Task 6 - Safety Review and Licensing
On the Job Training on Stress Analysis

F.M. with Finite Element analysis - Different calculation techniques + Numerical examples (ANSYS Apdl) 2/2

Davide Mazzini – **Ciro Santus**

BEYOND THE RESEARCH

UNIVERSITÀ DI PISA

Pisa (Italy)
June 15 – July 14, 2015

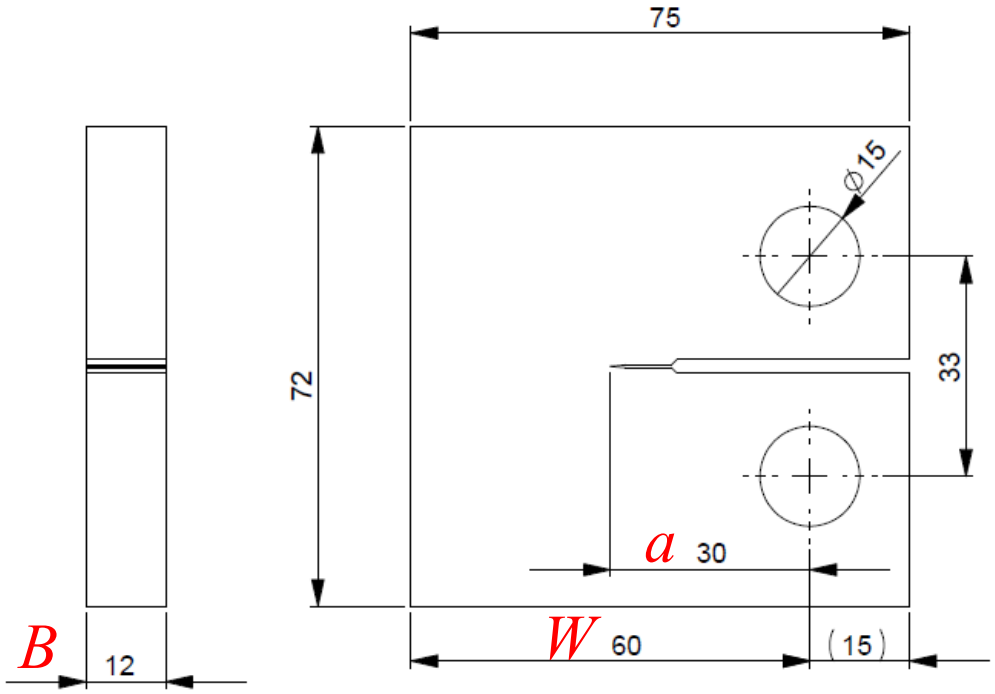
Content

- Different FE techniques
- ANSYS Workbench
  - Crack geometry element mesh preparation
  - $K_{I(II,III)}$ and J calculation
  - Examples
- ANSYS Apdl
  - The quarter point technique
  - Examples

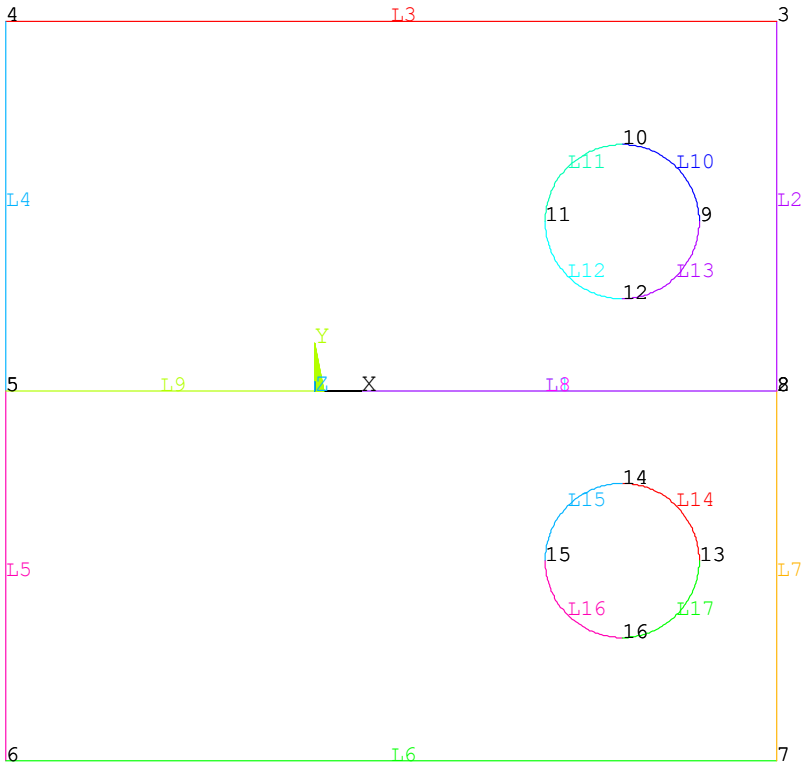
CT specimen (plane) modelling

$$P = 10\,\text{kN} \quad a = 30\,\text{mm}, \quad W = 60\,\text{mm}, \quad B = 12\,mm$$

$$K_I = \frac{P}{B\sqrt{W}} \frac{2+\alpha}{(1-\alpha)^{3/2}} (0.886 + 4.64\alpha - 13.32\alpha^2 + 14.72\alpha^3 - 5.64\alpha^4) = 971\,\text{MPa}\sqrt{\text{mm}}$$



CT specimen (plane) modelling



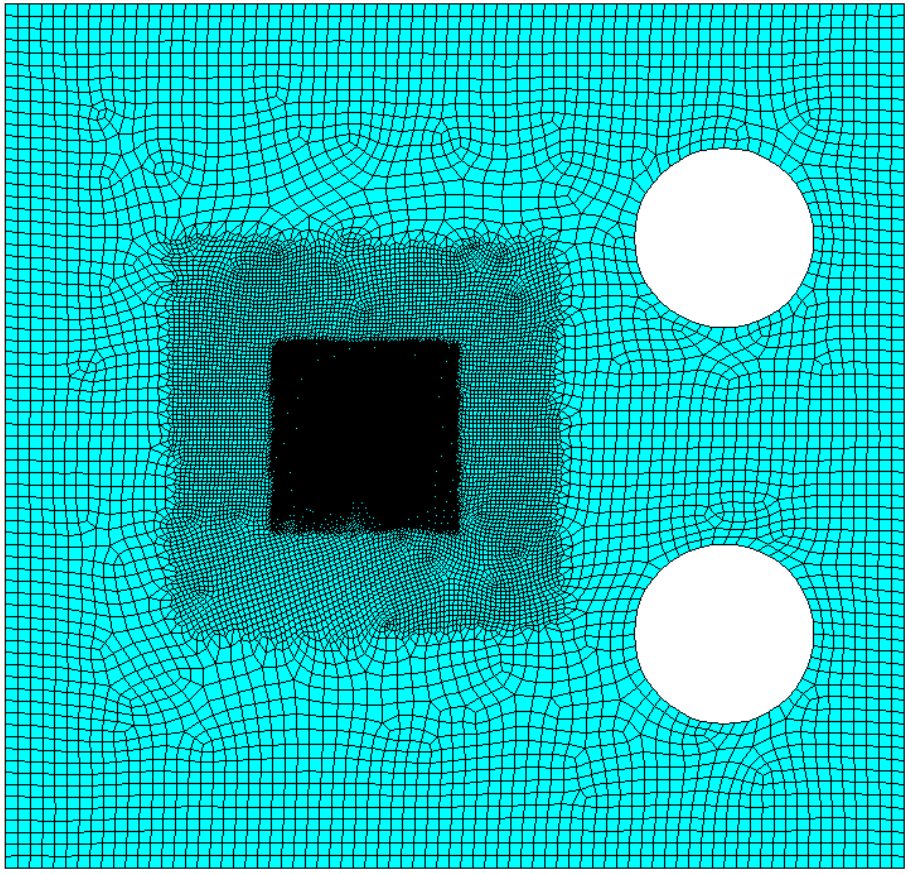

```
C*** parameters
! geometry
w = 60
wp = 15
h = 36
a = 30
ll = 33
df = 15
b = 12
! load
big_p = 10000
p = big_p/b
```

```
C*** keypoints
k,1,0.0,0.0
k,2,a+wp,0.0
k,3,a+wp,h
k,4,-(w-a),h
k,5,-(w-a),0.0
k,6,-(w-a),-h
k,7,a+wp,-h
k,8,a+wp,0.0

C*** lines
l,1,2
l,2,3
l,3,4
l,4,5
l,5,6
l,6,7
l,7,8
l,8,1
l,1,5

C*** areas
al,1,2,3,4,9
al,9,5,6,7,8

cyl4,a,ll/2,df/2
cyl4,a,-ll/2,df/2

asba,1,3
asba,2,4
```

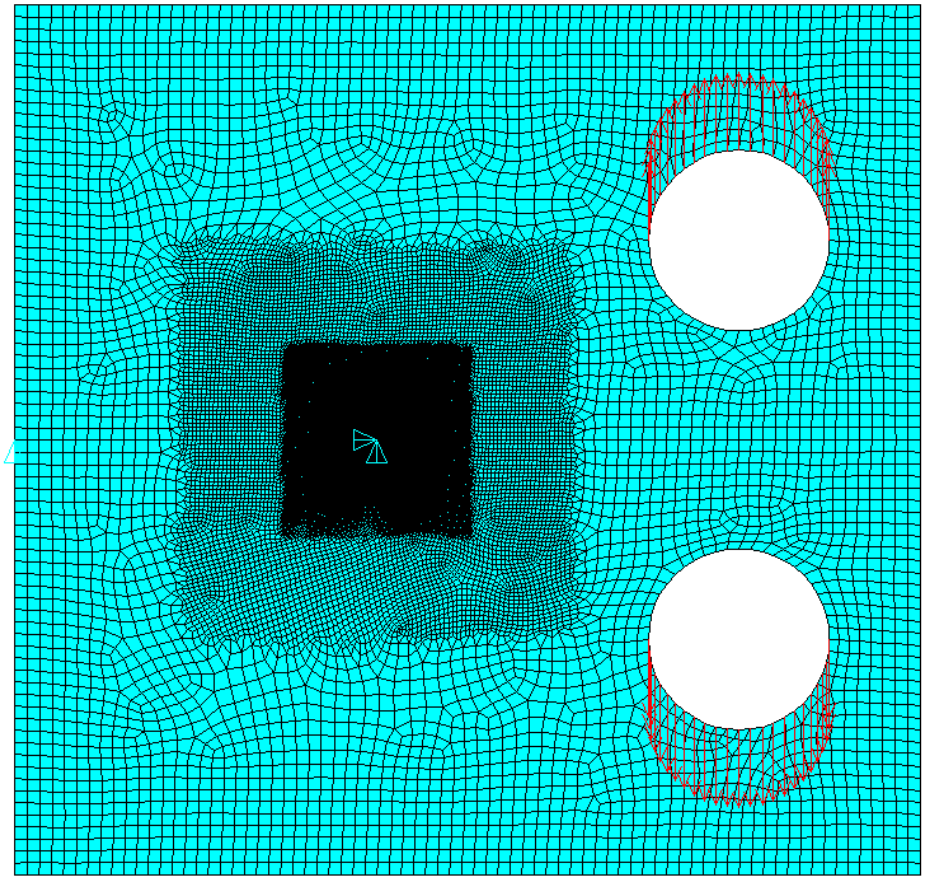
CT specimen (plane) modelling

Multiple and ‘nested’ refinements around the crack tip

Total: 304605 nodes
Element size at innermost refinement:
0.012 mm



CT specimen (plane) modelling



Plane load applied at pins:

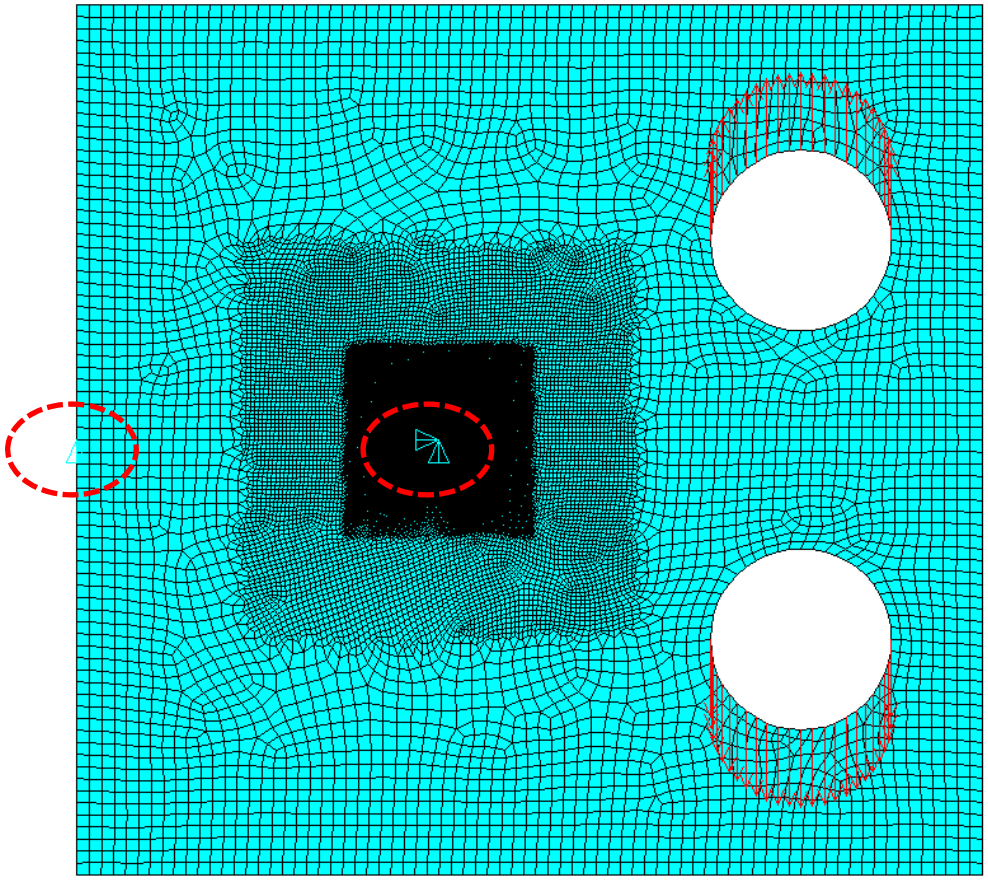
$$p = \frac{P}{B} = 833\ \text{N/mm}$$

Load on each node on the half circle:

$$p_i = \frac{p}{nn}$$

nn: number of nodes on the half circle

CT specimen (plane) modelling



In-plane contraints

to remove any lability

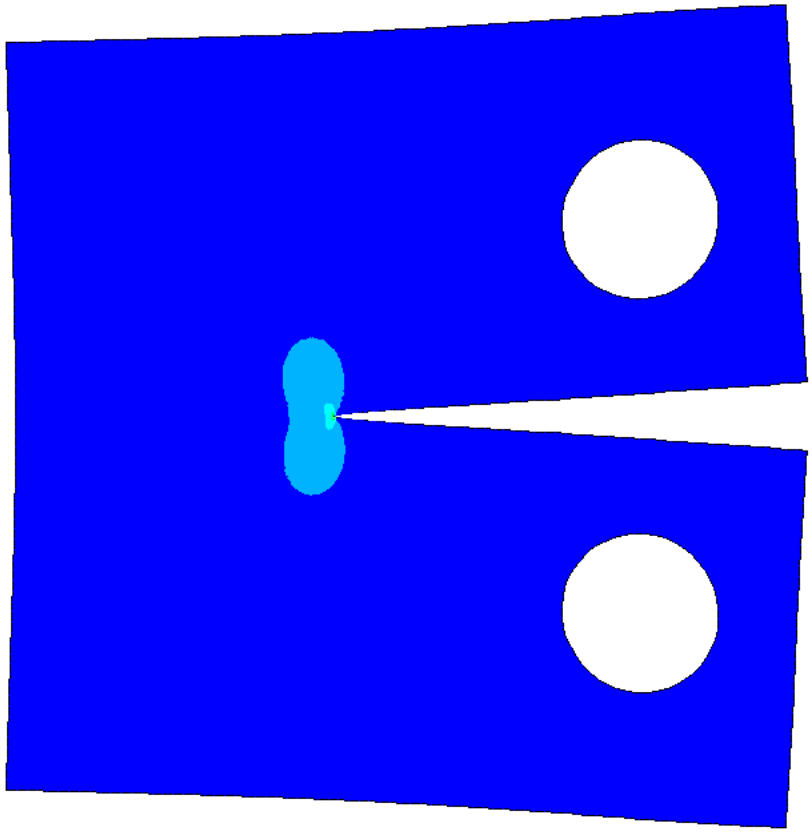
No rotation

No rigid motion

CT specimen (plane) modelling

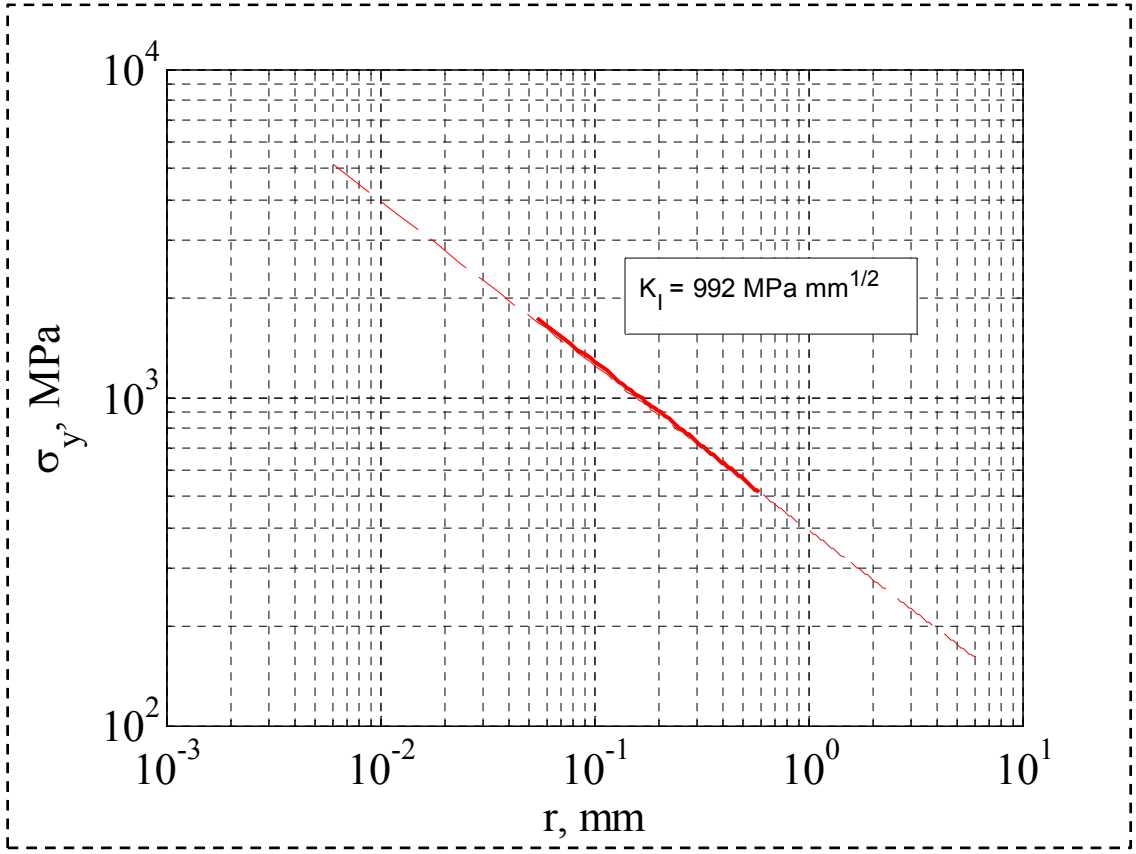
Stress singularity

The maximum value just related to the innermost element size



CT specimen (plane) modelling – Asymptotic calculation

log-log scales



MATLAB "polyfit": p_1, p_2

$p_1 = -0.5$ (approx.)

$K_I = 10^{p_2 + \frac{1}{2}\log_{10}(2\pi)}$

$K_I(\text{ANSYS-MATLAB}) = 992\,\text{MPa}\sqrt{\text{mm}}$

$K_I(\text{ASTM}) = 971\,\text{MPa}\sqrt{\text{mm}}$

$\Delta\% = 2.2\%$

Question:

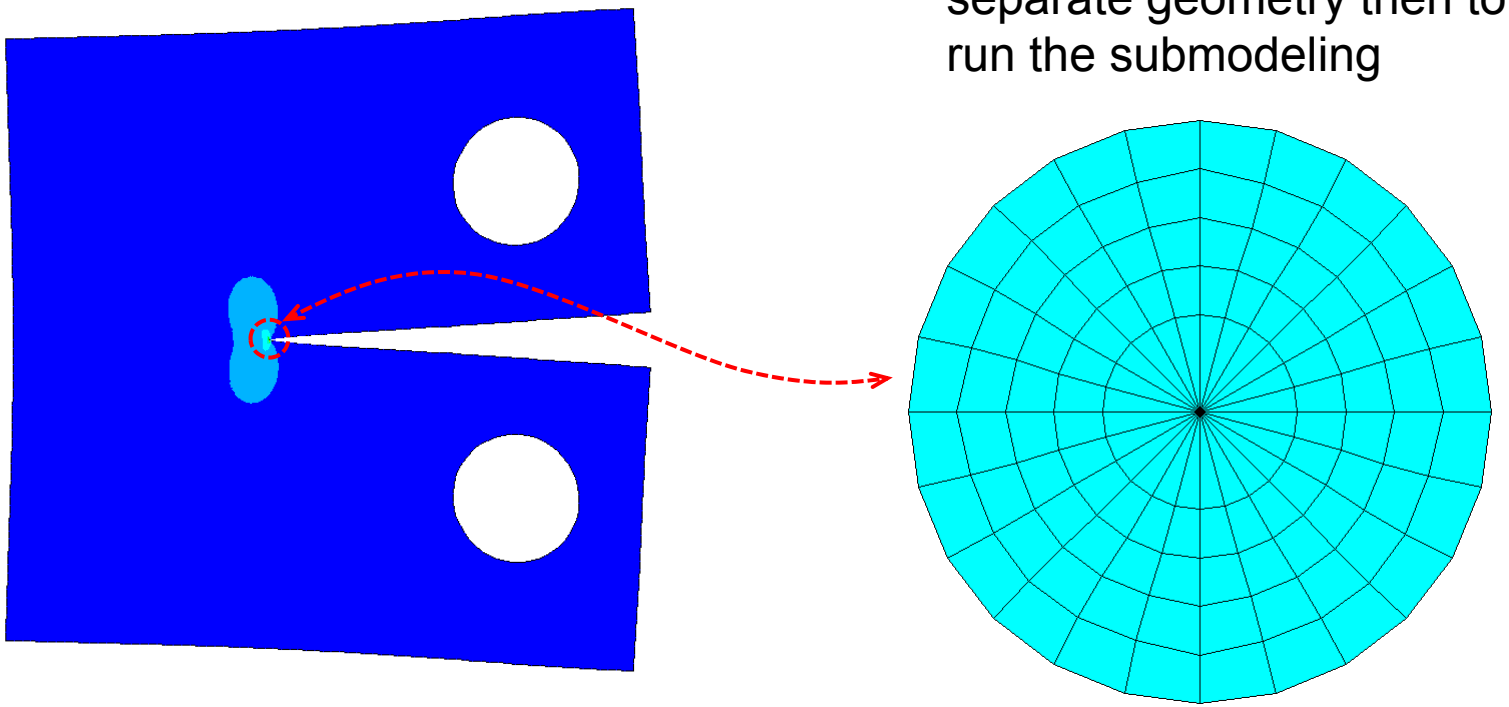
By switching from Pl. Strain to Pl. Stress, is the K_I result expected to change? Why?

```
C*** element type and material properties

! plane stress k3=0
! axisymmetric k3=1
! plane strain k3=2
k3 = 2
et,1,182,,,k3

mp,ex,1,205000
mp,prxy,1,0.3
```

CT specimen (plane) modelling – KCAL command

1st step: calculate the solution as previous

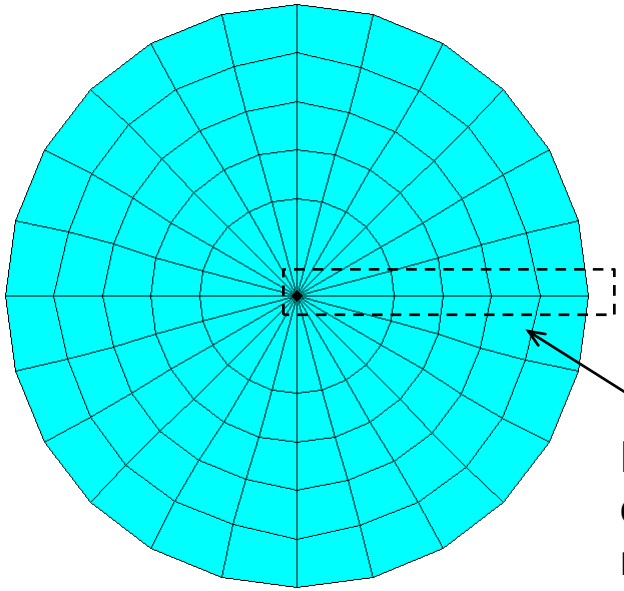
2nd step: prepare a spiderweb mesh, on a separate geometry then to run the submodeling



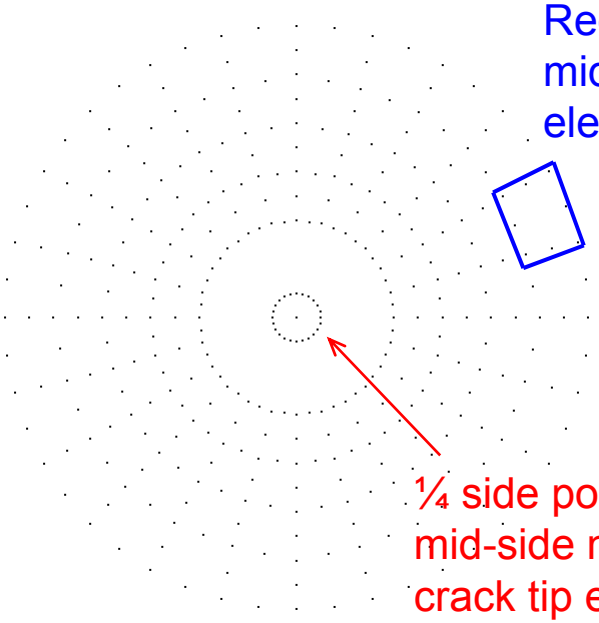
CT specimen (plane) modelling – KCAL command

ANSYS command for mid-side node at quarter point:

```
KSCON,1,r/3,1,6
```



Distinct overlying nodes (crack side)

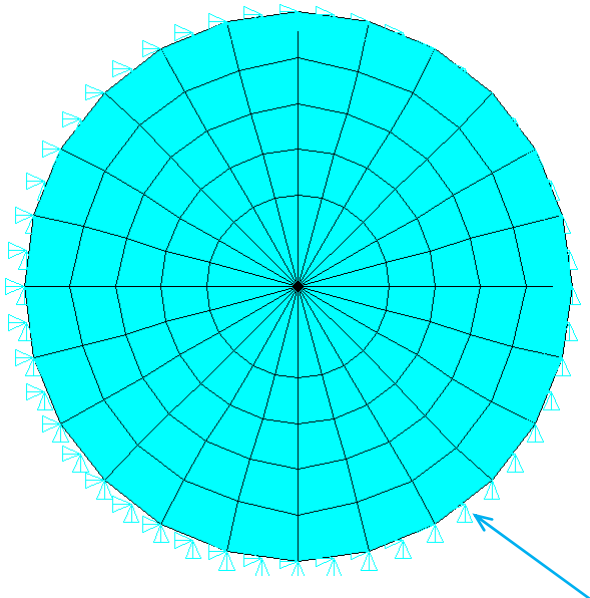


Regular mid-side elements

¼ side position of mid-side nodes for crack tip elements

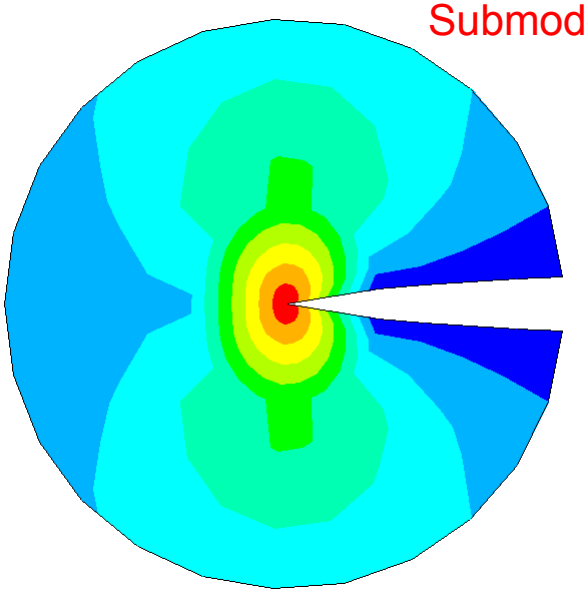
CT specimen (plane) modelling – KCAL command

3rd step: interpolate the displacements, apply to the submodel boundary, and run the solution



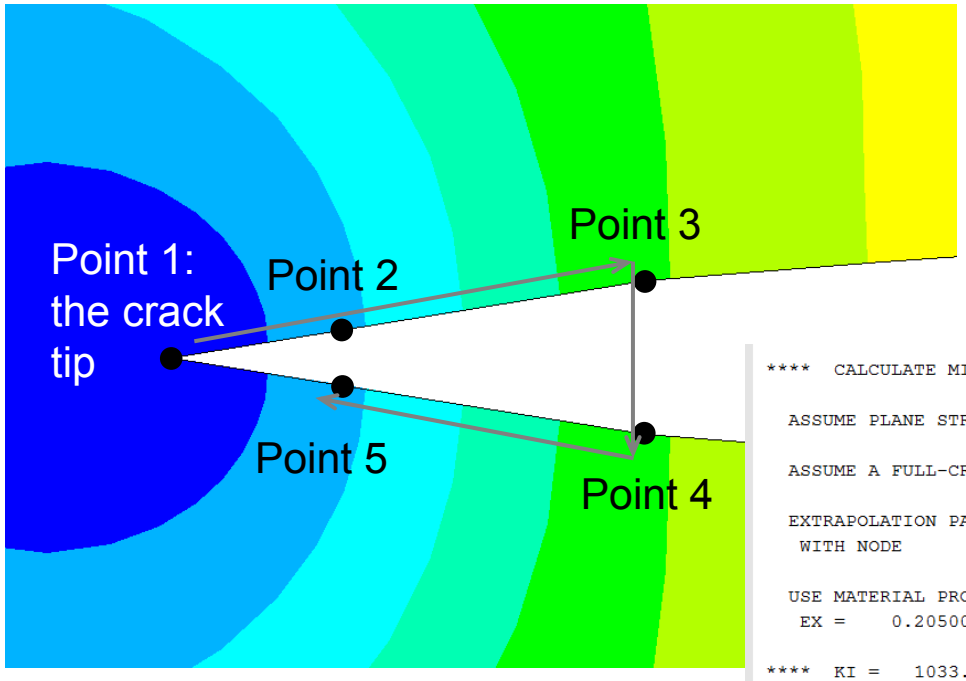
Displacements derived from the full model

Submodel solution



CT specimen (plane) modelling – KCAL command

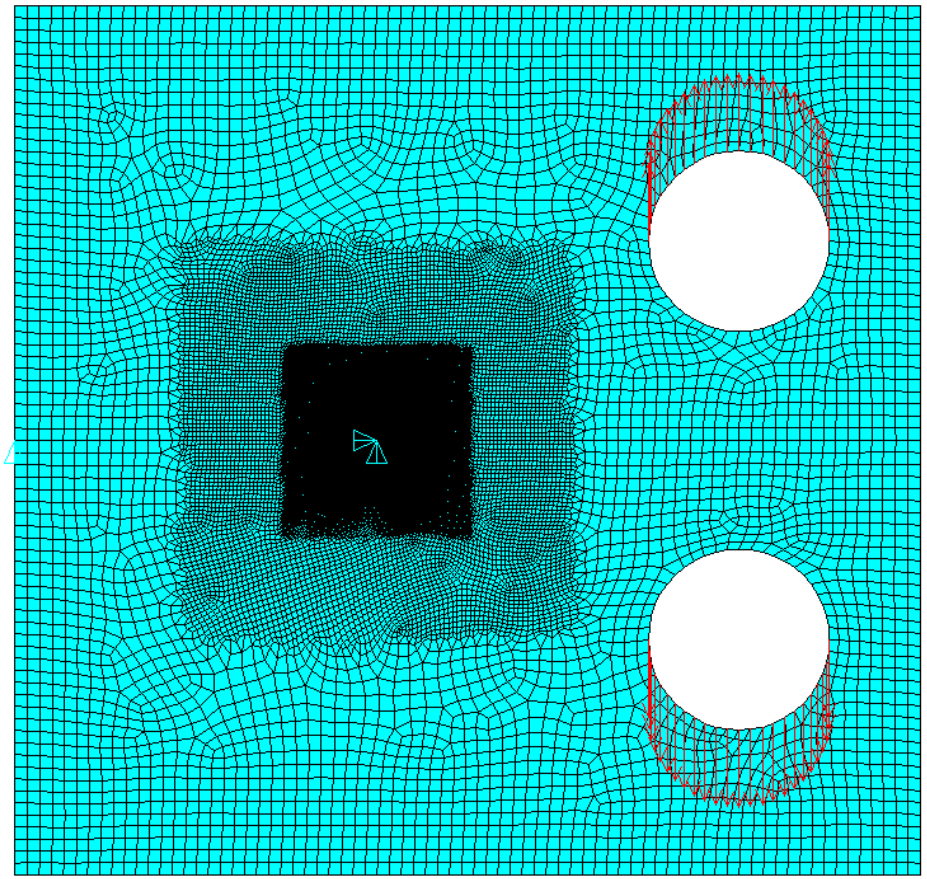
4th step: define an ANSYS path and run the KCAL command




```
****  CALCULATE MIXED-MODE STRESS INTENSITY  FACTORS  ****

 ASSUME PLANE STRAIN CONDITIONS

 ASSUME A FULL-CRACK MODEL (USE 5 NODES)

 EXTRAPOLATION PATH IS DEFINED BY NODES:       1       3       4     294     295
  WITH NODE       1 AS THE CRACK-TIP NODE

 USE MATERIAL PROPERTIES FOR MATERIAL NUMBER     1
  EX =    0.20500E+06   NUXY =    0.30000      AT TEMP =  0.0000

****  KI =   1033.3     ,   KII =  0.34648E-01,   KIII =   0.0000      ****
```

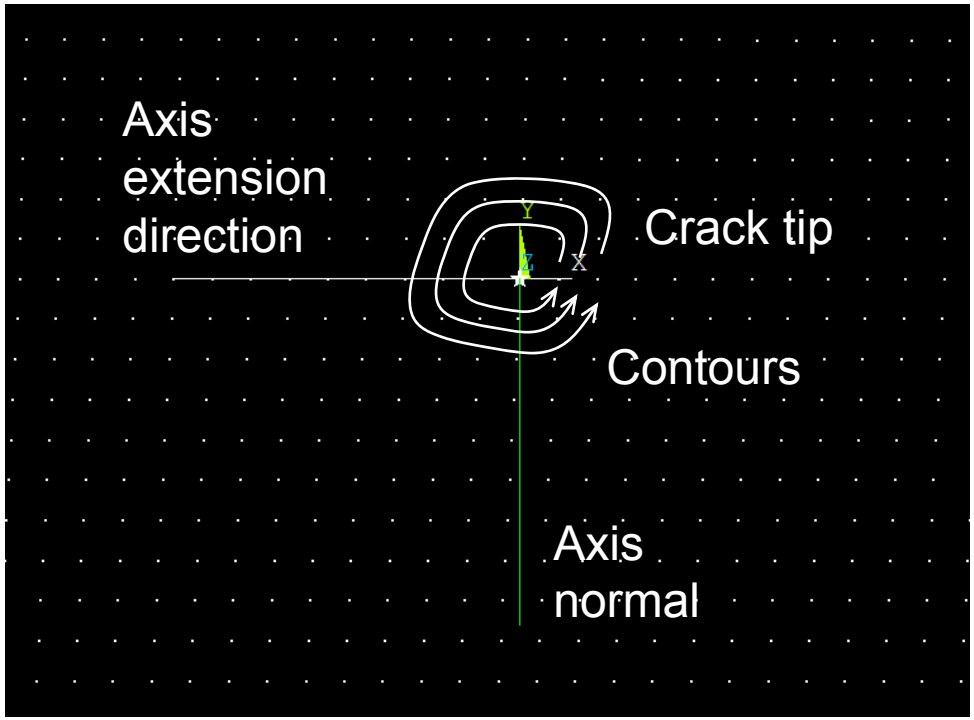
CT specimen (plane) modelling – CINT,SIFS command

1st step: prepare the model as previous



CT specimen (plane) modelling – CINT,SIFS command

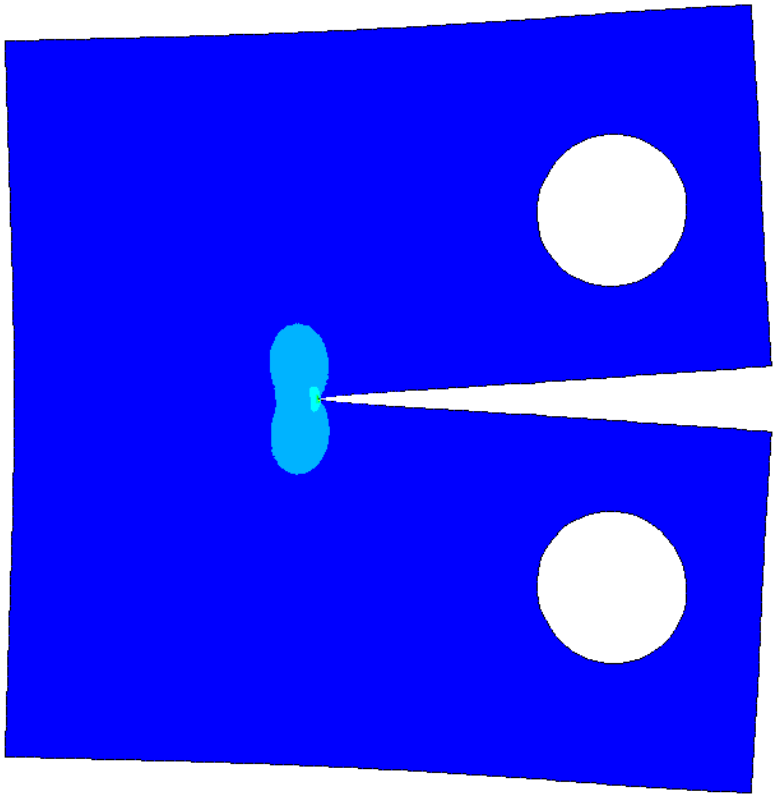
2nd step: define the crack tip for the CINT command and input other CINT options




```
C*** solution
/solu
C*** CINT with SIFS options
cint,new,1
cint,type,sifs

x_c = 0.0
y_c = 0.0
theta = 180
local,11,0,x_c,y_c,,theta

nsel,s,loc,x,x_c
nsel,r,loc,y,y_c
cm,CRACK_TIP_NODE_CM,node
alls

cint,ctnc,CRACK_TIP_NODE_CM
cint,normal,11,2
cint,symm,0
cint,ncon,10

solve
finish
```

CT specimen (plane) modelling – CINT,SIFS command

3rd step: just run the solution



CT specimen (plane) modelling – CINT,SIFS command

4th step: issue the `prcint` command to see the results

```
C*** CINT output
prcint,1,,K1
```

PRCINT Command

File

```
***** POST1 K1          RESULT LISTING *****

CrackID =   1
 Crack Front Node =      1
  Contour Values =    942.46      1027.3      1032.6      1033.6
  Contour Values =    1034.0      1034.2      1034.3      1034.4
  Contour Values =    1034.4      1034.4
```

Different results for the contours, convergence of results after the first few contours

PRCINT Command

File

```
***** POST1 K2          RESULT LISTING *****

CrackID =   1
 Crack Front Node =      1
  Contour Values =   -14.127     -0.11317     -0.12838     -0.55933E-01
  Contour Values =  -0.20951E-01 -0.67804E-02 -0.44983E-02  0.72401E-03
  Contour Values =   0.40783E-02  0.30702E-02
```

K2 values (that should be zero) give an indication of the accuracy

CT specimen (plane) modelling – Results comparison

$$K_{\mathrm{I}}(\mathrm{ASTM}) = 971\,\mathrm{MPa}\sqrt{\mathrm{mm}}$$

$$K_{\mathrm{I}}(\mathrm{ANSYS\text{-}Asympt.}) = 992\,\mathrm{MPa}\sqrt{\mathrm{mm}} \quad \Delta\% = 2.2\%$$

$$K_{\mathrm{I}}(\mathrm{ANSYS\text{-}KCALC}) = 1033\,\mathrm{MPa}\sqrt{\mathrm{mm}} \quad \Delta\% = 6.4\%$$

$$K_{\mathrm{I}}(\mathrm{ANSYS\text{-}CINT}) = 1034\,\mathrm{MPa}\sqrt{\mathrm{mm}} \quad \Delta\% = 6.5\%$$

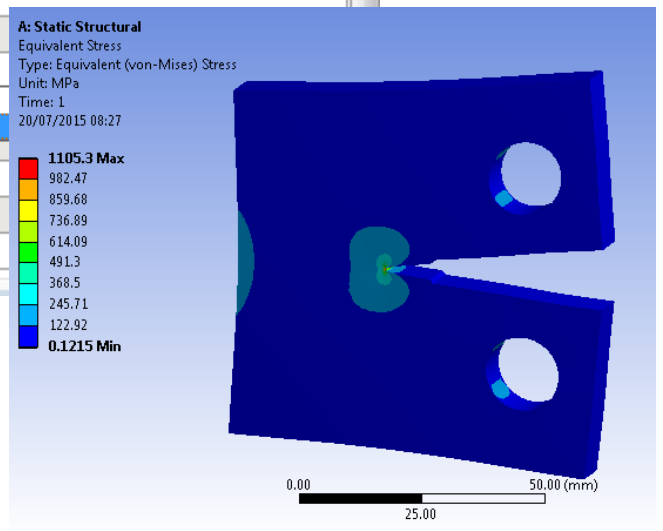
Which one could, or should, be the most accurate?

Why, any possible explanation?

CT specimen (3D) modelling, ANSYS Wb – ANSYS Apdl

Details of "Analysis Settings"

Large Deflection	Off
Inertia Relief	Off
Fracture	On
Restart Controls	
Nonlinear Controls	
Output Controls	
Analysis Data Management	
Solver Files Directory	C:\Users\Santus\Desktop\FM_Classes\Class7_Santus\ANSYS_Wb\CT_Specime...
Future Analysis	None
Scratch Solver Files ...	
Save MAPDL db	No
Delete Unneeded Fi...	No / Yes
Nonlinear Solution	No
Solver Units	Active System
Solver Unit System	nmm
Visibility	



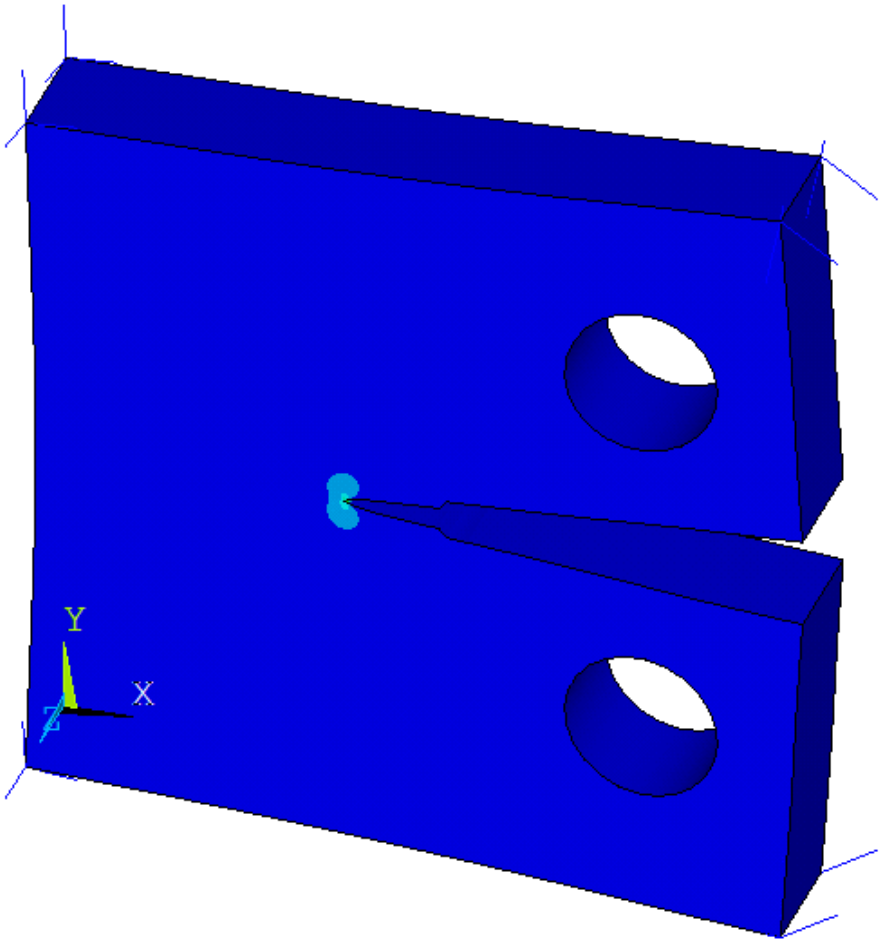
Save to ANSYS APDL environment

Exercise:

Calculate K_I at the *crack front* nodes by means of the CINT,SIFS command in the Apdl 3D environment.

Same results as Workbench PreCracked-Mesh procedure?

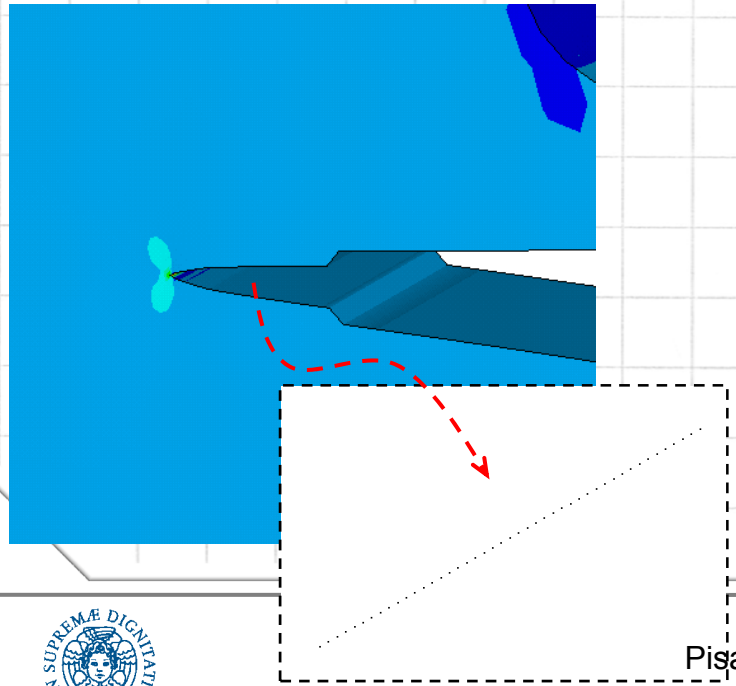
From ANSYS Workbench



ANSYS Workbench -> ANSYS Apdl

```
! CM selection of the nodes at the crack front
x_c = 30.0
y_c = 36.0
z_c = 6.0
!z_c = 0.0
theta = 180
local,21,0,x_c,y_c,z_c,theta
nsel,s,loc,x,0.0
nsel,r,loc,y,0.0
cm,CRACK_TIP_NODE_CM,node
alls
```

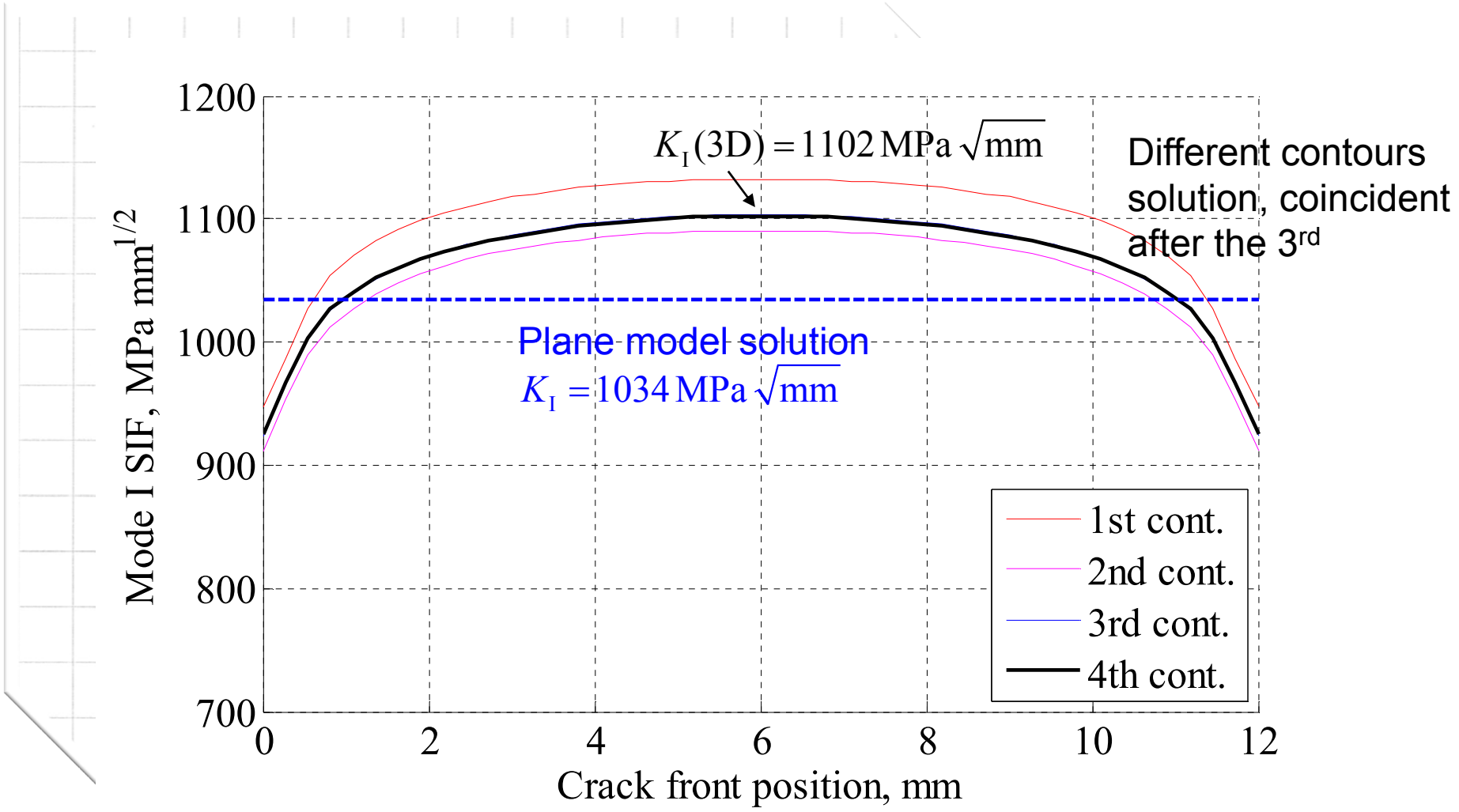
Crack front component
CM selection



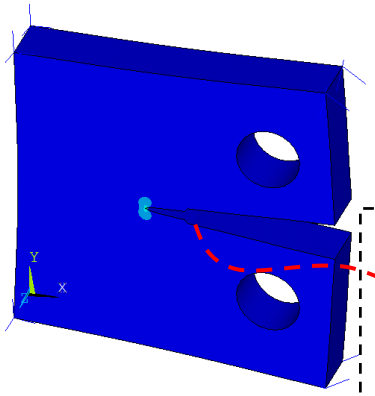
```
C*** solution
/solu

C*** CINT with SIFS options
cint,new,1
cint,type,sifs
cint,ctnc,CRACK_TIP_NODE_CM
cint,normal,21,2
cint,symm,0
cint,ncon,4

! solution
solve
finish
```

CINT commands

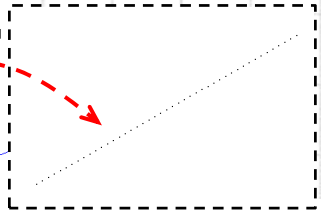
ANSYS Workbench -> ANSYS Apdl



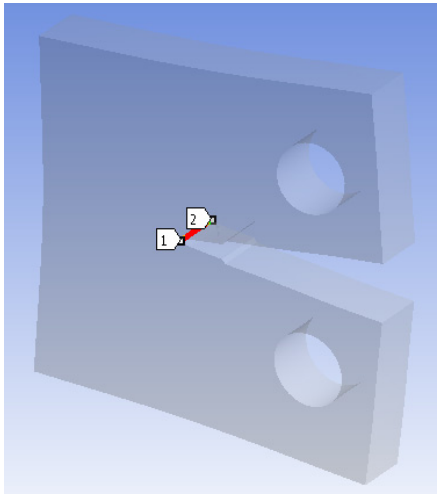
ANSYS Workbench -> ANSYS Apdl



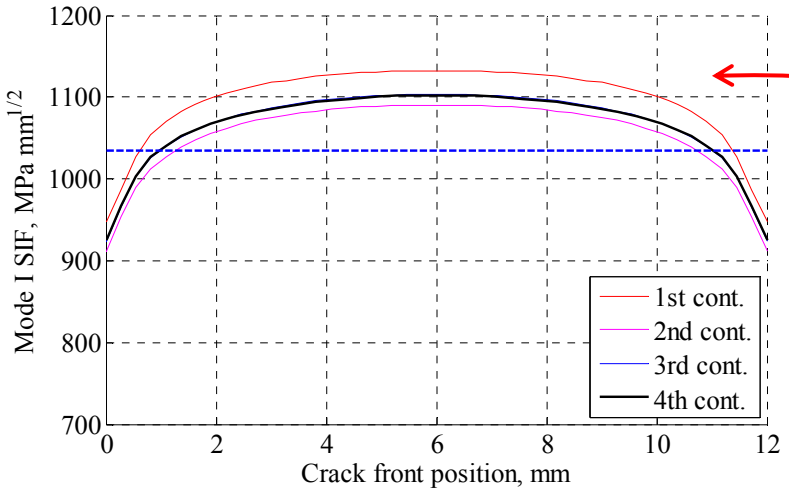
Workbench then Adpl to manually introduce the CINT command

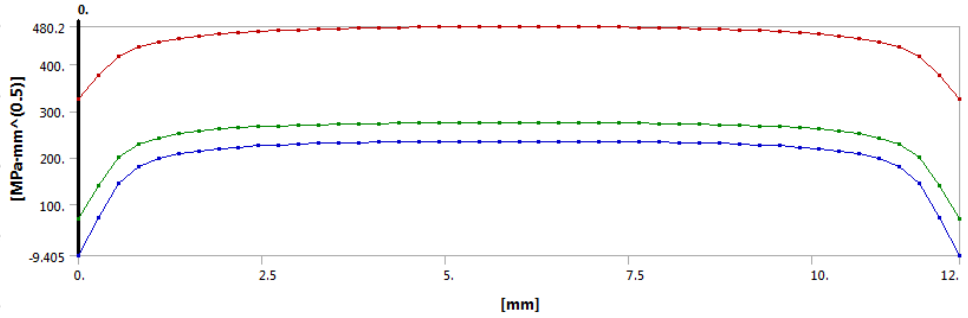


Workbench with PreMeshed-crack



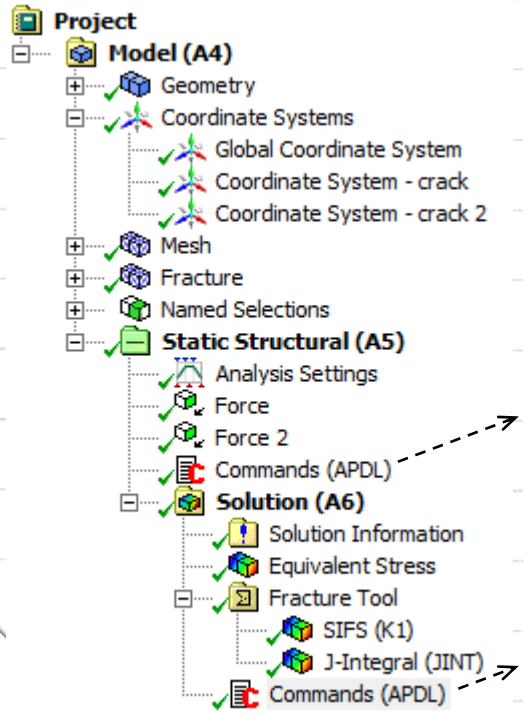
Same solution, so why this difference?





ANSYS Workbench -> ANSYS Apdl

Adpl Commands can be manually introduced in the Wb environment




```
!   Commands inserted into thi
!   These commands may superse

!   Active UNIT system in Work
!   NOTE:  Any data that requi
!                See Solving U


! CM selection of the nodes at
x_c = 30.0
y_c = 36.0
z_c = 6.0
!z_c = 0.0
theta = 180
local,21,0,x_c,y_c,z_c,theta
nsel,s,loc,x,0.0
nsel,r,loc,y,0.0
cm,CRACK_TIP_NODE_CM,node
alls

C*** CINT with SIFS options
cint,new,1
cint,type,sifs
cint,ctnc,CRACK_TIP_NODE_CM
cint,normal,21,2
cint,symm,0
cint,ncon,4
```

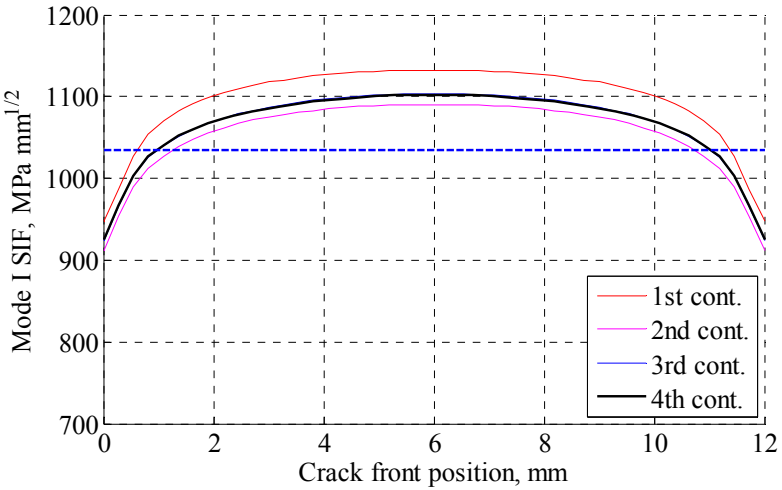
```
/header,off,off,off,off,off,off
/output,CINT_results_fromWb,dat
prcint,1,,K1
/output
/header
```

Results redirected to an output text file

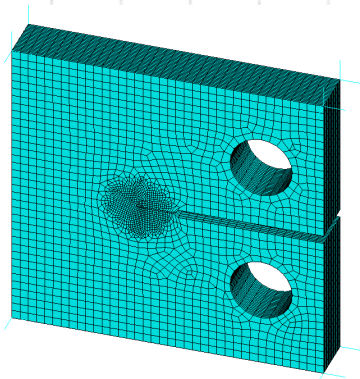
CINT_results_fromWb.dat

```
1
2  ***** POST1 K1          RESULT LISTING *****
3
4   CrackID =   1
5    Crack Front Node =   49678
6     Contour Values =    947.41       912.34       924.54       925.64
7    Crack Front Node =  183663
8     Contour Values =    986.23       953.07       965.69       966.80
9    Crack Front Node =   44349
10    Contour Values =    1027.3       989.04       1002.5       1003.7
11   Crack Front Node =  183661
12    Contour Values =    1054.0       1011.8       1025.2       1026.4
13   Crack Front Node =   44350
14    Contour Values =    1069.9       1027.4       1040.2       1041.2
15   Crack Front Node =  183664
```

Same results as after Wb to Apdl

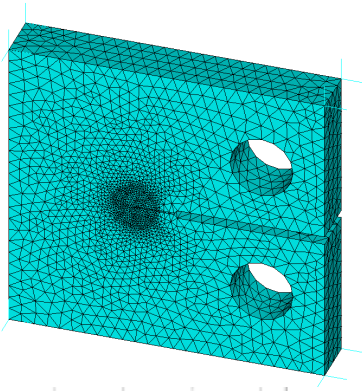


ANSYS Workbench -> ANSYS Apdl



Workbench to Adpl,
Hexahedrons mesh

OK



Workbench with
PreMeshed-crack

NO