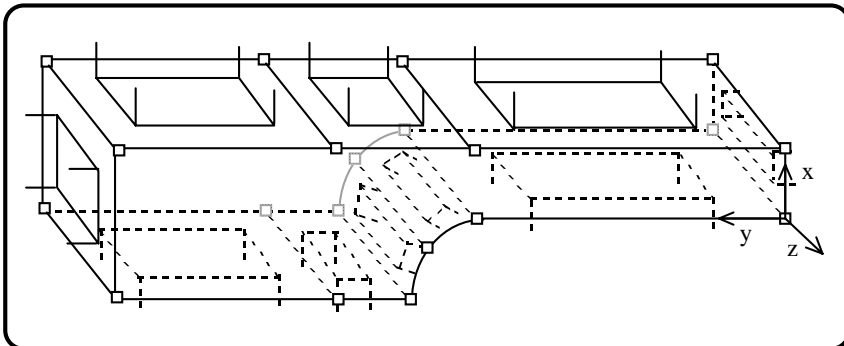
- add an extra point: **-1, 4, 0**
(make sure to clear the previous points from the dialog box)
- **Create A Circular Arc** using two arc points and the center point just added



Select All
points
curves

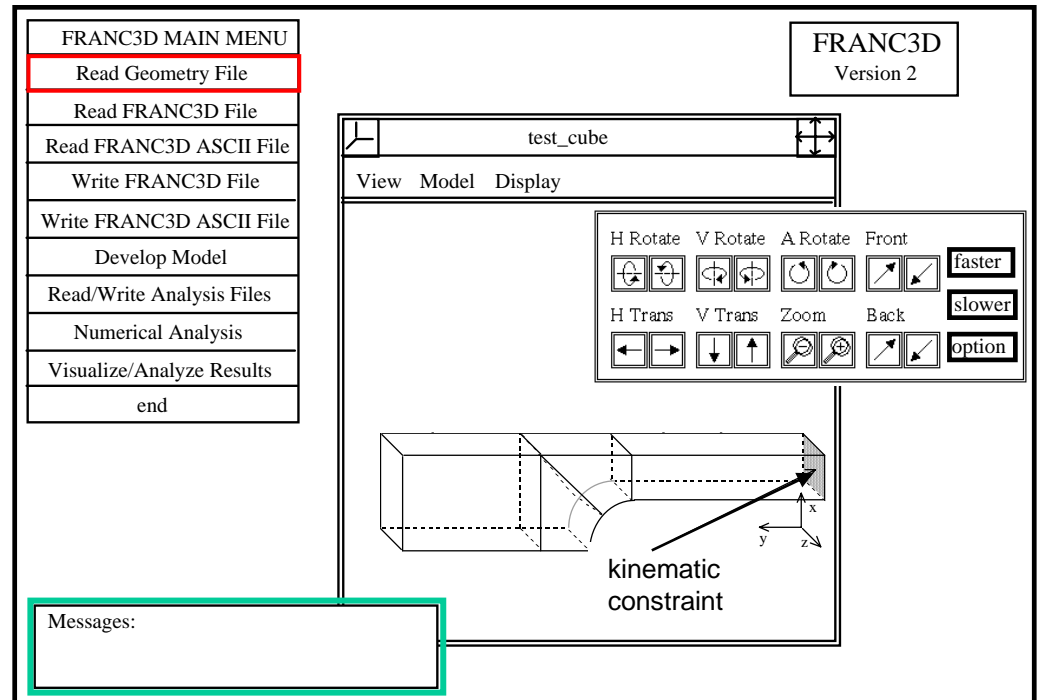
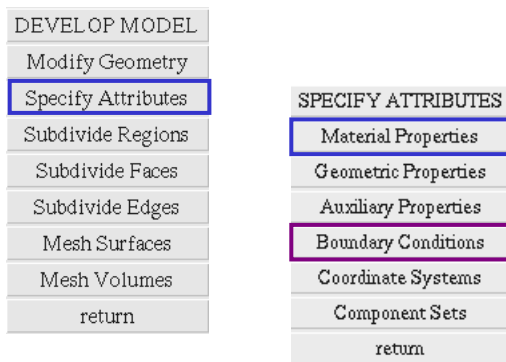


- collect all curves and points except the arc center and extrude in the Cartesian Z-direction to -1.0 and then ensure that all surface normals point outward and **add the missing faces**



FRANC3D / OSM 3D Tutorial

- save the OSM restart file and save the FRANC3D solid model file
- read the solid model file (**Geometry File**) into FRANC3D (if there are no error messages in the **Message/Information window** – the model is correct)
- select **Develop Model** and then **Specify Attributes**



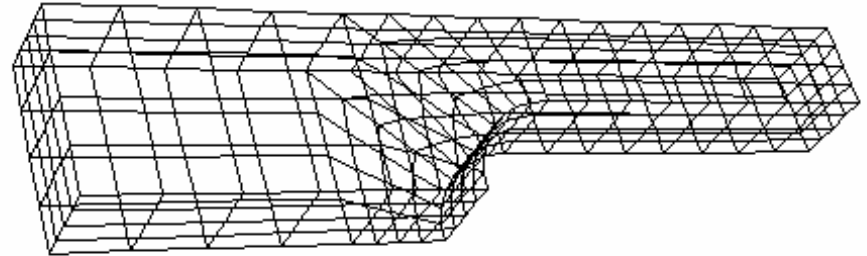
- set linear elastic material properties:
 $E=10,000$ and $\nu=0.33$

- create 2 face boundary conditions:
 - x,y,z displacements set to 0.0
 - y traction set to 1.0

FRANC3D / OSM 3D Tutorial

- discretize the model

- subdivide the edges
- mesh the surfaces
- mesh the volume (for FEM analysis only)
Use all triangular surface mesh and all tetrahedral volume mesh.



initial coarse surface mesh

- stress analysis

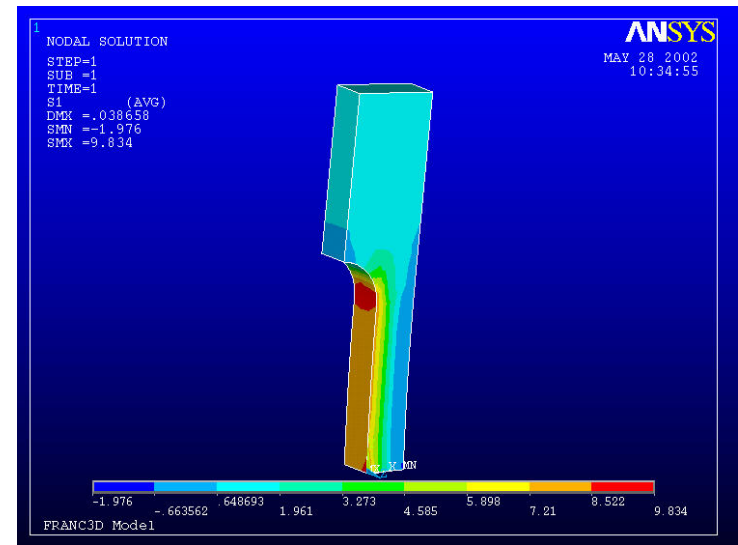
- for BEM, write the linear BES file and run (cge_bes -file tutorial.bes)
- for FEM, write the quadratic ANSYS file and run

- import results

- for BES, read tutorial.l.besout & tutorial.l.con
- for ANSYS, read the .dsp and .str files created by doing a nodal listing in ANSYS and saving file

- visualize results

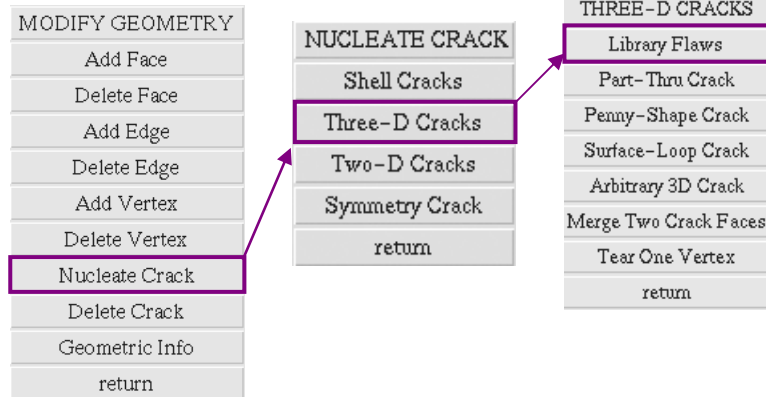
- for BES, use FRANC3D to display deformed shape and stress contours
- for ANSYS, use either ANSYS or FRANC3D to display results



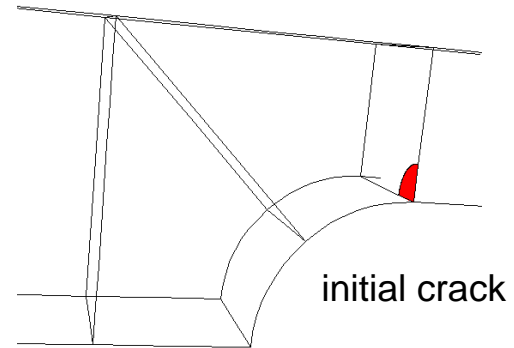
ANSYS post processor display

FRANC3D / OSM 3D Tutorial

- nucleate a crack in the model



show library of flaws
read flaw from file
show flaw in model
create flaw in model
cancel



- set values for **a** and **b** both to **0.25**
- set Rotations: Theta XY (deg) to **90**
- set **Translations: Along Y Axis to 4**
- select **Calculate** to compute the point locations (displayed in the model as red boxes).
- select **Accept** if the crack is located correctly

Num Points: 16

a: 0.25
b: 0.25

Curve fit: bsp p2 p3 p4

Rotations:

Theta XY (deg): 90
Theta XZ (deg): 0
Theta YZ (deg): 0

Translations:

Along X Axis: 0
Along Y Axis: 4
Along Z Axis: 0

Front Points:

	X	Y	Z
1			
2			
3			
4			
5			
6			
7			
8			
9			
10			
11			

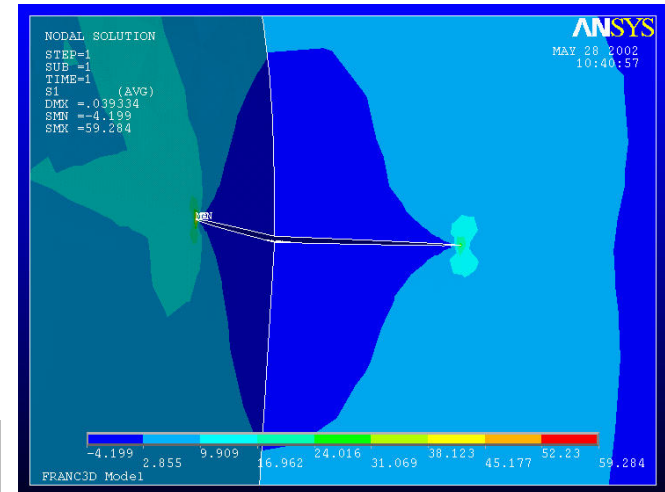
Edge Points:

	X	Y	Z
1			
2			

Select Shape **Calculate** Write File **Accept** Cancel

FRANC3D / OSM 3D Tutorial

- discretize the cracked model
 - subdivide the edges around the crack
 - mesh the surfaces around the crack
 - mesh the volume (for FEM analysis only)
- stress analysis, import results, visualize results
- compute fracture parameters and grow the crack



ANSYS post processor display of crack mouth

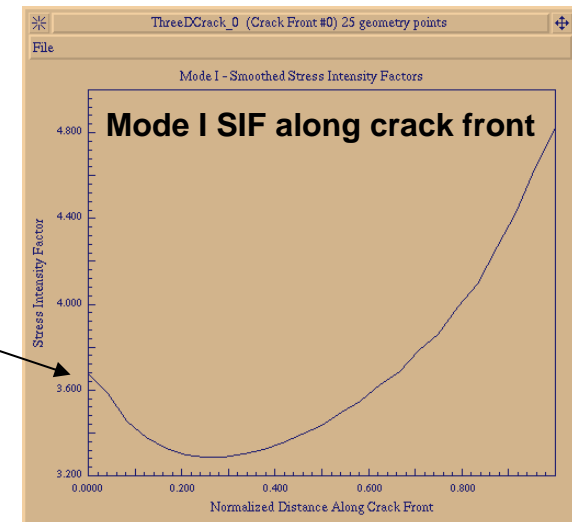
VISUALIZE/ANALYZE RESULTS
Deformation & Contour
Surface Line Plot
Point Information
3D Fracture Analysis
2D Fracture Analysis
Fracture Initiation
return

STRESS INTENSITY FACTORS	
<input type="checkbox"/> points on geometry edges	number of points
<input type="checkbox"/> points along mesh edges	<input type="text" value="24"/>
<input type="checkbox"/> plot SIF	<input type="checkbox"/> plot smoothed SIF
<input type="checkbox"/> print average SIFs	<input type="checkbox"/> save SIF history
<input type="button" value="Accept"/> <input type="button" value="Cancel"/>	

3D FRACTURE ANALYSIS
Stress Intensity Factor
Orthotropic SIF's
FEM J-Integral
Propagate Selected Crack
Propagate All Cracks
Show SIF History
Predict Fatigue Life
File Crack Info
Crack Description
return

- compute stress intensity factors for selected crack front

- compute SIFs along the geometry edge of the crack front and plot the values



FRANC3D / OSM 3D Tutorial

propagate the crack

- determine new front points
- smooth new points by fitting to a polynomial
- add and tear faces, edges, vertices to grow the crack

3D FRACTURE ANALYSIS
Stress Intensity Factor
Orthotropic SIF's
FEM J-Integral
Propagate Selected Crack
Propagate All Cracks
Show SIF History
Predict Fatigue Life
File Crack Info
Crack Description
return

CRACK PROPAGATION MODELS

direction Sigma_Max G_Max S_Min Planar

extension power of K1 K1-K1c / K1max-K1c

Accept Cancel

Sigma_Max – maximum hoop stress criterion provides orientation

CRACK EXTENSION MODEL: POWER OF KI

extension = $\max_ext * (K1^b) / \text{MAX}(K1^b)$

maximum extension

value of b

Accept Cancel

Crack Extension Model – provides for an increase in crack 'length' ('b' – comparable to Paris Law parameter 'n')

POLYNOMIAL FITTING

single piecewise user defined

polynomial order

Accept Cancel

Points defining the new crack front are smoothed by fitting them to a polynomial.

determine new front points

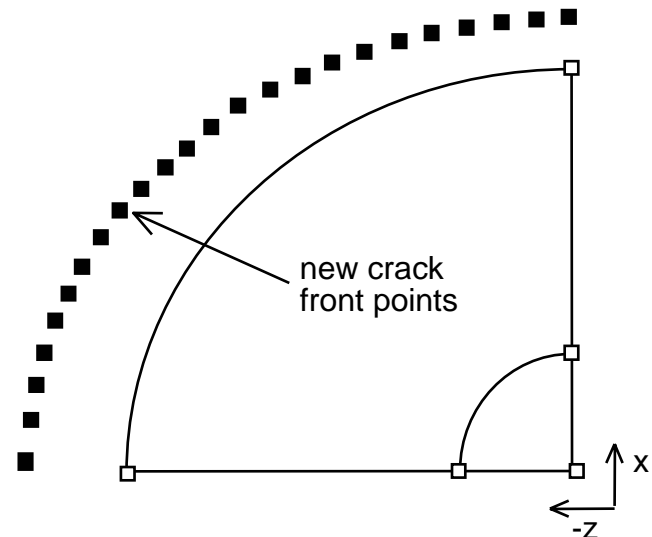
read crack increments from file

display new/fitted front points

add and tear edges and faces

cancel

Grow the crack geometry & topology.

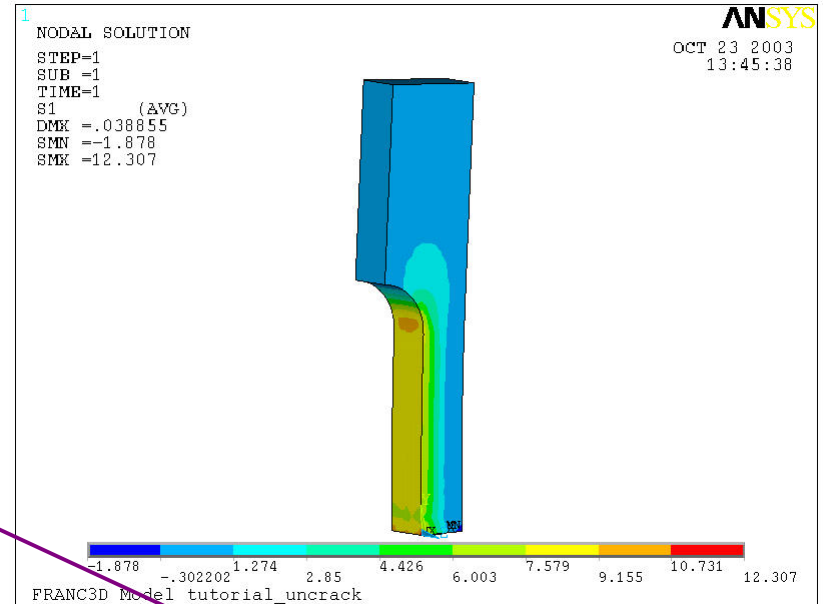


FRANC3D / OSM 3D Tutorial

- illustrate crack face traction MRP and superposition
 - ansys stress analysis of uncracked model
 - OSM conversion of .cdb and .str files to MRP
 - attach MRP to crack face for BEM analysis

- Create Radial-Edge Database
- Hiligh Geometry Points/Edges
- Extract Geometry Features
- Translate ANSYS .cdb**
- Translate ANSYS .igs

tutorial_uncrack.cdb and .str files converted by OSM creating: tutorial_uncrack_res_str.mrp



FRANC3D: define boundary conditions

- BOUNDARY CONDITIONS
- Define Load Case
- Define MRP BC's**
- Coupled Sets
- Contact Surfaces
- New Boundary Condition
- Edit Boundary Condition
- Delete Boundary Condition
- Attach Boundary Condition
- Detach Boundary Condition
- Show Boundary Condition
- Hiligh Non-Default BC's
- Reset BC's to default values
- return

MRP: User Defined Distributed BC's

MRP name: tutorial_uncrack_res_str.mrp

Number of nodes: 14146

Number of elements: 9338

Number of BCs per node: 6

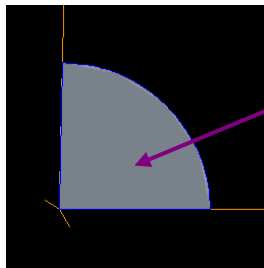
Number of BCs per element: 0

Define Nodes Define Elements

Define Node BCs Define Element BCs

interpolate in volume extrapolate to surface

Import Export Accept Cancel



attach face boundary condition to crack face

BOUNDARY CONDITIONS FOR A FACE

BC Name: crack_face Layer: 1

Coord Sys: default global sys global Cartesian surface local user defined

Load Case: default load case MRP BC: tutorial_uncrack_res_str.mrp

x (global or user Cartesian [green]) or n (surface local [red])

displacement traction 0 rotation moment 0

y (global or user Cartesian [blue]) or u (surface local [green])

displacement traction 0 rotation moment 0

z (global or user Cartesian [red]) or v (surface local [blue])

displacement traction 0 rotation moment 0

Accept Prev Layer Next Layer Prev Load Case Next Load Case Cancel

FRANC3D / OSM 3D Tutorial

- after propagating the crack

- discretize again
- stress analysis again
- post process again
- propagate again

- fatigue life prediction

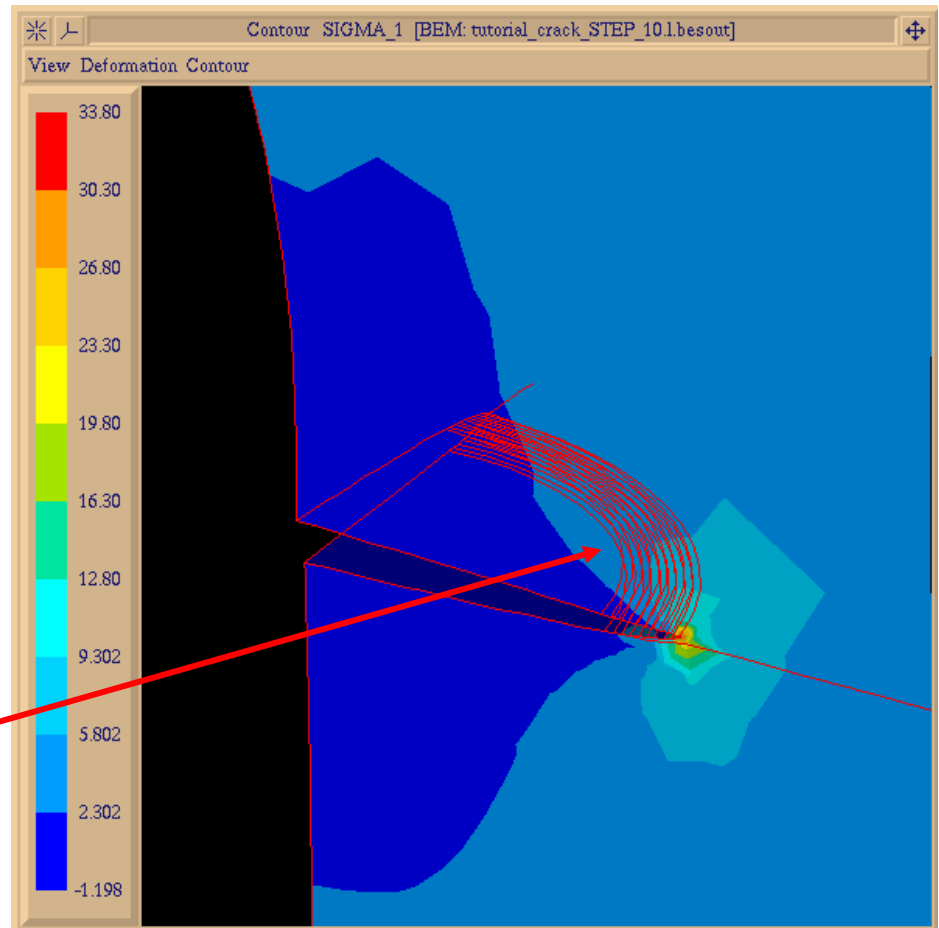
- stress intensity factor history
- FRANC3D computes fatigue life or use an external code

- risk assessment or redesign

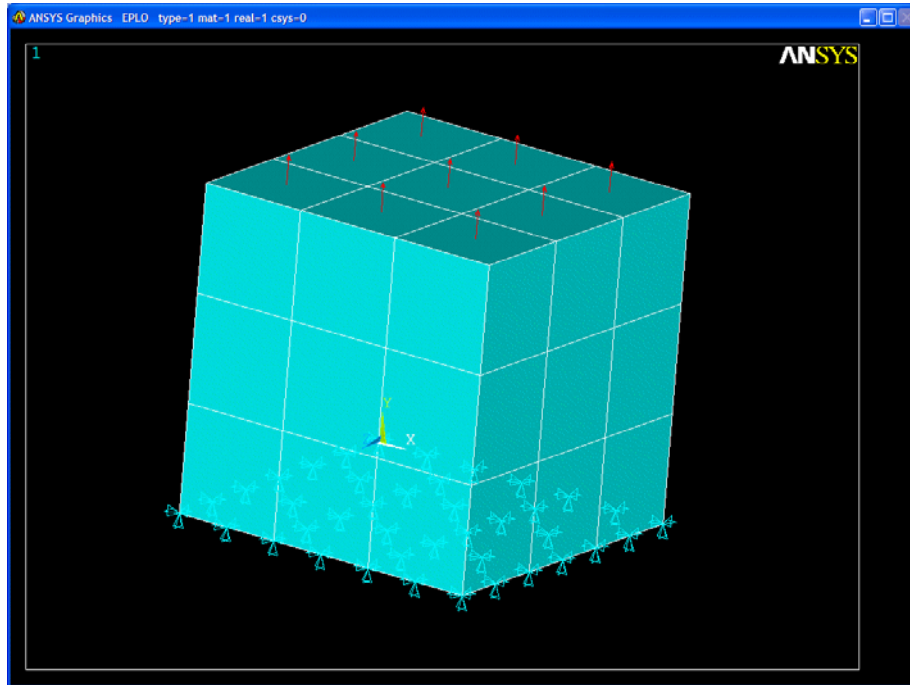
- automated analyses

- for BES only (FEM in the future)
- `franc3d -b -f tutorial_crack.fys -c crack_growth_model`

10 steps of crack growth

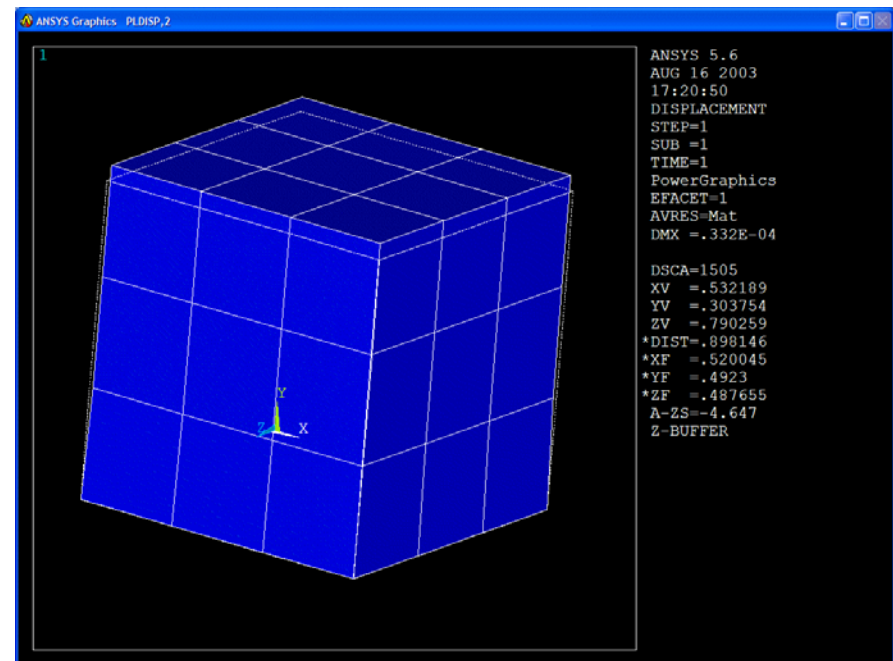


Simple Cube Example – Hands-on-training



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

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

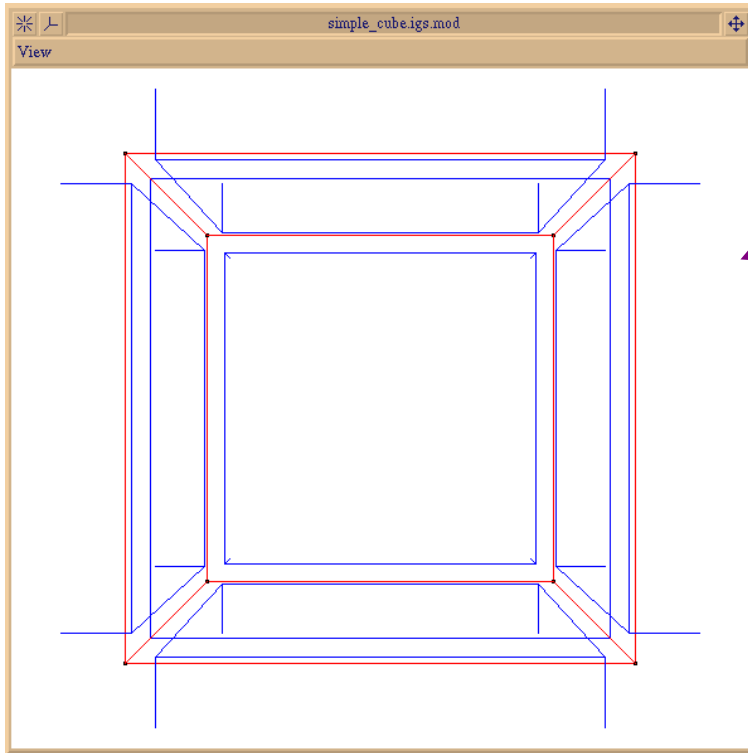


Simple Cube Example

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

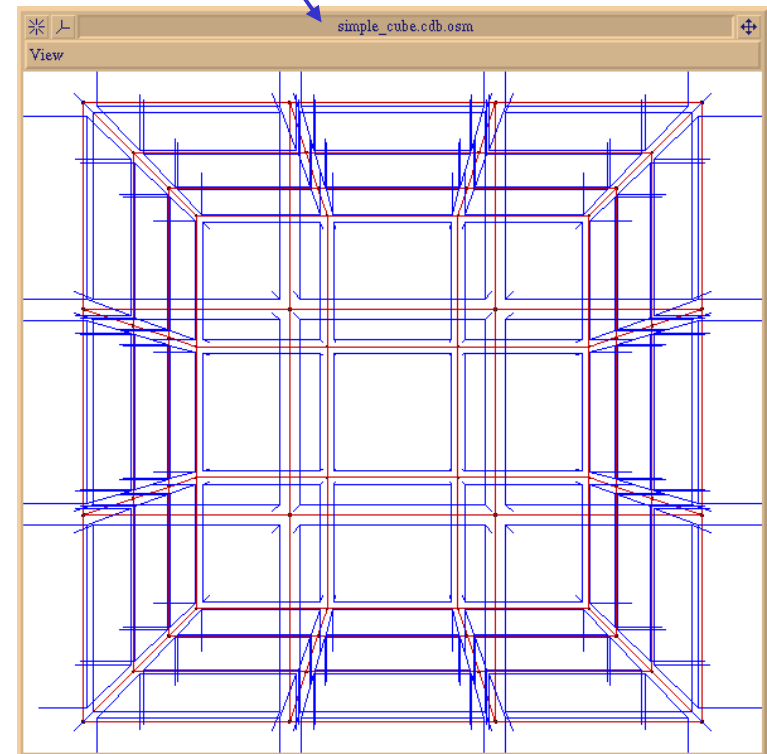


iges file contains only geometry
and if a solid model does not exist
there is no iges information

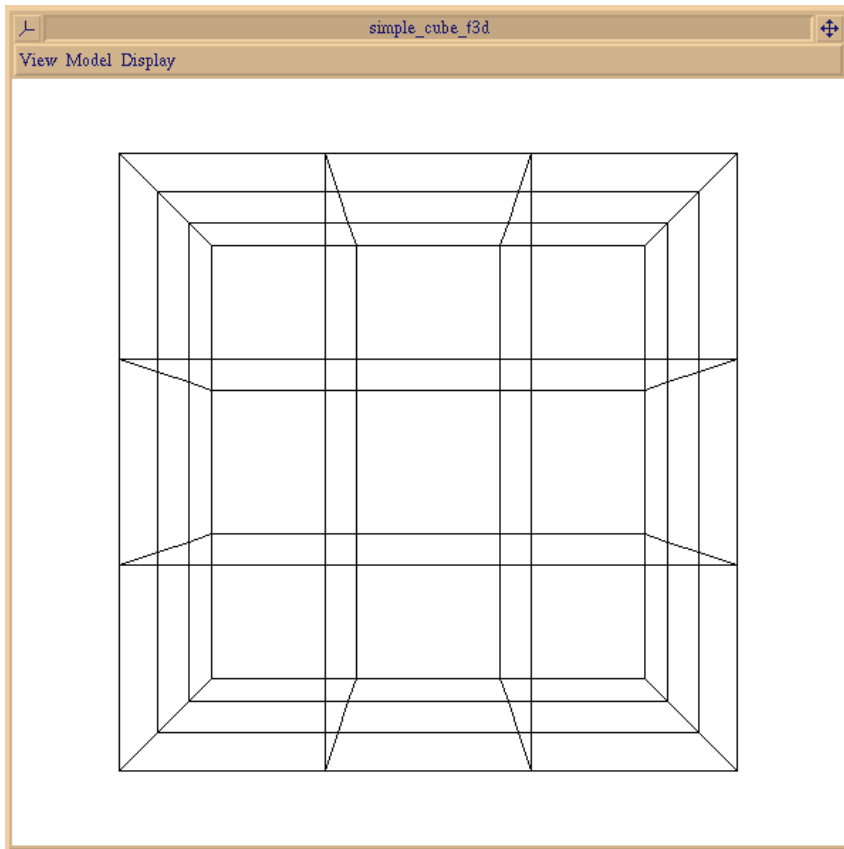
cdb file contains finite element
mesh information along with
boundary conditions and materials

- Create Radial-Edge Database
- Highlight Geometry Points/Edges
- Extract Geometry Features
- Translate ANSYS .cdb
- Translate ANSYS .igs

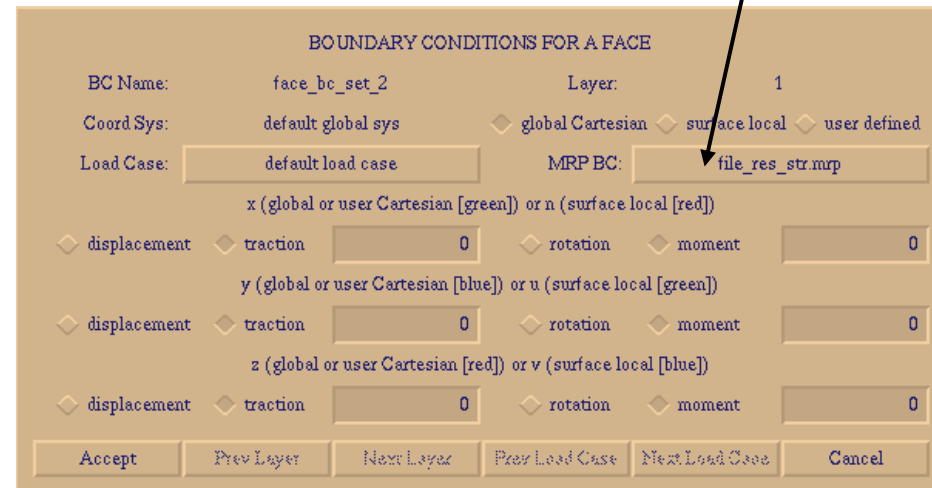
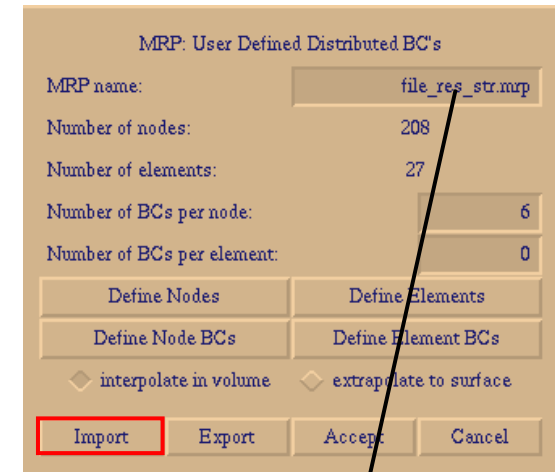
OSM can convert ANSYS
iges and cdb files



Simple Cube Example



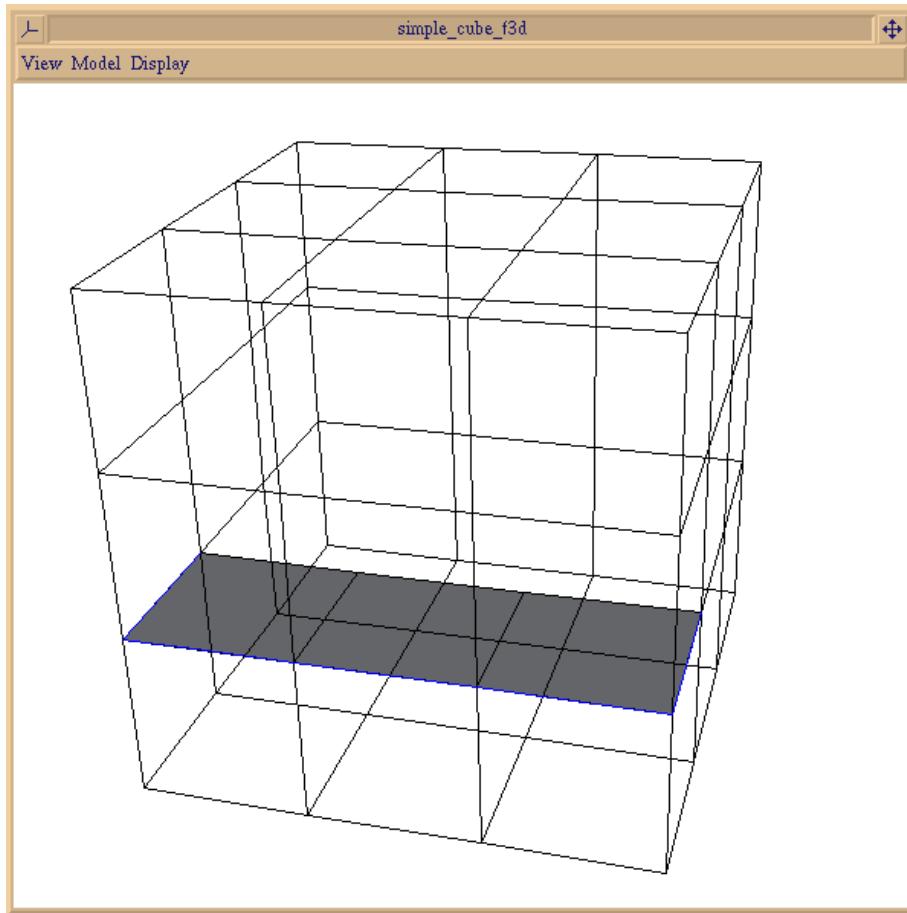
OSM reads the .str file and saves a _res_str.mrp file that contains the mesh information from the .cdb file as well as the stress results



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



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

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

