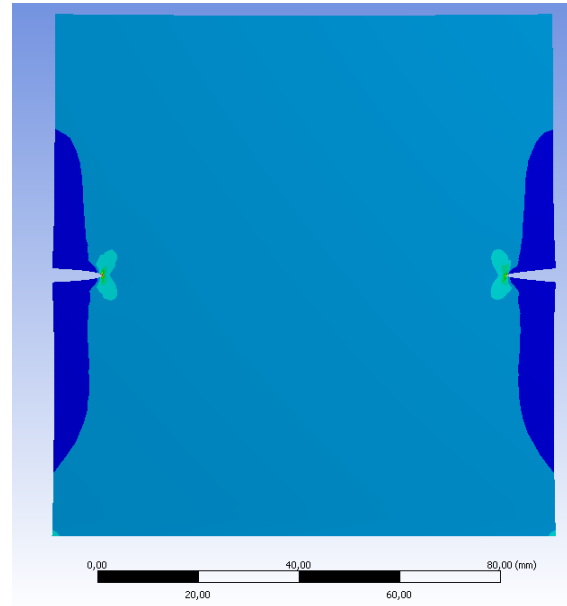
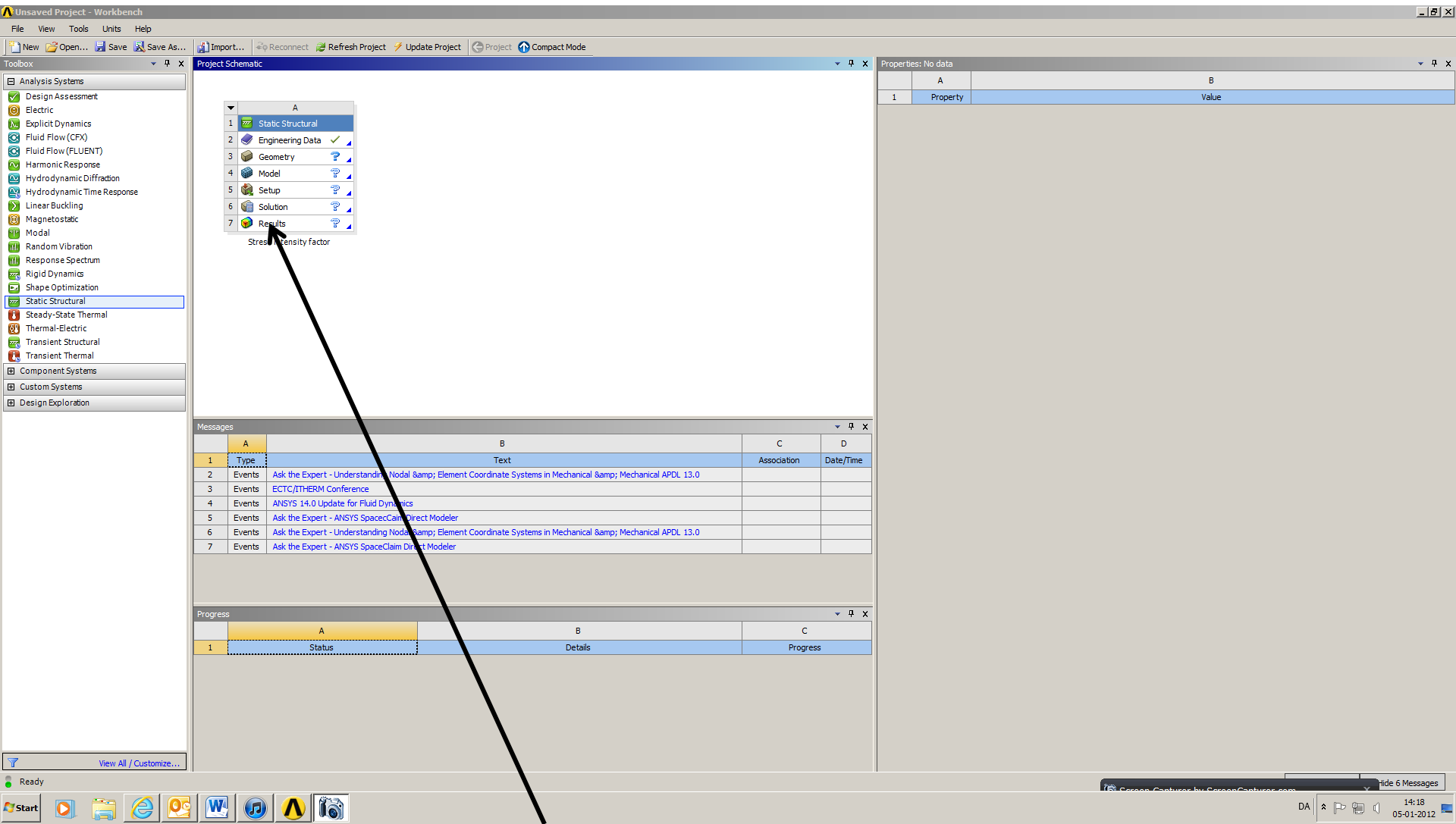


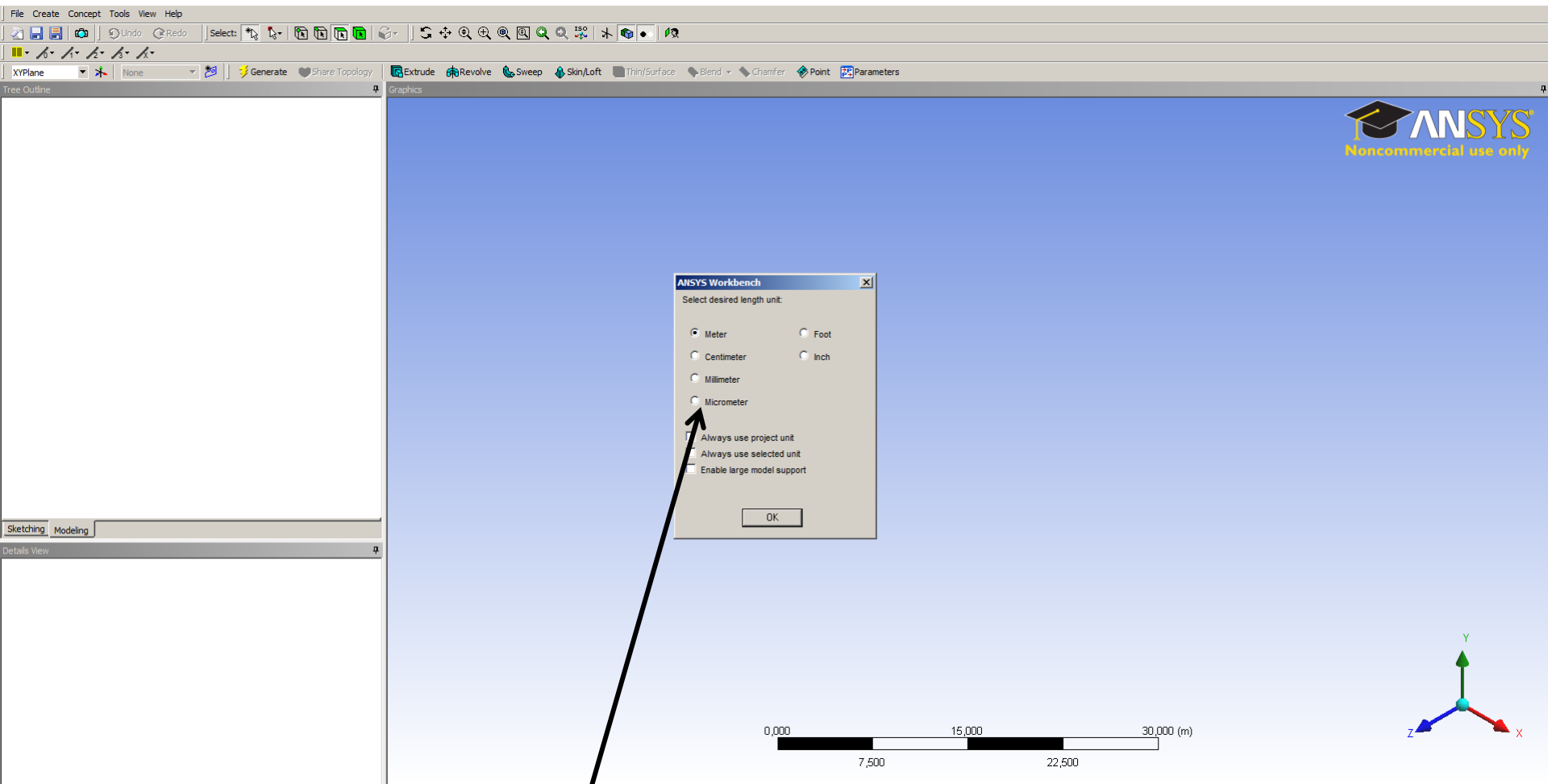
Calculation of the Stress intensity factor with CINT command in 3D Workbench 13.0



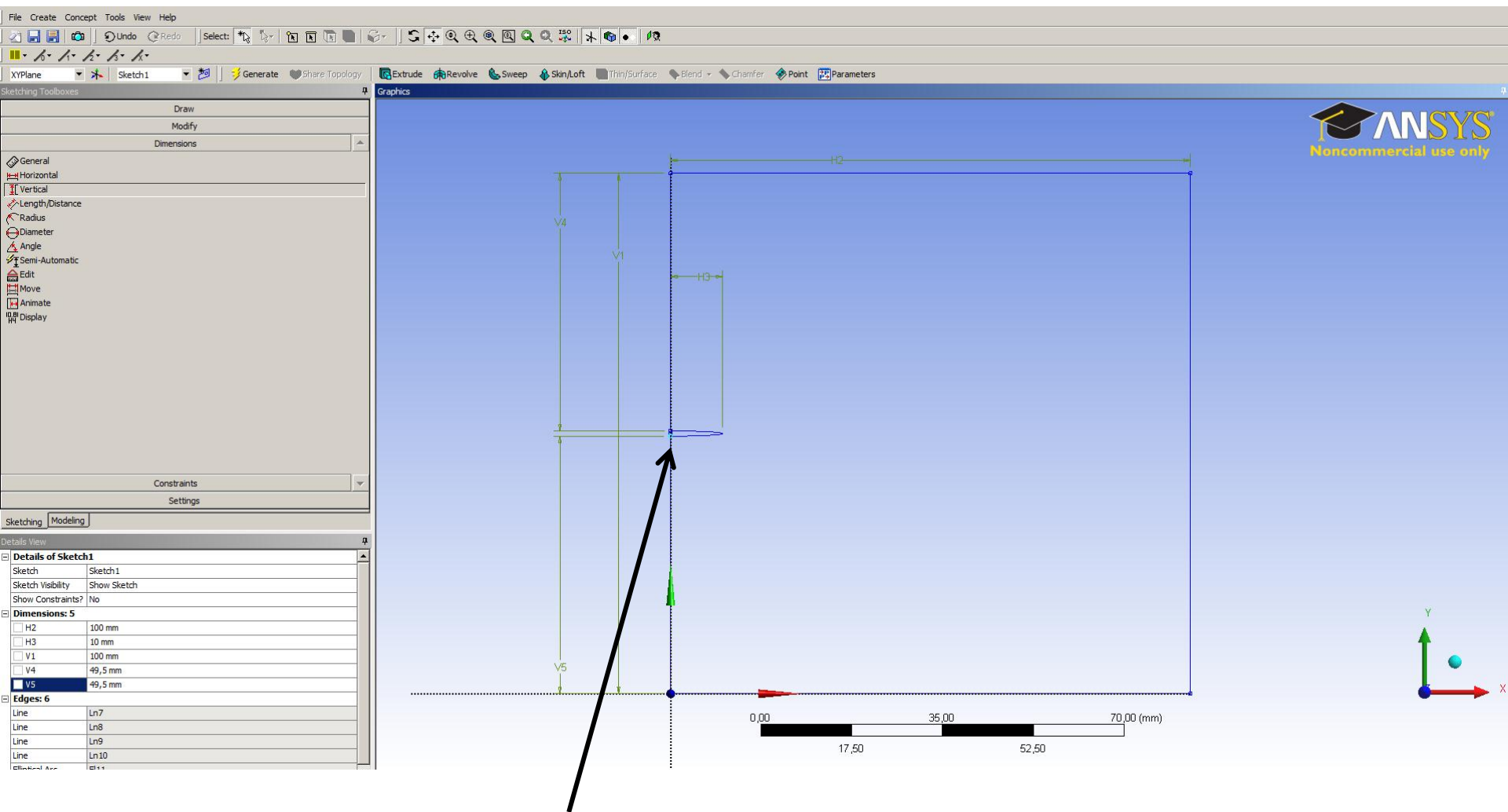
Aalborg Universitet esbjerg
Søren Heide Lambertsen



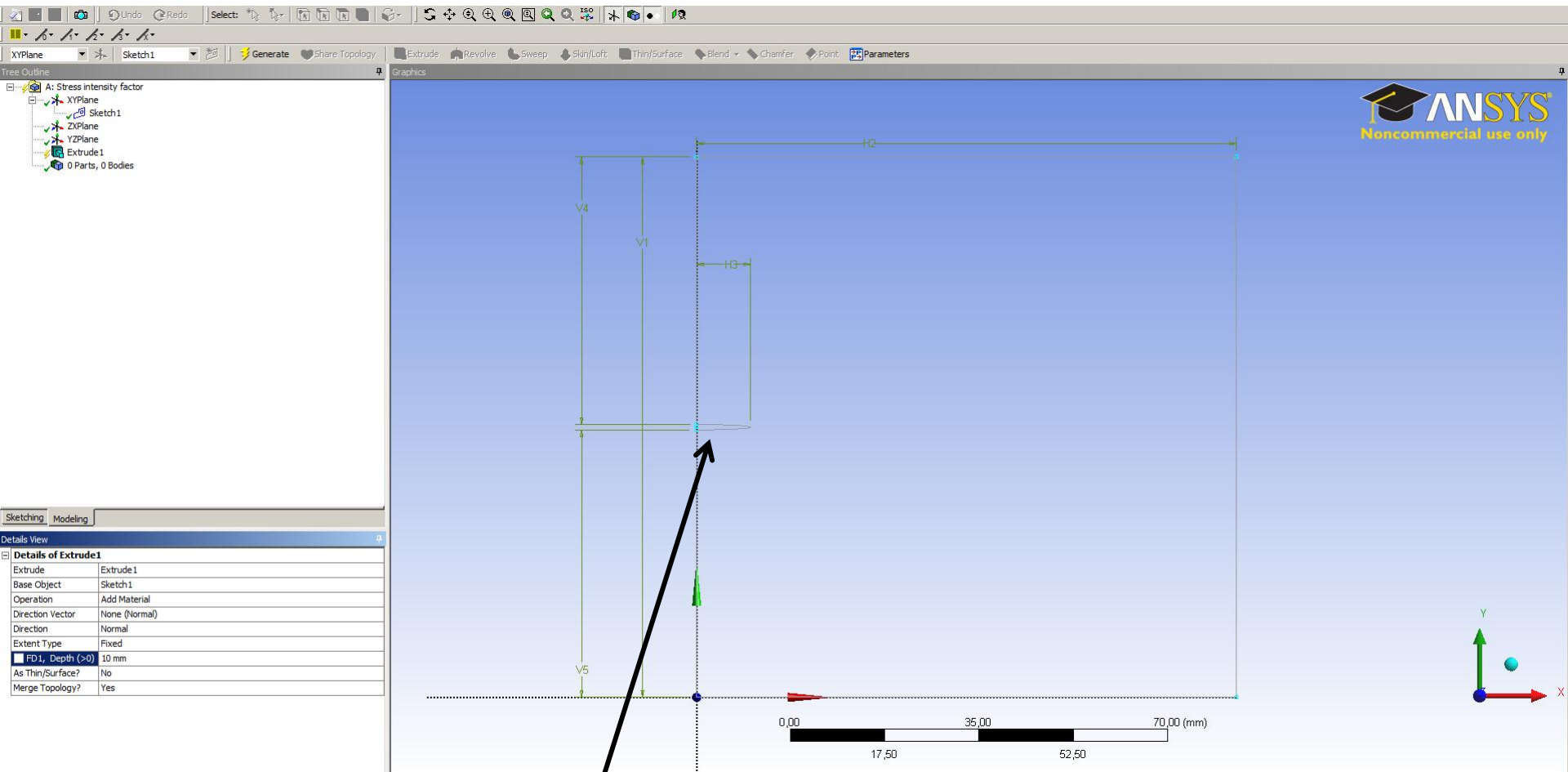
Start a new Static Structural



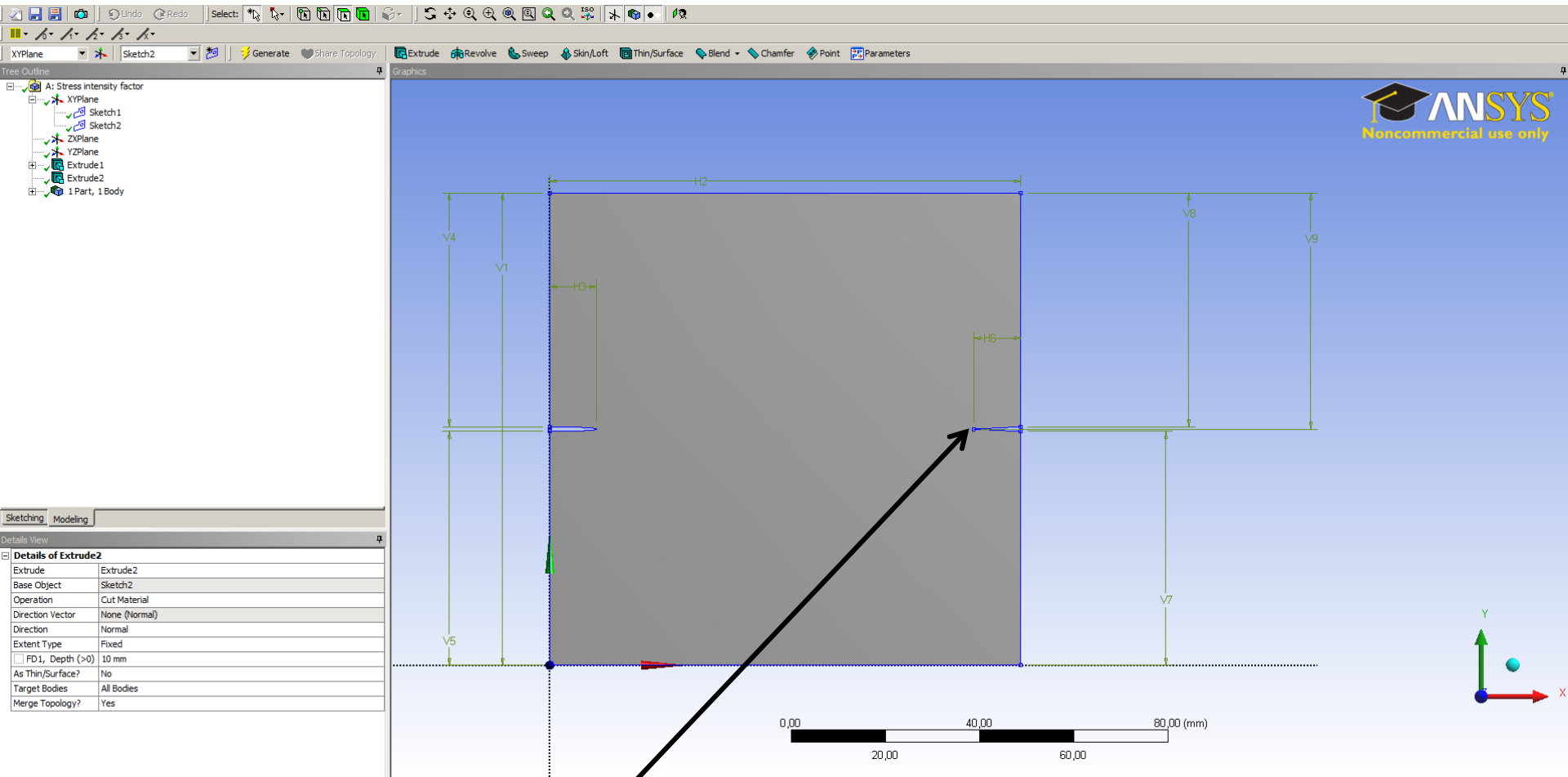
Chose mm as length unit



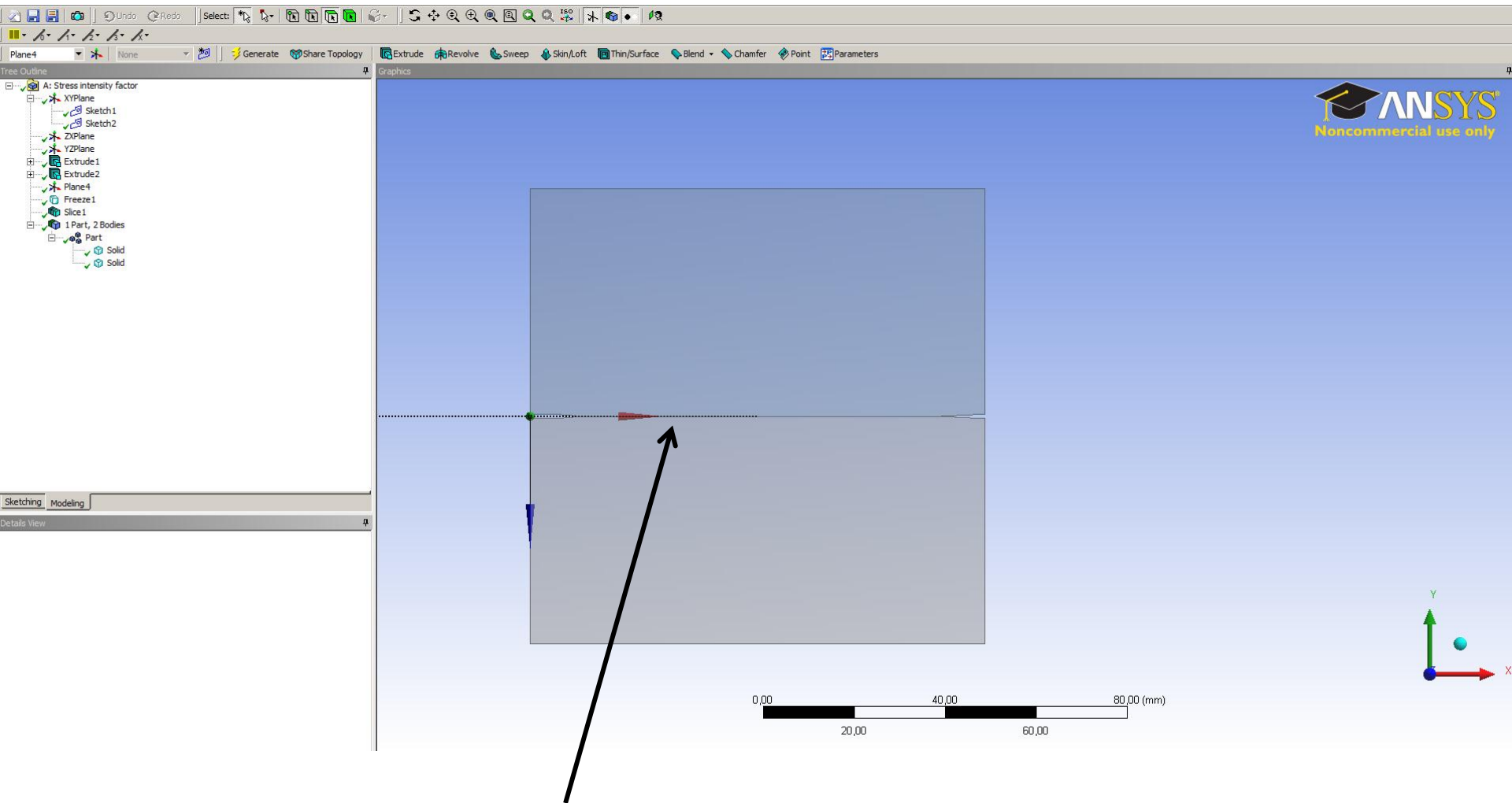
Build the geometry and model the crack



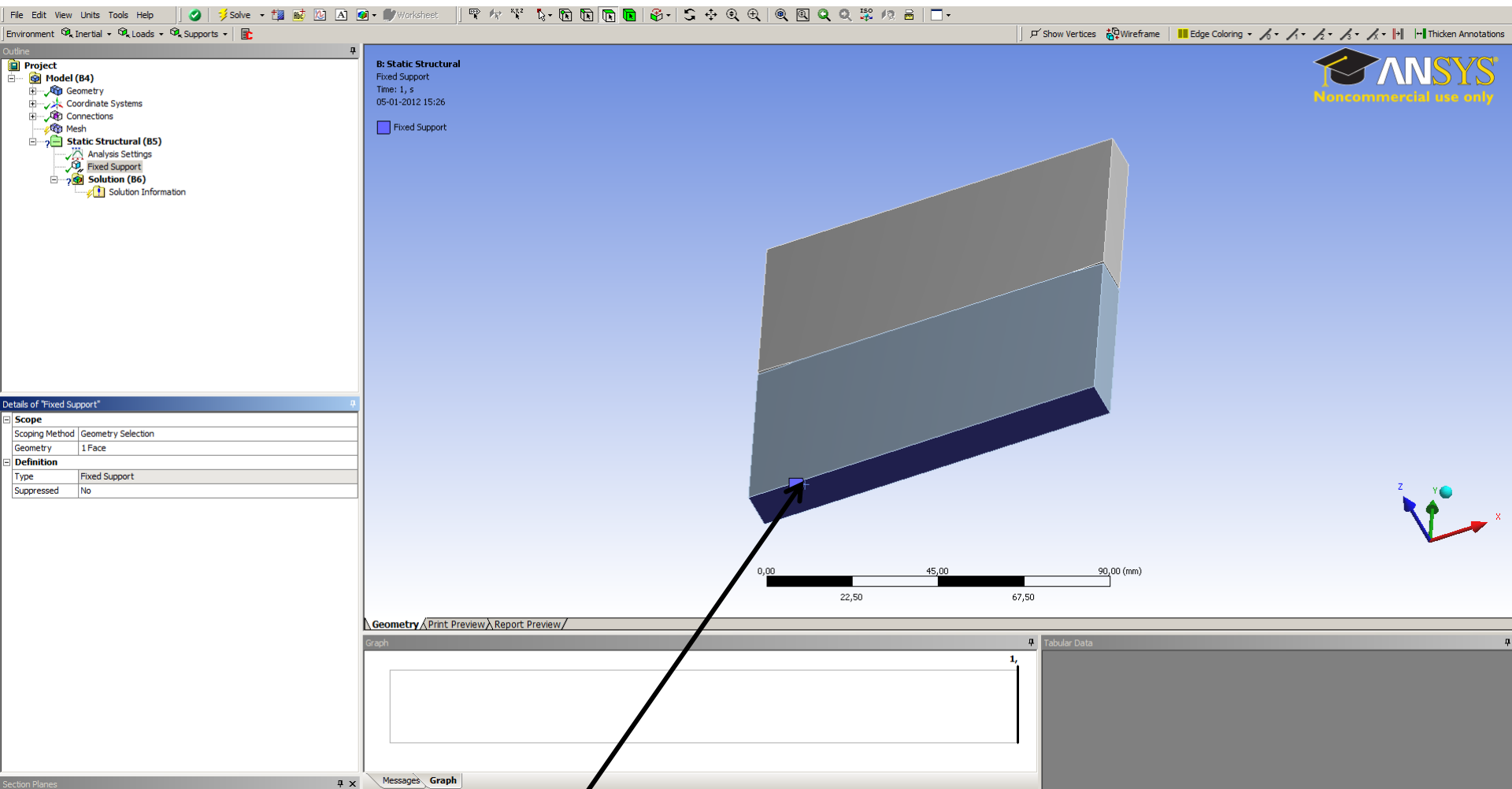
The crack on this side is made with a ellipse.



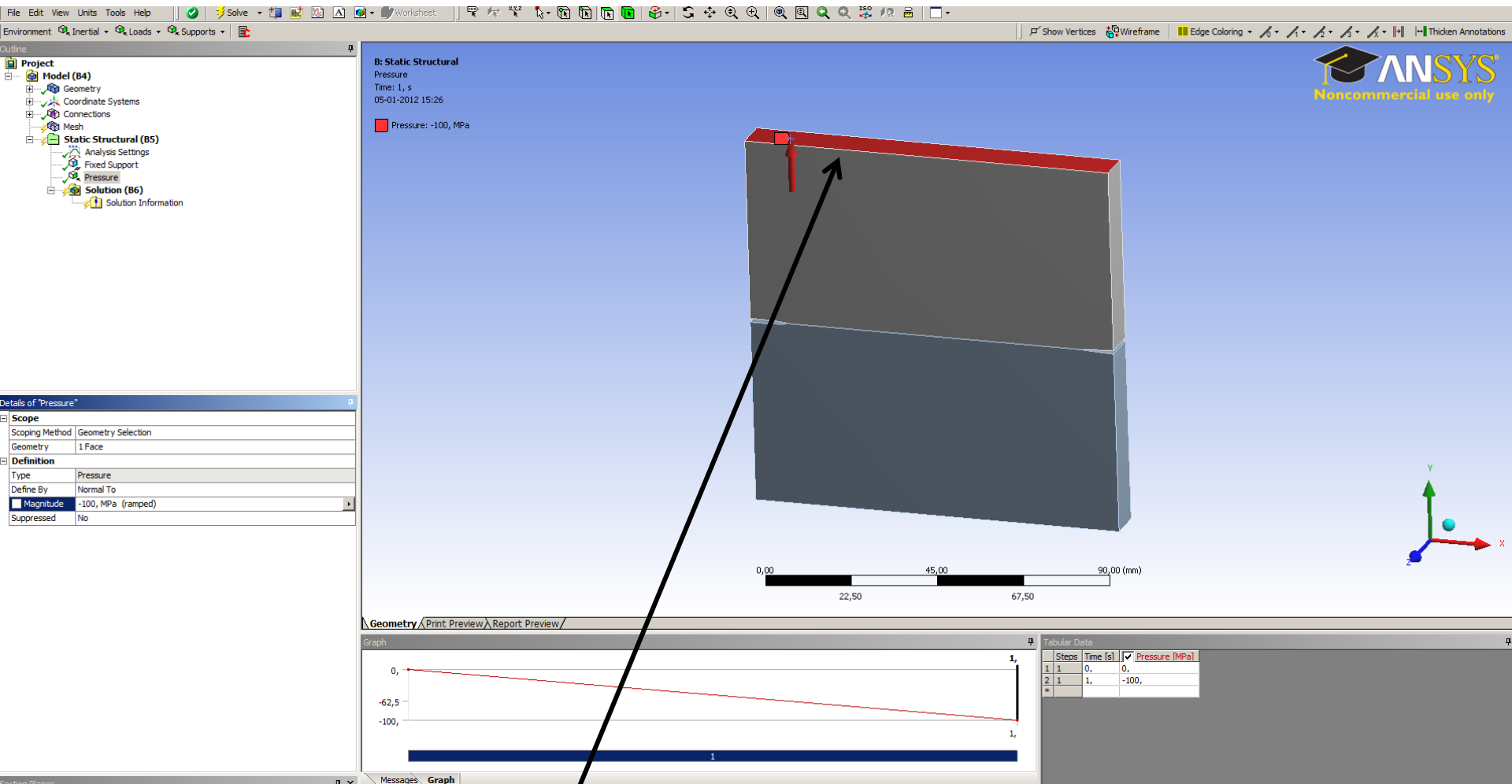
At the other side the crack is made of straight lines.



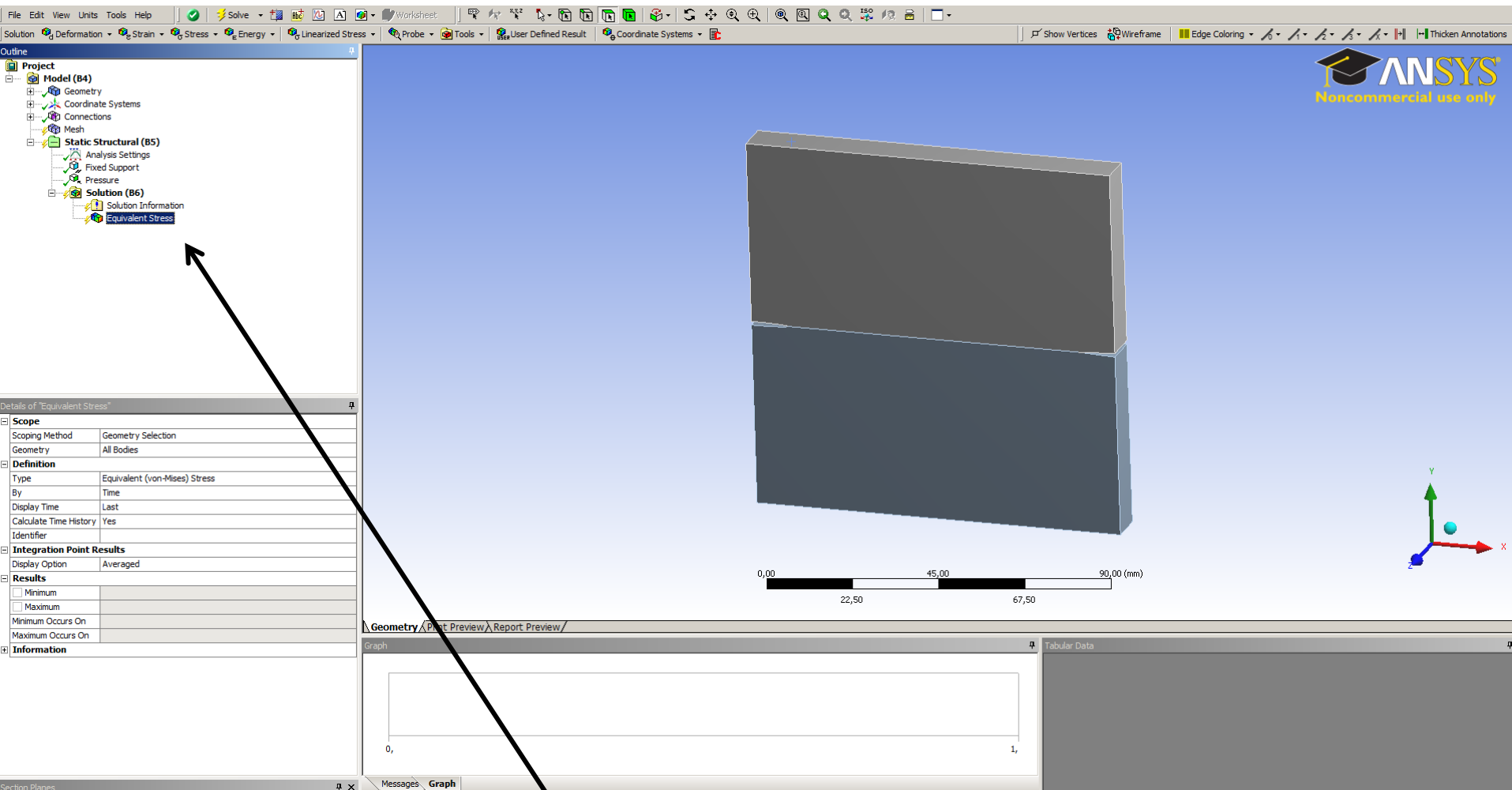
The model is sliced in two parts to get a better mesh control.



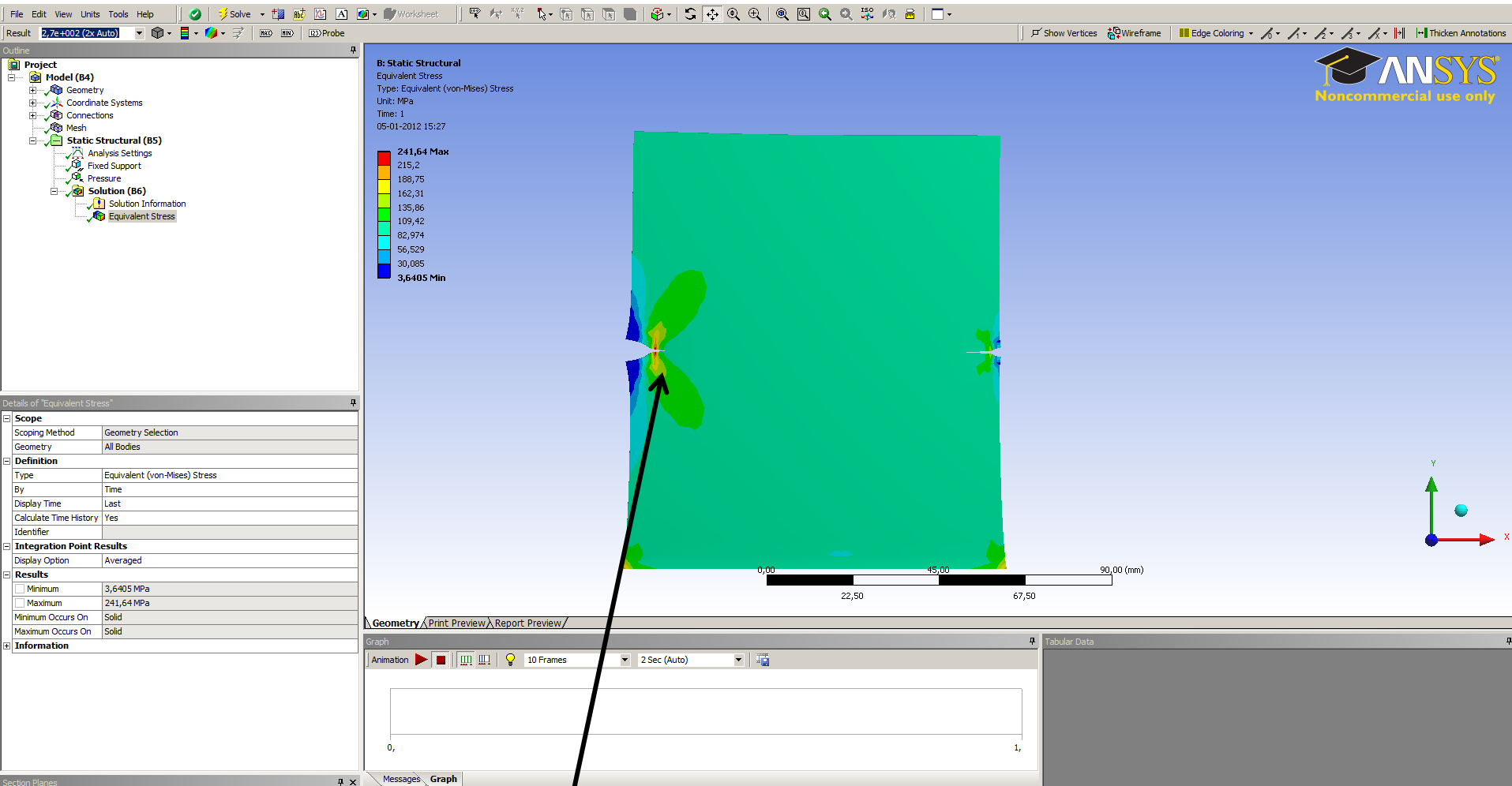
Apply a fixed supported on the model



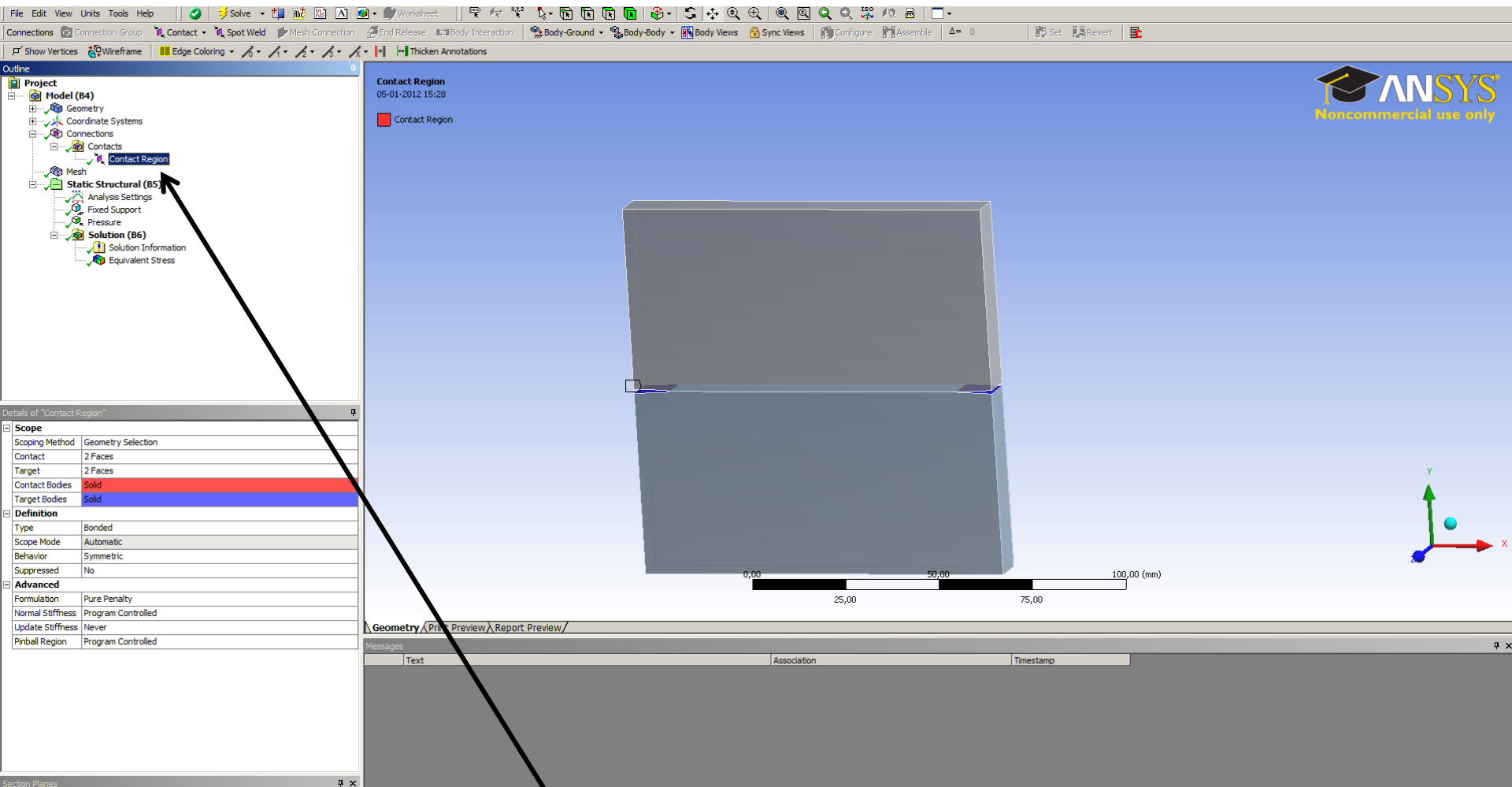
And add a pressure load at 100 MPa that the other side



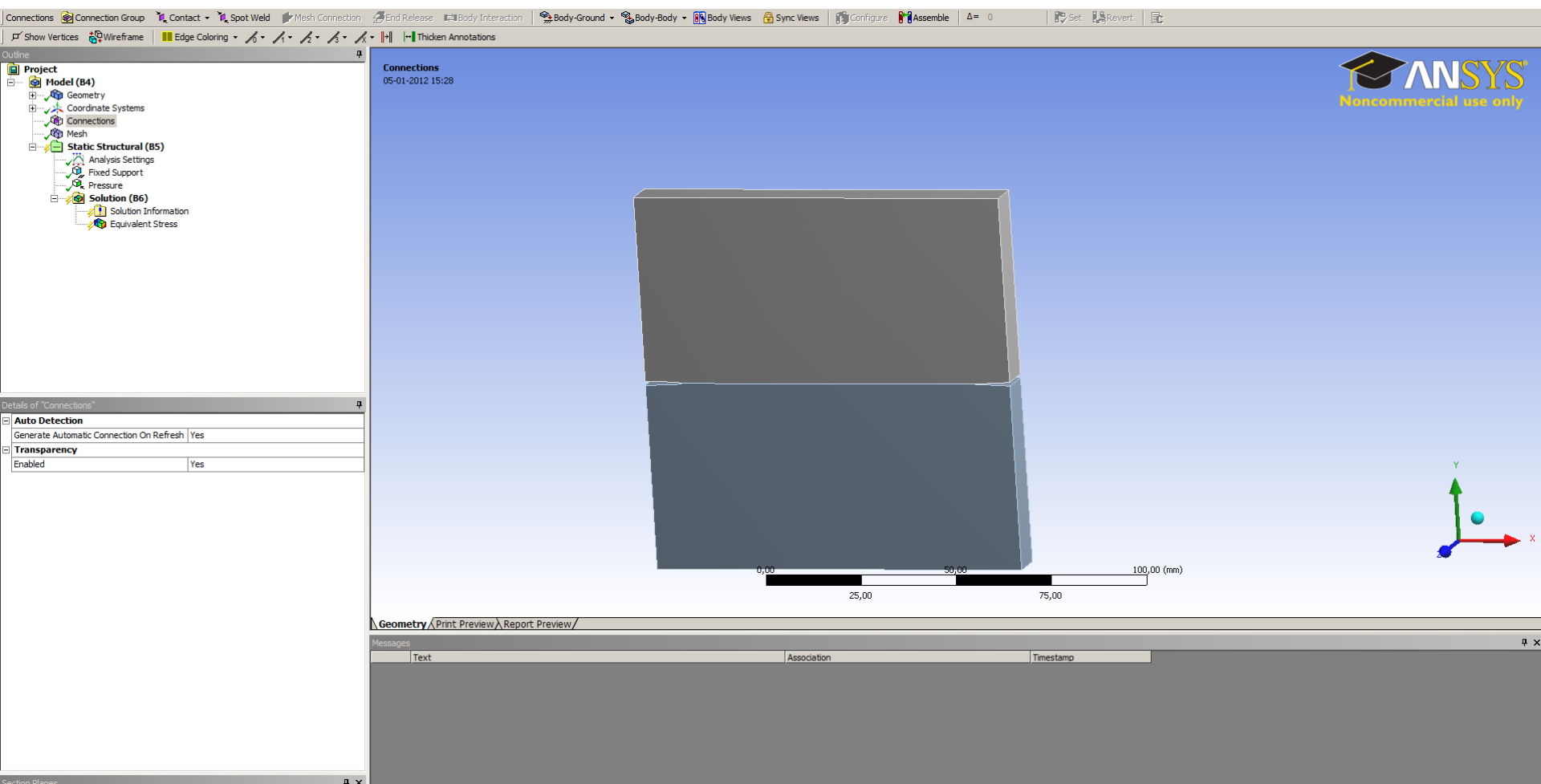
Then click the solve button and plot the stress.



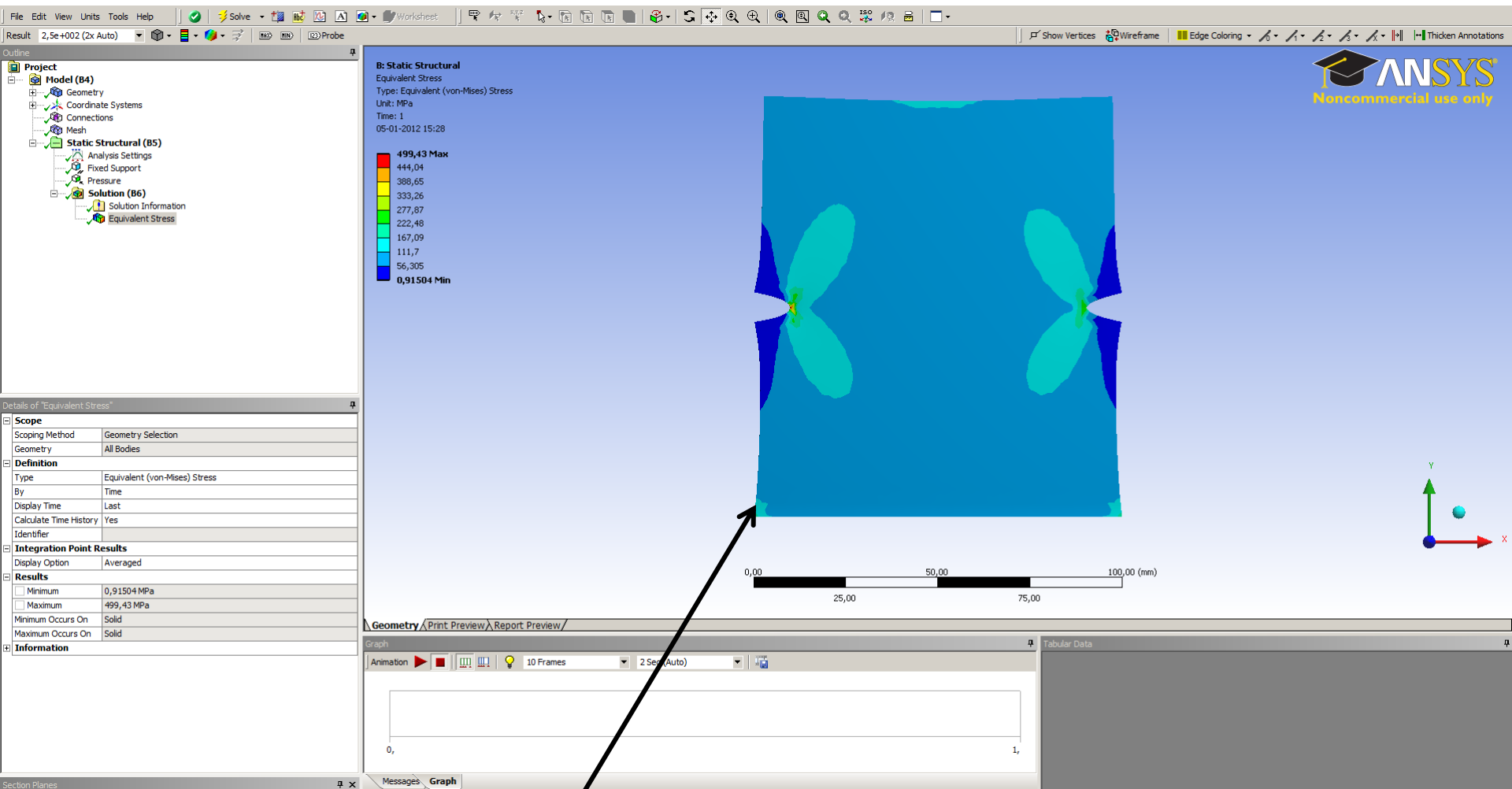
Stress distribution is not as it is expected and the deformation of the crack either. The problem is that workbench sometime add contact elements automatically and these had to be removed.



Remove the contact elements.



Then click on the solve bottom.



Now the stress and deformation is more as it is expected to be.

The screenshot displays the ANSYS software interface. On the left, the Outline tree shows a project structure with 'Model (B4)', 'Coordinate Systems', 'Global Coordinate System', 'Coordinate System', 'Connections', 'Mesh', 'Static Structural (B5)', 'Analysis Settings', 'Fixed Support', 'Pressure', 'Solution (B6)', 'Solution Information', and 'Equivalent Stress'. The 'Details of "Coordinate System"' panel is open, showing the following information:

Definition	
Type	Cartesian
Coordinate System ID	Manual
Coordinate System ID	44

Origin

Define By	Geometry Selection
Geometry	Click to Change
Origin X	90, mm
Origin Y	50, mm
Origin Z	10, mm

Principal Axis

Axis	X
Define By	Global X Axis

Orientation About Principal Axis

Axis	Y
Define By	Default

Directional Vectors

Axis	Y
Define By	Default

Transformations

Base Configuration	Absolute
Rotate Y	180, °
Transformed Configuration	[90, 50, 10,]

The main window shows a 3D model of a crack in a blue material. A coordinate system is defined with its origin at the crack tip. The X-axis is horizontal, the Z-axis is vertical, and the Y-axis is normal to the crack plane. A scale bar at the bottom indicates dimensions from 0,000 to 10,000 (mm). The ANSYS logo and 'Noncommercial use only' text are visible in the top right corner.

Then add a coordinate system for each crack. It is important that the y axis is normal to the crack plane. Also add a name for the coordinate system, in this example the name 44 is used

ANSYS Noncommercial use only

Project Outline:

- Model (B4)
 - Geometry
 - Coordinate Systems
 - Global Coordinate System
 - Coordinate System
 - Coordinate System 2
 - Connections
 - Mesh
 - Static Structural (B5)
 - Analysis Settings
 - Fixed Support
 - Pressure
 - Solution (B6)
 - Solution Information
 - Equivalent Stress

Details of "Coordinate System 2"

Definition	
Type	Cartesian
Coordinate System ID	Manual
Coordinate System ID	33

Origin	
Define By	Geometry Selection
Geometry	Click to Change
Origin X	10, mm
Origin Y	50, mm
Origin Z	10, mm

Principal Axis	
Axis	X
Define By	Global X Axis

Orientation About Principal Axis	
Axis	Y
Define By	Default

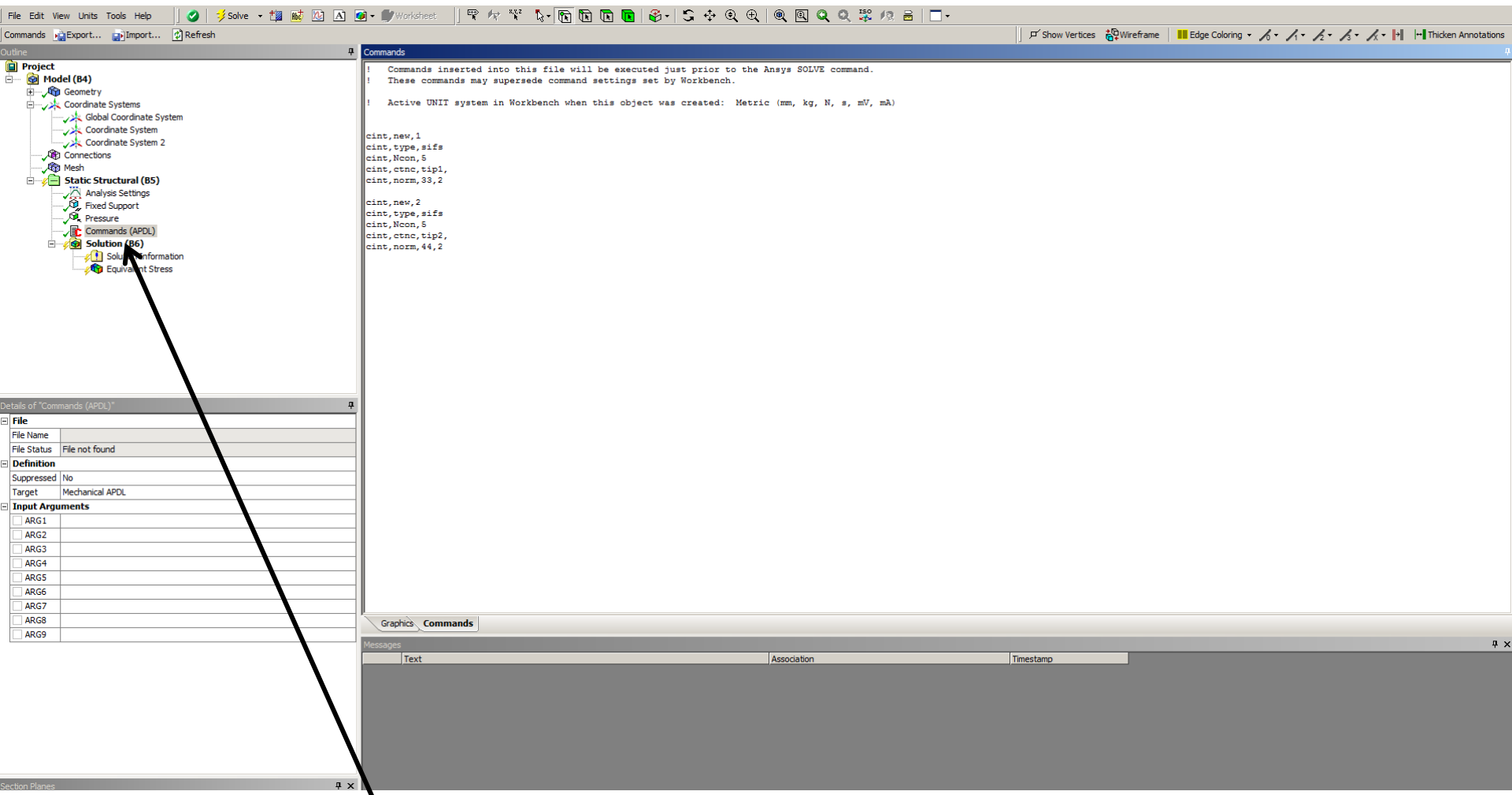
Directional Vectors	
Transformations	
Base Configuration	Absolute
Transformed Configuration	[10, 50, 10,]

Geometry / Print Preview / Report Preview

Messages

Text	Association	Timestamp
------	-------------	-----------

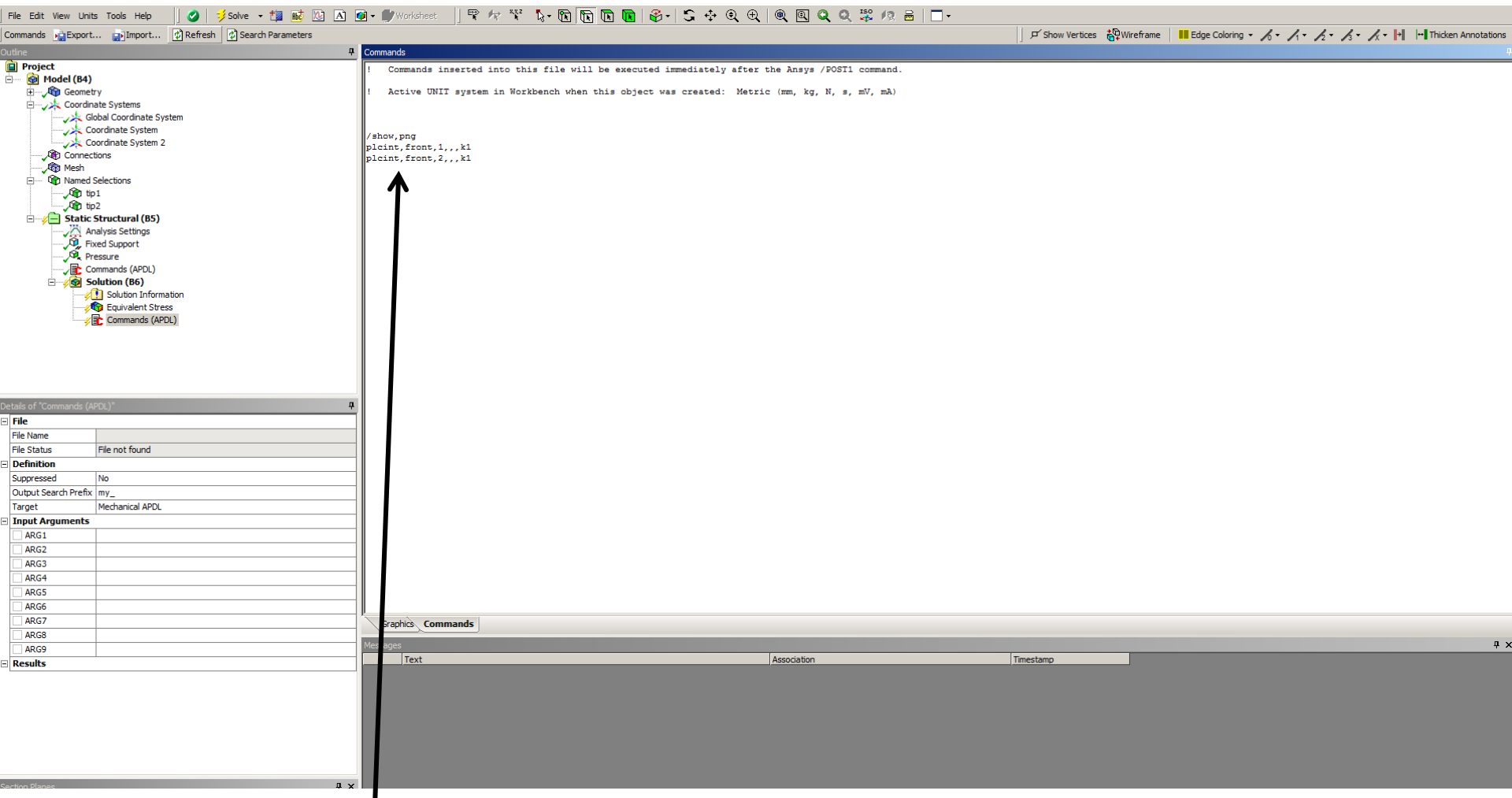
And the other crack coordinate system get the name 33



Then add a commands (APDL) under static structural. In the 2d crack tutorials there is a detail description of the commands.

The commands:

CINT,new,1	! CINT ID number.
CINT,type,sifs	! Type of calculation
CINT,norm,5	! Number of contours to be calculated.
CINT,ctnc,tip1	! Crack tip node component name
CINT,ncon,33,2	! Coordinate system number and Axis of coordinate system
CINT,new,2	
CINT,type,sifs	
CINT,norm,5	
CINT,ctnc,tip2	
CINT,ncon,44,2	



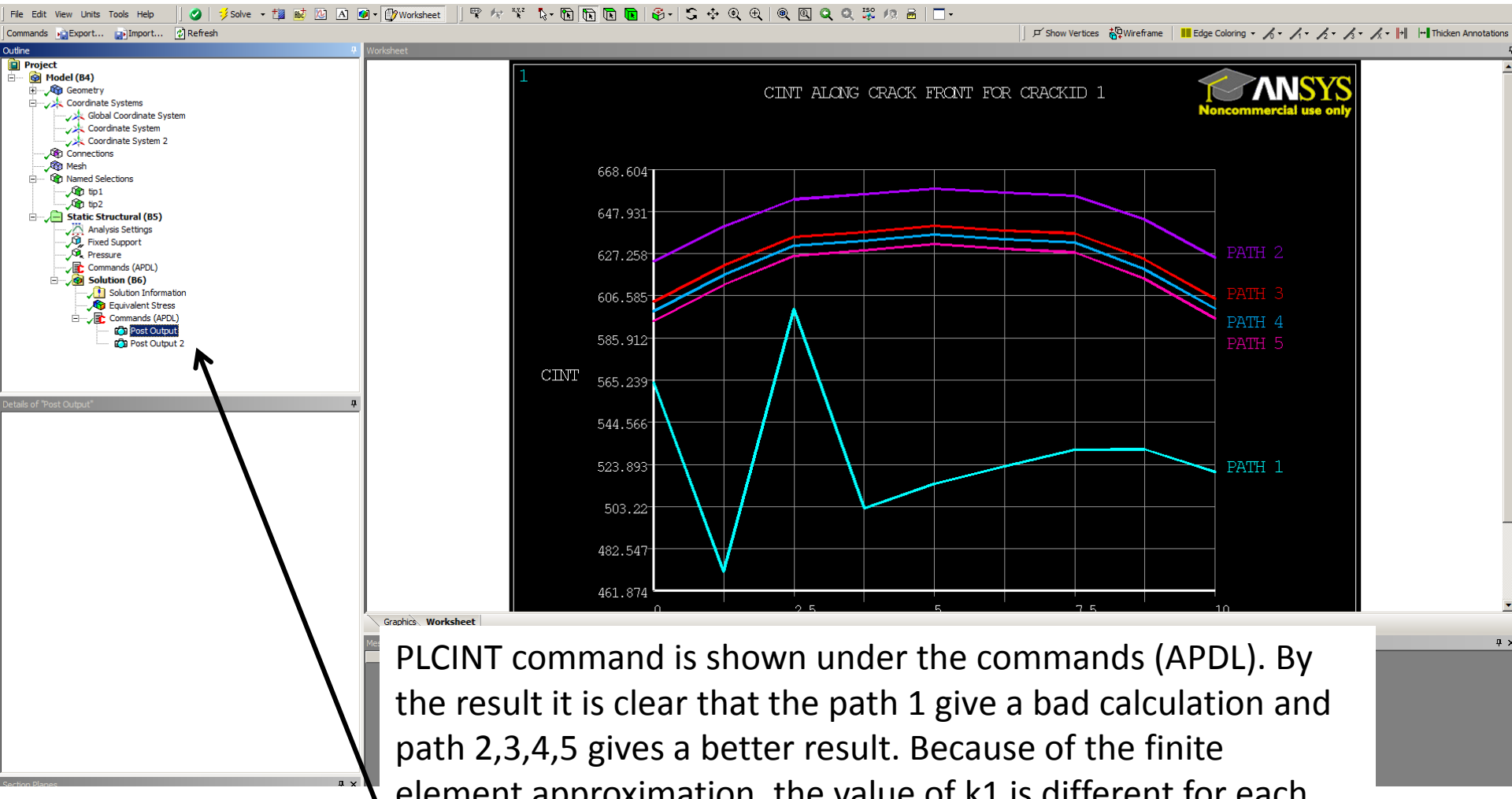
And then add a commands (APDL) to the solution and enter the commands to plot the results.

The commands:

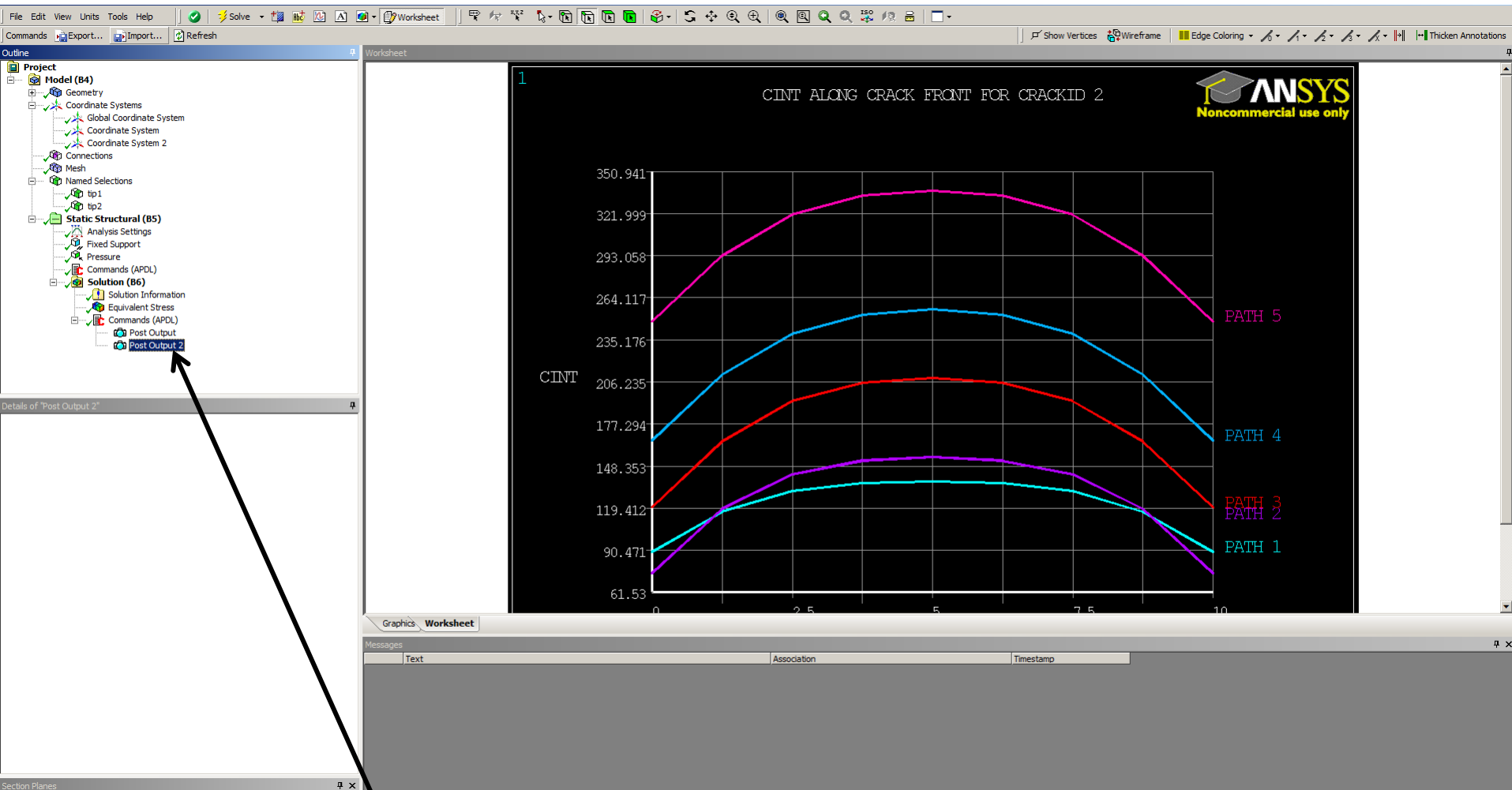
<code>/show,png</code>	! Show the PNG files
<code>PLCINT, front,1,,,k1</code>	! Plot result from the CINT commands id 1 the value of k1
<code>PLCINT, front,2,,,k1</code>	! Plot result from the CINT commands id 2 the value of k1

It is also possible to print the result in the solution information window by the command:

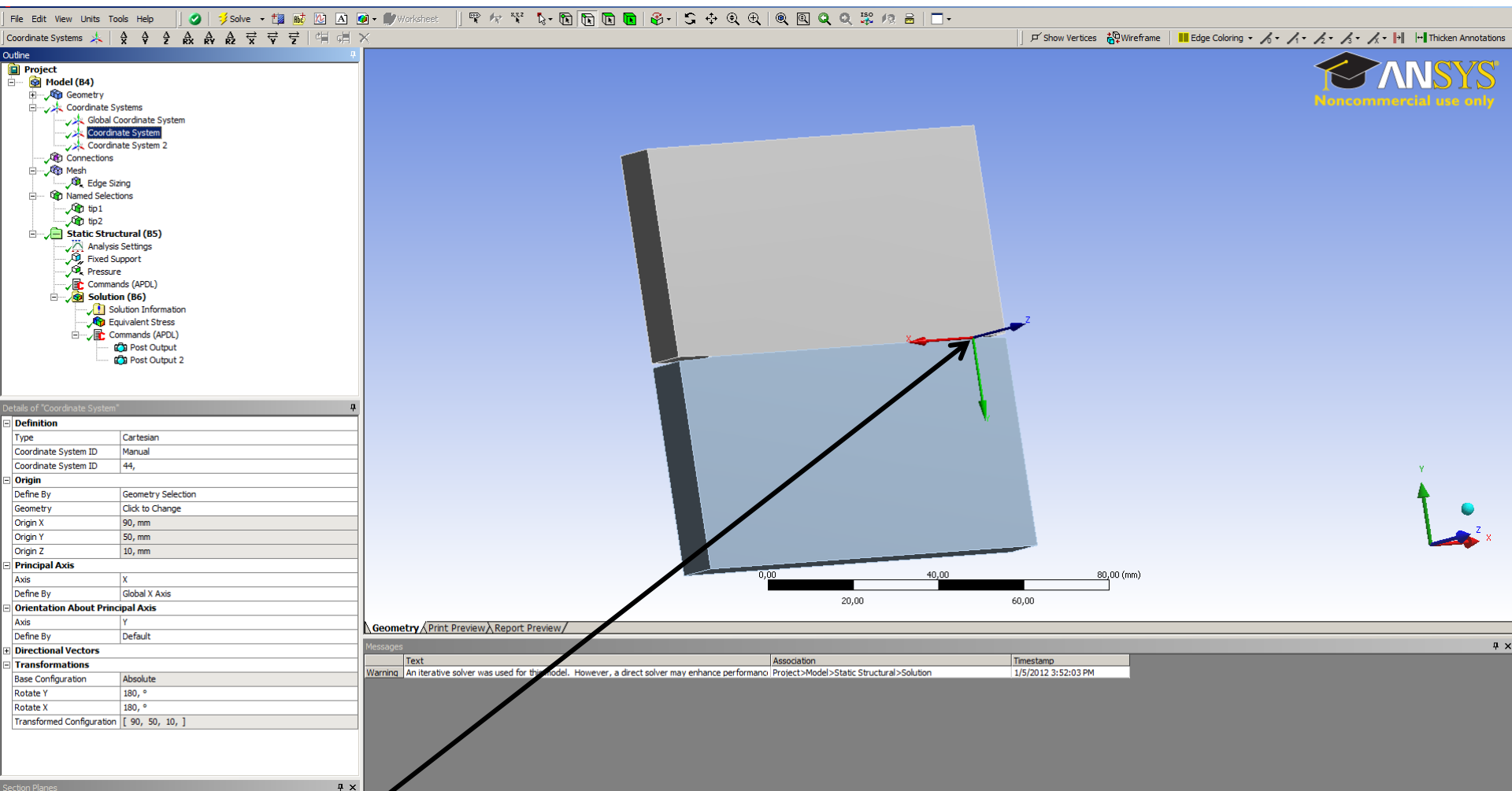
`PRCINT`



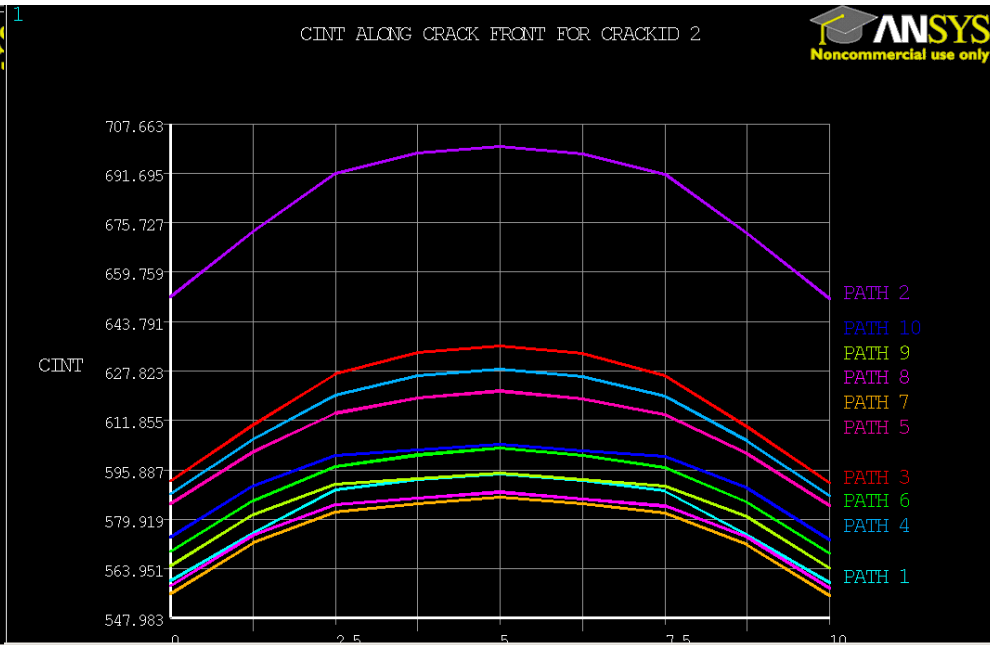
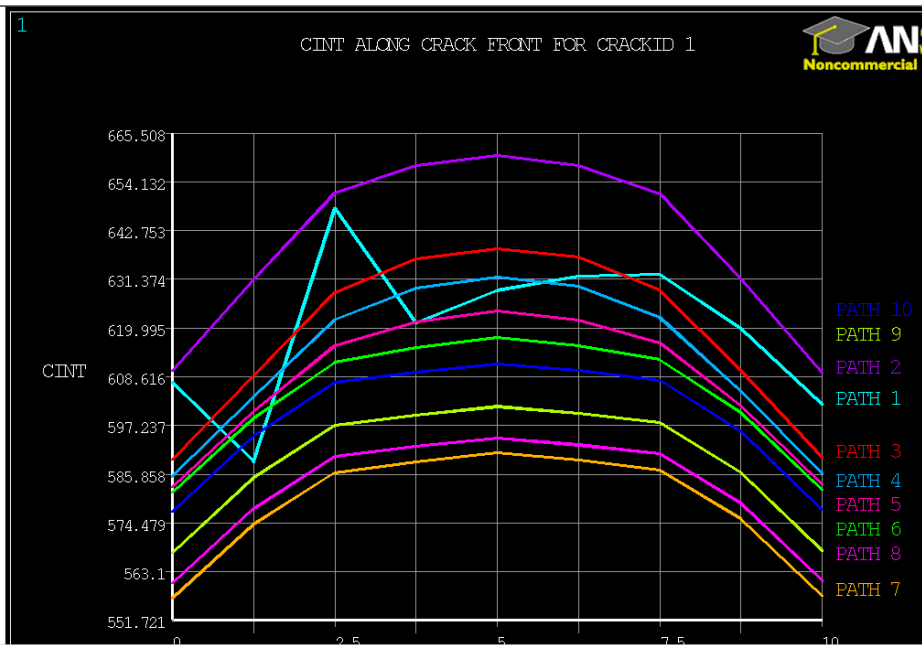
PLCINT command is shown under the commands (APDL). By the result it is clear that the path 1 give a bad calculation and path 2,3,4,5 gives a better result. Because of the finite element approximation, the value of k_1 is different for each path. The first path for K_1 is best discarded since it contains the singularity. The true value of K is usually estimated as the average of the remaining paths.



Crack calculation 2 gives a bad result. The reason is often that the y axis has to be switch because the J integral gives a negative result and therefore the K1 calculation is incorrect.



The coordinate system is changed, the contour number is set to 10.



Here is the result final result