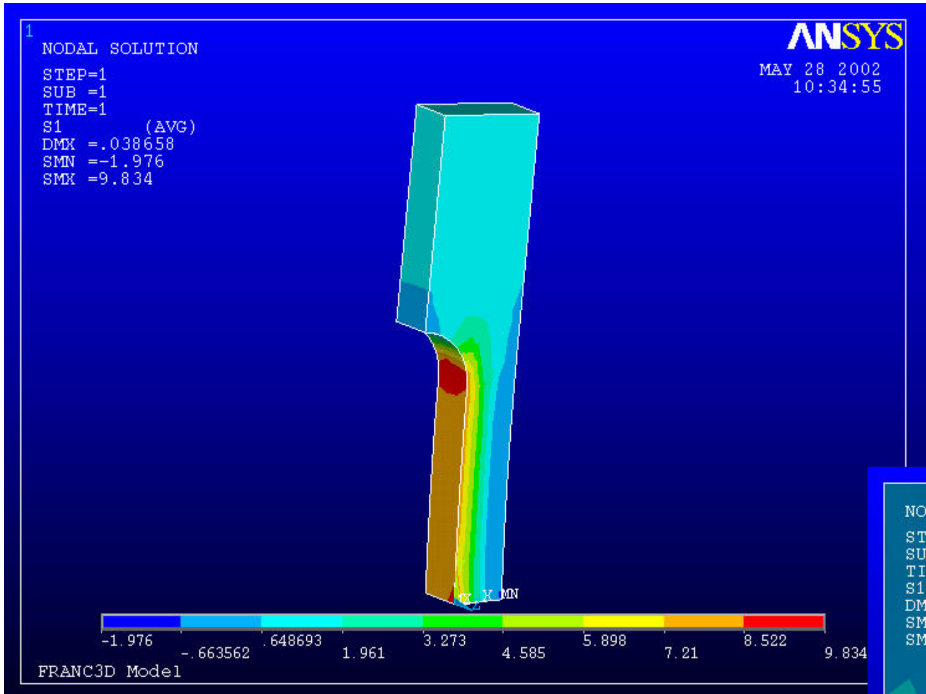
FRANC3D / OSM

Tutorial Slides

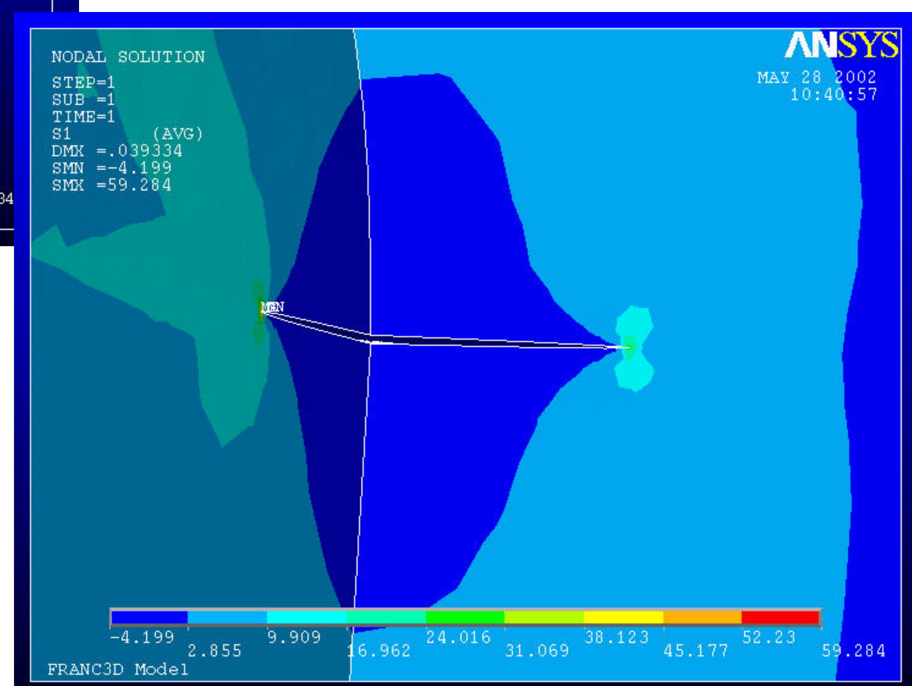
October, 2003

Cornell Fracture Group

Tutorial Example – Hands-on-training

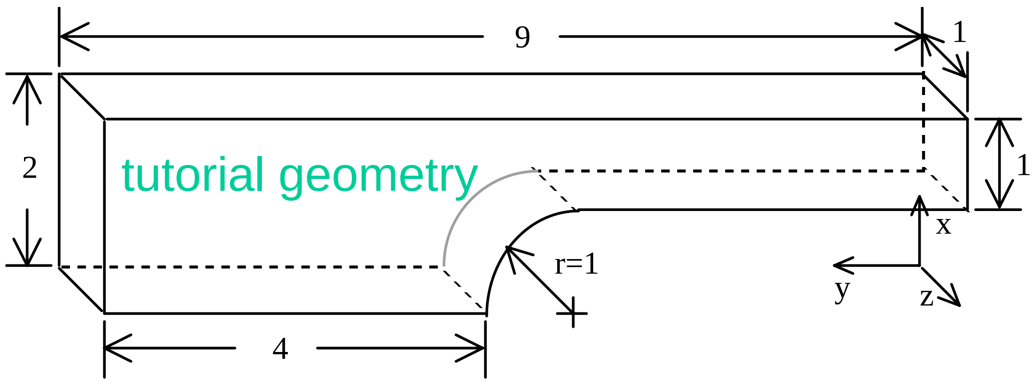


- learn how to use OSM to build simple models
- learn how to do uncracked stress analysis using FRANC3D
- learn how to nucleate a crack in FRANC3D



- 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)

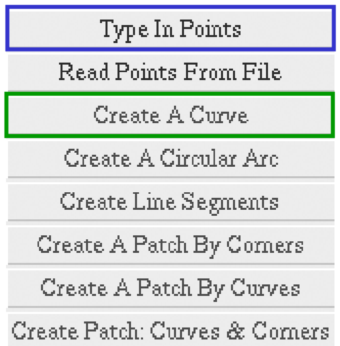
```
franc3d -b -f tutorial.fys -c crack_growth_model > junk &
```

• 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



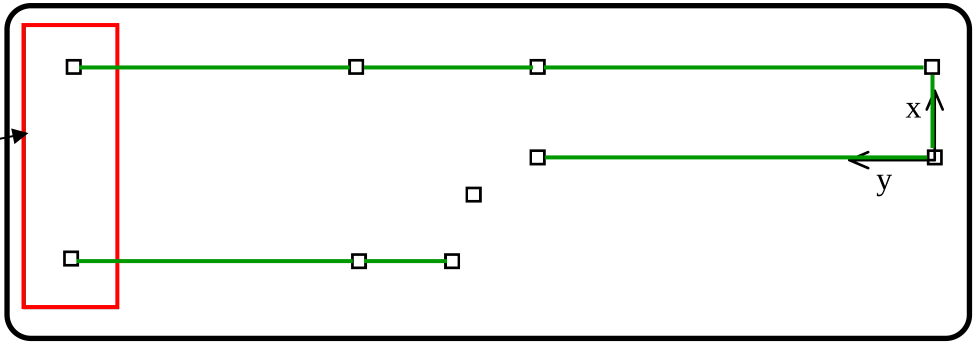
- generate a set of key points in the modeling window

- 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



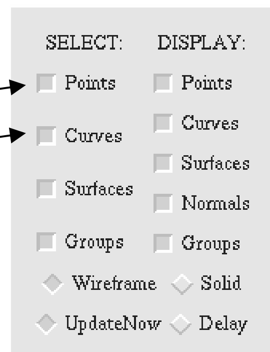
drag the mouse to select 2 points or click on the points while holding the shift key

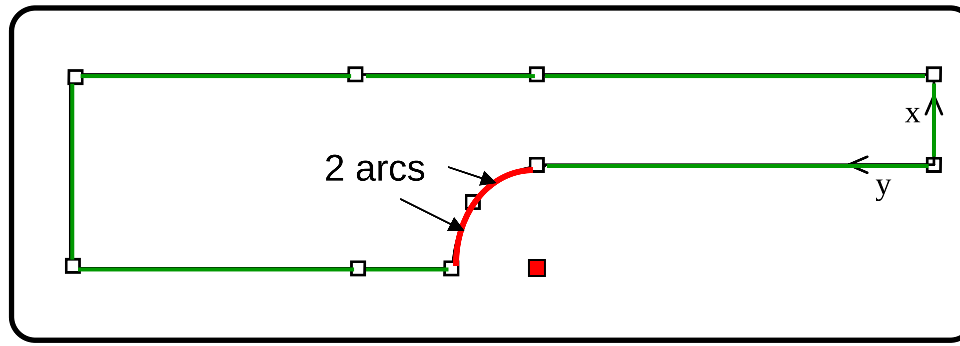
FRANC3D / OSM 3D Tutorial

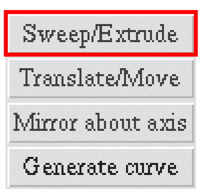
- add an extra point: -1, 4, 0
  (make sure to clear the previous points from the dialog box)

- Create A Circular Arc using two arc points and the center point just added

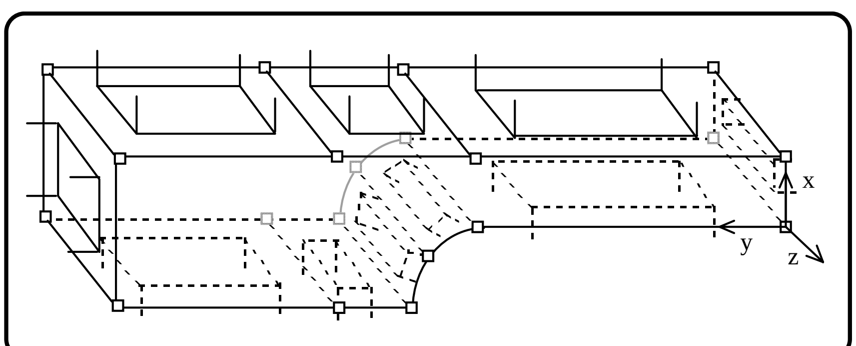
Select All
points
curves

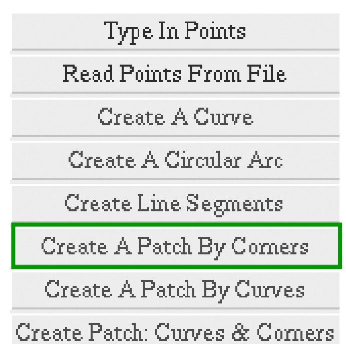


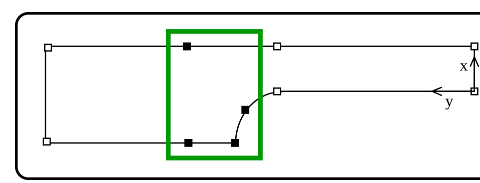


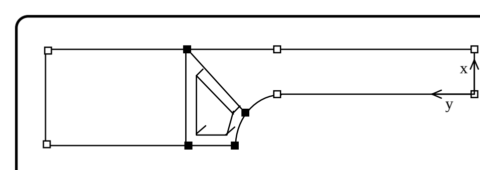


- collect all curves and points except the arc center and extrude in the Cartesian Z-direction to -1.0 and then ensure that all surface normals point outward and add the missing faces







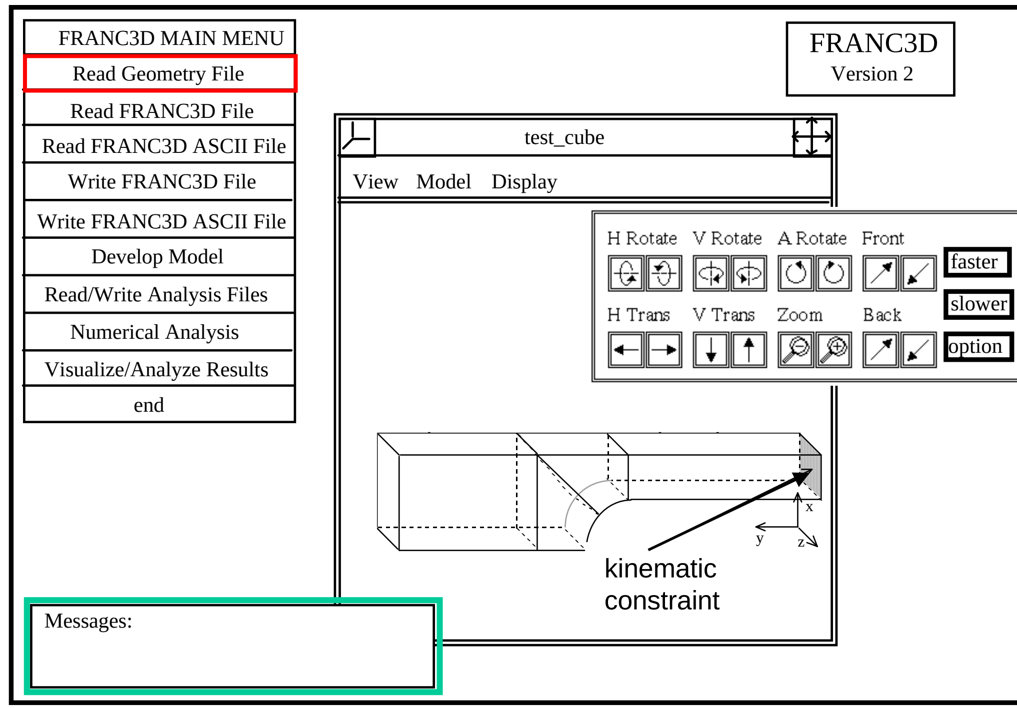


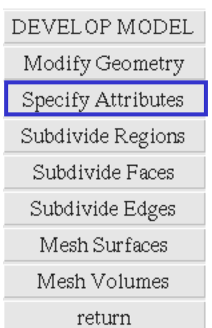
FRANC3D / OSM 3D Tutorial

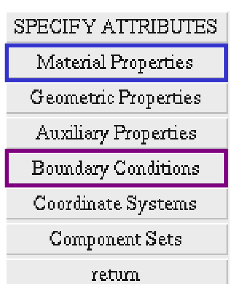
- save the OSM restart file and save the FRANC3D solid model file

- 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





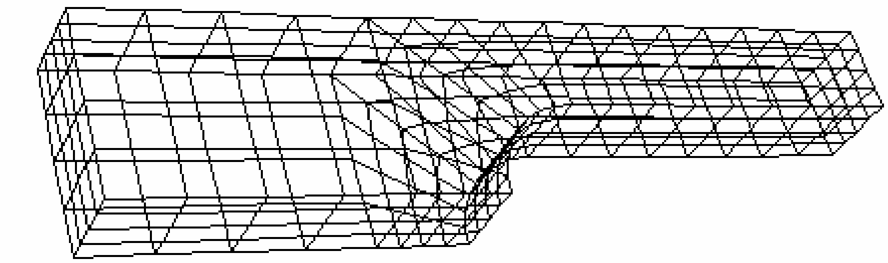


- set linear elastic material properties: E=10,000 and ν=0.33

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

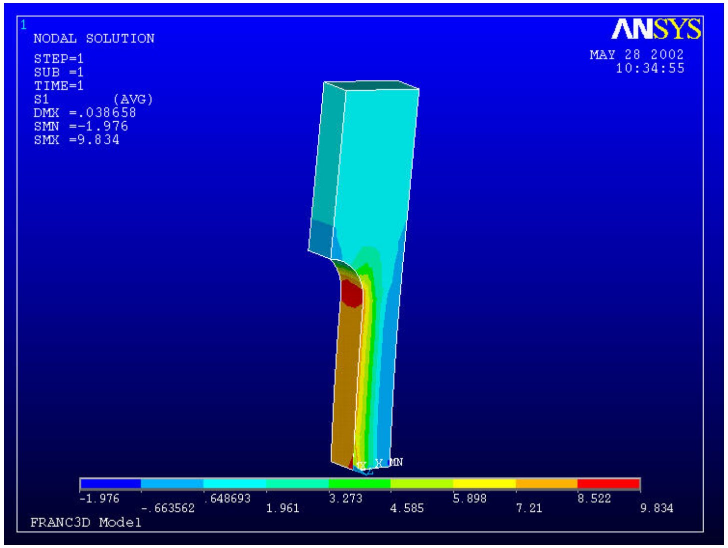
FRANC3D / OSM 3D Tutorial

- 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

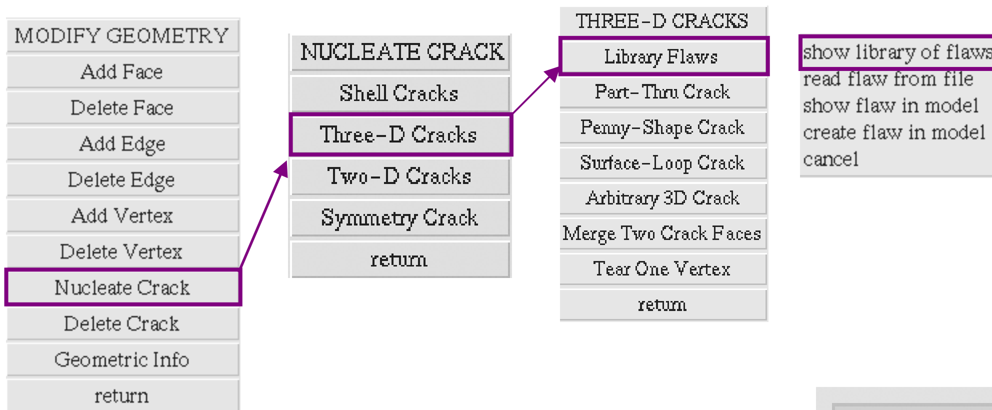


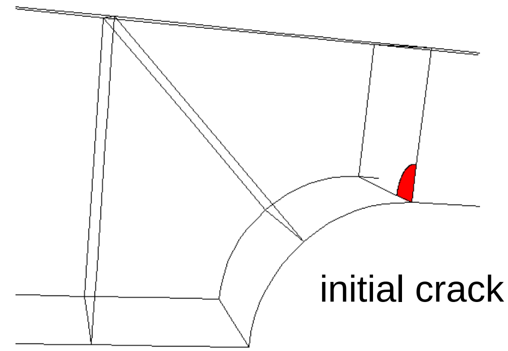
ANSYS post processor display

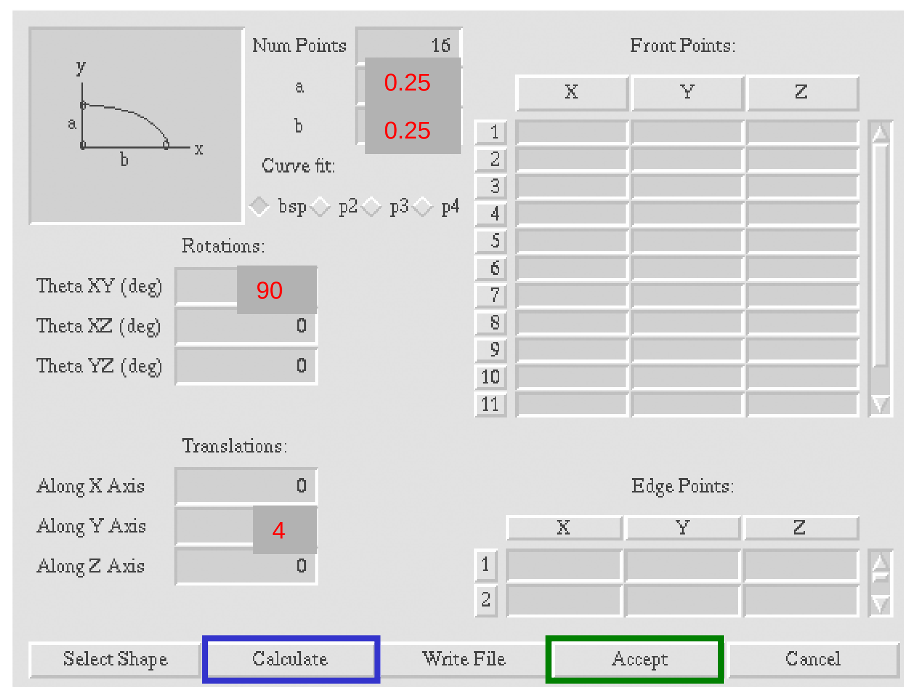
- import results
  - for BES, read tutorial.l.besout & tutorial.l.con
  - for ANSYS, read the .dsp and .str files created by doing a nodal listing in ANSYS and saving file
- visualize results
  - for BES, use FRANC3D to display deformed shape and stress contours
  - for ANSYS, use either ANSYS or FRANC3D to display results

FRANC3D / OSM 3D Tutorial

- nucleate a crack in the model



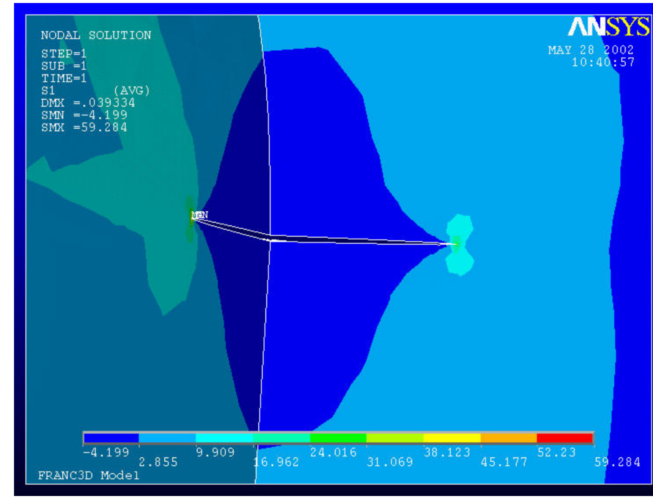




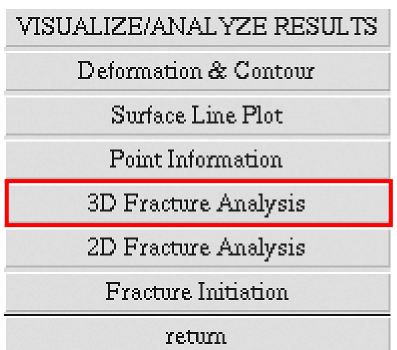
- set values for **a** and **b** both to 0.25
- set Rotations: Theta XY **(deg)** to 90
- set **Translations: Along Y Axis** to 4
- select **Calculate** to compute the point locations (displayed in the model as red boxes).
- select **Accept** if the crack is located correctly

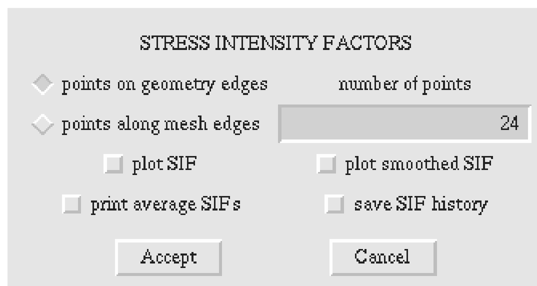
FRANC3D / OSM 3D Tutorial

- 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



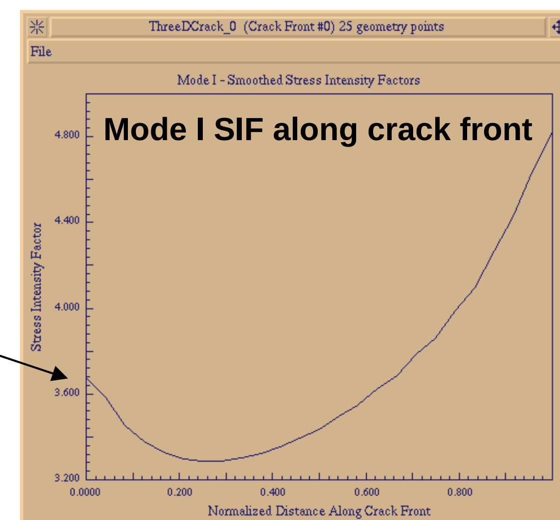
ANSYS post processor display of crack mouth





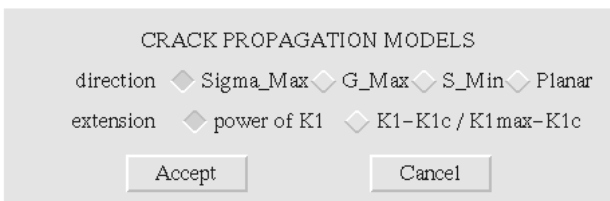
- compute stress intensity factors for selected crack front

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

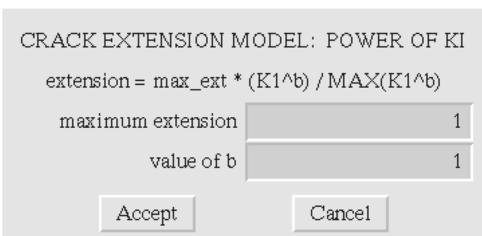


FRANC3D / OSM 3D Tutorial

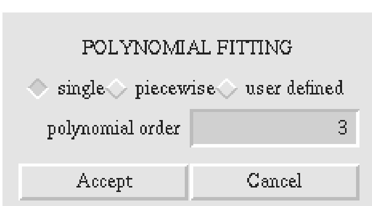
- propagate the crack
  - determine new front points
  - smooth new points by fitting to a polynomial
  - add and tear faces, edges, vertices to grow the crack



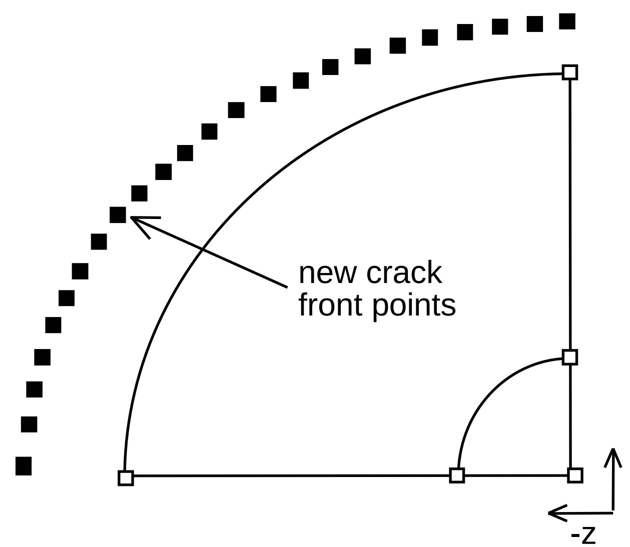
Sigma_Max – maximum hoop stress criterion provides orientation

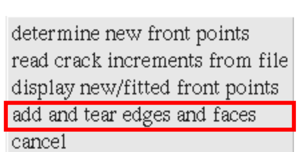


Crack Extension Model – provides for an increase in crack 'length' ('b' – comparable to Paris Law parameter 'n')



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

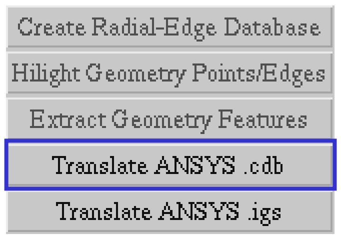




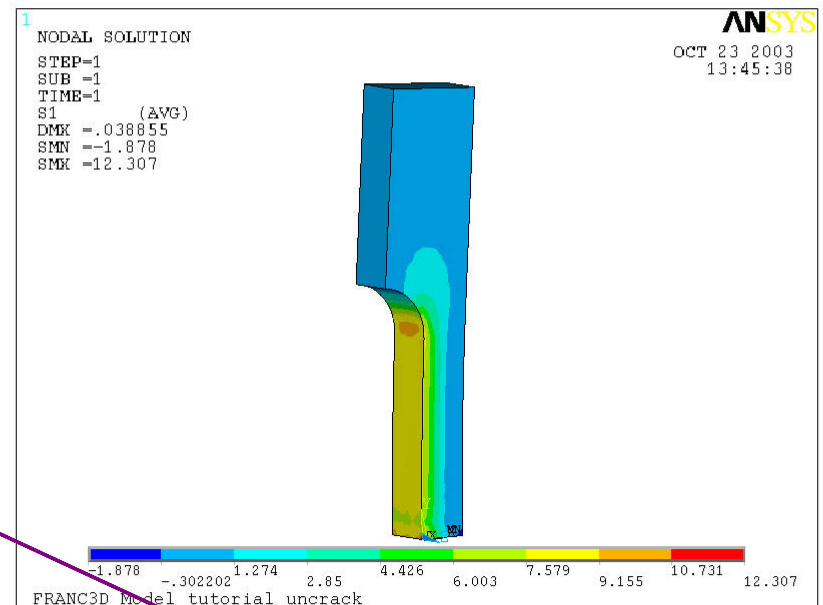
Grow the crack geometry & topology.

FRANC3D / OSM 3D Tutorial

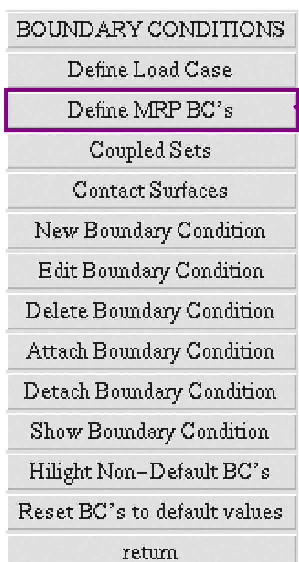
- illustrate crack face traction MRP and superposition
  - ansys stress analysis of uncracked model
  - OSM conversion of .cdb and .str files to MRP
  - attach MRP to crack face for BEM analysis

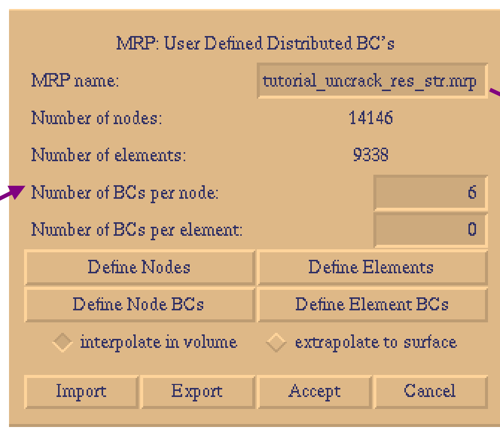


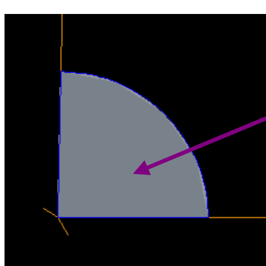
tutorial_uncrack.cdb and .str files converted by OSM creating: tutorial_uncrack_res_str.mrp



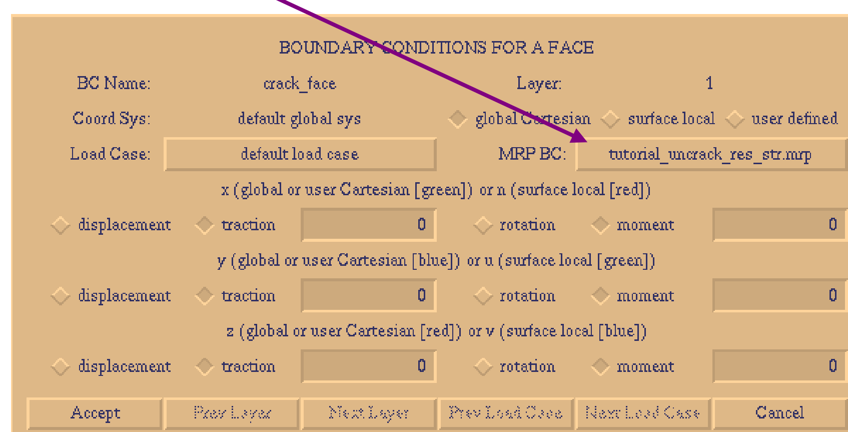
FRANC3D: define boundary conditions







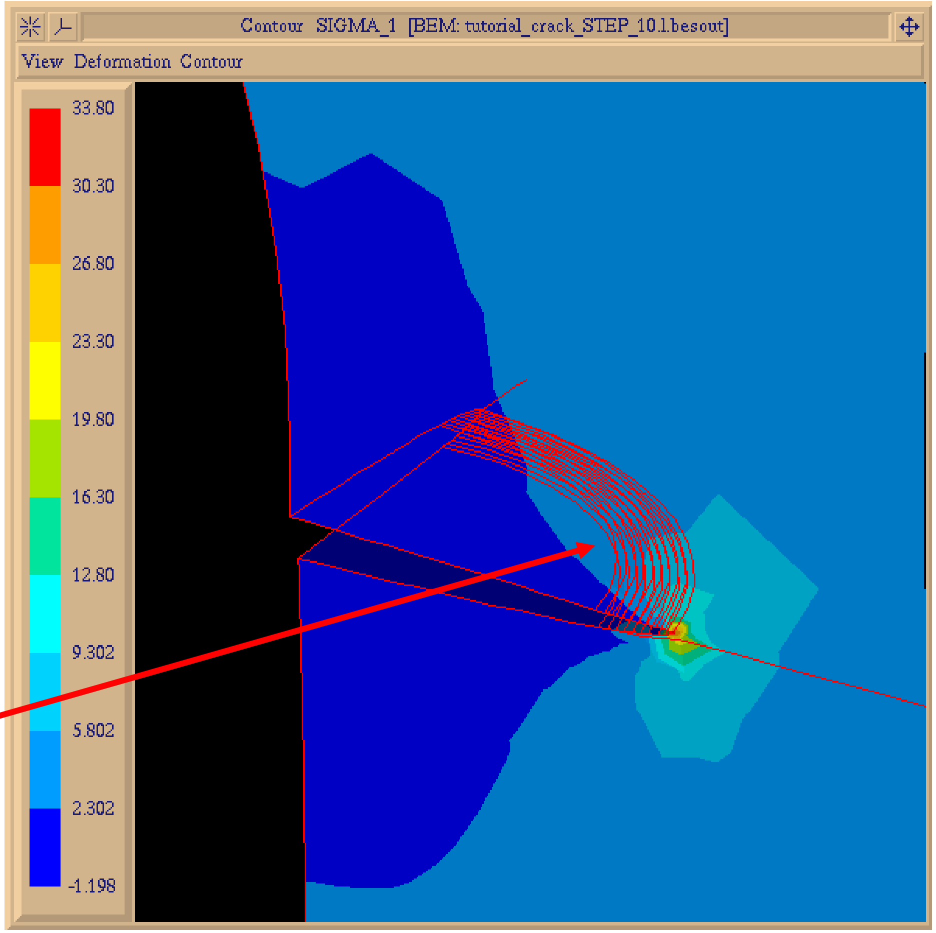
attach face boundary condition to crack face



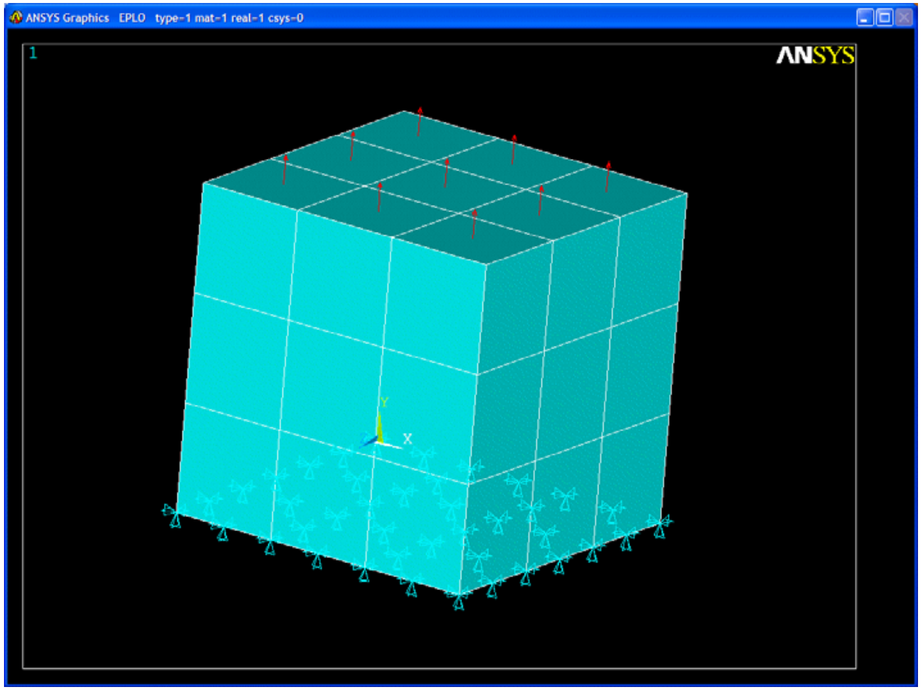
FRANC3D / OSM 3D Tutorial

- 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

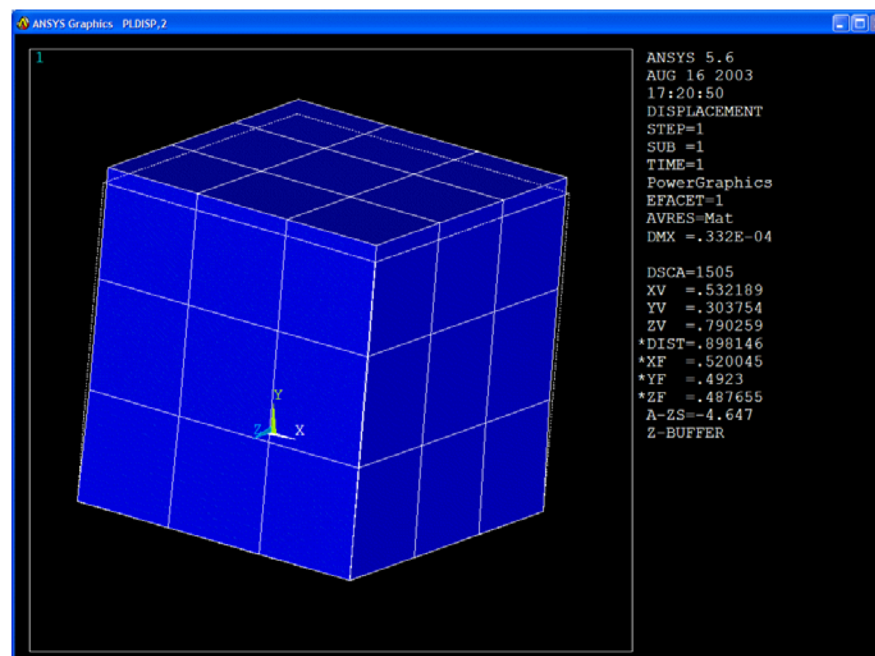
10 steps of crack growth



Simple Cube Example – Hands-on-training



- 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

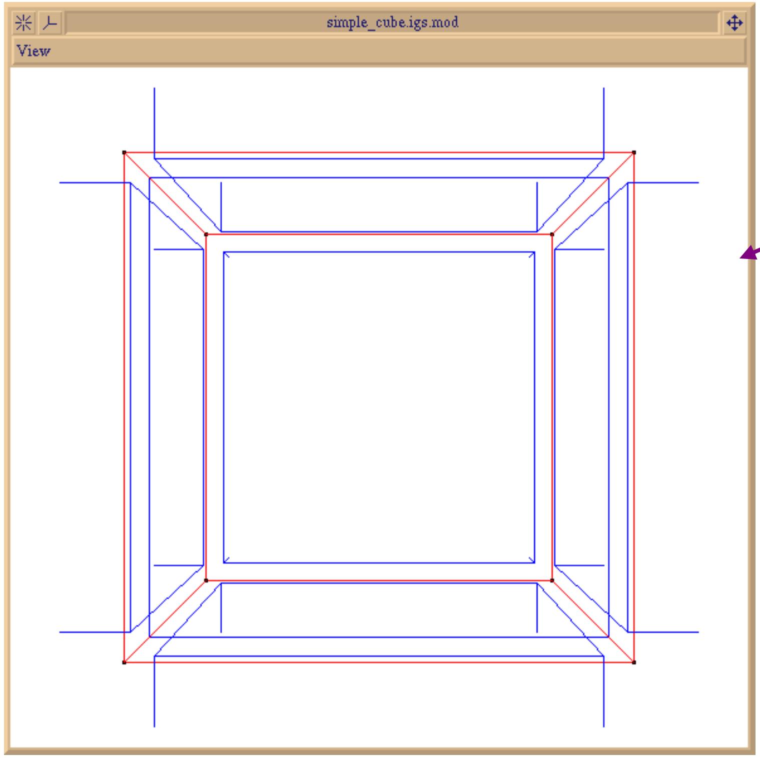
- discuss MRP’s generated by OSM from ANSYS .cdb file and from ANSYS nodal results saved as .dsp and .str files

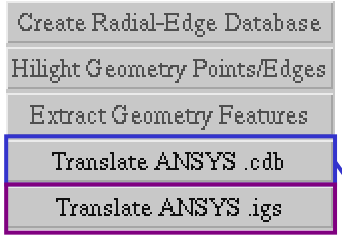
- read model into FRANC3D and apply MRP boundary conditions

- add a crack and analyze using BES

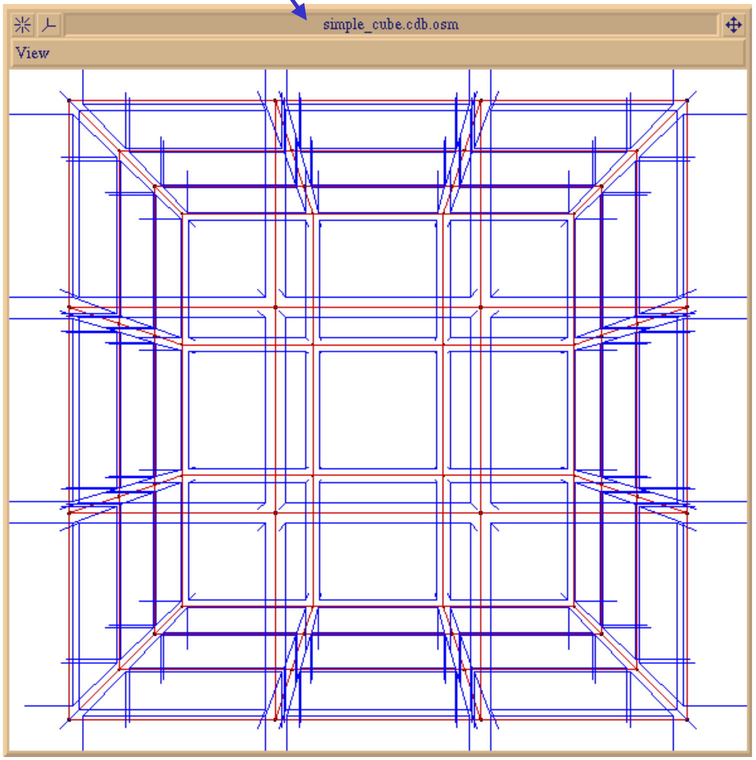
- compare stress intensity factors based on both BES and ANSYS results

Simple Cube Example





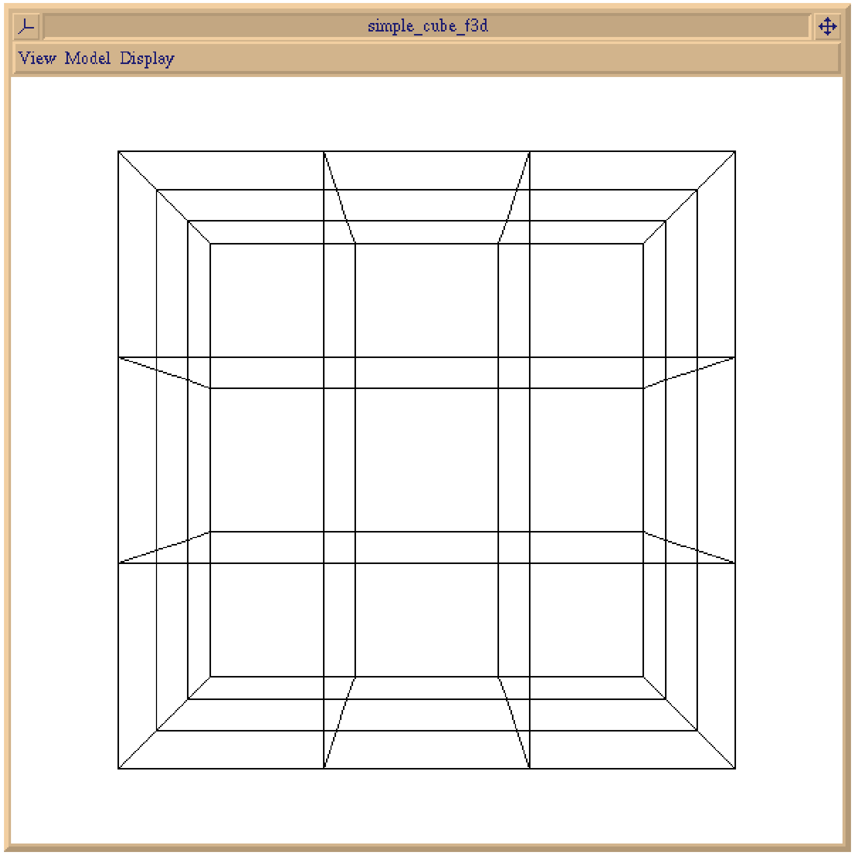
OSM can convert ANSYS iges and cdb files



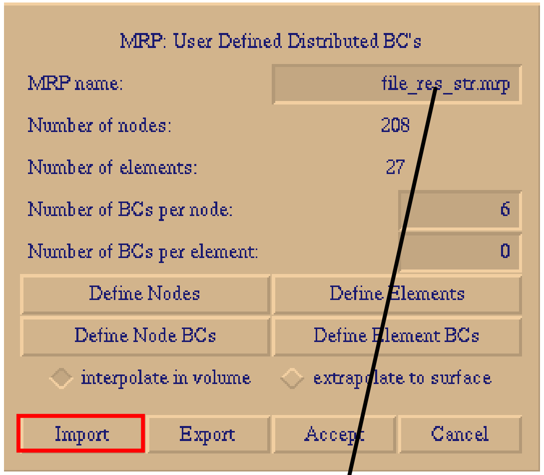
iges file contains only geometry and if a solid model does not exist there is no iges information

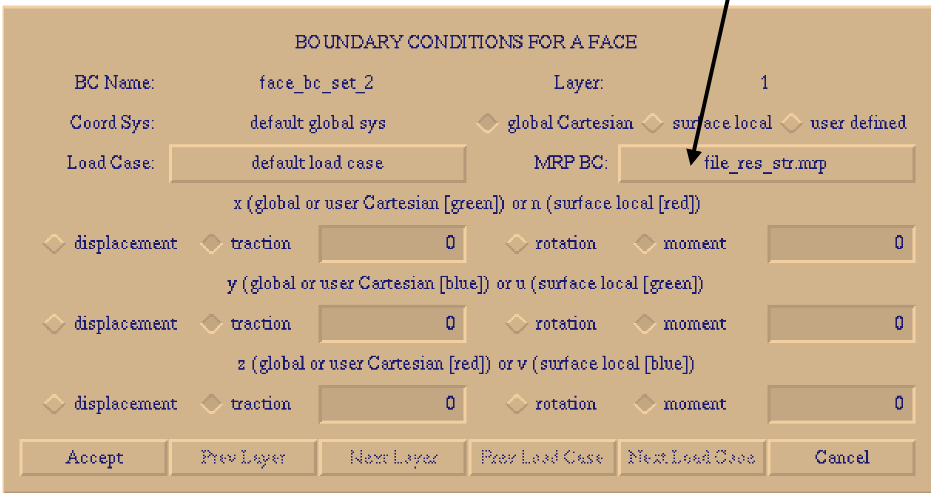
cdb file contains finite element mesh information along with boundary conditions and materials

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

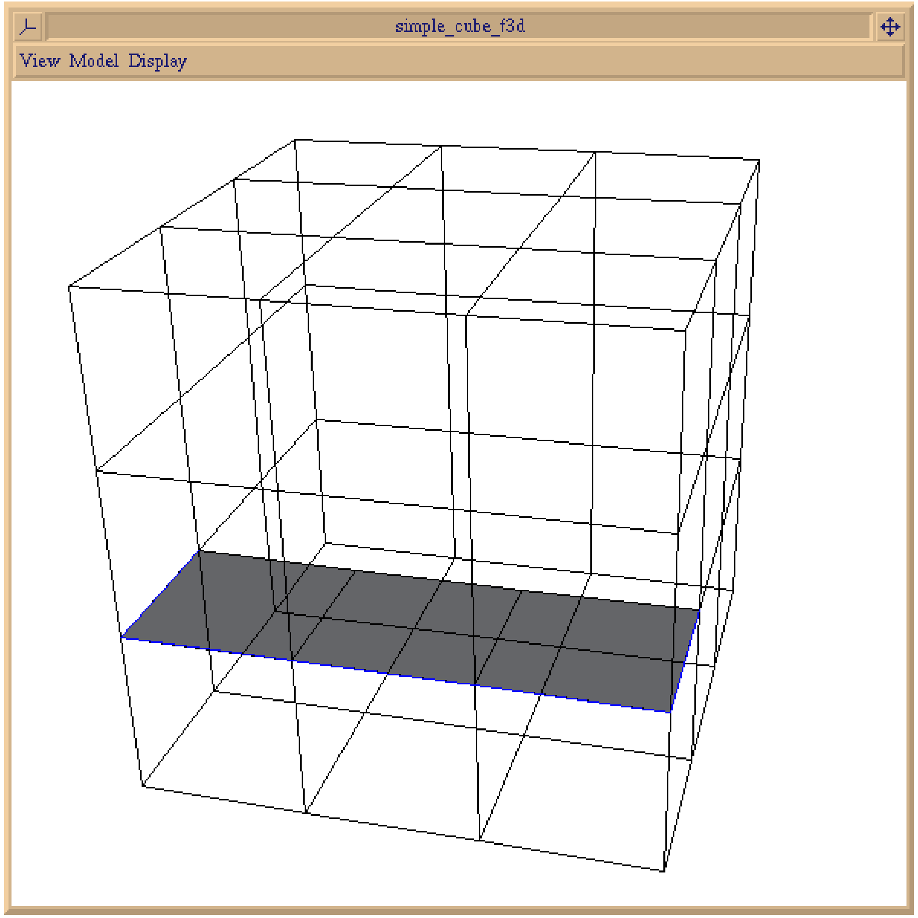




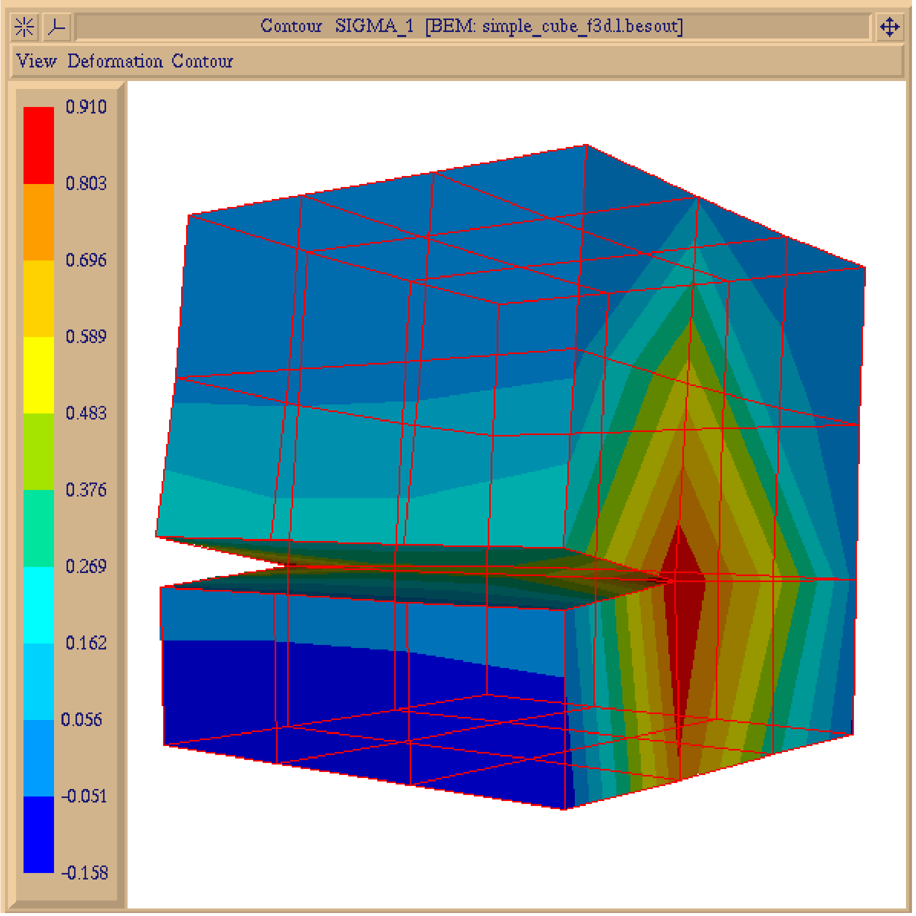
We can use the ANSYS mesh facets to define FRANC3D geometry, but it is not recommended for big models.

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.