CHAPTER 1
INTRODUCTION

1.1 Background Study

During the event of fire, the high temperature may cause wood to burn and metal works to melt. Since steel is one of the most widely used building materials, a building may collapse if the steel that supports the structure melts. This may worsen the situation in the event of fire because the occupants that are trapped inside the building are threatened not only by the fire and smoke, but also the risk of the building collapsing and cause increased number of deaths. Hence, fire protection of buildings should not be taken lightly.

In Malaysia there is an act that monitors the fire protection measures in every building. All the buildings in Malaysia must obey the standards that have been set up by the Jabatan Bomba dan Penyelamat Malaysia. In UK, there is also a body called Health, Safety and Environment (HSE) Department of UK that monitors the fire protection system in a building. These examples are good indicators of the importance of fire safety. The fundamental purpose of fire protection is to prevent the fire from spreading to another neighboring building, to allow the occupants to safely evacuate the building and to reduce the damage to the building structure. A building may collapse due to fire as structural steel loses an appreciable part of its load carrying ability when its temperature exceeds 500°C. (Jimenez, 2006)

Basically, fire protection can be divided into two types which are active and passive fire protection. Active fire protection requires a certain amount of motion and response in order to make it work. The means of active fire protection are detecting the fire, alerting the occupants of the building about the fire, controlling the movement of the smoke and stopping the fire by blocking the existence of oxygen. Good examples of active fire protection are fire suppression and fire detection. Passive fire protection on the other hand consists of the fire protection by means of using fire rated partitions such as door, ceiling and wall, slow down the growth of fire which may delay the structure from collapse.

One of the widely used passive fire protection nowadays is intumescent coating. It is widely used due to its convenience, efficiency and low cost as a result of improved technology. In addition to that, the coating can be applied on most construction materials that are vulnerable to fire such as steel and wood. The coating can be simply applied or painted on the existing paint. When the coating is exposed to fire or certain amount of heat, it will expand up to 60 times of its original thickness and the expanded coating is called char. This thick layer will insulate the structure from fire and avoid the structure from being burnt and collapse.

However, considering the 60 times expansion from the coating's original thickness in the plane of the interface, it is too much. The stress created at the beam-char interface may disturb the heat insulation function of the intumescent coating because proper expansion cannot take place. In addition to the very high surrounding temperature, the stress created may have the effect of premature failure on the char.

1.2 Problem Statement

The intumescent coating can be simply applied on a structure or on existing paint. Depending on the type of coating, it can provide heat insulation layer ranging from 30 to 120 minutes to the structure it is protecting. When the coating is exposed to fire or excessive heat, chemical reaction will take place and the coating will expand around 60 times of its original thickness and will form a heat barrier layer consists of porous and char.

Ideally the expansion of the coating does not only occur in positive z-direction only, instead the expansion also takes place in x and y-direction as shown in Figure 1. Since the coating can expand 60 times in z-direction, the same swelling will also take place in *x*- and *y*-direction. However in *x*- and *y*-direction, the coating is constrained by the strong bonding at the interface between the coating and the steel substrate (refer to Appendix 1-1). Based on the Figure 1, a significant value of stress is created on the beam-char interface and close attention needs to be paid on the potential stress concentrated area. The stress created may compensates the fire-retardant and heat insulative properties of the coating. Hence, the stress created on the structure-char interface needs to be analyzed so that proper solution can be taken to avoid premature failure of the structure or heat insulation failure by the intumescent coating. For this project, several beams will be taken as the sample steelwork structure and the coating is assumed to be uniformly applied on the structure.

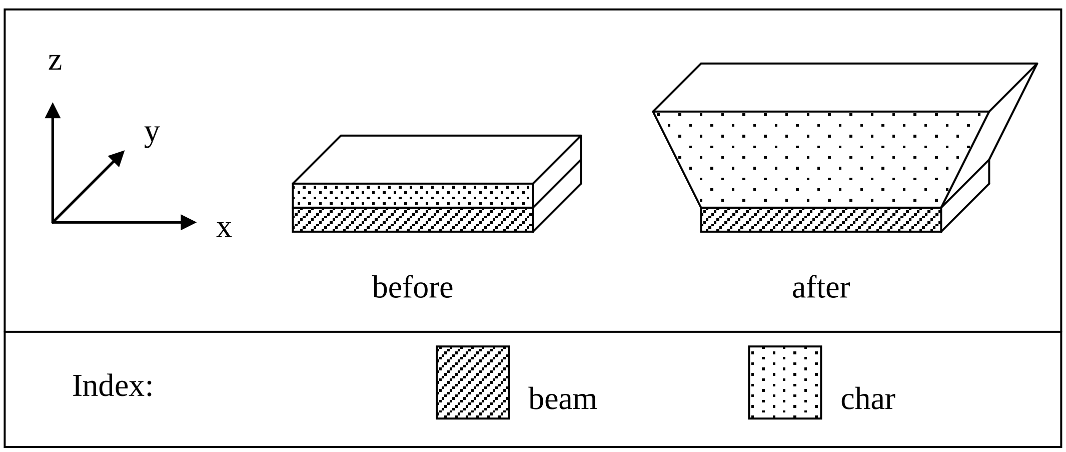


Figure 1: The char before and after the expansion.

1.3 Objectives and Scope of Study

The objective of this project is to analyze the stress that will be created on the beam resulting from char development when the intumescent coating expands. First the stress concentrated area will be identified by using analytical calculations in 1- dimensional case. A basic calculation by using Euler's Formula also done to analyze the tenderncy of char to buckle. Then, the simulation of the expansion will be analyzed using ANSYS.

CHAPTER 2
LITERATURE REVIEW

2.1 Intumescent Coating

Intumescent coating is a type of fire retardant method that is widely used nowadays. The coating can be applied on most construction materials such as wood and steel. Basically the coating will expand when it is exposed to flame or excessive heat to form a heat-insulation layer called char to prevent premature failure of the structure or from being burnt due to excessive heat. The expansion occurs due to three precisely blended components which are carbon supplier, acid source and expanding agent. As heat is applied, the chemical reaction will start with softening of the polymeric binder and release of an organic acid. After that, the polyols begin to carbonize and gas is produced by the decomposition of expanding agent (melamine). After certain time the molten mixture will start to swell and as time goes by the foamed char solidified. Upon solidification, the char will form a heat insulation foam-like layer. The layer will protect the structure from fire by reducing the heat transfer from fire to the structure and blocking the movement of oxygen into the char layer. Hence it will keep the fire away from the structure. The process of expansion of the intumescent coating to become char is shown in Figure 2.

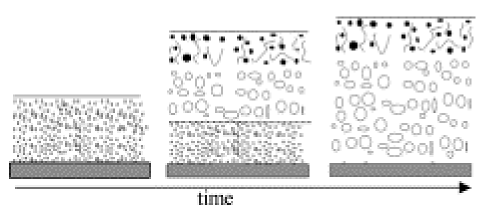


Figure 2: The timeline of the expansion of the coating (Duquesne, 2004).

2.2 Beams

Basically an off-shore platform has high potential of catching fire. Steel beams are the main component of an off-shore platform as shown in Figure 3 with high risk of catching fire. Therefore, most of the structures are coated with intumescent coating as a precaution method. There is a wide range of structure to be selected especially on steel beam/member. There are several types of steel member available in the market such as I-beam, hollow structure, C-channel, steel angle (L-shape) and many more.

Figure 3: An off-shore steel platform.

The most common type of steel beam in the market is the I-beam. The beam is also known as W-beam or double-T for the cross section shape of the beam that looks like letter I or H. This beam is very good in carrying load like shear or bending in the plane of the web but not good in handling torsion. Other than steel, the beam also can be made from other material such as aluminum. Commercially, there is a standard that determine the size and dimension of the beam. The notation for a beam is W*310* x *52*, where W represent the type of beam as W-beam, while *310* represent the depth of the beam,*d* and *52* represent the weight of the beam per unit length. The depth of the beam is shown in Figure 4. The unit of notation can either be in FPS unit or metric unit (Vable, 2002). An example of I-beam application in supporting a structure is shown in Figure 5.

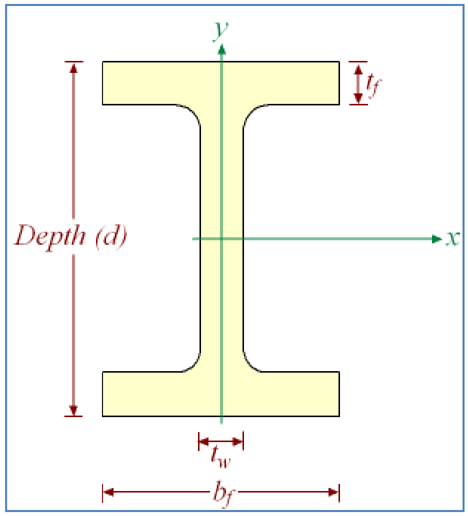


Figure 4: I-beam cross sectional dimension.

Figure 5: I-beam in structure.

Other types of common structure like C-channel and steel angle (L shape) have their own notation. C-channel for example, the notation is C*380*x*50* where C represents C-channel, *380* represents the depth of the channel,*d* and *50* represent the weight per unit length as shown in Figure 6. While for steel angle, the notation is L*203*x*203*x*19* where L represents the ‘L’ shape cross section, the first two numerals represent the length of its ‘a’ and ‘b’ legs as shown in Figure 7 and the last numeral represents the thickness of the beam.

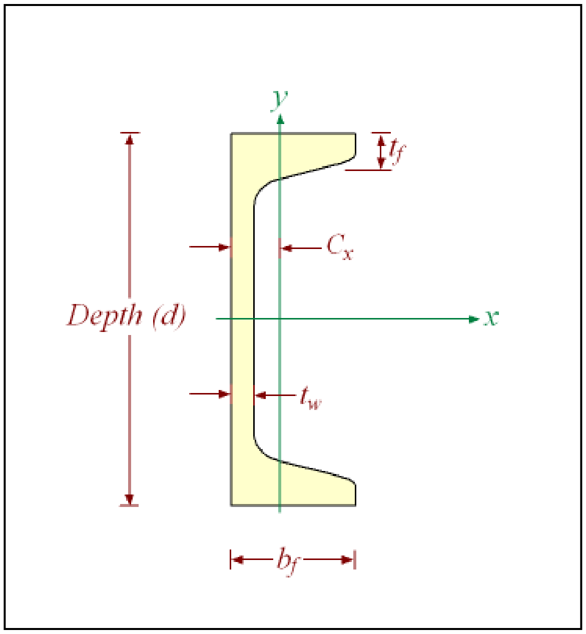


Figure 6: C-channel cross sectional dimension.

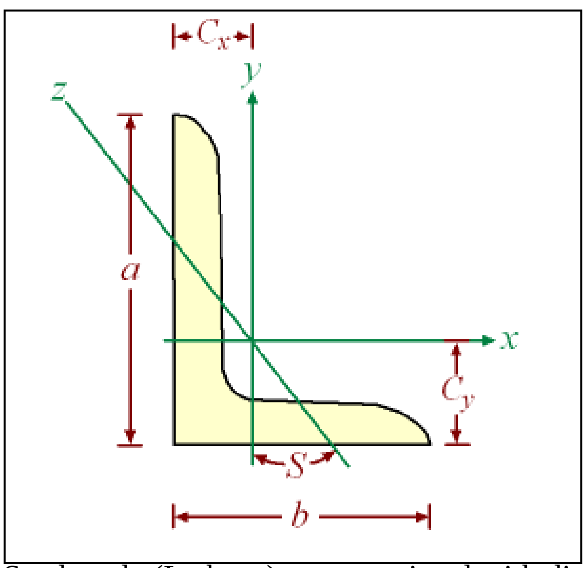


Figure 7: Steel angle (L-shape) cross sectional with dimension.

For this project, the beams that have been chosen to be modeled in ANSYS are I-beam, C-channel and Steel angle. The dimensions for those beams are shown in Table 1.

Table 1: The beam dimensions as modeled in the ANSYS

Beam	Dimension	
I-beam (W310x52)	Area, A	6617 mm²
	Depth, d	317.0 mm
	Web thickness, t_w	7.6 mm
	Flange width, b_f	167.0 mm
	Flange thickness, t_f	13.2 mm
C-channel (C380x50)	Area, A	6430 mm²
	Depth, d	381.0 mm
	Web thickness, t_w	10.2 mm
	Flange width, b_f	86.4 mm
	Flange thickness, t_f	16.5 mm
Steel angle (L203x203x19)	Area, A	7380 mm²
	Leg A	203 mm
	Leg B	203 mm
	Thickness	19 mm

Another type of steel structure is hollow structural section (HSS). This type of structure may be available in circular, rectangular and elliptical cross sectional shape. This structure can be differentiated with pipe or hollow structural steel by the designation of the dimension and its class. For rectangular HSS, the corner of the cross section is heavily chamfered as much as twice its thickness as shown in Figure 8. This type of structure is very good in carrying load that comes from multiple axis and also torsion. The elliptical cross sectional shape in Figure 9 is now becoming more popular as it is used for the aesthetic value.

Figure 8: Rectangular HSS

Figure 9: Elliptical HSS

2.3 ANSYS

Finite element method is a procedure that provides solution to people in solving complicated engineering problem. Some of the engineering problems that may be solved by using this method are stress analysis, thermal analysis, fluid flow and electromagnetic problems. In industry nowadays, there are many computer programs in that use this method in order to solve complicated engineering problem such as I-DEAS, Hyperworks, NASTRAN, LS Dyna etc. ANSYS is one of the earliest computer programs released to perform finite element method to solve static, dynamic, heat transfer, fluid flow and electromagnetism analyses (Moaveni, 2003). In addition to that, ANSYS is also capable of solving those analyses in 1-, 2- or 3-dimension. Generally, there are three basic steps in using the computer programs incorporating ANSYS which are started with preprocessing phase followed by solution phase, and post-processing phase. Preprocessing phase involved the early stage of the simulation including creating and discretize the model, setting the equation, assemble the elements and applying boundary condition and loads. ANSYS has many options available to discretize the model based on type of analysis and the degree of freedom. Creativity of the user is needed to model the boundary conditions and the load in ANSYS. Shear stress for example, needs to be modeled as force parallel to the surface applied on each node on the surface. The solution phase is where the computer program will solve the problem by using finite element method based on the specified equation defined in the preprocessing steps. Last but not least, the result of the simulation is analyzed in the postprocessing phase. These phases are almost the same for all software for all types of analysis. In industry however, they may use several computer programs to handle each of the steps at optimum speed and cost.

CHAPTER 3
METHODOLOGY

The methodology on completing this project starts with some literature review that was done on some related areas such as intumescent coating, beams and ANSYS software. The literature review is then followed by preliminary calculation and simulation in ANSYS. The flow chart of this project is shown in Figure 11 while all the detailed steps and the tool being used in order to complete this project are explained as follows.

3.1 Preliminary 1-Dimensional and Buckling Calculation

After having a good overview on the project with some literature reviews on the related topics, some basic calculation regarding the stress was done to predict the behavior of stress at the beam-char interface. For 1-dimension problem, the expansion of the coating is modeled as uniaxial load. The results of the calculation indicate whether the beam-char interface will face a tension or compression as well as the 1-dimension pattern of the stress. Apart from that, another basic calculation was done to measure the tendency of the char to buckle. The critical load before the char will buckle was calculated on the longest flange of all the beams considered in this project.

3.2 Pre-processing

Some of the data that has been collected before is used as the input for the pre-processing steps in ANSYS. The pre-processing steps that must be done in ANSYS are defining element type, defining element real constant, defining material properties and defining meshing control. In order to run the simulation of the beam-char interface in the ANSYS, a specific type of element has been chosen for meshing purpose which is the four-node shell element. This 2D element is a simple structural element that has 4 nodes with three degrees of freedom in translational *x*-, *y*- and *z*- directions and also known as 'SHELL 43' in ANSYS.

For this 2D element, the thickness of the element needs to be defined as the real constant. 2 mm has been set as the element thickness for 'SHELL 43'. The material properties that need to be defined in order to run this simulation are the density, Young's modulus and Poisson's ratio of Structural A36 steel alloy as shown in Table 2.

Table 2: Mechanical properties of the beam

Data	Value
Density	7.850×10^{-6} kg/mm^3
Poisson's ratio	0.3
Young's modulus	2×10^5 N/mm^2

While for defining meshing size control, the type of model and analysis was taken into consideration. Since this simulation does not require a very fine meshing, the minimum edge length has been set to be 5 mm. All the details for the steps taken on pre-processing are shown in the Appendix 3-1 to Appendix 3-3.

3.3 3D Modeling & Meshing

For this analysis, all of the beams being modeled in ANSYS have symmetrical shape. Hence, there is no need to model the whole beam because the full result can be achieved by reflecting the result in the symmetrical plane. For I-beam, only quarter of the beam was modeled because the beam is symmetrical in XY and YZ planes as shown in Figure 10. Hence, full result can be achieved by reflecting the obtained result twice in XY and YZ planes. For C-channel as in Figure 6 on the other hand, half of the beam need to be modeled because the beam is only symmetrical in XZ plane. The same methodology goes to steel angle because there is a symmetrical plane on the beam. By just modeling part of the beam which has symmetrical plane, the full result of the simulation can be achieved by reflecting the obtained result from the simulation in XZ plane. This strategy greatly reduces the time working on modeling as well as processing time later on. All the areas in the finished model will then be meshed with the specified element mentioned above. The method of meshing the model on the other hand, depends on the complexity of the model. Fortunately all the beams being modeled for this project are quite simple hence it will be meshed automatically by the software. If the model is complex like having so much areas, curves and holes mapped meshed may need to be considered. Detailed modeling and meshing steps are shown in Appendix 3-4 to Appendix 3-13.

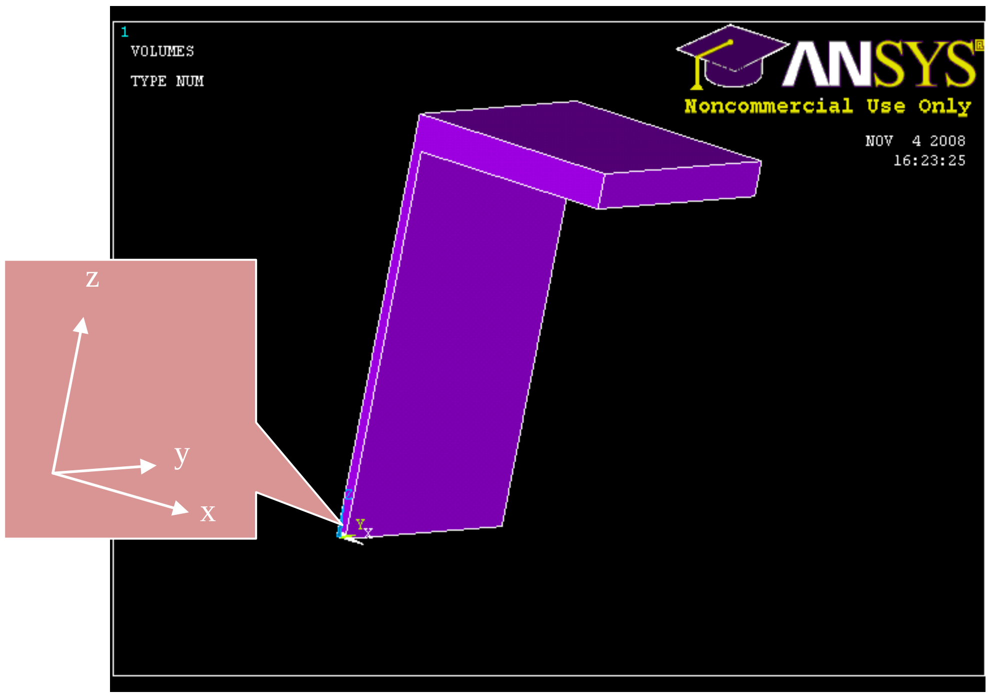


Figure 10: Symmetrical planes for I-beam

3.4 Applying Boundary Conditions and Loads

For this project, all the beams being modeled have almost the same boundary conditions to be applied on its model. The boundary conditions being applied on the model are basically on the symmetrical plane. All the symmetrical planes are set not to have any displacement in the direction normal to their surface. The expansion of the coating on the other hand is modeled as shear stress in ANSYS. Hence all the nodes at the beam interface that has been applied with the coating are applied with 10 N shear force pointing outward each interface. The load applied on the I-beam is shown in the Appendix 3-14 to Appendix 3-15.

3.5 Simulation and Post-Processing

After all the boundary conditions and loads have been applied on the modeled beam, the simulation started. Once the solution is done, the stress concentration area and deformation of the beam can be viewed in post-processing method. The step to start the simulation is shown in the Appendix 3-16.

3.6 ANSYS

The main tool for this project is a simulation software called ANSYS. Almost 60% of the project is done by using ANSYS. This software has been used as the tool of this project from the pre-processing step until the simulation and post-processing steps.

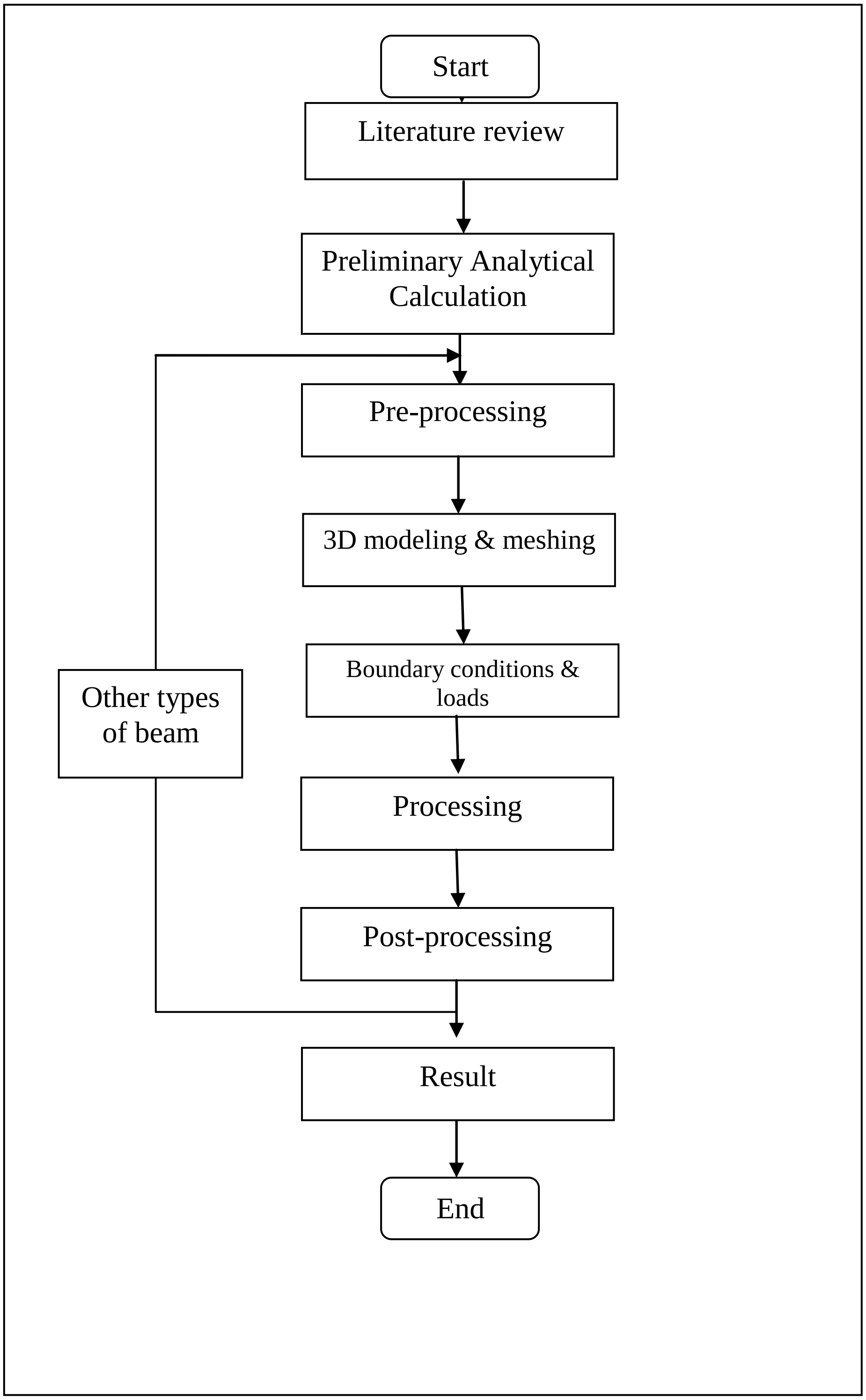


Figure 11: Flow chart of the project

CHAPTER 4
RESULT AND DISCUSSION

4.1 Preliminary Analytical Calculation

4.1.1 1-Dimensional stress calculation

The basic calculation on stress was done to predict the behavior of stress at the beam-char interface. Considering a beam with area, *A* subjected to a shear force as shown in Figure 12, the average shear stress experienced by the interface can be calculated using,

$$\tau_{avg}=\frac{V}{A} \quad \ldots\ldots\ldots\ldots\ldots\ldots.. (1)$$

τ_{avg} = average shear stress
V = shear load
A = surface area

Figure 12: A beam subjected to shear load, *V*

For the beam-char interface, the expansion of the intumescent coating parallel to the beam surface creates shear force on the interface pointing outward of the surface. Therefore, based on the free body diagram at the interface of the beam, the force diagram can be generated as shown in Figure 13.

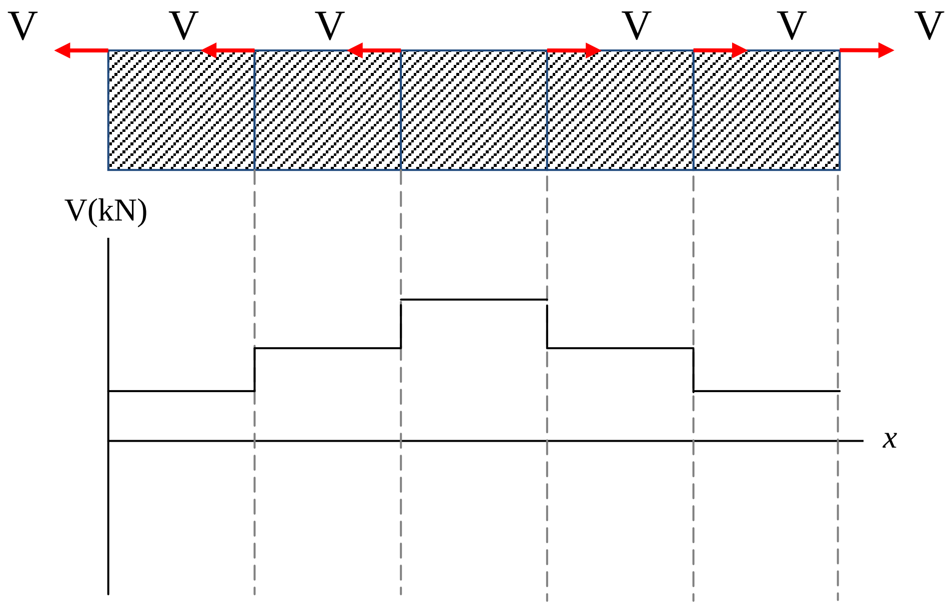


Figure 13: The force body diagram at the interface of the beam and its corresponding force diagram.

From the force diagram, it can be seen that the magnitude of the resultant internal force is highest in the middle of the beam. Based on Equation (1), it can be estimated that, the shear stress is highest in the middle of beam and the longer the beam, the higher the stress.

4.1.2 Buckling Calculation

Basic buckling analysis was done to calculate the critical compression load before the char can buckle. The critical load should be high enough so that the char will not buckle easily. Should the char buckle, it will affect the heat insulation properties because of the imperfect expansion and the char may also crack because of the bending. The critical load before the char will buckle was calculated by using Euler's Formula.

$$P_{cr} = \frac{\pi^2 EI}{L^2}$$

P_{cr}	= critical compression load
E	= Young's modulus
I	= moment of inertia
L	= length

For this buckling critical load calculation, graphite's Young's modulus has been used as approximation for char's Young's modulus and the value is 10 GPa. While for moment of inertia, the value of *b* is the thickness of the char and the value of *h* is considering the char (in yellow) that sticks on a 100 mm length of beam as shown in Figure 14. For the value of L, the longest flange of each beam was substituted into the equation.

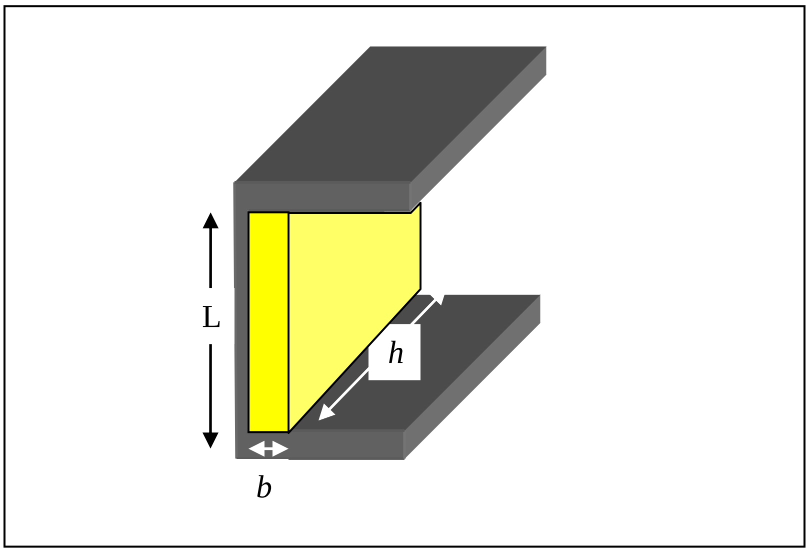


Figure 14: Dimension of char sticking on beam

I-beam:

$$P_{cr} = \frac{\pi^2 (10000MPa) \, {}^{(30^2 mm \times 100mm)}/_{12}}{317^2 mm} = 220.99kN$$

C-channel:

$$P_{cr} = \frac{\pi^2(10000MPa)\,{}^{(30^2mm \times 100mm)}/_{12}}{381^2mm} = 152.98kN$$

Steel angle:

$$P_{cr} = \frac{\pi^2(10000MPa)\,{}^{(30^2mm \times 100mm)}/_{12}}{203^2mm} = 538.88kN$$

From the calculation, it is observed that the critical load is quite high. Therefore, it is almost impossible for the char to buckle. In addition to that, the calculated critical load is only considering if the char is not sticking to any surface. As far as this project is concern, the char is sticking to the surface of the beam, therefore the critical compression load will be higher to overcome the bonding between the char and the surface of the beam. It is also observed that the critical compression load reduced as the length of the flange is longer. Therefore, as the length of the flange increased, the higher the possibility for the char to buckle. It is concluded that, the char would not buckle because the critical compression load is very high.

4.2 ANSYS 3D Modeling

Based on the dimensions of all the beams as in Table 1, quarter of the I-beam and half of C-channel and steel angle have been modeled in ANSYS. The beam was then meshed with four-node shell element also known as SHELL43 in ANSYS. The mesh was based on the minimum of 5 mm edge length meshing control specified in the preprocessing phase. The modeled and meshed beams are shown in Figure 14 to 19.

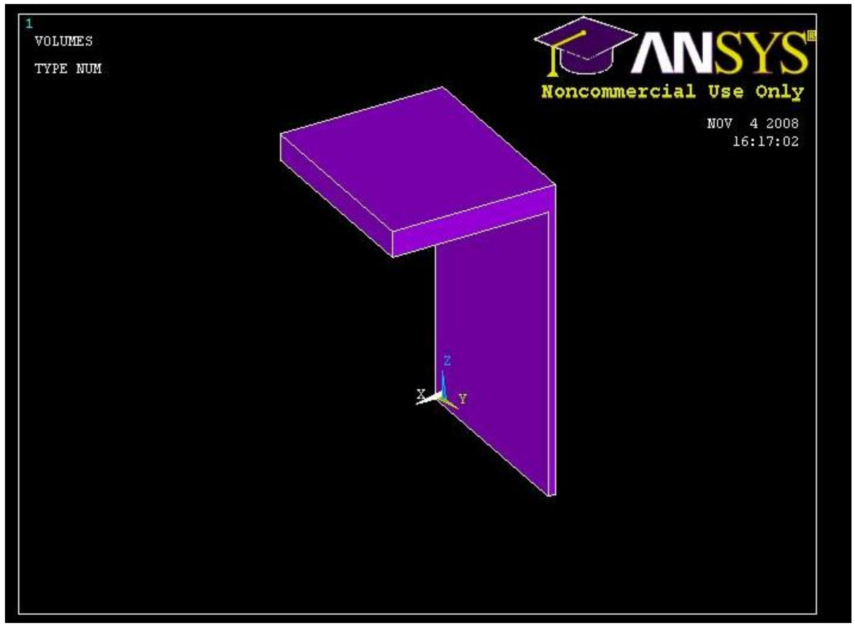


Figure 15: 3-D model of the I-beam in ANSYS

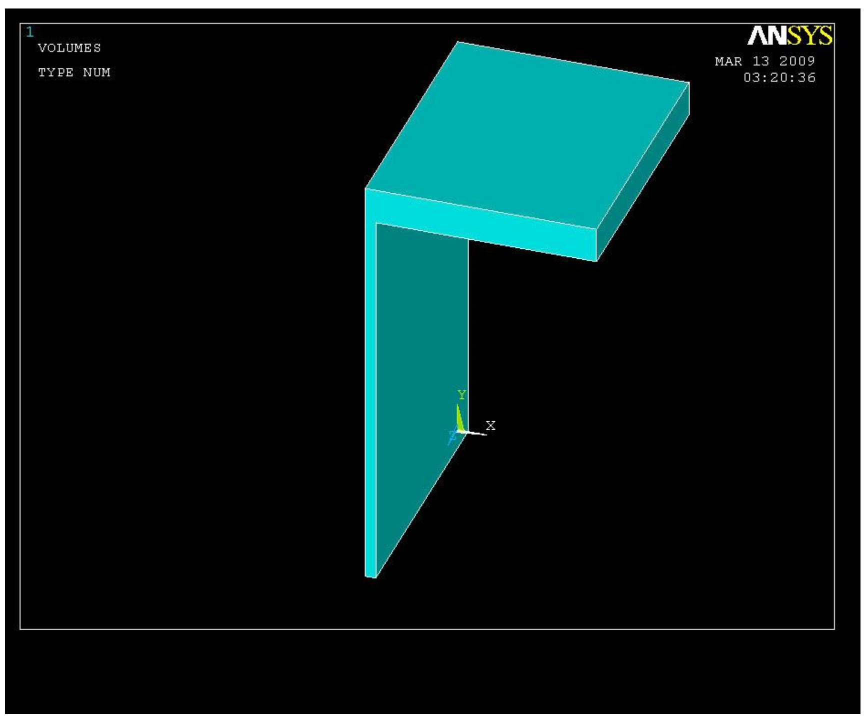


Figure 16: 3-D model of the C-channel in ANSYS

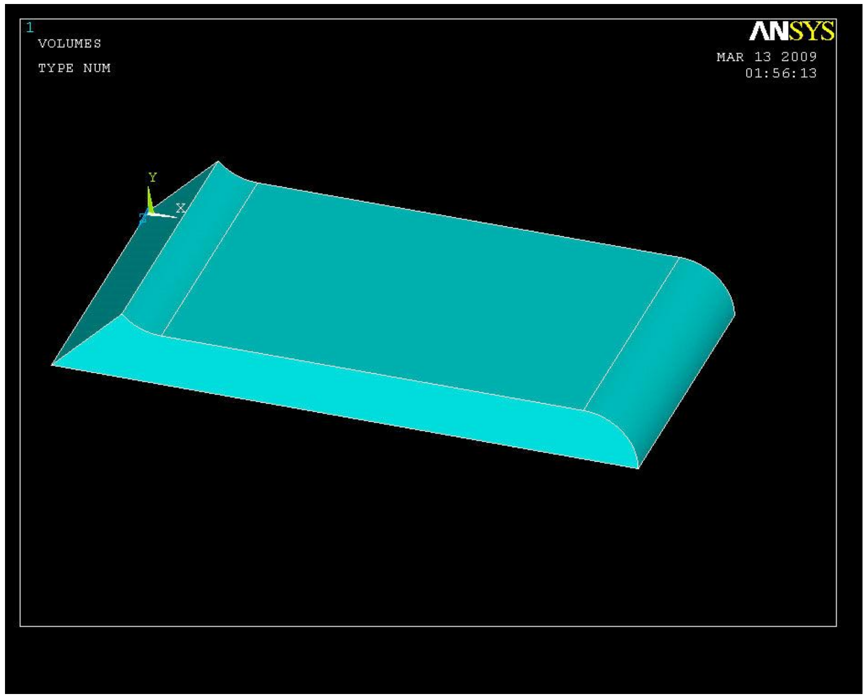


Figure 17: 3-D model of the steel angle in ANSYS

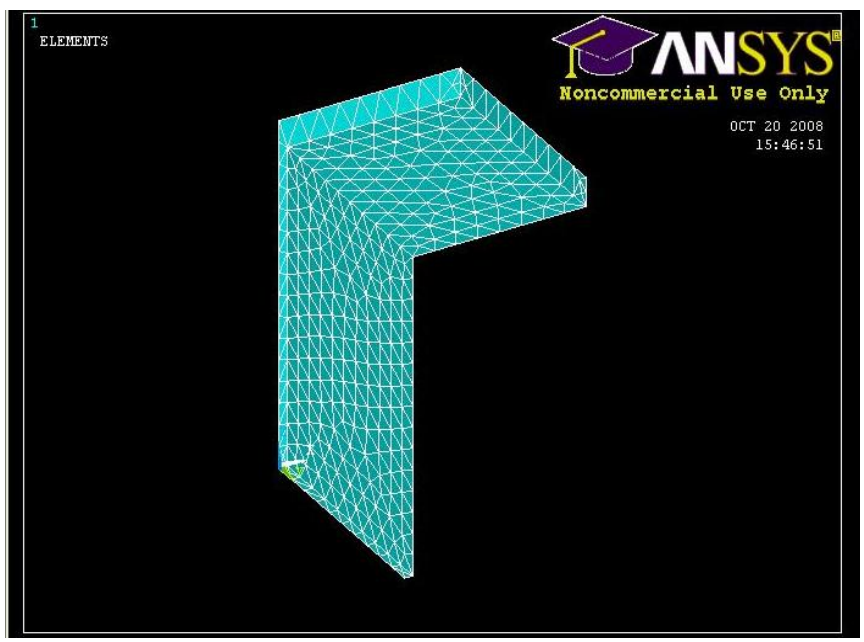


Figure 18: Meshed model of I-beam in ANSYS

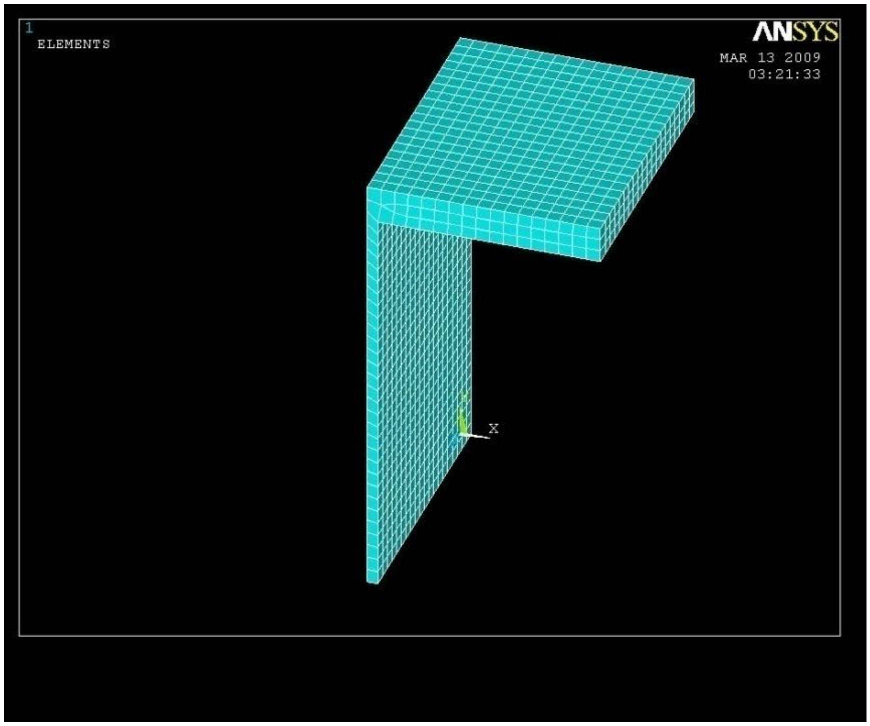


Figure 19: Meshed model of C-channel in ANSYS

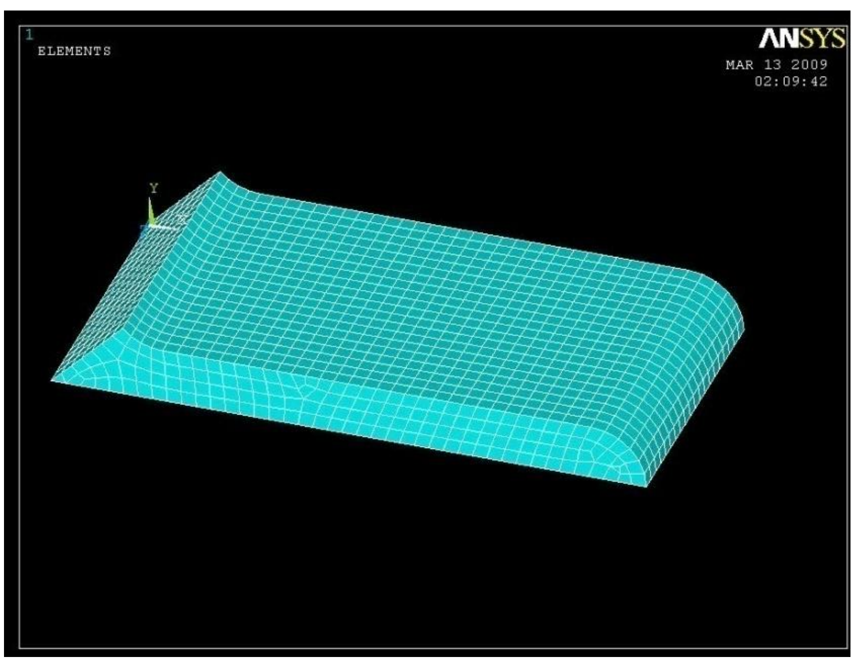


Figure 20: Meshed model of steel angle in ANSYS

4.3 ANSYS Simulation

Based on the simulation that has been done, the results of all the simulations of the beam are shown in Figure 20 to 25.

4.3.1 I-beam

Based on the simulation done on the I-beam, it is observed that the stress concentration area is located at the middle span of the I-beam as can be seen in the table above. The longer the flange of the beam, the higher the stress will be. This result is also parallel with the result obtained from the analytical calculation whereby the stress is expected to be concentrated at the middle span of the beam-char interface.

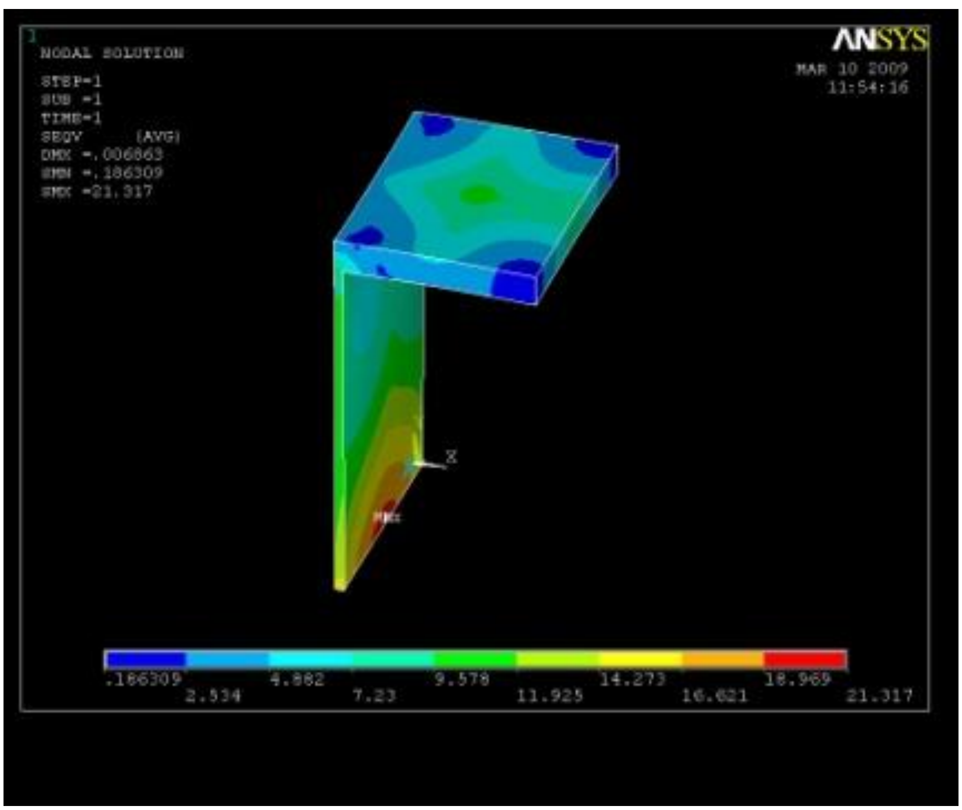


Figure 21: I-beam simulation result 1

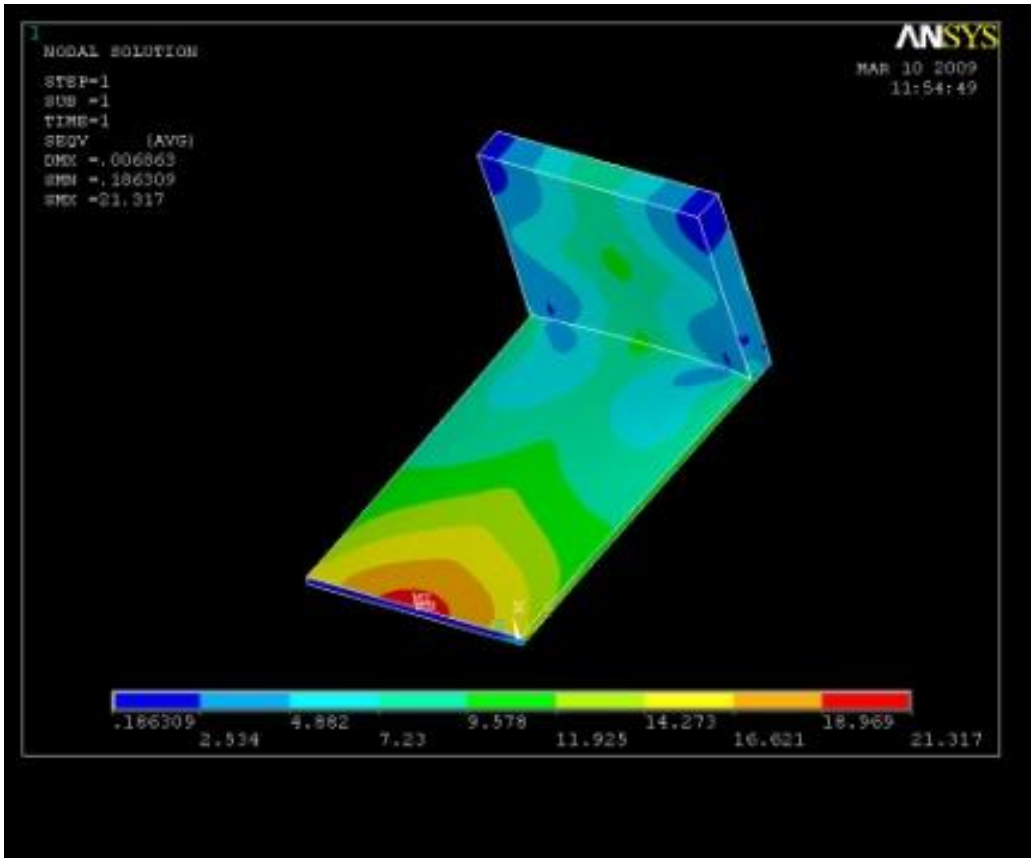


Figure 22: I-beam simulation result 2

From the figure, the stress on the interface is indicated by the color code. Blue color shows the minimum stress and maximum stress is indicated by red color. The minimum stress is 0.106MPa and the maximum stress is 21.317 MPa.

4.3.2 C-channel

Similar observation can be seen from the C-channel simulation where the stress concentration area is located at the middle span of the beam. It is also observed that the stress is higher if the length of the interface is longer. This result is also corresponding with the result obtained from the analytical calculation.

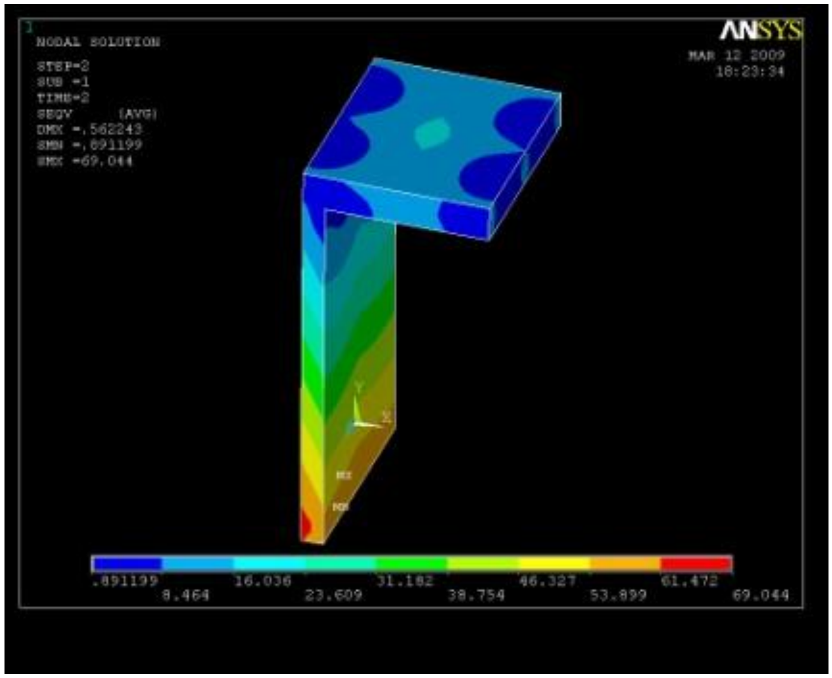


Figure 23: C-channel simulation result 1

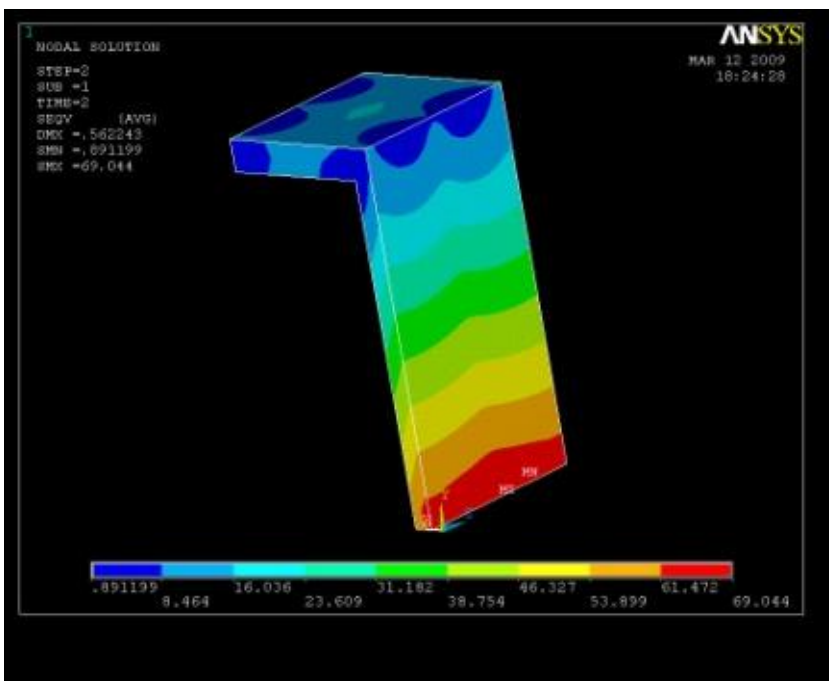


Figure 24: C-channel simulation result 2

From the figure, the stress on the interface is indicated by the color code. Blue color shows the minimum stress and maximum stress is indicated by red color. The minimum stress is 0.912MPa and the maximum stress is 69.04MPa.

4.3.3 Steel Angle

Simulation of Steel angle shows that the stress concentrated area is located in between the legs. This is because, the expansion of the coating in between the legs have less degree of freedom to expand compared to the expansion at the end of the legs.

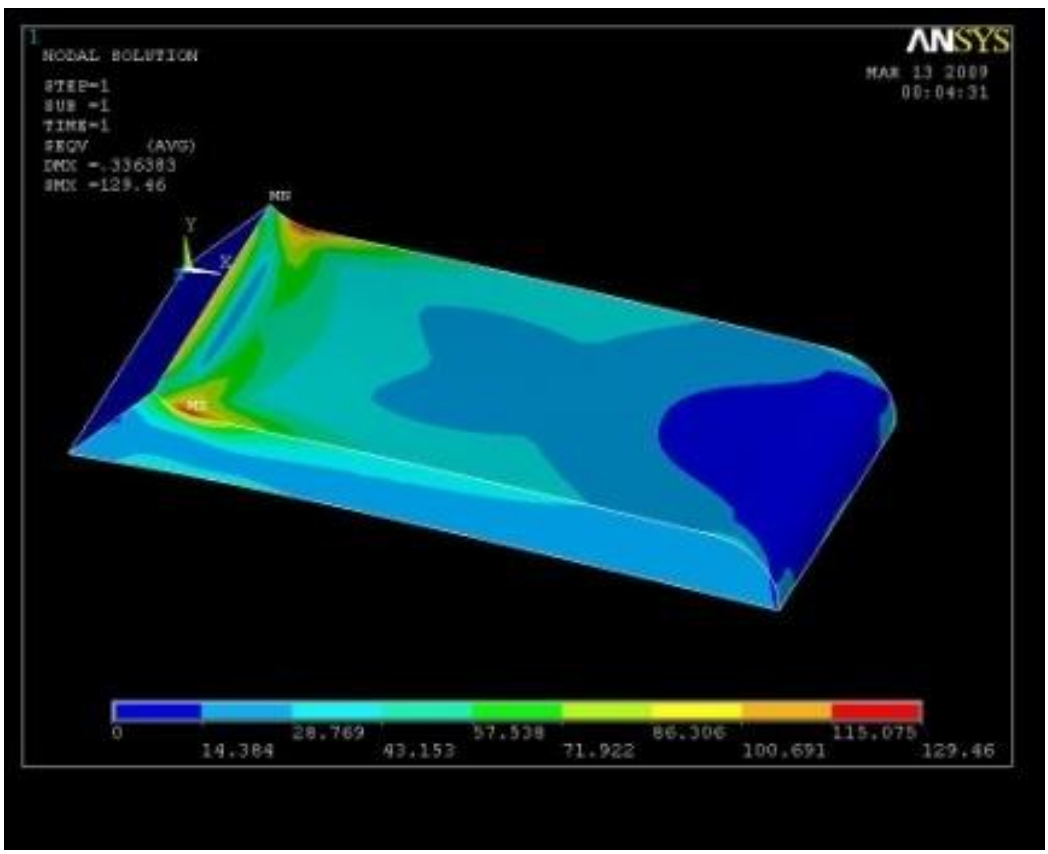


Figure 25: Steel angle simulation result 1

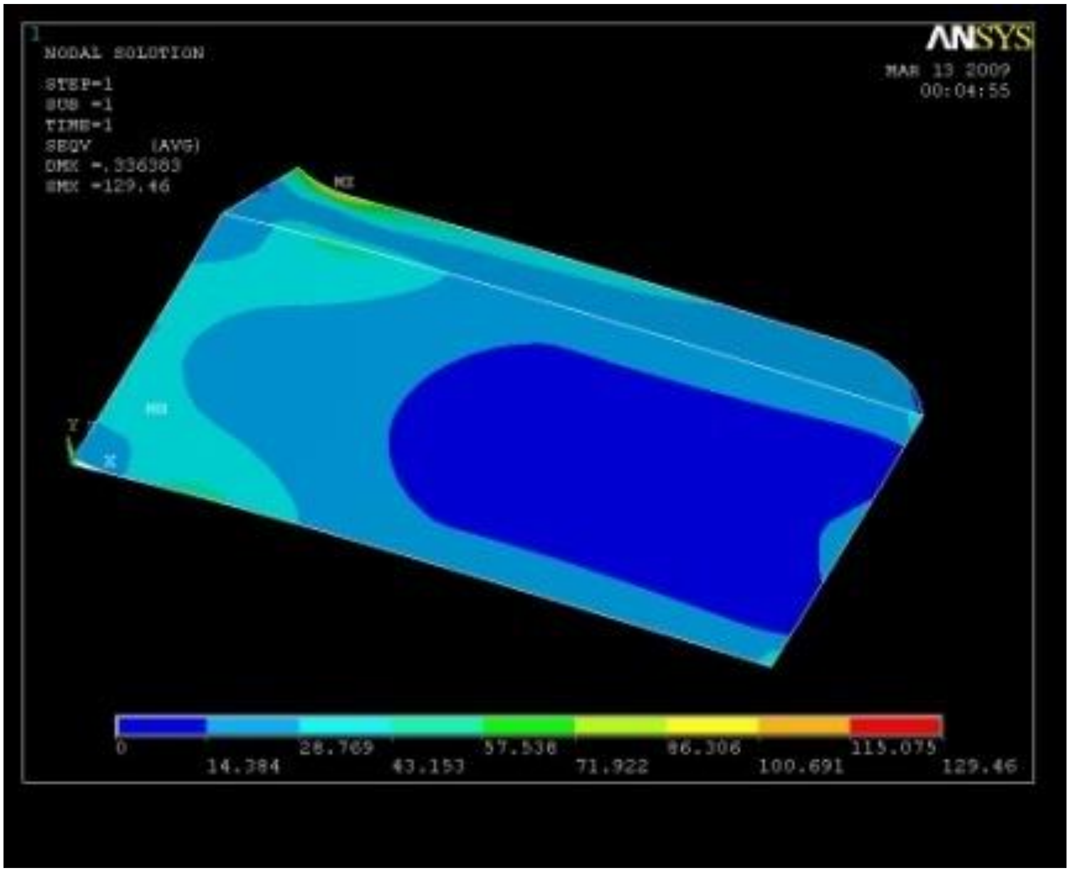


Figure 26: Steel angle simulation result 2

From the figure, the stress on the interface is indicated by the color code. Blue color shows the minimum stress and maximum stress is indicated by red color. The minimum stress is 0.337MPa and the maximum stress is 129.46MPa.

4.4 Discussion

From the result, it is observed that the stress at the interface is directly proportional to the length of the interface length as shown in Figure 13. It is also observed that the critical load of buckling, P_{cr} is inversely proportional to the length of the interface. It is proposed to wrap carbon-fiber mat on stress concentration area to reduce the stress experienced by the char as shown in Figure 27.

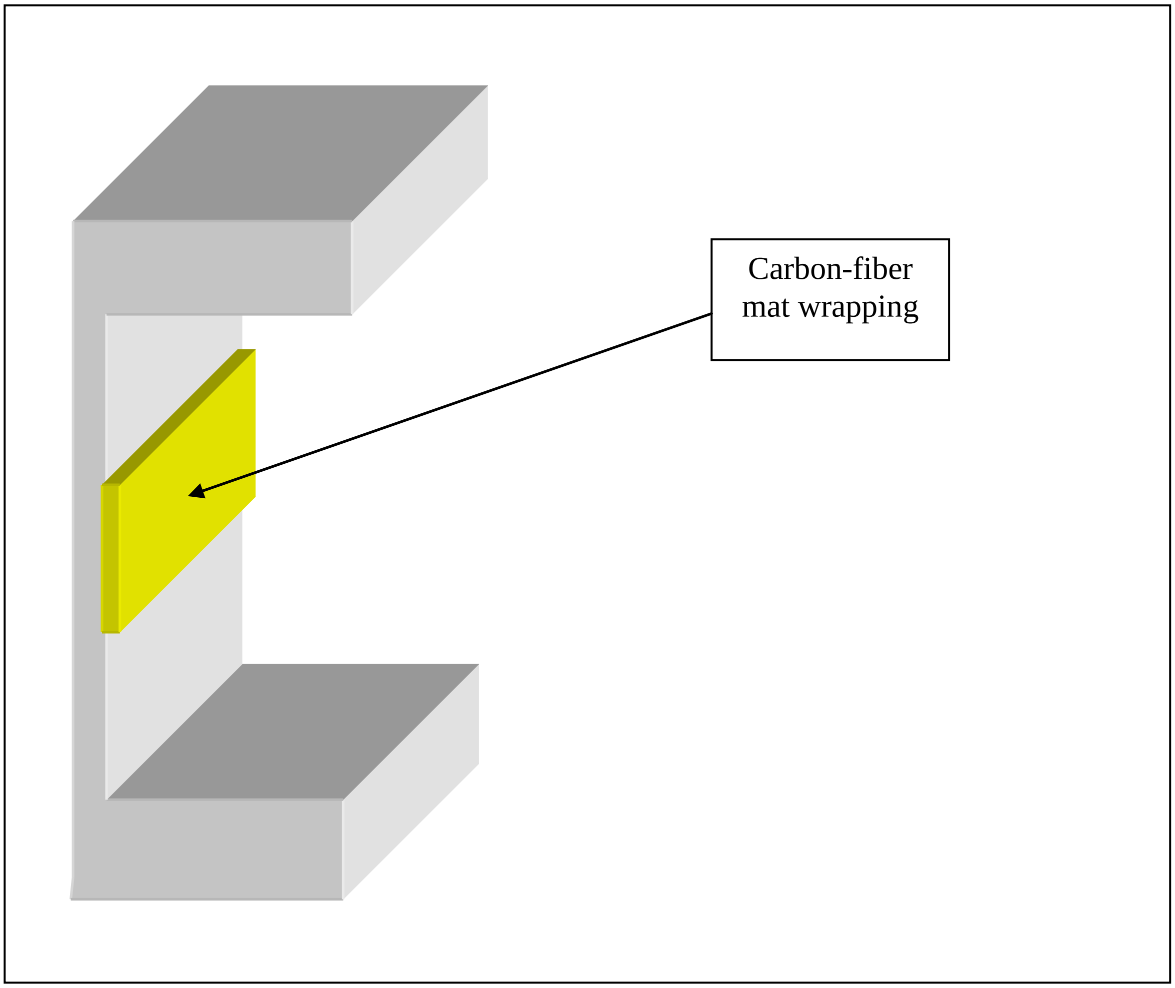


Figure 27: Carbon-fiber mat wrap on the stress concentration area

CHAPTER 5
CONCLUSIONS AND RECOMMENDATIONS

From the analytical calculation that has been done, it is observed that the interface of the char will face compression and the stress is higher in the middle of the beam-char interface. This result is parallel with the simulation done on ANSYS; it is observed that the stress is concentrated in the middle of the beam-char interface when the coating expanded. Another observation done on the result found that the stress is higher if the length of the interface is longer. Hence it is recommended to apply carbon-fiber mat on stress concentration area to reduce the stress experienced by the char. This can greatly reduced the cost rather than wrapping the whole beam with carbon-fiber mat to reduce the stress. As conclusion, the expansion of intumescent coating upon exposure to the fire or excessive heat will create a stress on the char in middle of the beam surface. The stress does not allow the intumescent coating to expand which will affect the fire insulation property. Hence by wrapping the critical area with carbon-fiber mat, it can reduce the stress experienced by the char.

Reference

(n.d.). Retrieved October 5, 2008, from http://www.louisianasteel.com/shapes.html

Callister, W. D. (2003). *Material Science and Engineering: An Introduction.* John Wiley & Sons.

Duggal, S. K. (2000). *Design of Steel Structure.* New Delhi: McGraw Hill.

Duquesne, S. (2004). Intumescent paints: fire protective coatings for metallic substrates . *Surface and Coatings Technology* , 302-307.

Grainger, S. (1998). *Engineerin Coatings: design and application.* London: Abington Publishing.

Hibbeler, R. C. (2005). *Mechanics of Material.* Singapore: Prentice Hall.

Jimenez, M. (2006). Characterization of the performance of an intumescent fire protective coating. *Surface and Coatings Technology* , 979-987.

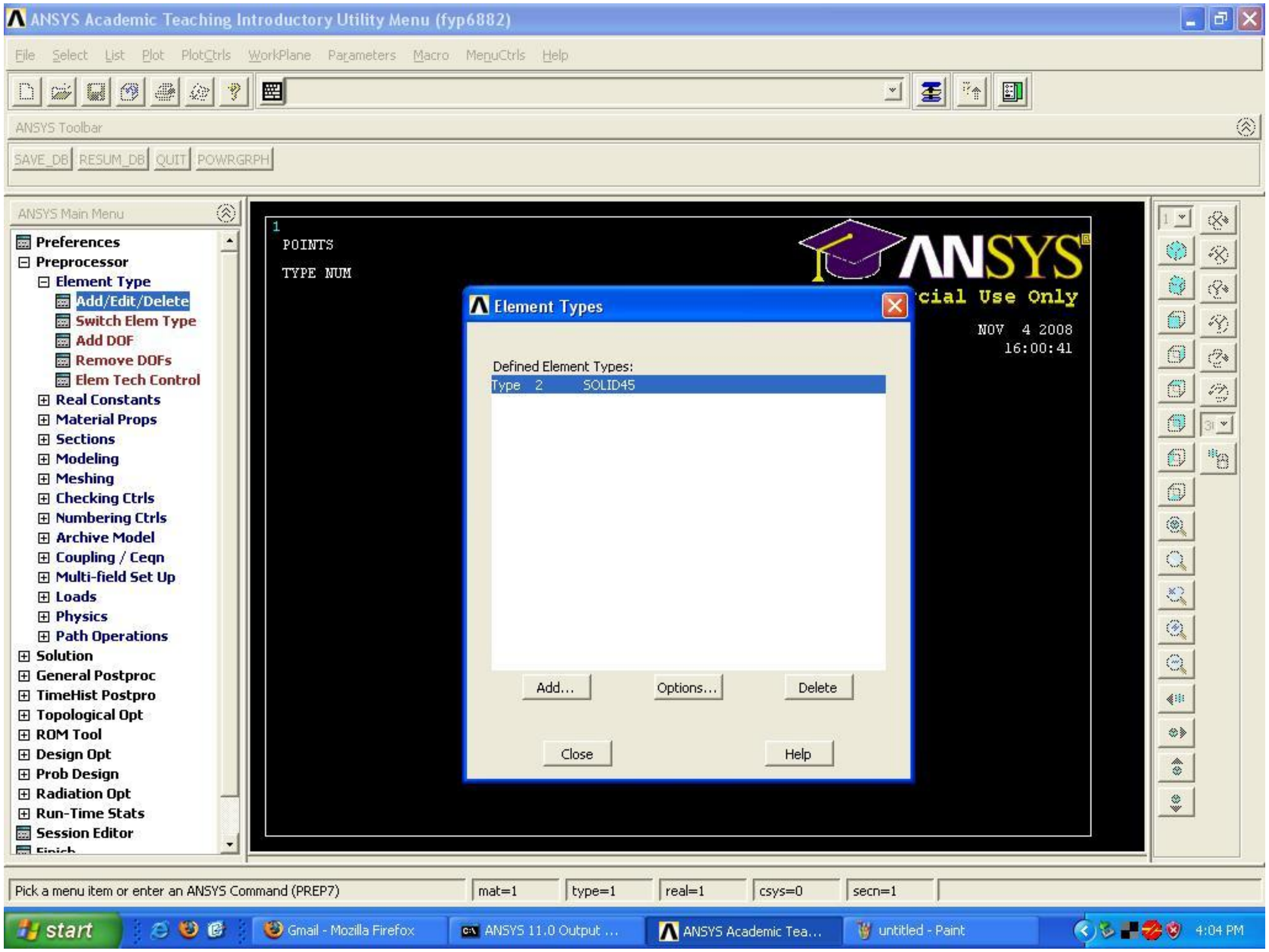
Moaveni, S. (2003). *Finite Element Analysis: Theory And Application With Ansys.* New Jersey: Pearson Education Inc.

Reshedko, P. (2003). *Coated Metal: Structure and Properties of Metal-Coating Compositions.* Springer.

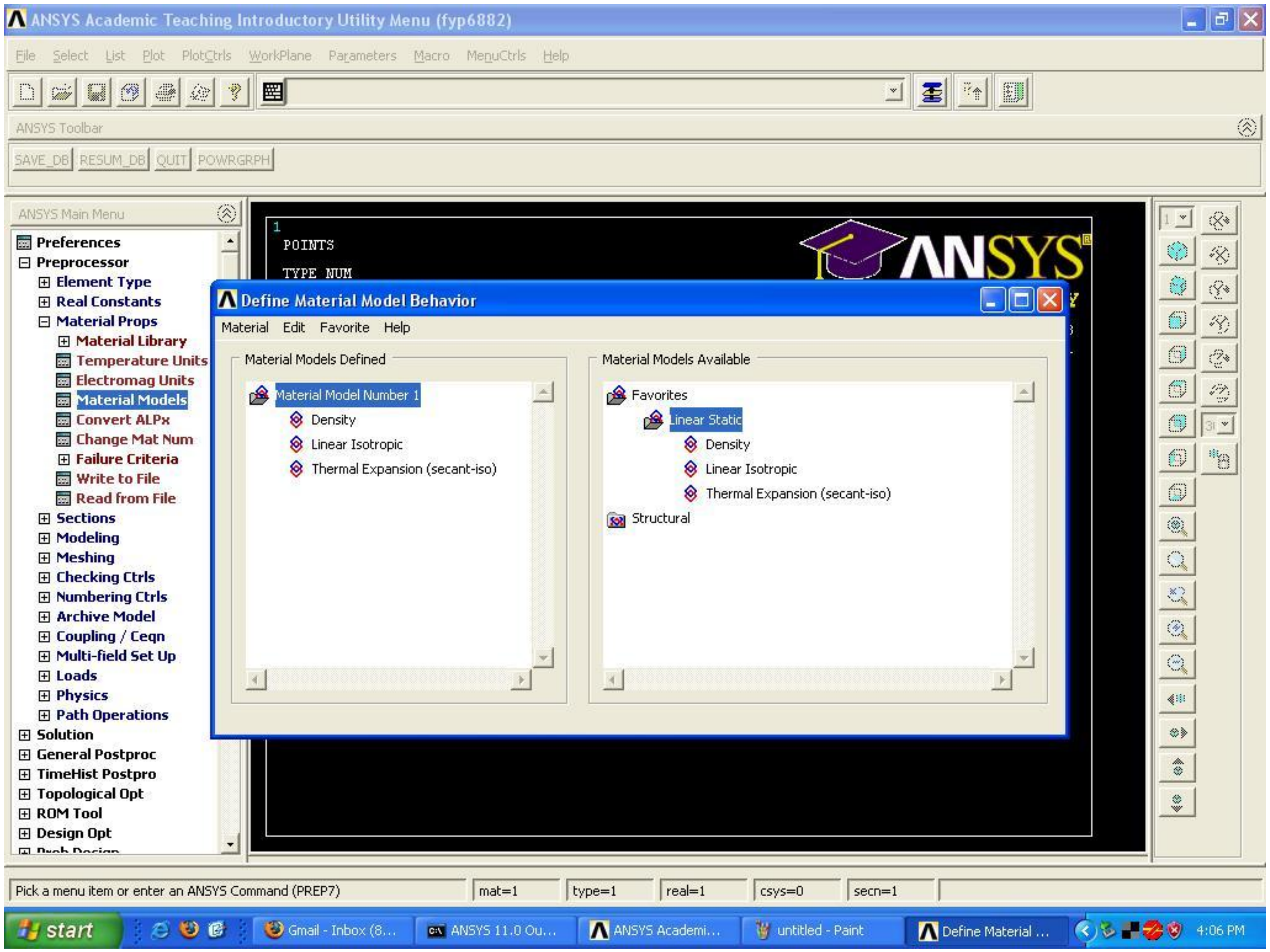
Vable, M. (2002). *Mechanics of Material.* Oxford University Press.

Wikipidea. (2008, October). Retrieved November 2008, from http://en.wikipedia.org/wiki/I-beam

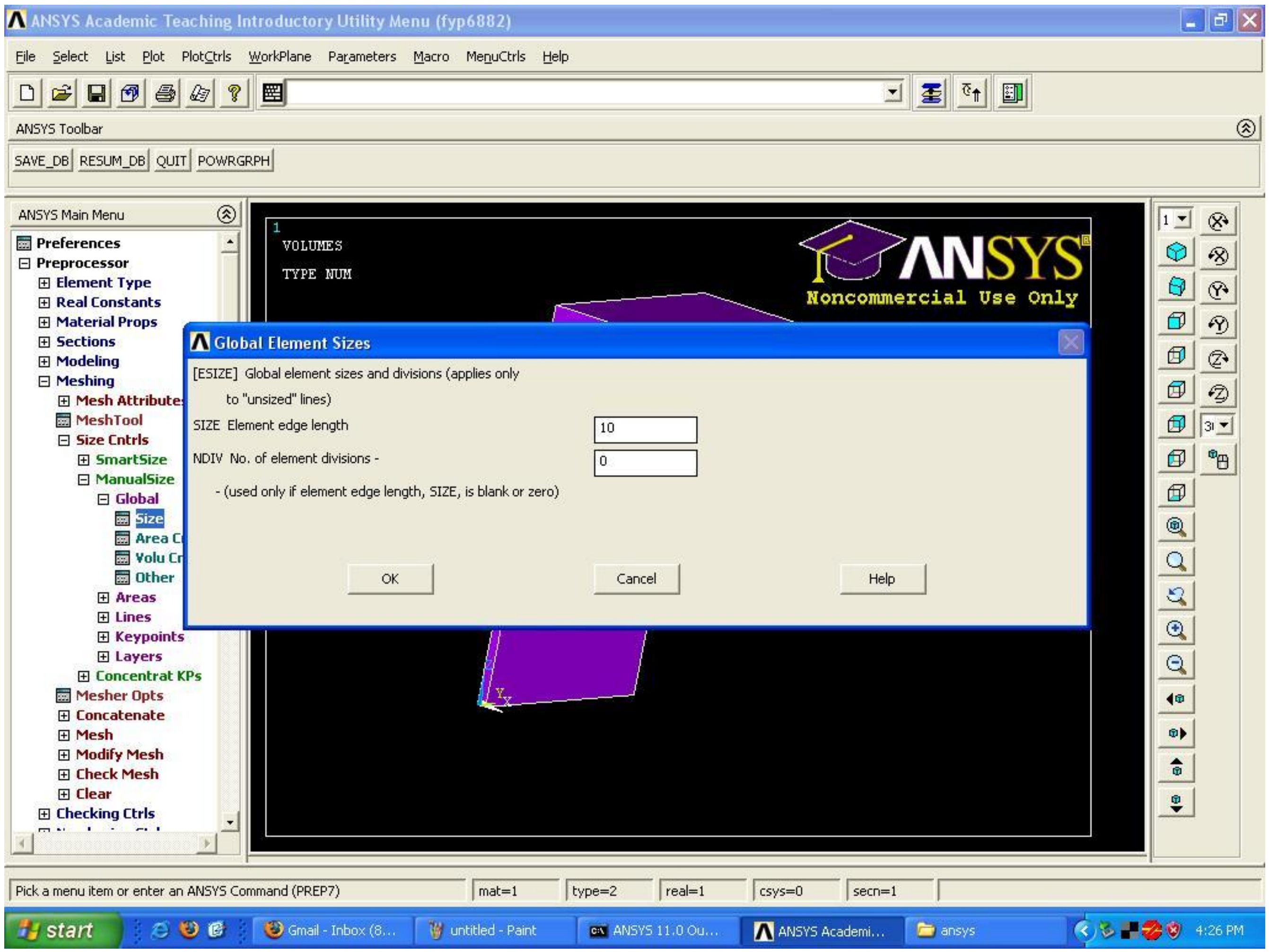
APPENDICES



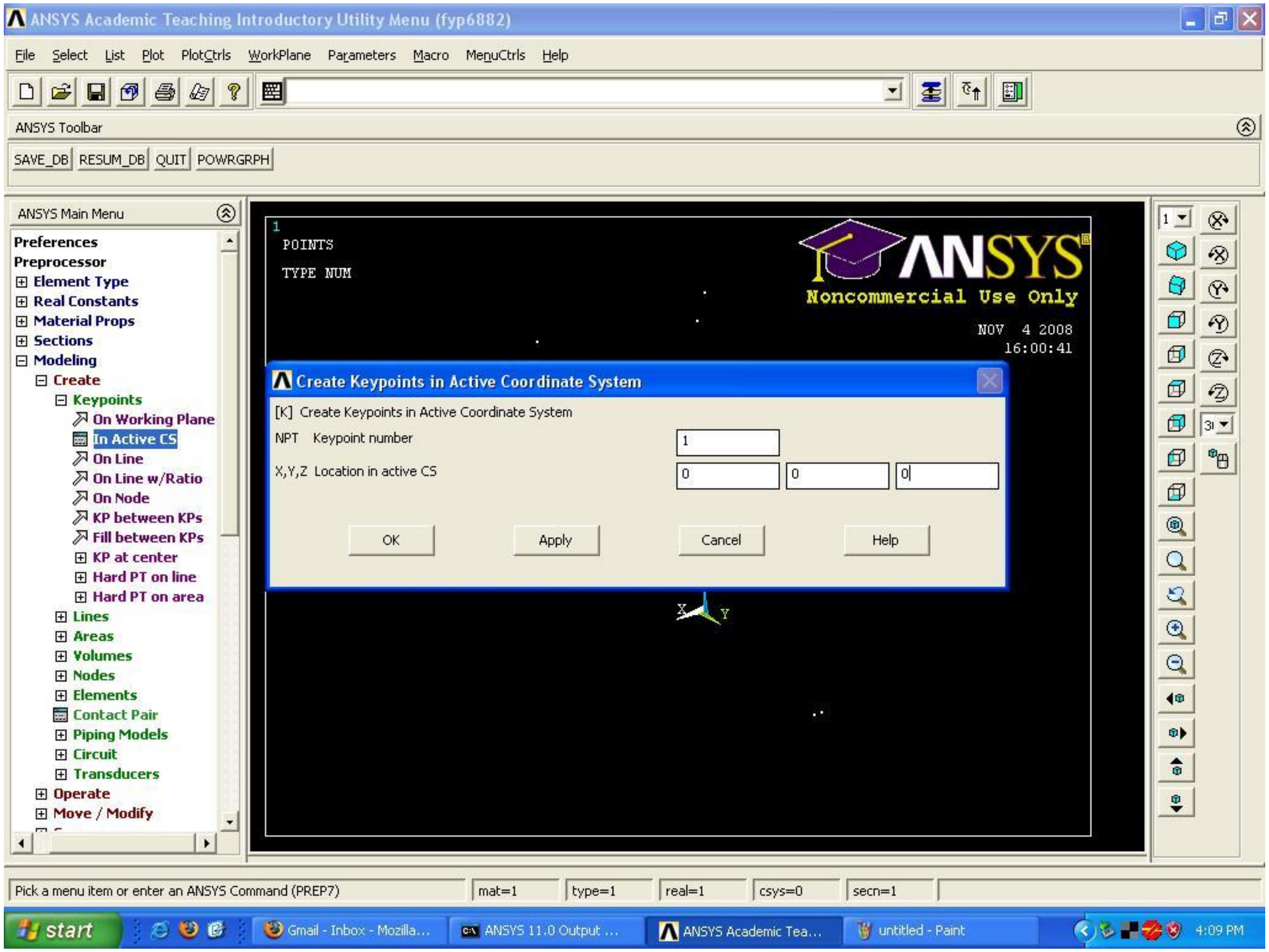
Defining element type



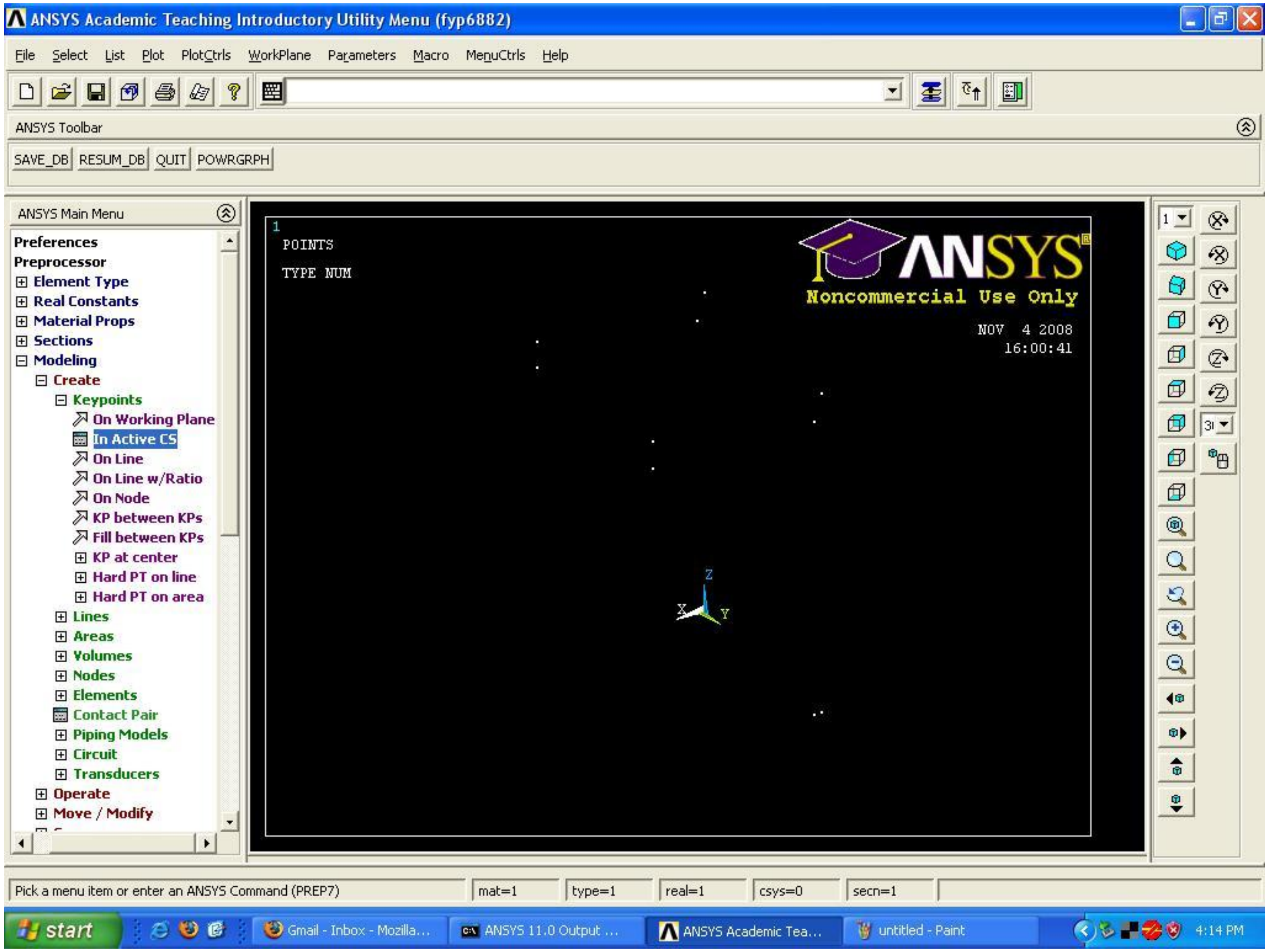
Defining material properties



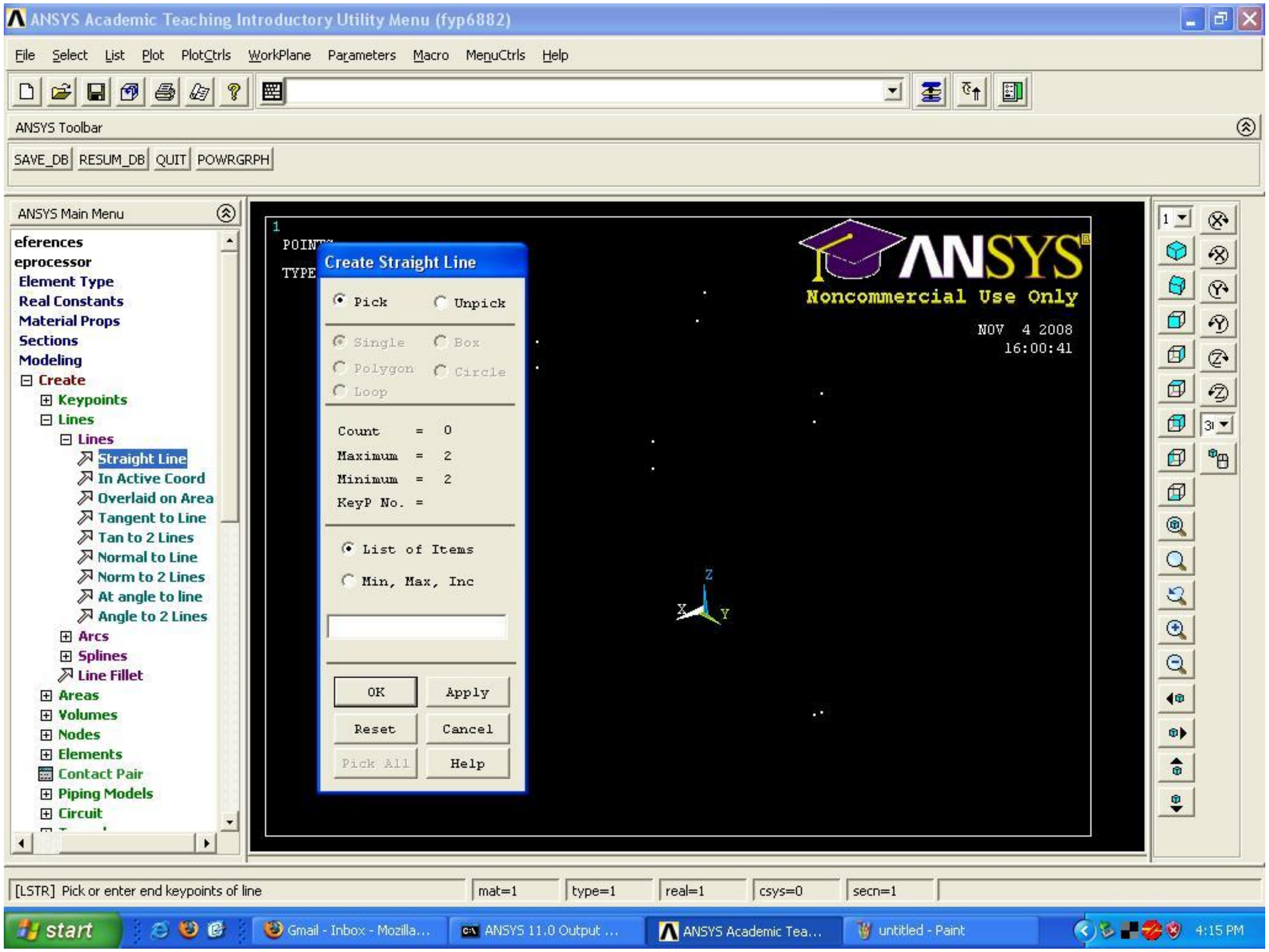
Defining element size



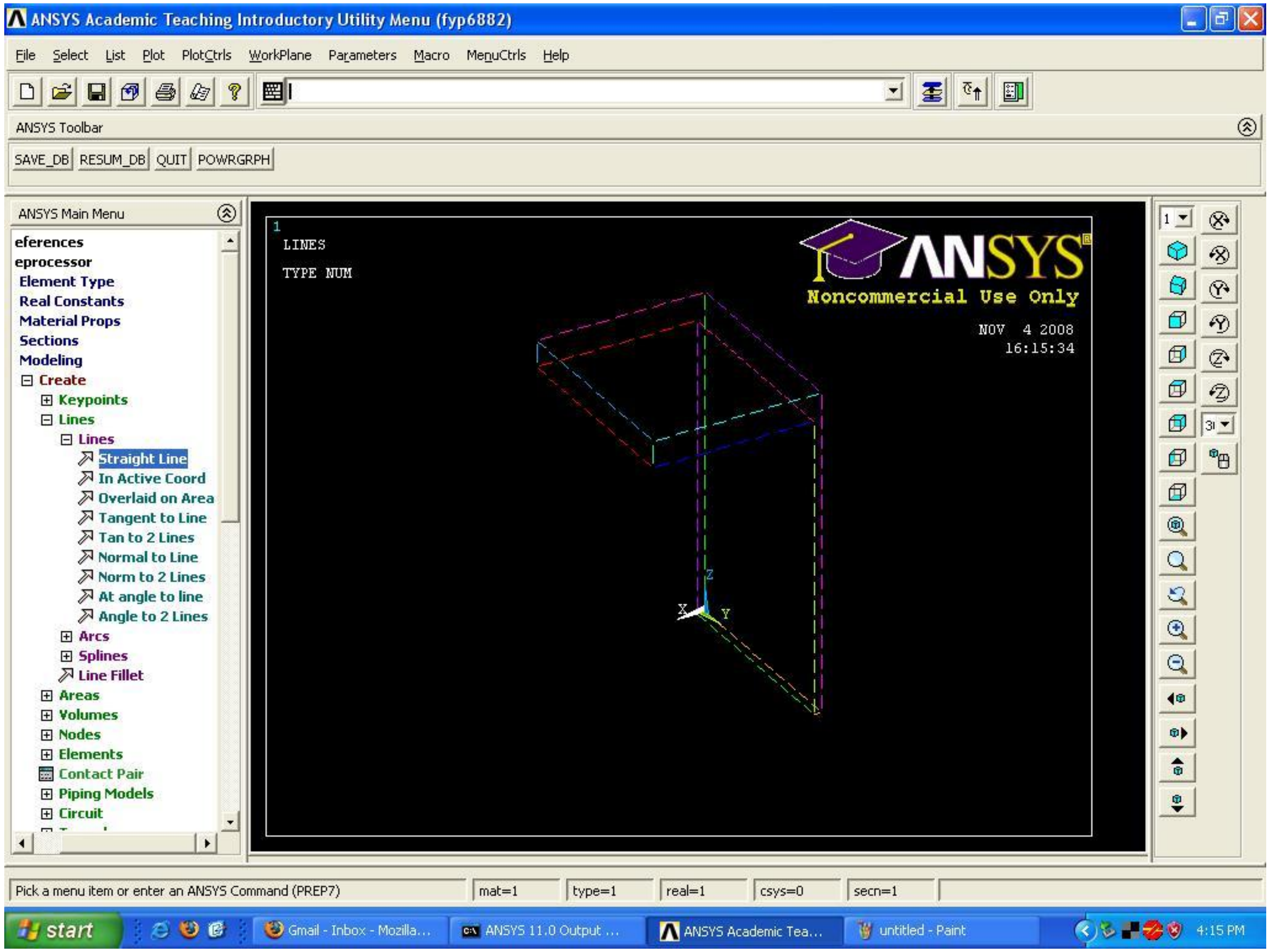
Creating keypoint



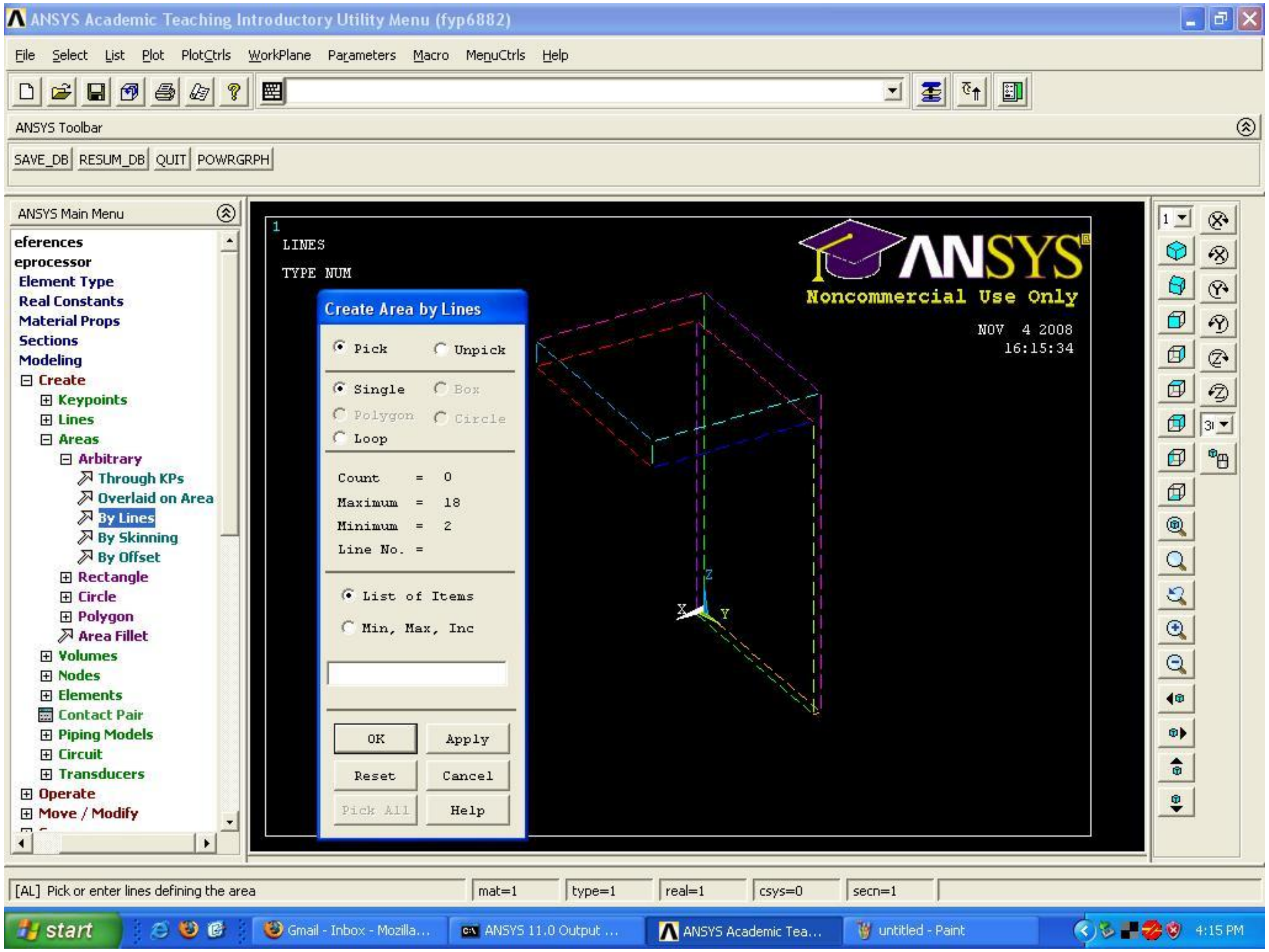
Created keypoints



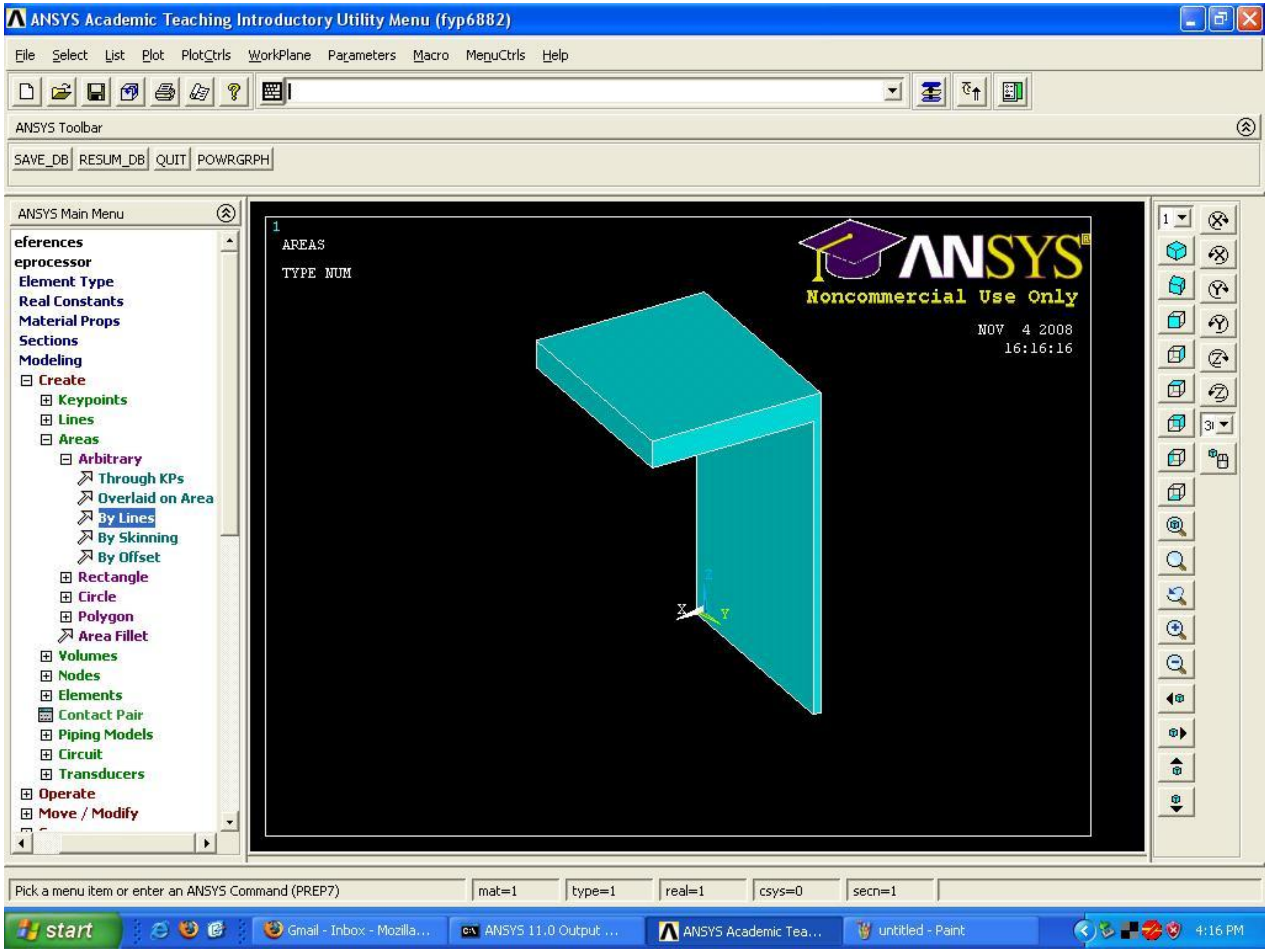
Creating line



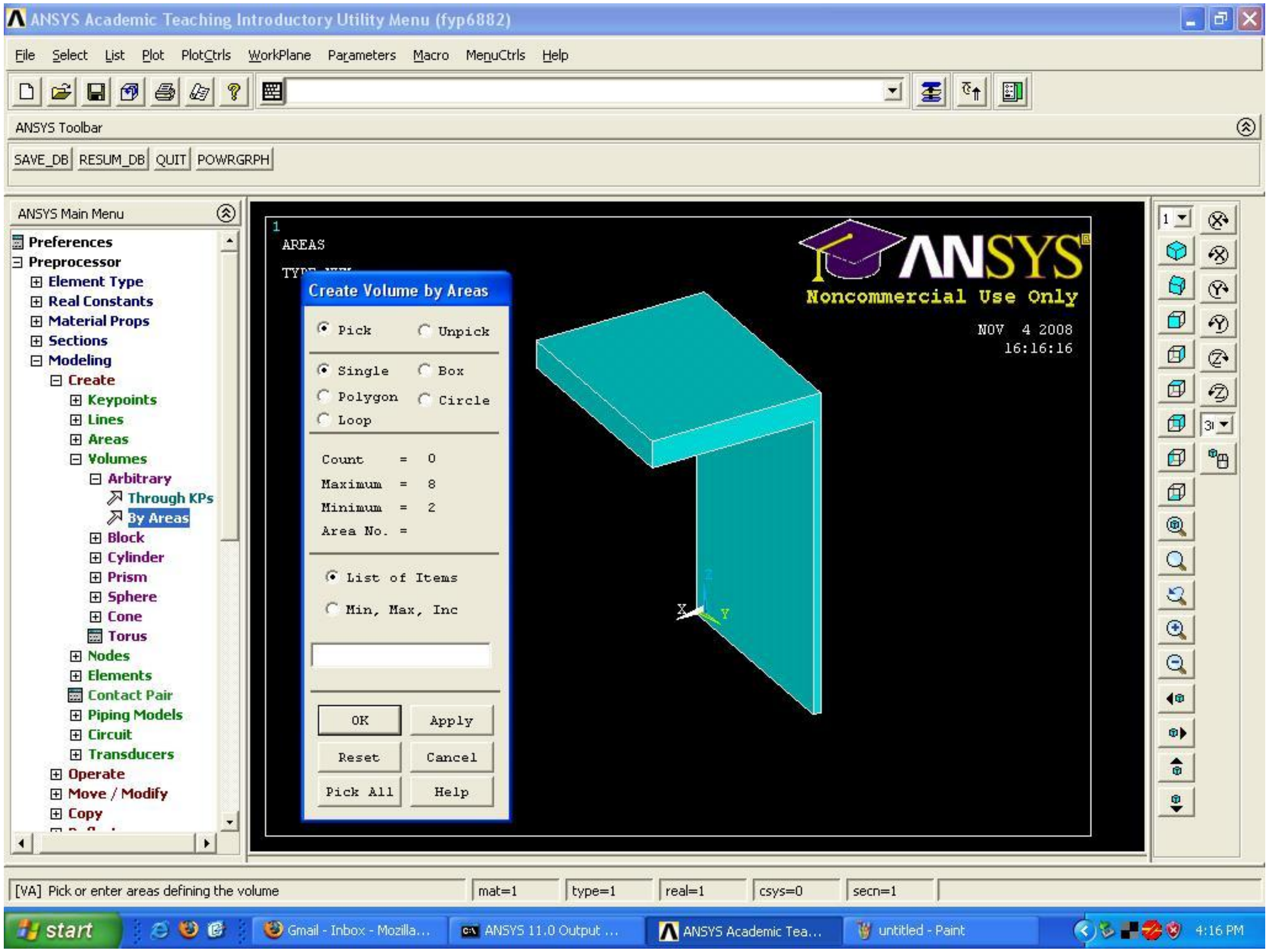
Created lines


ANSYS Academic Teaching Introductory Utility Menu (fyp6882)
File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help
ANSYS Toolbar
SAVE_DB RESUM_DB QUIT POWRGRPH
ANSYS Main Menu
eferences
eprocessor
Element Type
Real Constants
Material Props
Sections
Modeling
Create
Keypoints
Lines
Areas
Arbitrary
Through KPs
Overlaid on Area
By Lines
By Skinning
By Offset
Rectangle
Circle
Polygon
Area Fillet
Volumes
Nodes
Elements
Contact Pair
Piping Models
Circuit
Transducers
Operate
Move / Modify
LINES
TYPE NUM
ANSYS
Noncommercial Use Only
NOV 4 2008
16:15:34
Create Area by Lines
Pick
Unpick
Single
Box
Polygon
Circle
Loop
Count = 0
Maximum = 18
Minimum = 2
Line No. =
List of Items
Min, Max, Inc
OK
Apply
Reset
Cancel
Pick All
Help
[AL] Pick or enter lines defining the area
mat=1
type=1
real=1
csys=0
secn=1
start
Gmail - Inbox - Mozilla...
ANSYS 11.0 Output ...
ANSYS Academic Tea...
untitled - Paint
4:15 PM

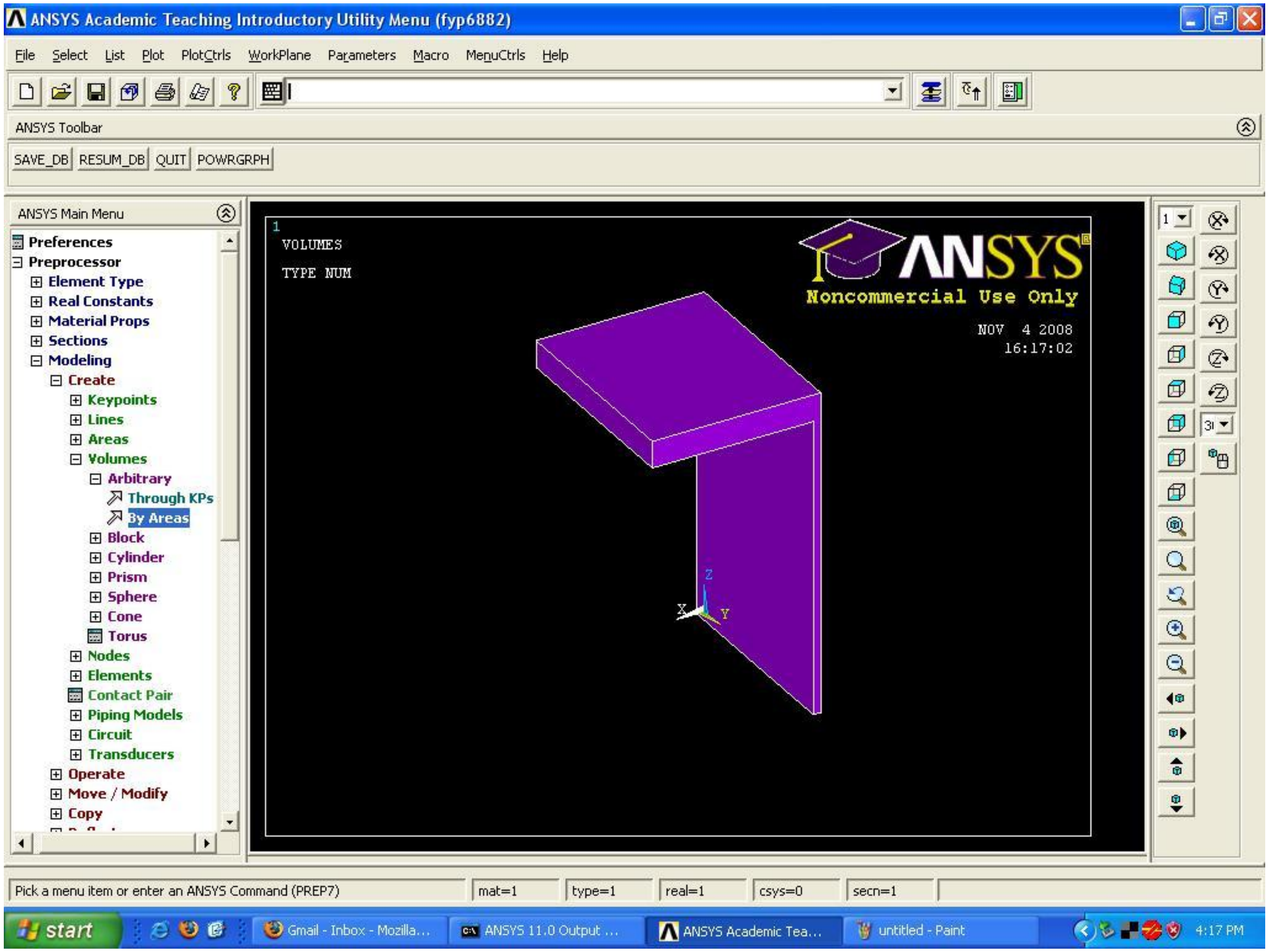
Create area



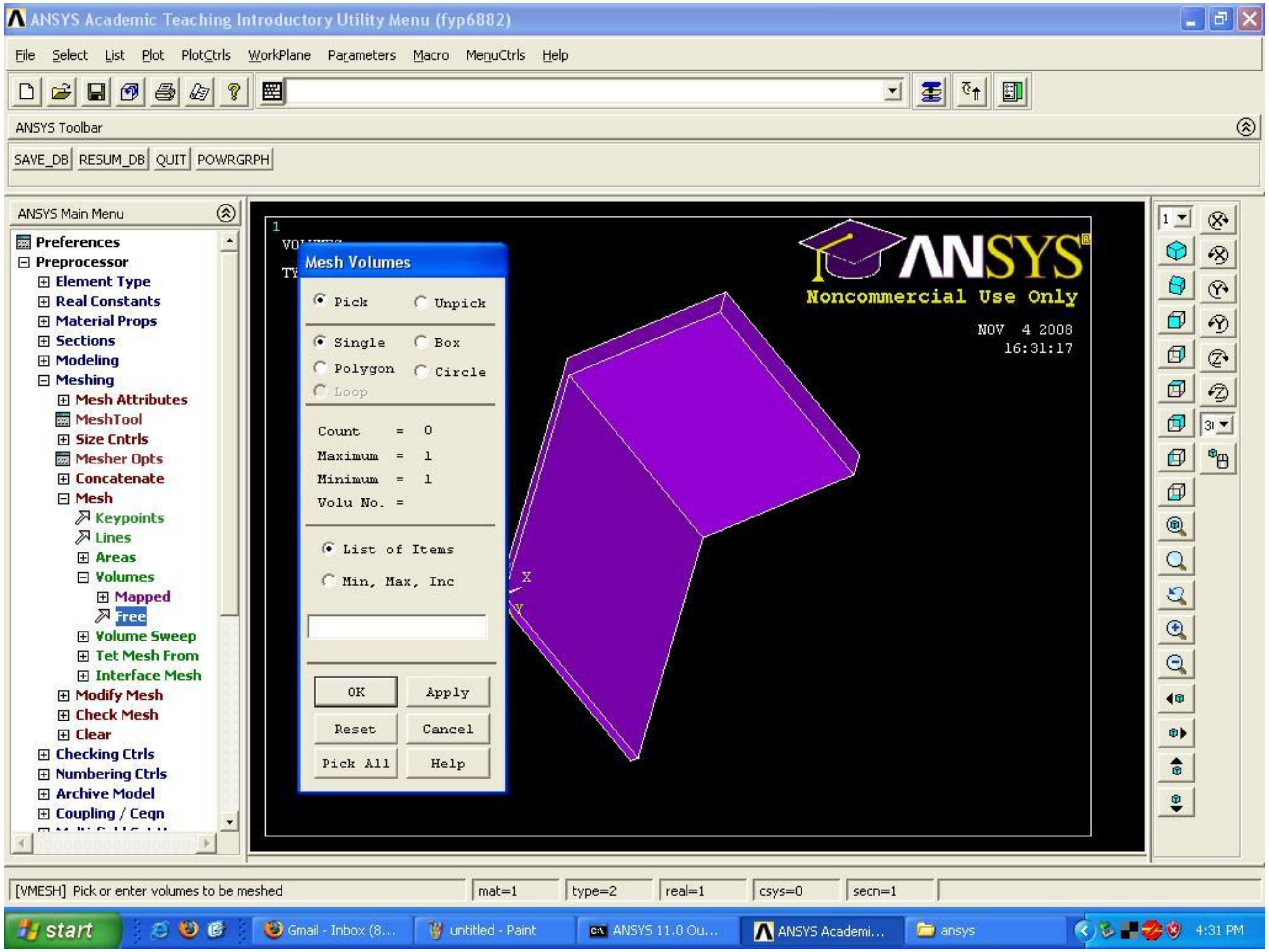
Created areas



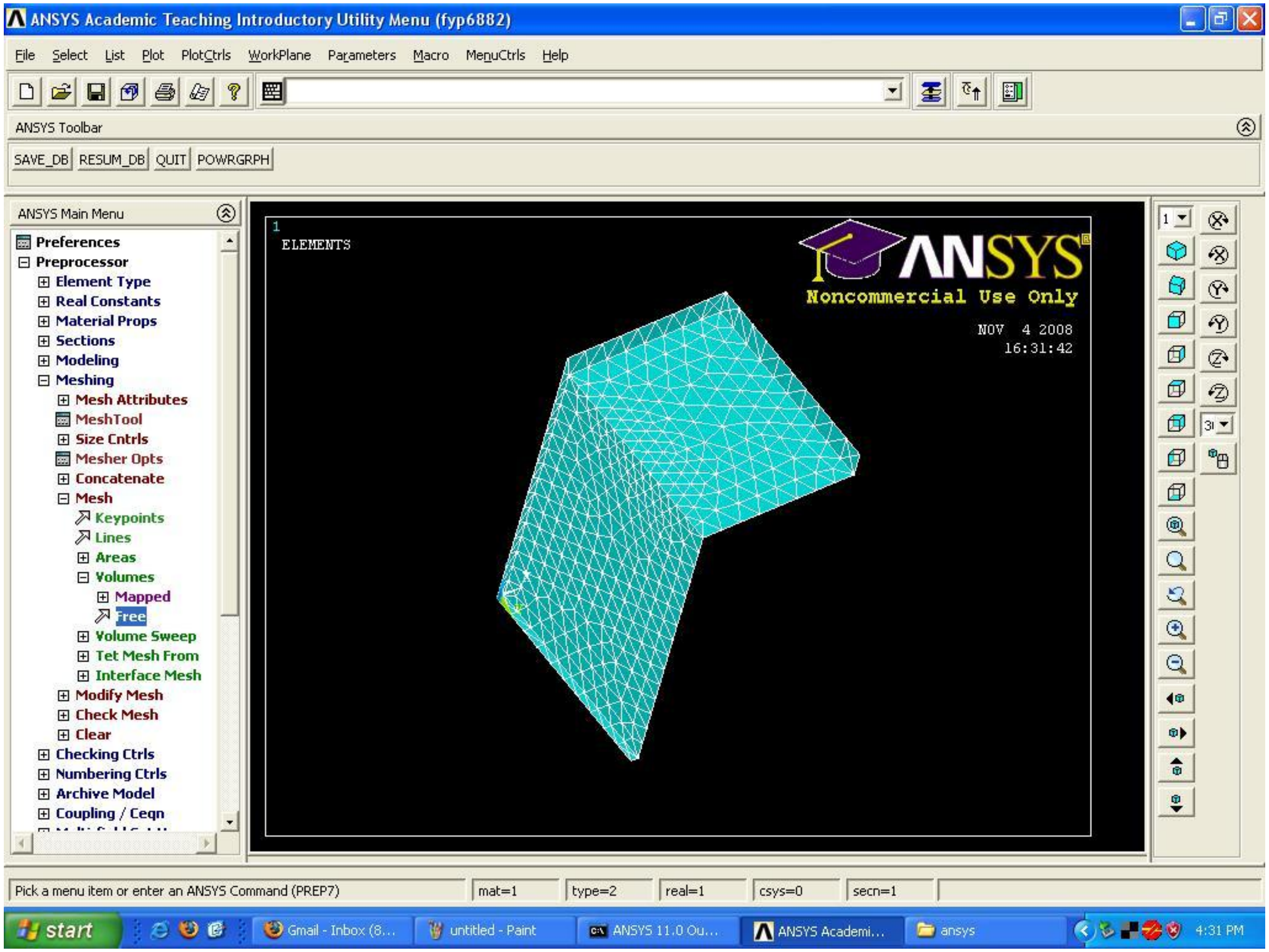
Create volume



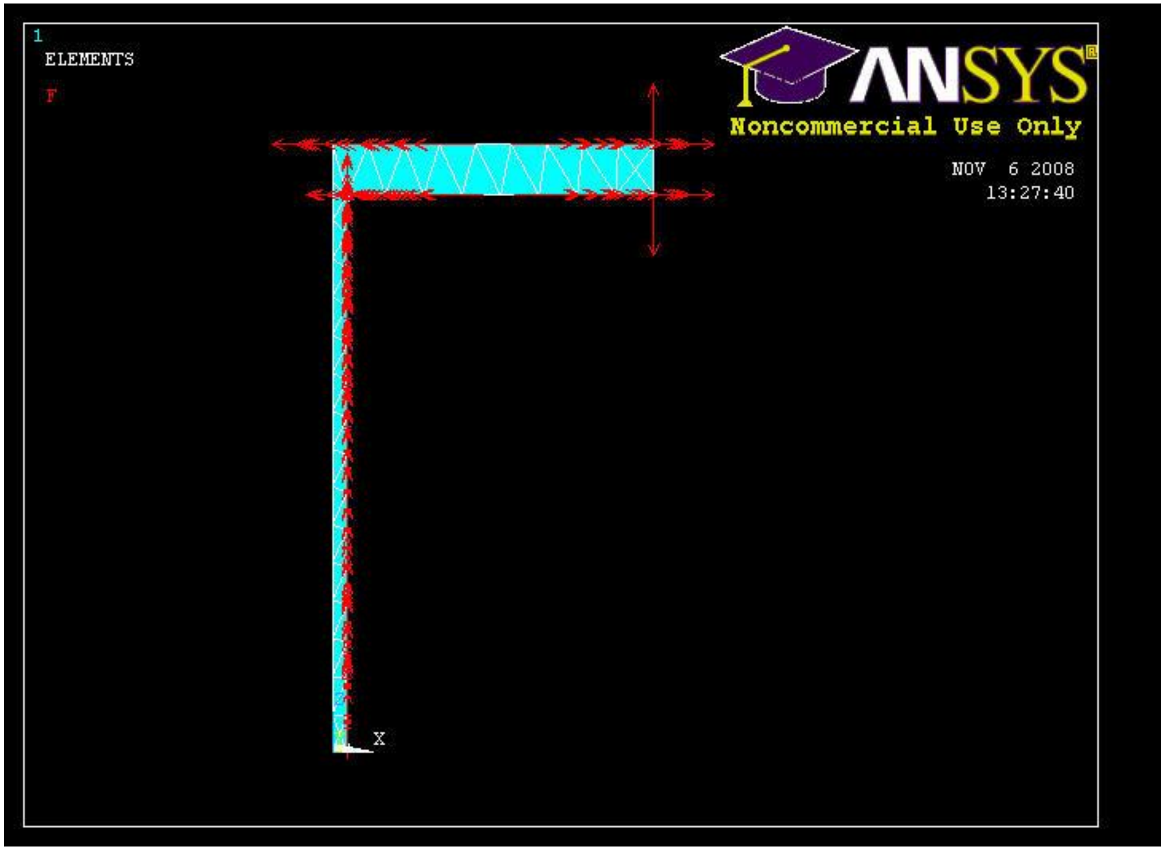
Created volumes


ANSYS Academic Teaching Introductory Utility Menu (fyp6882)
File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help
ANSYS Toolbar
SAVE_DB RESUM_DB QUIT POWRGRPH
ANSYS Main Menu
Preferences
Preprocessor
Element Type
Real Constants
Material Props
Sections
Modeling
Meshing
Mesh Attributes
MeshTool
Size Cntrls
Mesher Opts
Concatenate
Mesh
Keypoints
Lines
Areas
Volumes
Mapped
Free
Volume Sweep
Tet Mesh From
Interface Mesh
Modify Mesh
Check Mesh
Clear
Checking Ctrls
Numbering Ctrls
Archive Model
Coupling / Ceqn
Mesh Volumes
Pick Unpick
Single Box
Polygon Circle
Loop
Count = 0
Maximum = 1
Minimum = 1
Volu No. =
List of Items
Min, Max, Inc
OK Apply
Reset Cancel
Pick All Help
ANSYS
Noncommercial Use Only
NOV 4 2008
16:31:17
[VMESH] Pick or enter volumes to be meshed
mat=1
type=2
real=1
csys=0
secn=1

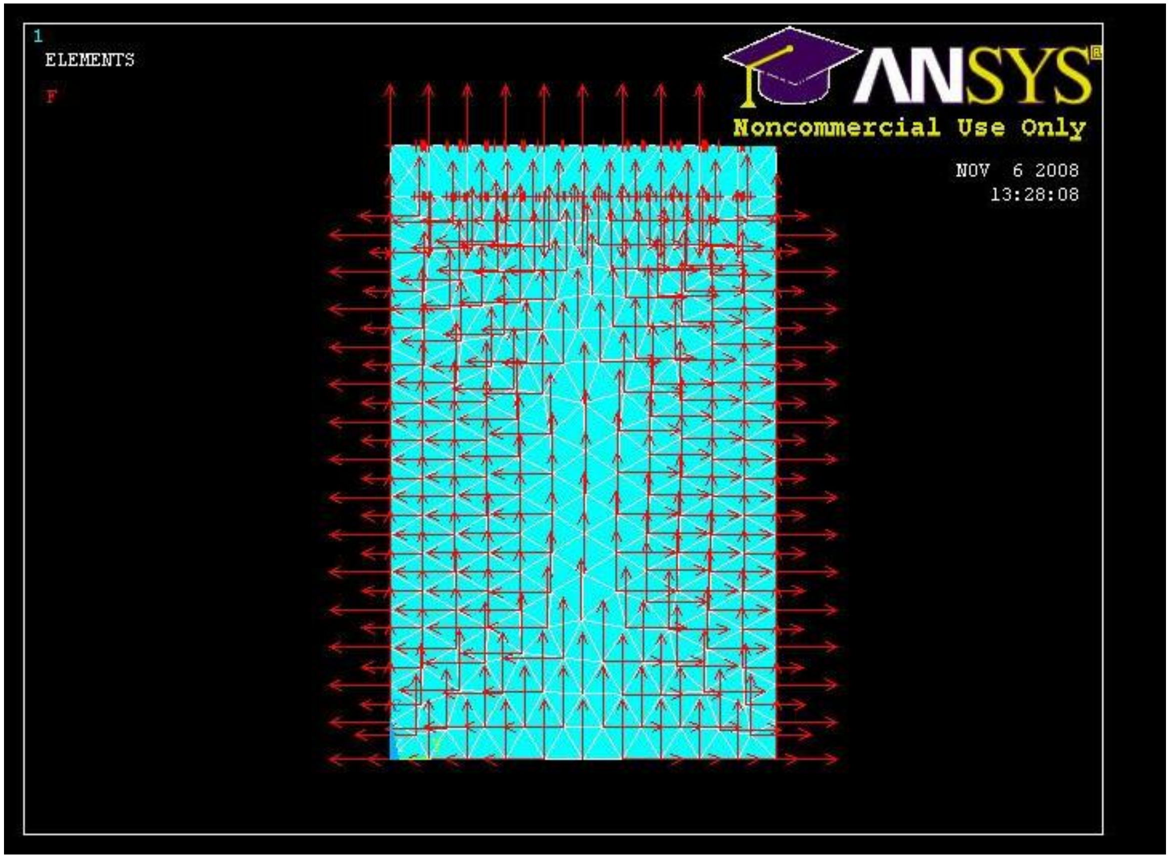
Meshing


ANSYS Academic Teaching Introductory Utility Menu (fyp6882)
File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help
ANSYS Toolbar
SAVE_DB RESUM_DB QUIT POWRGRPH
ANSYS Main Menu
Preferences
Preprocessor
Element Type
Real Constants
Material Props
Sections
Modeling
Meshing
Mesh Attributes
MeshTool
Size Cntrls
Mesher Opts
Concatenate
Mesh
Keypoints
Lines
Areas
Volumes
Mapped
Free
Volume Sweep
Tet Mesh From
Interface Mesh
Modify Mesh
Check Mesh
Clear
Checking Ctrls
Numbering Ctrls
Archive Model
Coupling / Ceqn
ELEMENTS
ANSYS
Noncommercial Use Only
NOV 4 2008
16:31:42
Pick a menu item or enter an ANSYS Command (PREP7)
mat=1
type=2
real=1
csys=0
secn=1
start
4:31 PM

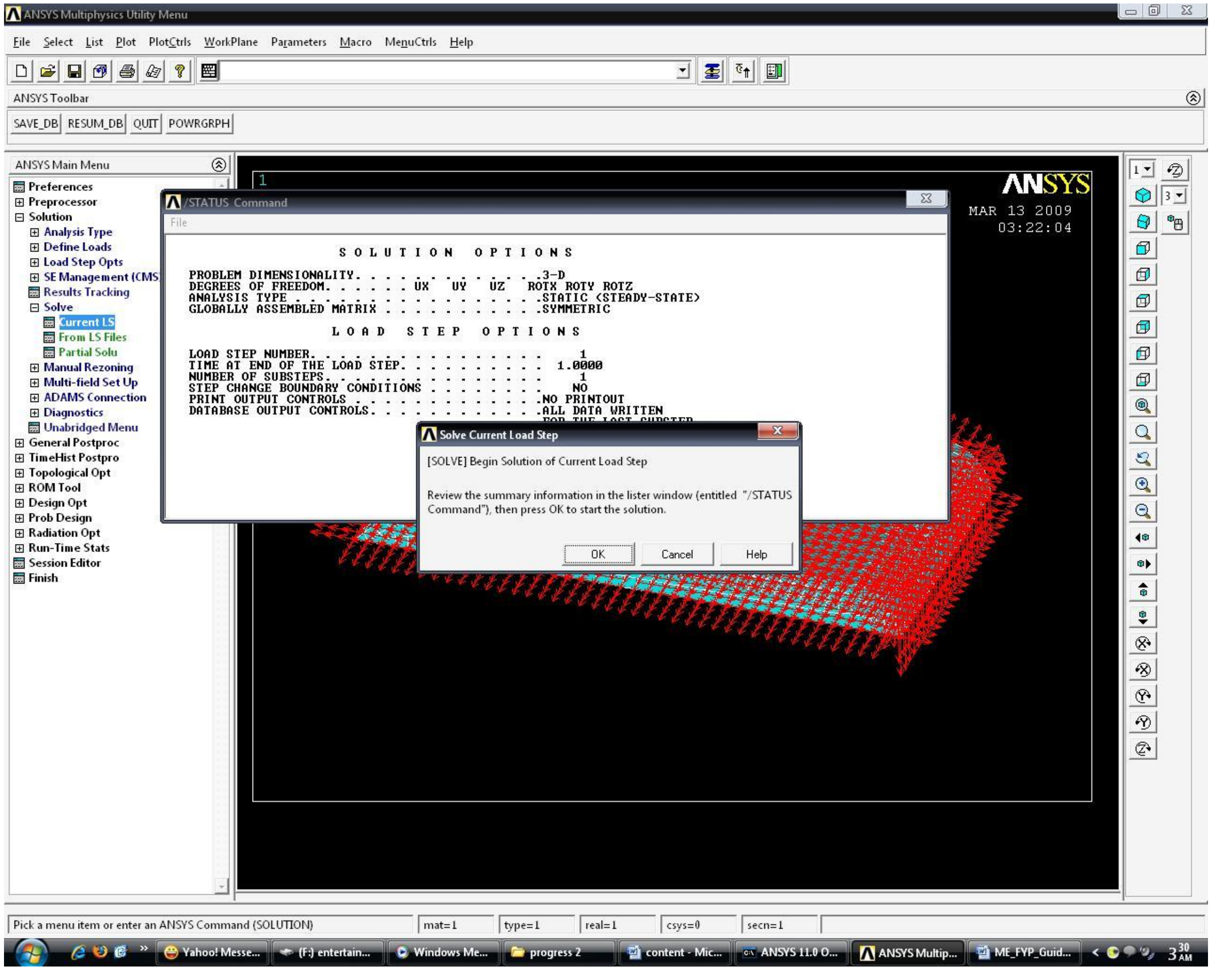
Meshed



The stress from side view



The stress from front view



To run the simulation

First semester Gantt chart

No.	**Detail/Week** Process	**1**	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**	**12**	**13**	**14**
1	Selection of Project Topic	■	■												
2	Preliminary Research Work		■	■	■										
3	Submission of Preliminary Report				■										
4	Project Work : Type of intumescent coating search Type of structure search					■	■	■							
5	Completion of Progress Report							■	■						
6	Seminar preparation							■	■						
7	Project work: Preliminary analytical calculation for 1-D and 2-D									■	■	■	■	■	■
8	Completion of Interim Report Final Draft												■	■	
9	Oral Presentation preparation													■	■

Second semester Gantt chart

No.	Detail/Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14
1	Project work: Preliminary analytical calculation for 1-D and 2-D														
2	Completion of Progress Report 1														
3	Project Work: 3-D simulation by using ANSYS														
4	Completion of Progress Report 2														
5	Seminar preparation														
6	Project work: Continuation of 3-D simulation by using ANSYS														
7	Poster Exhibition														
8	Completion of Dissertation (soft bound)														
9	Oral Presentation														
10	Completion of Project Dissertation (Hard Bound)														

Process