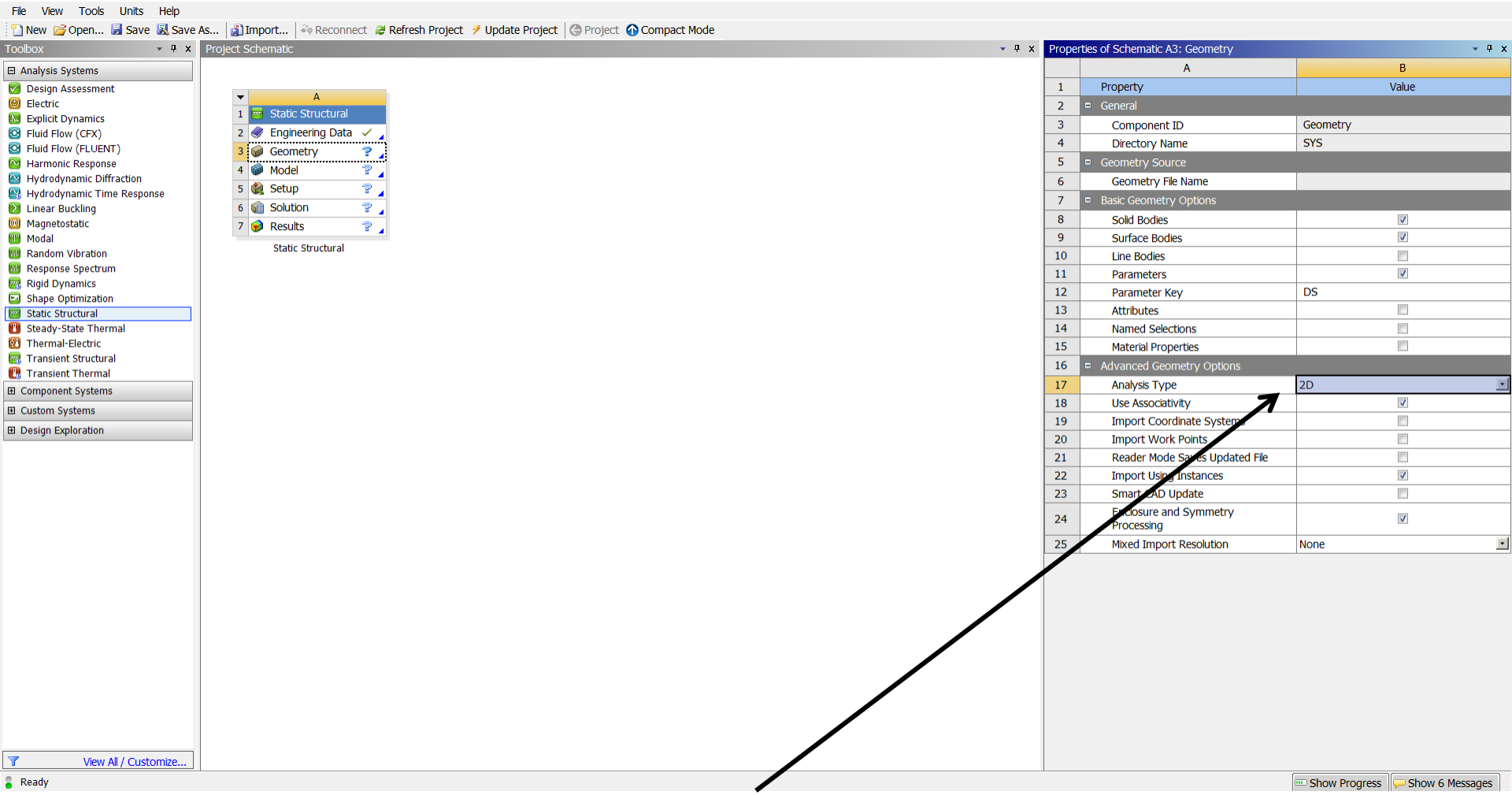
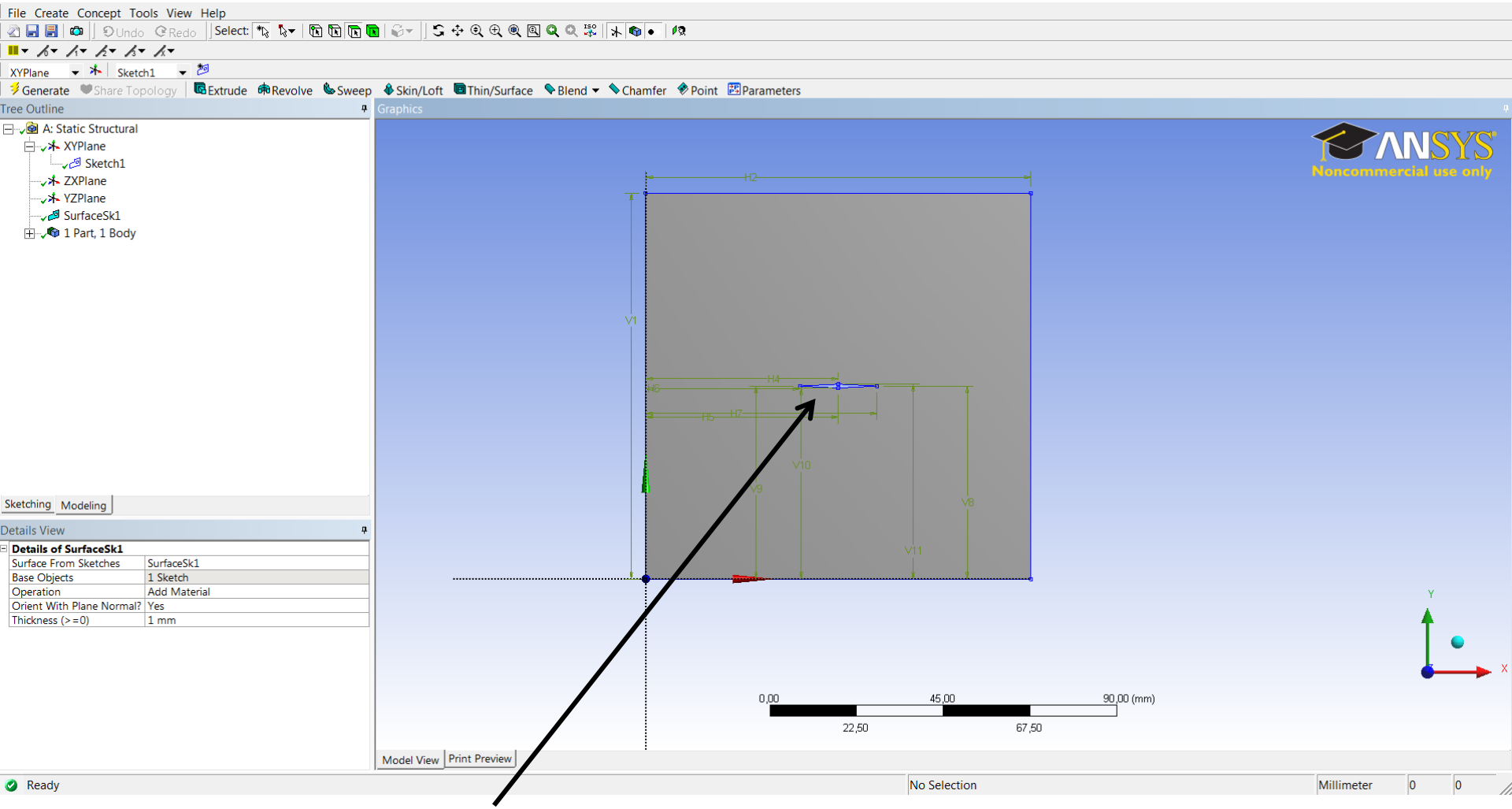
Calculation of the Stress intensity factor with CINT command in 2D Workbench 13.0

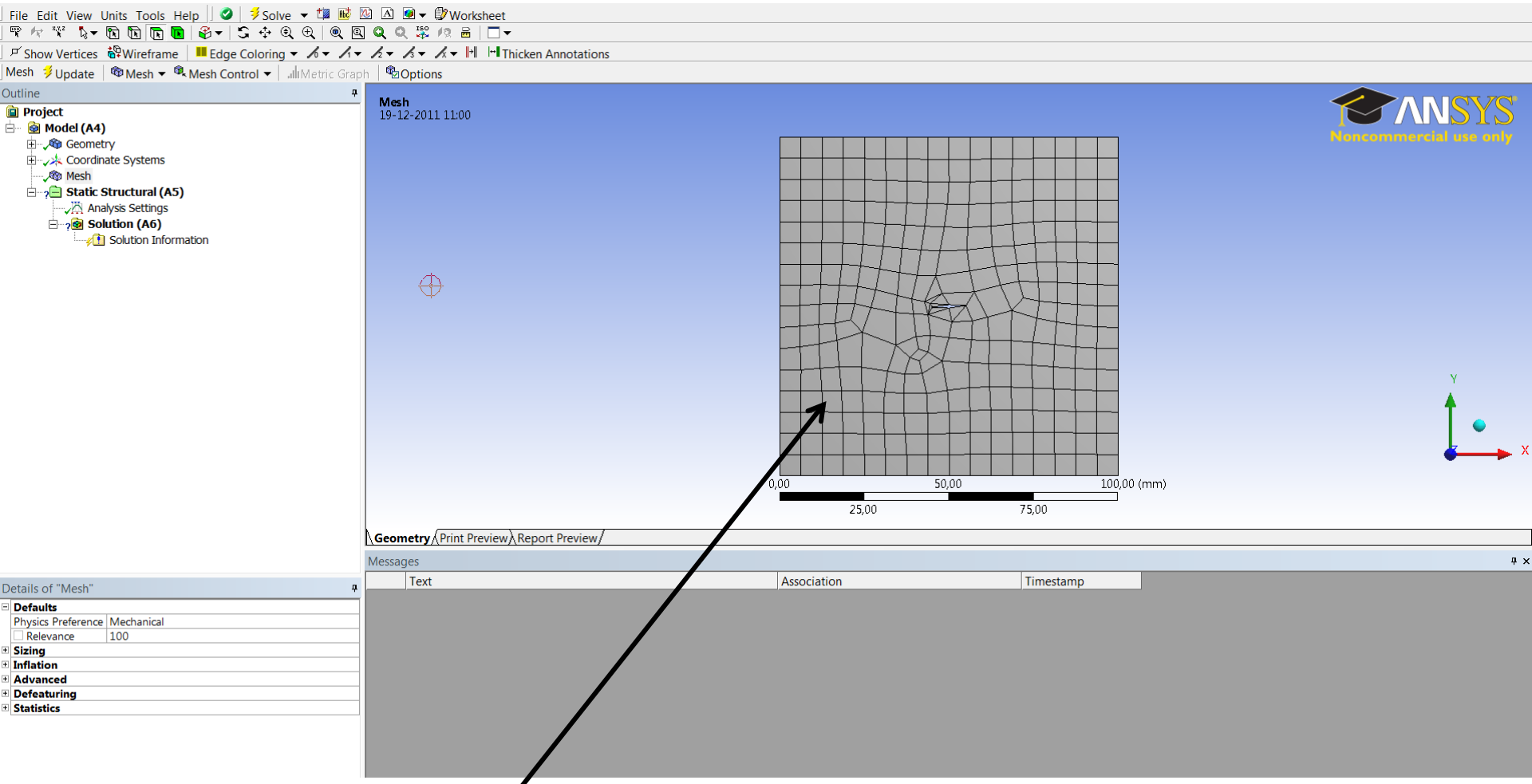
Aalborg Universitet esbjerg
Søren Heide Lambertsen



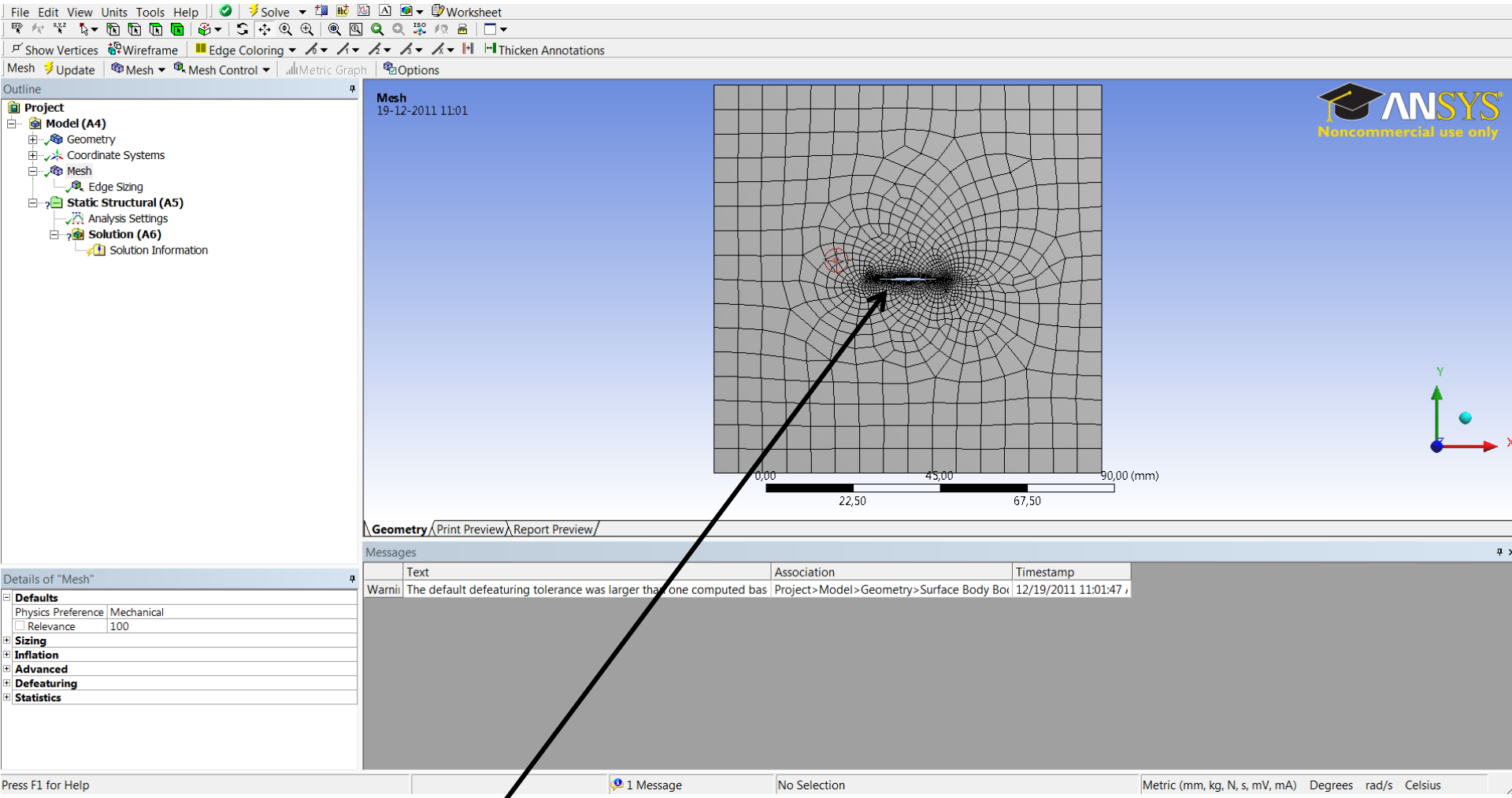
Select Analysis type to 2D and then open the Design Modeler.



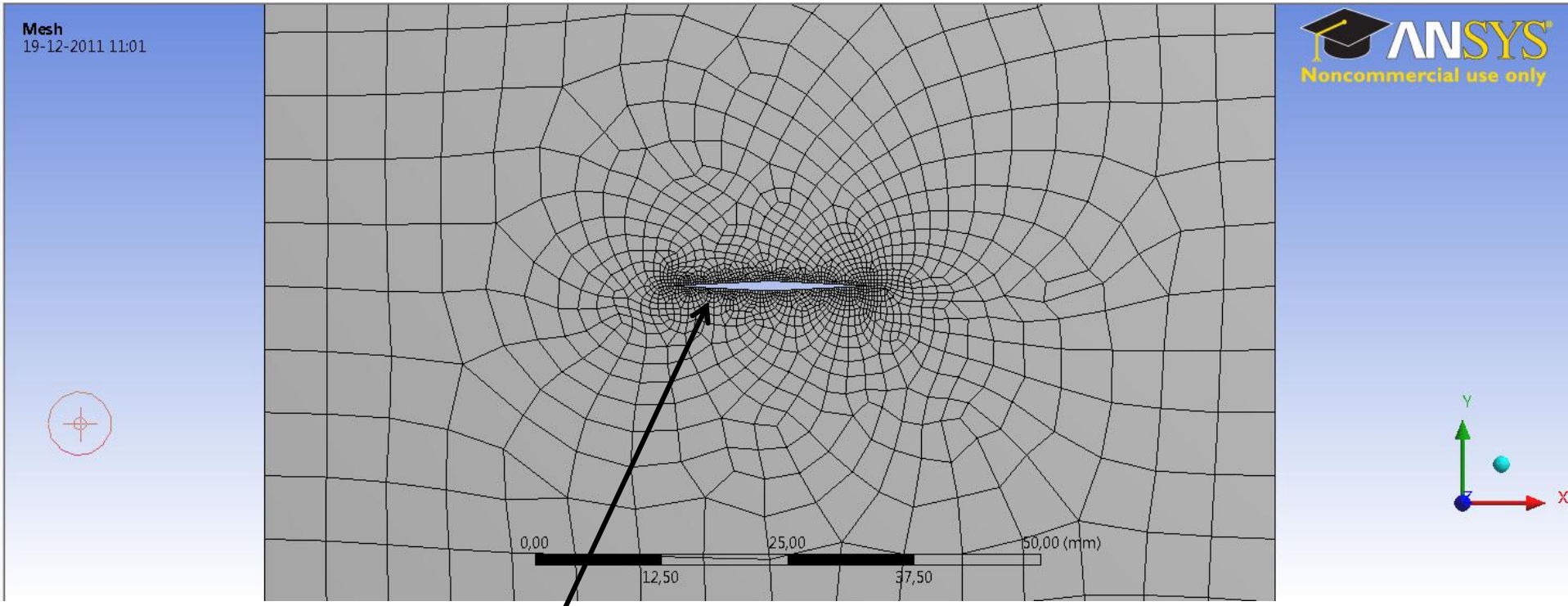
Create a crack in the geometry and close the Design modeler.



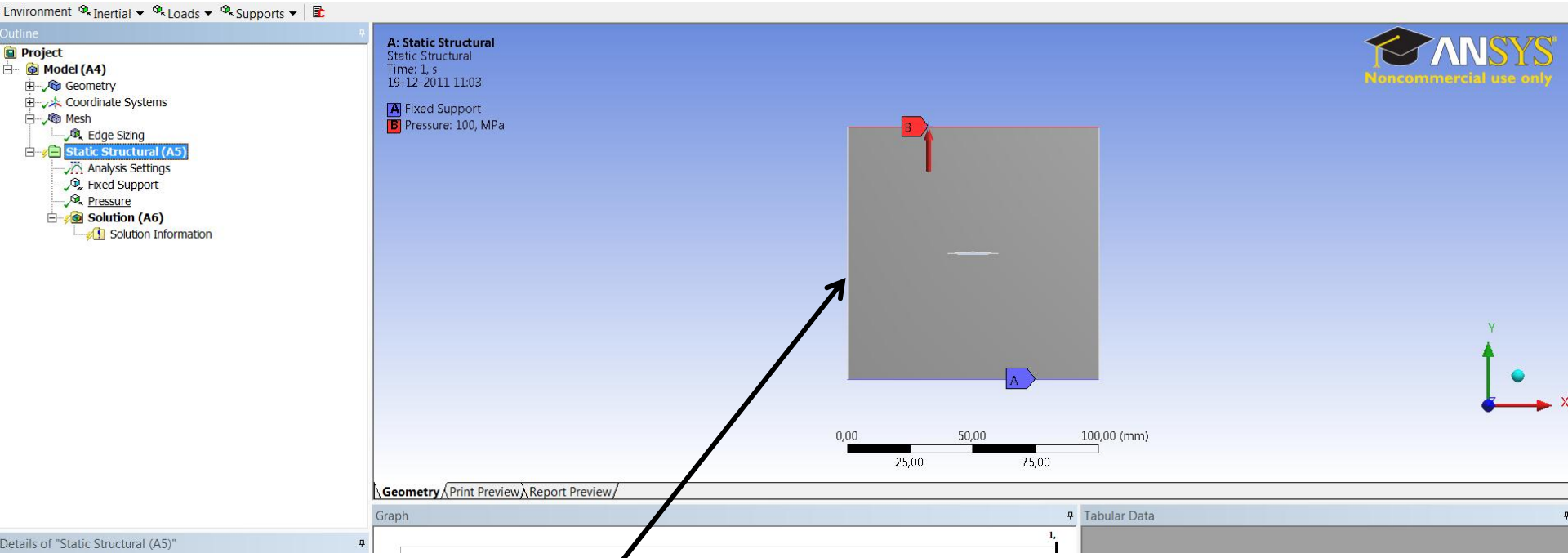
Open the model and refine the mesh. Radial/ring patterned mesh is not required.



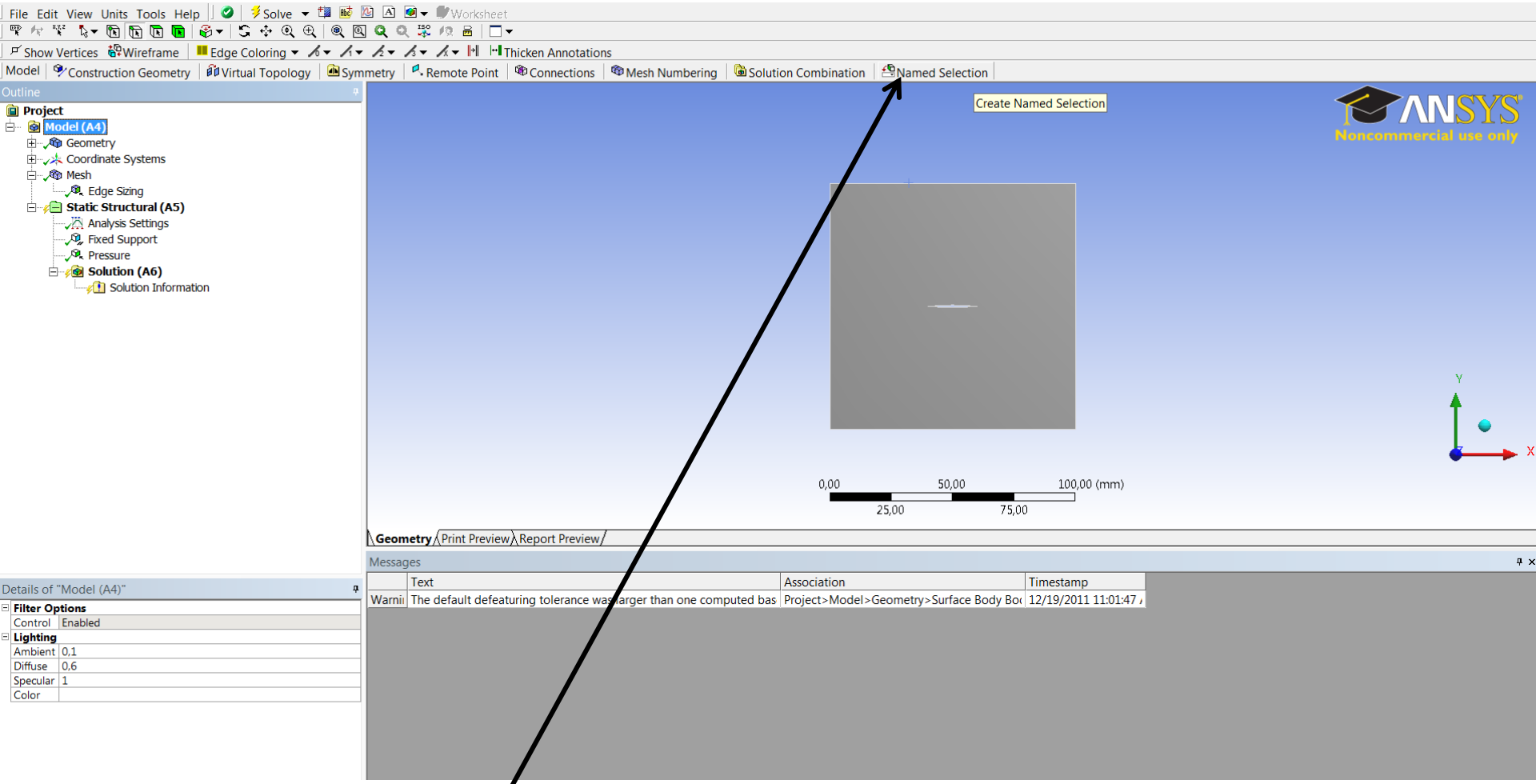
In this example a edge mesh sizing is set to 0,5 mm and behavior to Hard.


Mesh
19-12-2011 11:01
ANSYS
Noncommercial use only
0,00
12,50
25,00
37,50
50,00 (mm)
Y
X

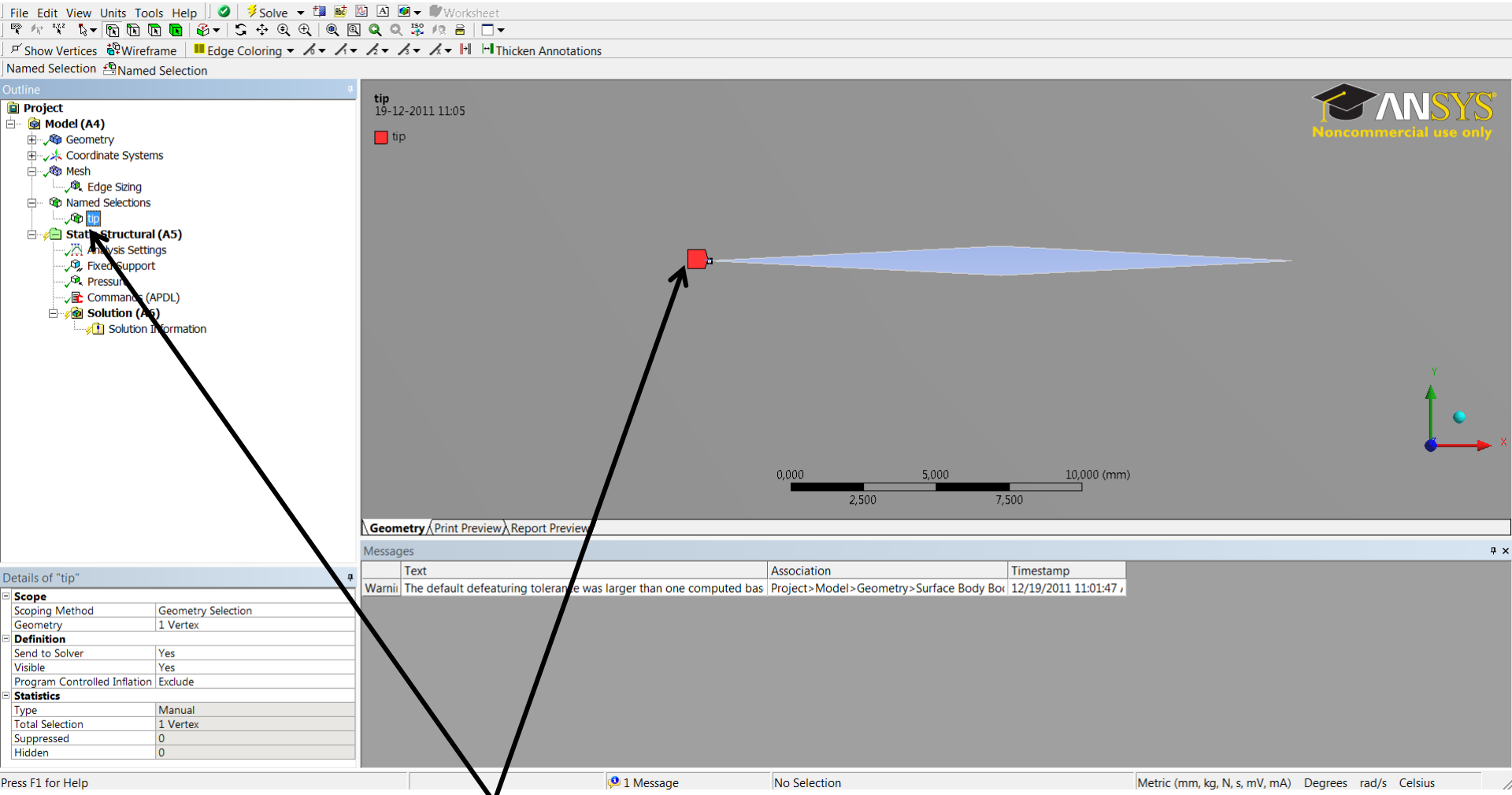
The mesh.



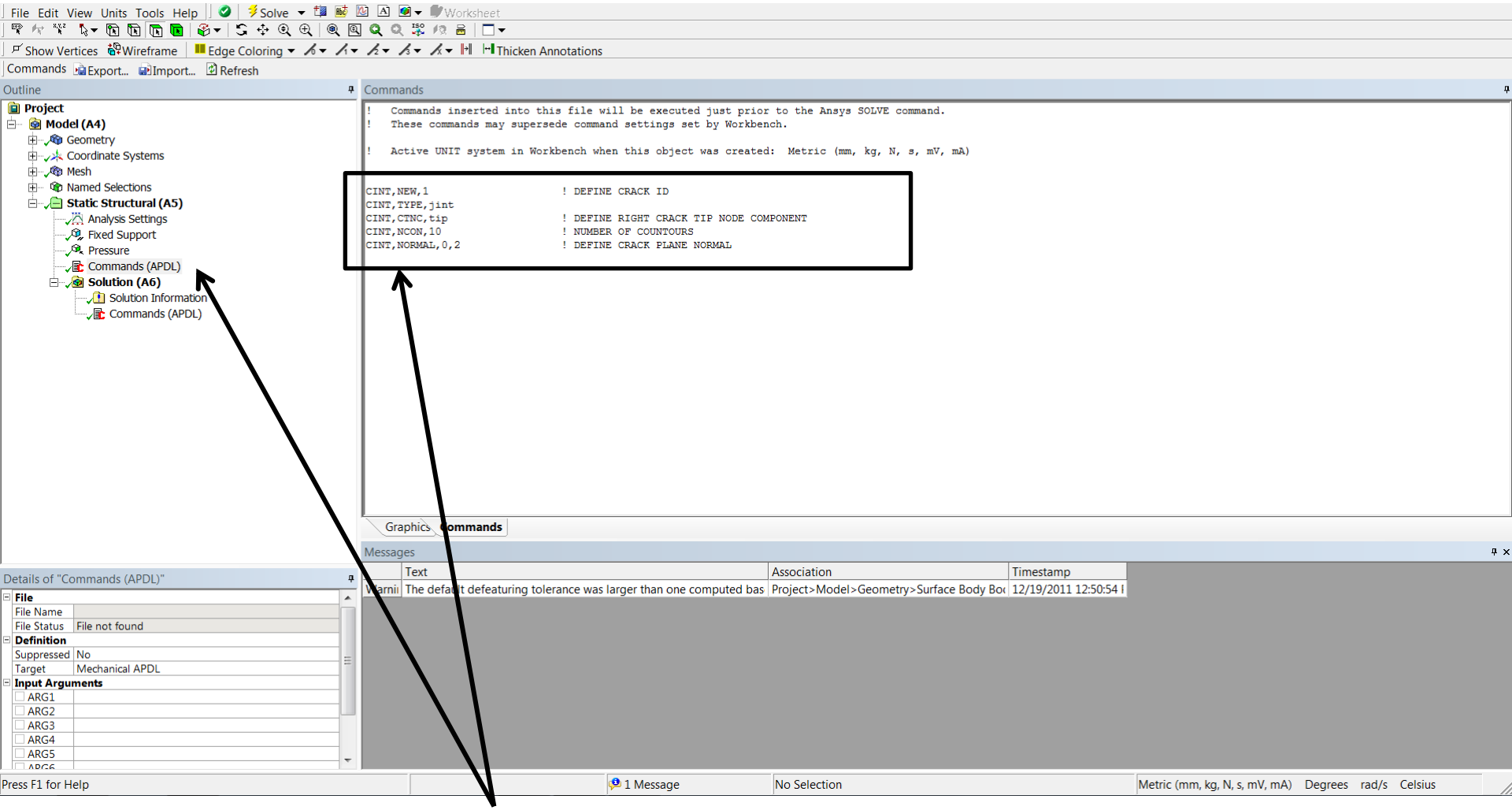
Then set the boundary condition.



Select Named Selection.



Select the crack tip and rename the Selection.



Start a Command window and write the CINT commands.

Start a new calculation and name the calculation.

Cint,new,1

Enter the type of calculation. In this example the valve of the j integral is calculated.

Cint,type,jint

Define the crack tip, in this example the crack tip is named tip

Cint,ctnc,tip

Enter the coordinate system her the x=1 y=2 z=3. In this example the coordinate is 0 with is the default coordinate system and 2 is the y axis with is normal to the crack direction.

Cint,normal,0,2

There are many other parameters there can be changed, in the example is command is the simplest setup possible. A extra command can be the contours number below the number is set to 10.

Cint,ncon,10

Start a new calculation and name the calculation.

Cint,new,1

Enter the type of calculation. In this example the valve of the j integral is calculated.

Cint,type,jint

Define the crack tip, in this example the crack tip is named tip

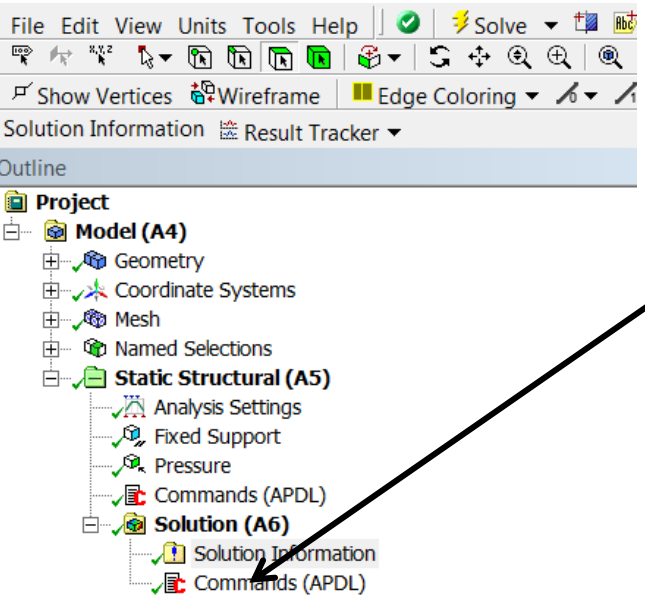
Cint,ctnc,tip

Enter the coordinate system her the x=1 y=2 z=3. In this example the coordinate is 0 with is the default coordinate system and 2 is the y axis with is normal to the crack direction.

Cint,normal,0,2

There are many other parameters there can be changed, in the example is command is the simplest setup possible. A extra command can be the contours number below the number is set to 10.

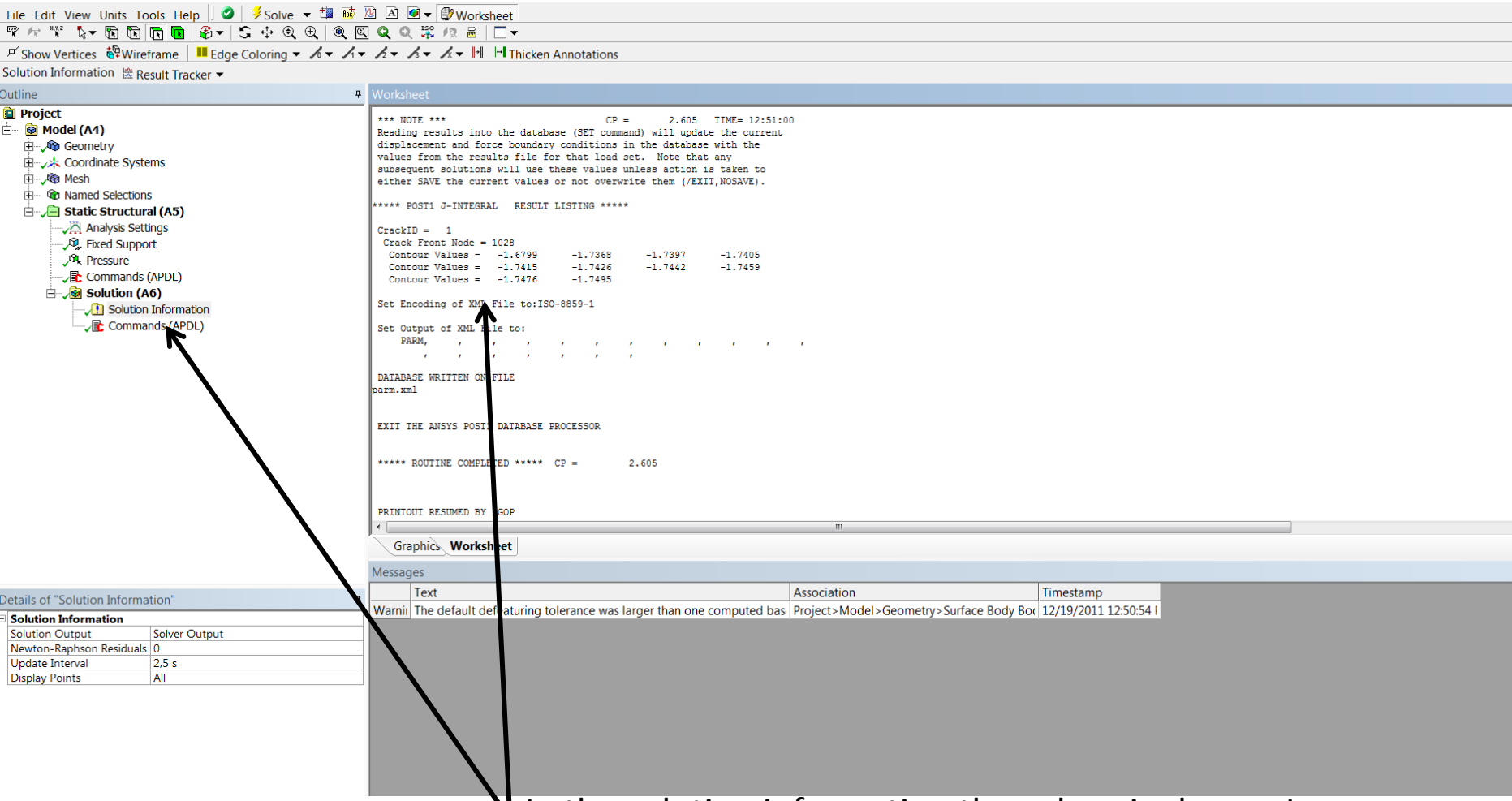
Cint,ncon,10



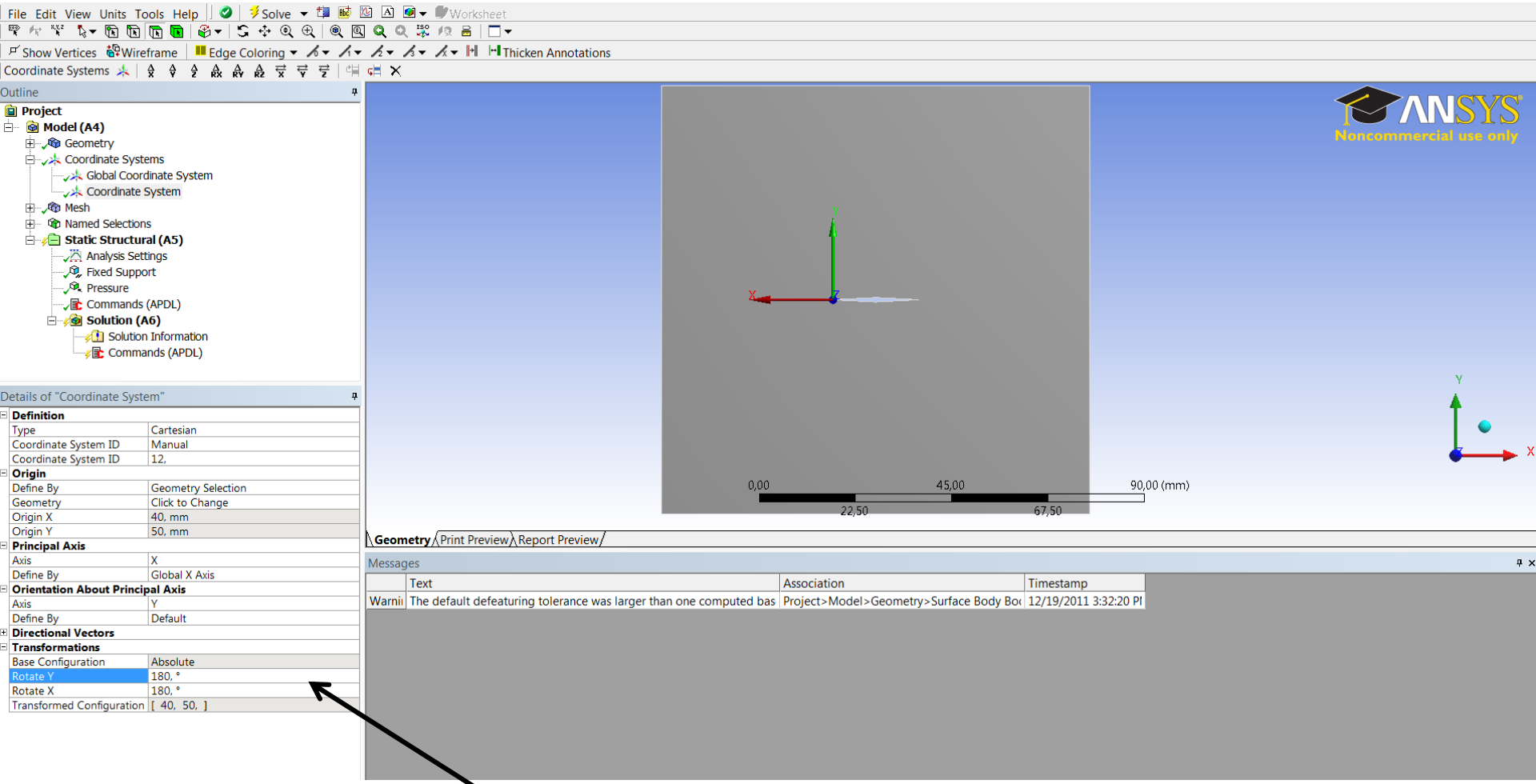
Start a command window under solution and enter the command for printing the j integral solution.

To print the calculation number 1

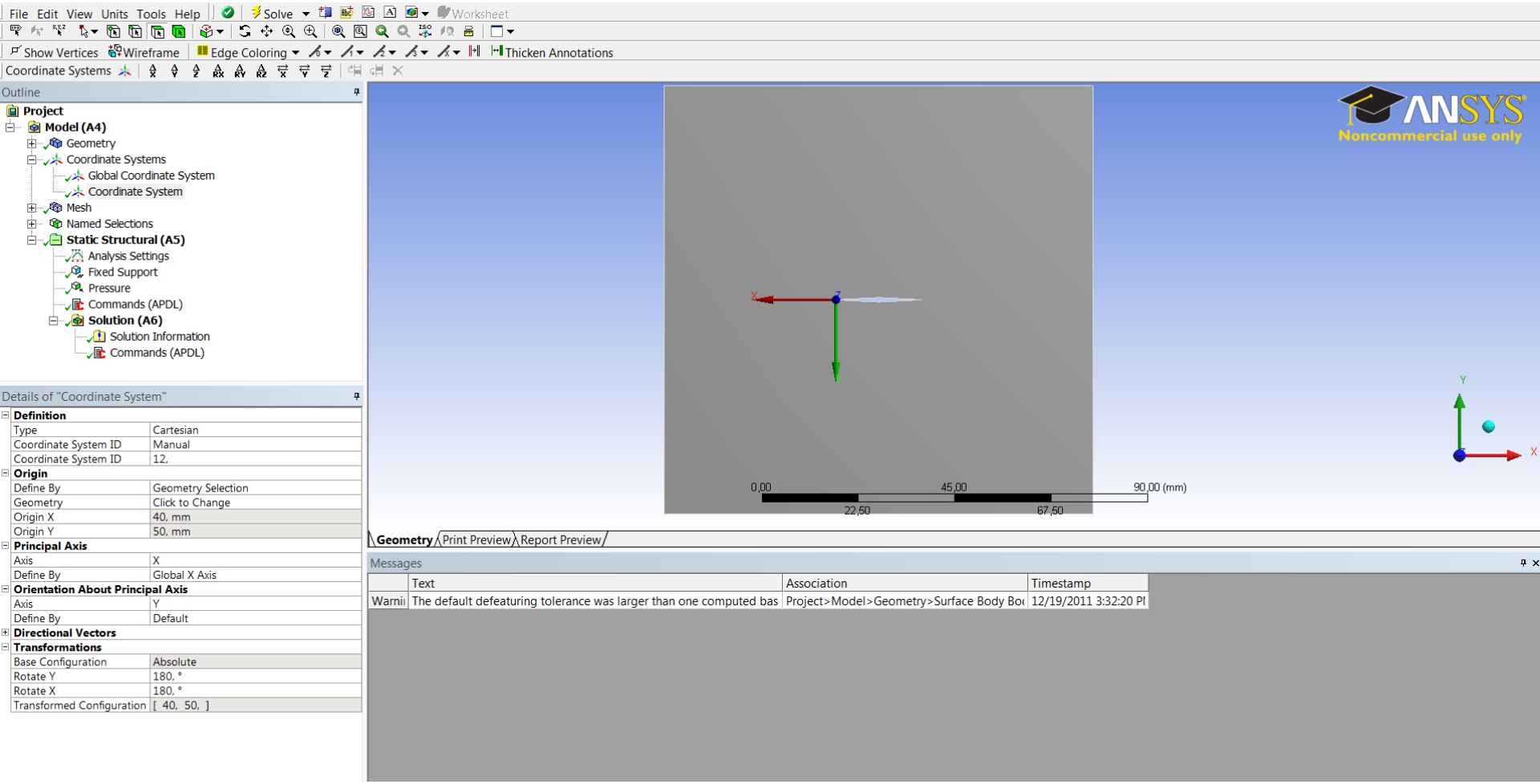
Print,1,,JINT



In the solution information the values is shown. In the example the values is negatives, to changes this switch the normal plane direction. How to se below.



Changes the direction of y axis.


File Edit View Units Tools Help
Solve
Worksheet
Show Vertices
Wireframe
Edge Coloring
Thicken Annotations
Coordinate Systems
Outline
Project
Model (A4)
Geometry
Coordinate Systems
Global Coordinate System
Coordinate System
Mesh
Named Selections
Static Structural (A5)
Analysis Settings
Fixed Support
Pressure
Commands (APDL)
Solution (A6)
Solution Information
Commands (APDL)
Details of "Coordinate System"
Definition
Type Cartesian
Coordinate System ID Manual
Coordinate System ID 12,
Origin
Define By Geometry Selection
Geometry Click to Change
Origin X 40, mm
Origin Y 50, 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
Rotate Y 180, °
Rotate X 180, °
Transformed Configuration [40, 50,]
ANSYS
Noncommercial use only
0,00
45,00
90,00 (mm)
22,50
67,50
Geometry
Print Preview
Report Preview
Messages
Text
Association
Timestamp
Warni
The default defeaturing tolerance was larger than one computed bas
Project>Model>Geometry>Surface Body Bo
12/19/2011 3:32:20 PI

Like this.

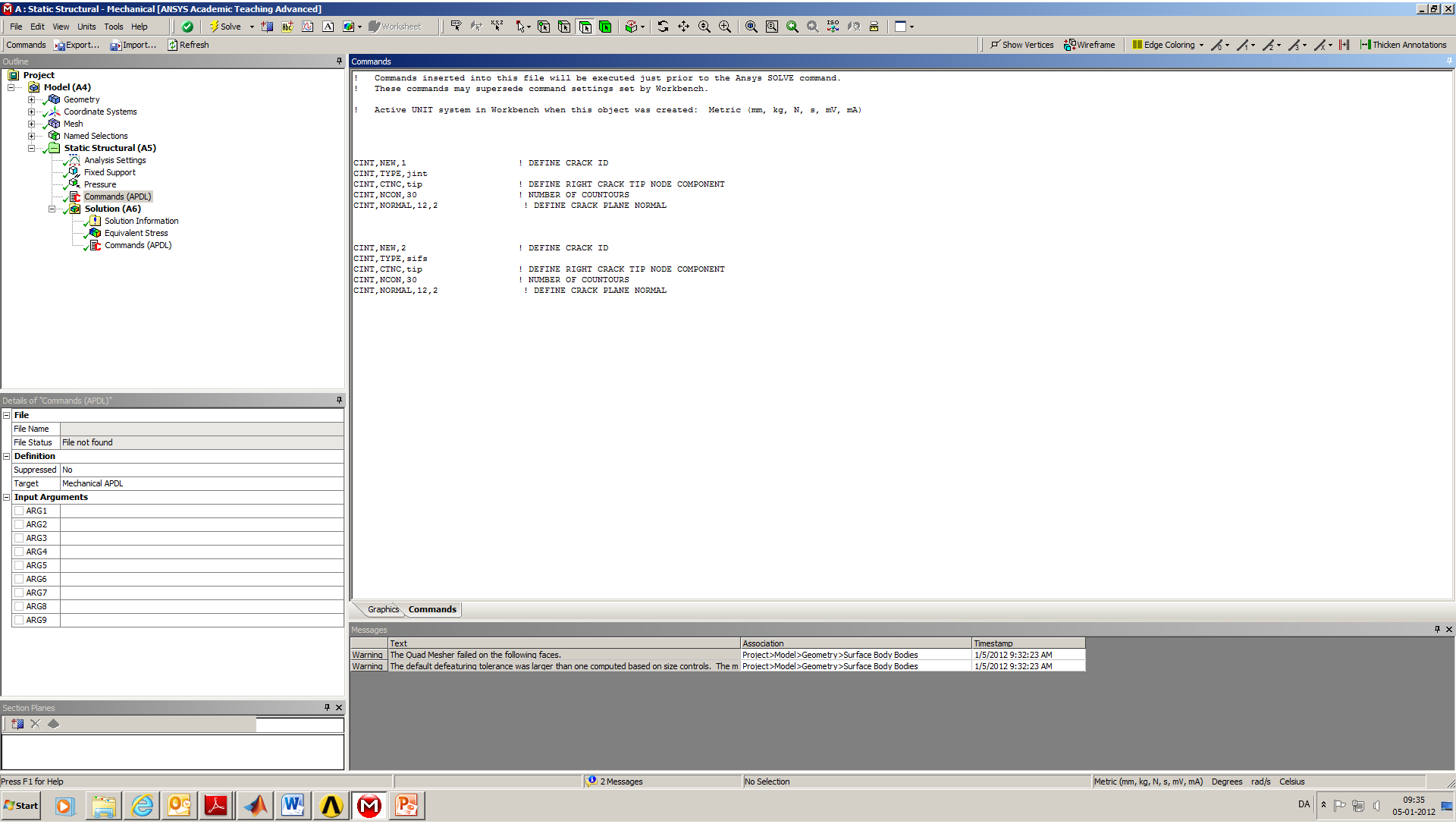
Calculation of the stress intensity factor. It is important that the coordinate system is correct, if the j-integral is negative switch the direction like in the example.

We add a new command. Changes to new,2

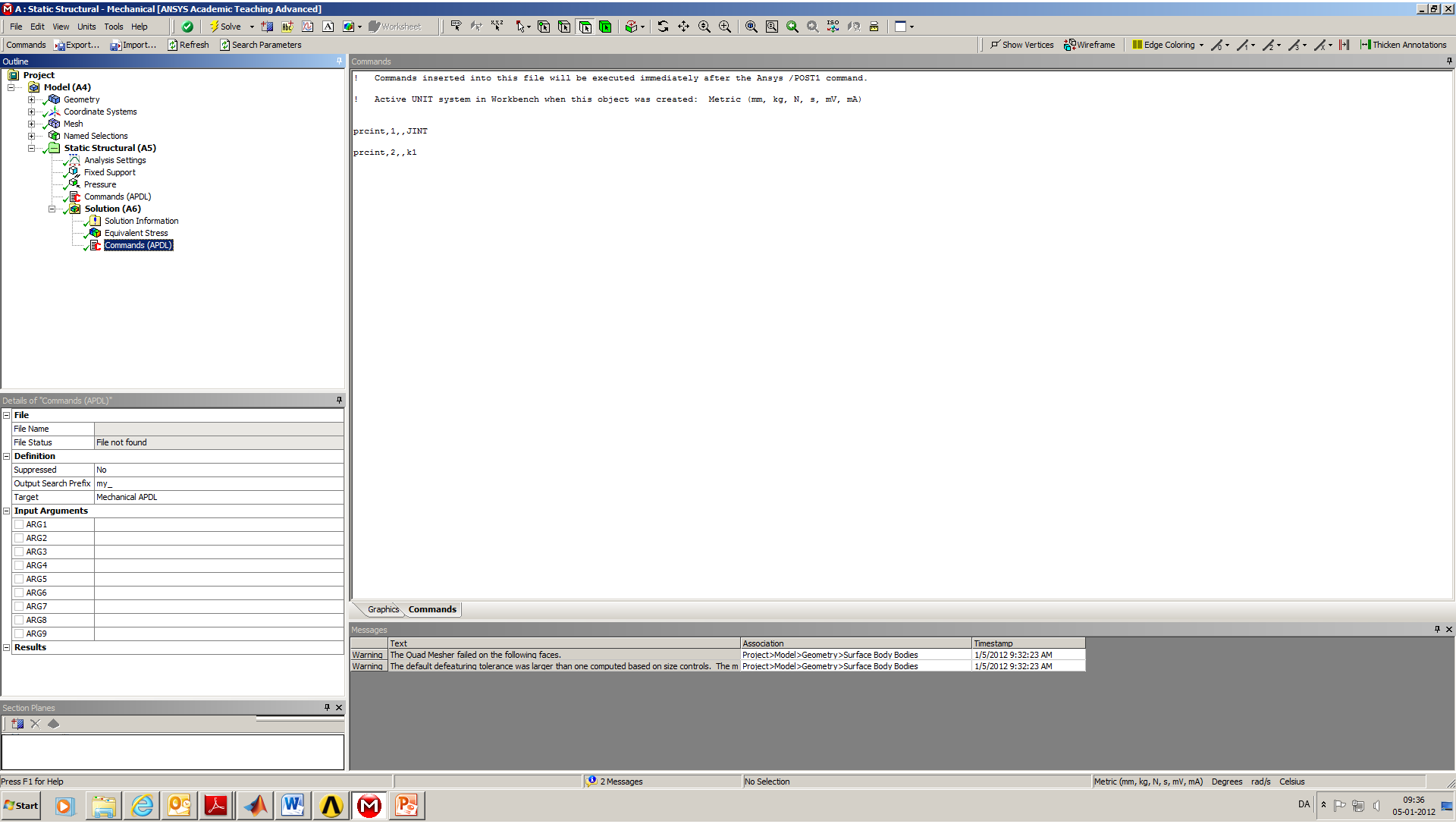
Cint,new,2
Cint,type,sifs
Cint,ctnc,tip
Cint,ncon,10
Cint,normal,0,2

And also print the result from calculation 2**.**

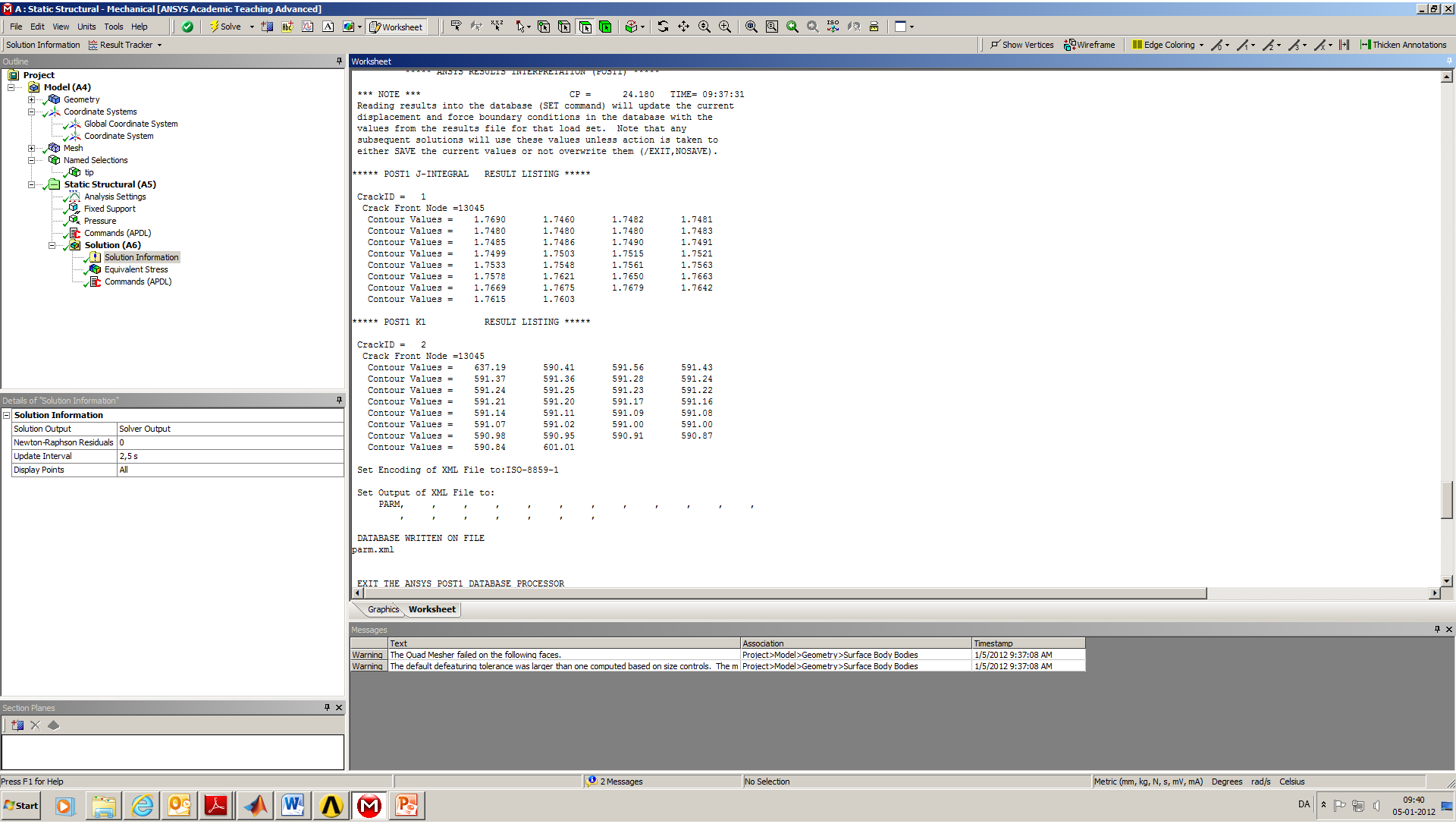
Prcint,2,,K1


A : Static Structural - Mechanical [ANSYS Academic Teaching Advanced]
File Edit View Units Tools Help
Solve
Worksheet
Commands Export... Import... Refresh
Show Vertices Wireframe Edge Coloring Thicken Annotations
Outline
Project
Model (A4)
Geometry
Coordinate Systems
Mesh
Named Selections
Static Structural (A5)
Analysis Settings
Fixed Support
Pressure
Commands (APDL)
Solution (A6)
Solution Information
Equivalent Stress
Commands (APDL)
Details of "Commands (APDL)"
File
File Name
File Status File not found
Definition
Suppressed No
Target Mechanical APDL
Input Arguments
ARG1
ARG2
ARG3
ARG4
ARG5
ARG6
ARG7
ARG8
ARG9
Section Planes
Commands
! Commands inserted into this file will be executed just prior to the Ansys SOLVE command.
! These commands may supersede command settings set by Workbench.
! Active UNIT system in Workbench when this object was created: Metric (mm, kg, N, s, mV, mA)
CINT,NEW,1 ! DEFINE CRACK ID
CINT,TYPE,jint
CINT,CTNC,tip ! DEFINE RIGHT CRACK TIP NODE COMPONENT
CINT,NCON,30 ! NUMBER OF COUNTOURS
CINT,NORMAL,12,2 ! DEFINE CRACK PLANE NORMAL
CINT,NEW,2 ! DEFINE CRACK ID
CINT,TYPE,sifs
CINT,CTNC,tip ! DEFINE RIGHT CRACK TIP NODE COMPONENT
CINT,NCON,30 ! NUMBER OF COUNTOURS
CINT,NORMAL,12,2 ! DEFINE CRACK PLANE NORMAL
Graphics Commands
Messages
Text Association Timestamp
Warning The Quad Mesher failed on the following faces. Project>Model>Geometry>Surface Body Bodies 1/5/2012 9:32:23 AM
Warning The default defeaturing tolerance was larger than one computed based on size controls. The m Project>Model>Geometry>Surface Body Bodies 1/5/2012 9:32:23 AM
Press F1 for Help
2 Messages
No Selection
Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius
Start
DA
09:35
05-01-2012

Like this.


A : Static Structural - Mechanical [ANSYS Academic Teaching Advanced]
File Edit View Units Tools Help
Solve
Worksheet
Commands Export... Import... Refresh Search Parameters
Show Vertices Wireframe Edge Coloring Thicken Annotations
Outline
Project
Model (A4)
Geometry
Coordinate Systems
Mesh
Named Selections
Static Structural (A5)
Analysis Settings
Fixed Support
Pressure
Commands (APDL)
Solution (A6)
Solution Information
Equivalent Stress
Commands (APDL)
Commands
! Commands inserted into this file will be executed immediately after the Ansys /POST1 command.
! Active UNIT system in Workbench when this object was created: Metric (mm, kg, N, s, mV, mA)
prcint,1,,JINT
prcint,2,,k1
Graphics Commands
Details of "Commands (APDL)"
File
File Name
File Status File not found
Definition
Suppressed No
Output Search Prefix my_
Target Mechanical APDL
Input Arguments
ARG1
ARG2
ARG3
ARG4
ARG5
ARG6
ARG7
ARG8
ARG9
Results
Section Planes
Messages
Text Association Timestamp
Warning The Quad Mesher failed on the following faces. Project>Model>Geometry>Surface Body Bodies 1/5/2012 9:32:23 AM
Warning The default defeaturing tolerance was larger than one computed based on size controls. The m Project>Model>Geometry>Surface Body Bodies 1/5/2012 9:32:23 AM
Press F1 for Help
2 Messages
No Selection
Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius
Start
DA
09:36
05-01-2012

Like this.



$$K1 = 591\ MPa \cdot \sqrt{mm}$$

Calculated Stress intensity factor

$$K1 = 100MPa \cdot \sqrt{10mm \cdot \pi} = 560.5\ MPa \cdot \sqrt{mm}$$

$$K1 = 100MPa \cdot \sqrt{10mm \cdot \pi} \cdot \frac{1 - \frac{a}{2w} + \frac{0.326a^2}{w^2}}{\sqrt{1 - \frac{a}{w}}} = 572.2\ MPa \cdot \sqrt{mm}$$