3D Truss Analysis using MATLAB

Deepak Raina

M15ME003

Guided By: Dr. Akshay Prakash

MECHANICAL ENGINEERING DEPARTMENT

INDIAN INSTITUTE OF TECHNOLOGY, JODHPUR (RAJ.), INDIA

INTRODUCTION

- A *Truss* is essentially a triangulated system of straight interconnected structural elements.

- The Purpose of this program is to analyze all 2D/3D Trusses with all degrees of freedom using stiffness method (matrix analysis) under any kind of concentrated nodal loadings (F_x, F_y, F_z) and to submit values of supportive reactions, nodal displacements, axial forces and element stresses and strain as MATLAB Output.

- General feature of this program includes one ‘m-file’ and an ‘Excel’ input file, which are required to run this program. Using this program is very easy and user-friendly.

Applications of Trusses

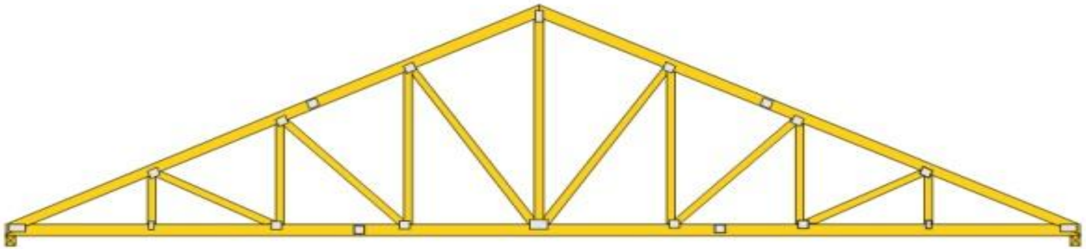


Fig1: Roof Top Truss

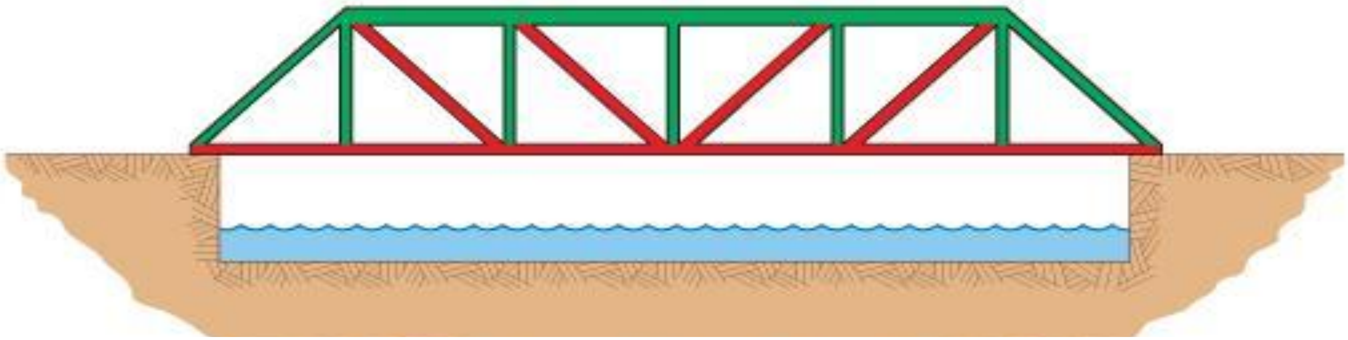


Fig2: Bridges

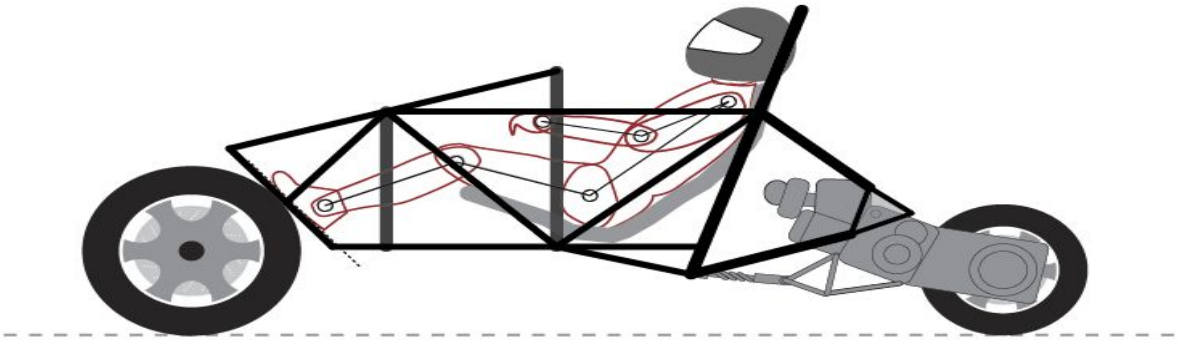


Fig3: Roll Cage of Racing Cars

Fig4: Supporting Towers

Finite Element Model for 1D Truss

The element, which we simply call a bar element, is particularly useful in the analysis of both two- and three-dimensional frame or truss structures.

Assumptions:

1. The bar is geometrically straight.
2. The material obeys Hooke's law.
3. Forces are applied only at the ends of the bar.
4. The bar supports axial loading only; bending, torsion, and shear are not transmitted to the element via the nature of its connections to other elements.
5. The bar is connected to other structural members via pins (2-D) or ball-and-socket joints (3-D)

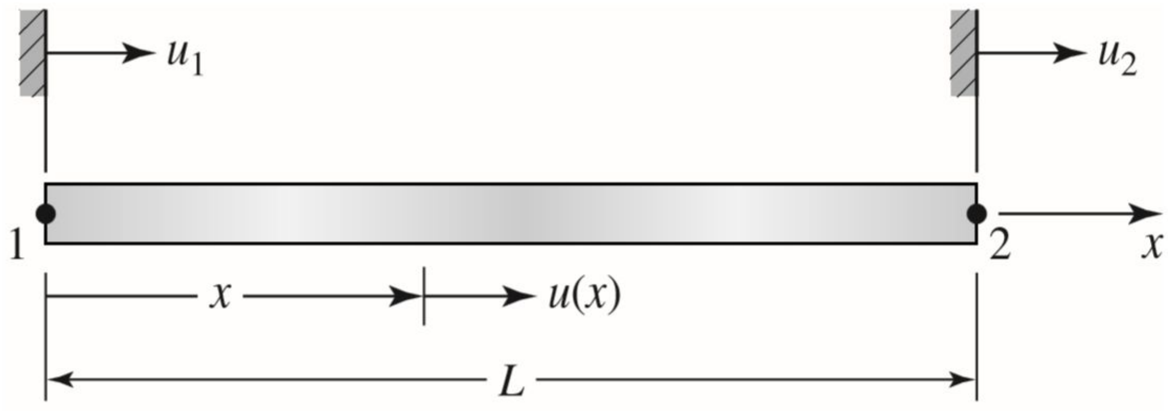


Fig 5: A bar (or truss) element

The elementary strength of materials that the deflection δ of an elastic bar of length L and uniform cross-sectional area A when subjected to axial load P is given by

$$\boxed{\delta = \frac{PL}{AE}}$$

where E is the modulus of elasticity of the material.

The equivalent spring constant of an elastic bar will be given by

$$k = \frac{P}{\delta} = \frac{AE}{L}$$

In uniaxial loading, as in the bar element, we need consider only the normal strain component, defined as

$$\varepsilon_x = \frac{du}{dx}$$

$$\varepsilon_x = \frac{u_2 - u_1}{L}$$

The axial stress, by Hooke's law, is then

$$\boxed{P = \sigma_x A = \frac{AE}{L}(u_2 - u_1)}$$

Let the applied nodal forces be f_1 and f_1. Then,

$$f_1 = -\frac{AE}{L}\,(u_2 - u_1)$$

$$f_2 = \frac{AE}{L}\,(u_2 - u_1)$$

In Matrix Form,

$$\boxed{\frac{\boldsymbol{AE}}{\boldsymbol{L}}\begin{bmatrix} \mathbf{1} & \mathbf{-1} \\ \mathbf{-1} & \mathbf{1} \end{bmatrix}\begin{bmatrix} \boldsymbol{u_1} \\ \boldsymbol{u_2} \end{bmatrix} = \begin{bmatrix} \boldsymbol{f_1} \\ \boldsymbol{f_2} \end{bmatrix}}$$

Here the **Stiffness Matrix for Bar Element** is given by

$$\boxed{\boldsymbol{k_e} = \frac{\boldsymbol{AE}}{\boldsymbol{L}}\begin{bmatrix} \mathbf{1} & \mathbf{-1} \\ \mathbf{-1} & \mathbf{1} \end{bmatrix}}$$

Note:

1. The Element Stiffness Matrix for bar element is symmetric.
2. The stiffness matrix is one-dimensional. Application of this element formulation is to analyse two- and three-dimensional structures

Finite Element Model for 2D Truss

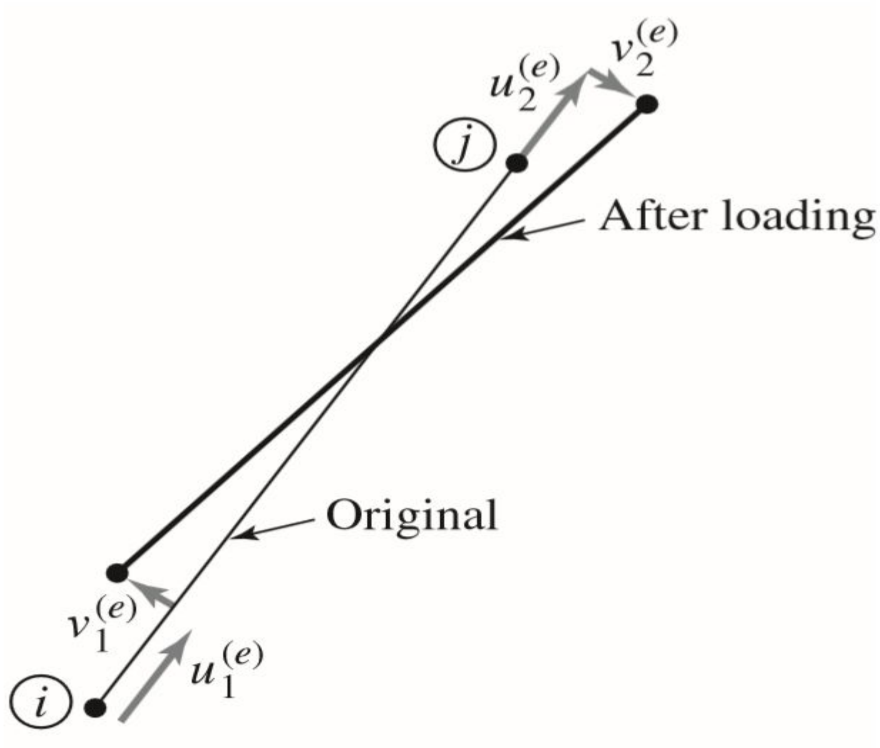


Fig7: General Displacements of Bar element in 2D

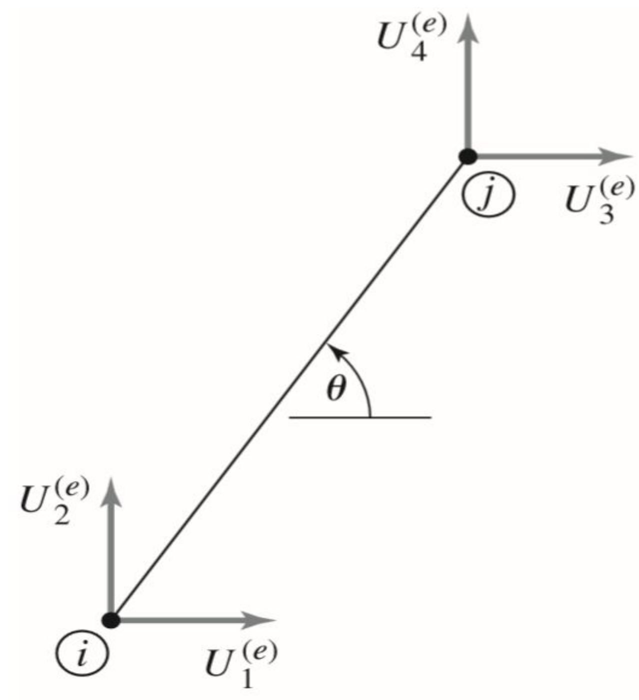


Fig8: Bar element Global Displacements

Here,

$$\boxed{\begin{aligned} u_1^{(e)} &= U_1^{(e)}\cos\theta + U_2^{(e)}\sin\theta \\ u_2^{(e)} &= U_3^{(e)}\cos\theta + U_4^{(e)}\sin\theta \end{aligned}}$$

which can be written in matrix form as

$$\begin{Bmatrix} u_1^{(e)} \\ u_2^{(e)} \end{Bmatrix} = \begin{bmatrix} \cos\theta & \sin\theta & 0 & 0 \\ 0 & 0 & \cos\theta & \sin\theta \end{bmatrix} \begin{Bmatrix} U_1^{(e)} \\ U_2^{(e)} \\ U_3^{(e)} \\ U_4^{(e)} \end{Bmatrix} = [R] \begin{Bmatrix} U_1^{(e)} \\ U_2^{(e)} \\ U_3^{(e)} \\ U_4^{(e)} \end{Bmatrix}$$

where

$$[R] = \begin{bmatrix} \cos\theta & \sin\theta & 0 & 0 \\ 0 & 0 & \cos\theta & \sin\theta \end{bmatrix}$$

is the transformation matrix of element axial displacements to global displacements.

Recalling the bar element equations expressed in the element frame as

$$\frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_1^{(e)} \\ u_2^{(e)} \end{Bmatrix} = \begin{bmatrix} k_e & -k_e \\ -k_e & k_e \end{bmatrix} \begin{Bmatrix} u_1^{(e)} \\ u_2^{(e)} \end{Bmatrix} = \begin{Bmatrix} f_1^{(e)} \\ f_2^{(e)} \end{Bmatrix}$$

Using the Previous two equations, we get

$$\begin{bmatrix} k_e & -k_e \\ -k_e & k_e \end{bmatrix} \begin{bmatrix} \cos\theta & \sin\theta & 0 & 0 \\ 0 & 0 & \cos\theta & \sin\theta \end{bmatrix} \begin{Bmatrix} U_1^{(e)} \\ U_2^{(e)} \\ U_3^{(e)} \\ U_4^{(e)} \end{Bmatrix} = \begin{Bmatrix} f_1^{(e)} \\ f_2^{(e)} \end{Bmatrix}$$

or

$$\begin{bmatrix} k_e & -k_e \\ -k_e & k_e \end{bmatrix} [R] \begin{Bmatrix} U_1^{(e)} \\ U_2^{(e)} \\ U_3^{(e)} \\ U_4^{(e)} \end{Bmatrix} = \begin{Bmatrix} f_1^{(e)} \\ f_2^{(e)} \end{Bmatrix}$$

Now we have transformed the equilibrium equations from 1D displacements to 2D displacements , the RHS equations are still expressed in the 1D coordinate system. So we will pre-multiply both sides of Equation by $[R]^T$, the transpose of the transformation matrix; that is,

$$[R]^T \begin{bmatrix} k_e & -k_e \\ -k_e & k_e \end{bmatrix} [R] \begin{Bmatrix} U_1^{(e)} \\ U_2^{(e)} \\ U_3^{(e)} \\ U_4^{(e)} \end{Bmatrix} = \begin{bmatrix} \cos\theta & 0 \\ \sin\theta & 0 \\ 0 & \cos\theta \\ 0 & \sin\theta \end{bmatrix} \begin{Bmatrix} f_1^{(e)} \\ f_2^{(e)} \end{Bmatrix} = \begin{Bmatrix} f_1^{(e)}\cos\theta \\ f_1^{(e)}\sin\theta \\ f_2^{(e)}\cos\theta \\ f_2^{(e)}\sin\theta \end{Bmatrix}$$

The RHS of equation can further be written as:

$$[R]^T \begin{bmatrix} k_e & -k_e \\ -k_e & k_e \end{bmatrix} [R] \begin{Bmatrix} U_1^{(e)} \\ U_2^{(e)} \\ U_3^{(e)} \\ U_4^{(e)} \end{Bmatrix} = \begin{Bmatrix} F_1^{(e)} \\ F_2^{(e)} \\ F_3^{(e)} \\ F_4^{(e)} \end{Bmatrix}$$

Here, **Element Stiffness matrix in 2D** Frame is given by

$$[K^{(e)}] = [R]^T \begin{bmatrix} k_e & -k_e \\ -k_e & k_e \end{bmatrix} [R]$$

or

$$\boxed{[K^{(e)}] = k_e \begin{bmatrix} c^2 & sc & -c^2 & -sc \\ sc & s^2 & -sc & -s^2 \\ -c^2 & -sc & c^2 & sc \\ -sc & -s^2 & sc & s^2 \end{bmatrix}}$$

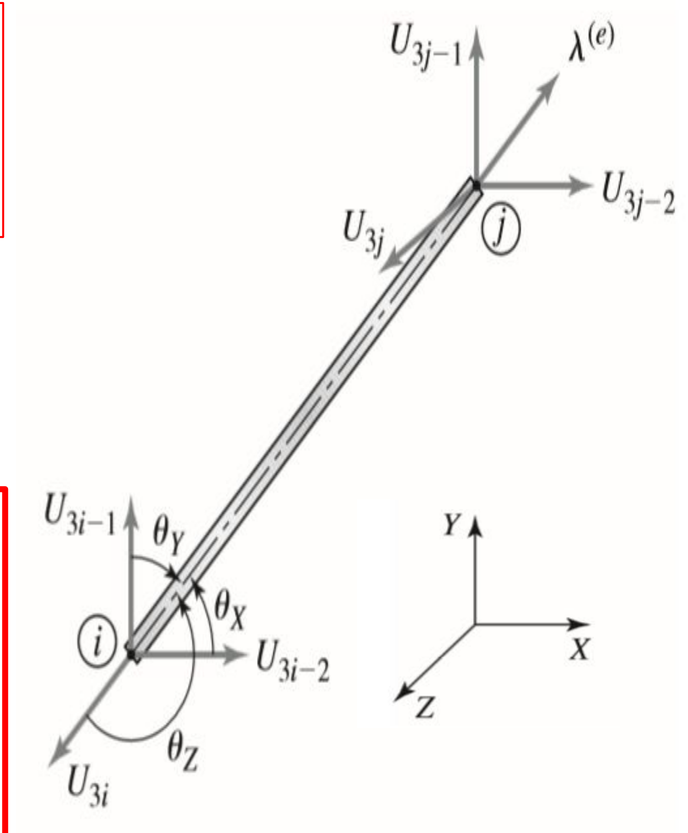
Where $c = cos\theta = \frac{x_j - x_i}{L}$, $s = sin\theta = \frac{y_j - y_i}{L}$

and, L = $\sqrt{(x_j - x_i)^2 + (y_j - y_i)^2}$

Finite Element Model for 3D Truss

The element displacements are expressed in components in the 3-D global system as :

$$
\boxed{\begin{aligned}
u_1^{(e)} &= U_1^{(e)} \cos\theta_x + U_2^{(e)} \cos\theta_y + U_3^{(e)} \cos\theta_z \\
u_2^{(e)} &= U_4^{(e)} \cos\theta_x + U_5^{(e)} \cos\theta_y + U_6^{(e)} \cos\theta_z
\end{aligned}}
$$



Following the similar procedure as 2D Truss, the **3-D stiffness matrix** for one-dimensional bar element is given by

$$
\boxed{[K^{(e)}] = k_e \begin{bmatrix}
c_x^2 & c_x c_y & c_x c_z & -c_x^2 & -c_x c_y & -c_x c_z \\
c_x c_y & c_y^2 & c_y c_z & -c_x c_x & -c_y^2 & -c_y c_z \\
c_x c_z & c_y c_z & c_z^2 & -c_x c_z & -c_y c_z & -c_z^2 \\
-c_x^2 & -c_x c_x & -c_x c_z & c_x^2 & c_x c_y & c_x c_z \\
-c_x c_y & -c_y^2 & -c_y c_z & c_x c_y & c_y^2 & c_y c_z \\
-c_x c_z & -c_y c_z & -c_z^2 & c_x c_z & c_y c_z & c_z^2
\end{bmatrix}}
$$

where

$$
\begin{aligned}
c_x &= \cos\theta_x \\
c_y &= \cos\theta_y \\
c_z &= \cos\theta_z
\end{aligned}
$$

Direct Assembly Of Global Stiffness Matrix

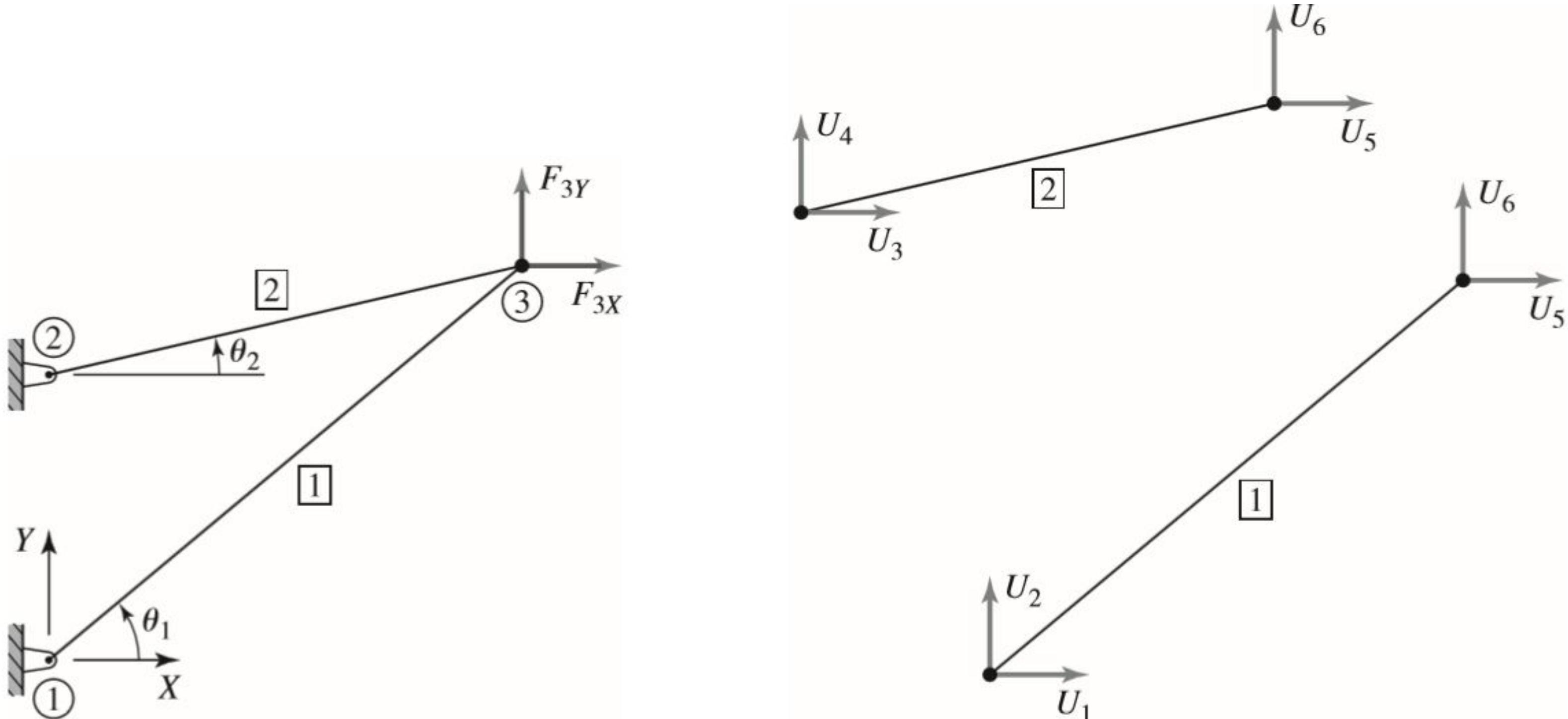


Fig 9 : A Two-element Truss

$$[K^{(1)}] = \begin{array}{c} \\ \left[\begin{matrix} k_{11}^{(1)} & k_{12}^{(1)} & k_{13}^{(1)} & k_{14}^{(1)} \\ k_{21}^{(1)} & k_{22}^{(1)} & k_{23}^{(1)} & k_{24}^{(1)} \\ k_{31}^{(1)} & k_{32}^{(1)} & k_{33}^{(1)} & k_{34}^{(1)} \\ k_{41}^{(1)} & k_{42}^{(1)} & k_{43}^{(1)} & k_{44}^{(1)} \end{matrix}\right] \begin{matrix} \text{U1} \\ \text{U2} \\ \text{U5} \\ \text{U6} \end{matrix} \\ \begin{matrix} \text{U1} & \quad \text{U2} & \quad \text{U5} & \quad \text{U6} \end{matrix} \end{array}$$

$$[K^{(2)}] = \begin{array}{c} \\ \left[\begin{matrix} k_{11}^{(2)} & k_{12}^{(2)} & k_{13}^{(2)} & k_{14}^{(2)} \\ k_{21}^{(2)} & k_{22}^{(2)} & k_{23}^{(2)} & k_{24}^{(2)} \\ k_{31}^{(2)} & k_{32}^{(2)} & k_{33}^{(2)} & k_{34}^{(2)} \\ k_{41}^{(2)} & k_{42}^{(2)} & k_{43}^{(2)} & k_{44}^{(2)} \end{matrix}\right] \begin{matrix} \text{U3} \\ \text{U4} \\ \text{U5} \\ \text{U6} \end{matrix} \\ \begin{matrix} \text{U3} & \quad \text{U4} & \quad \text{U5} & \quad \text{U6} \end{matrix} \end{array}$$

The stiffness matrices, given by 2 elements together, will form the 6 ×6 system matrix containing 36 terms given by

$$
\begin{array}{c c}
\begin{array}{cccccc} U_1 & U_2 & U_3 & U_4 & U_5 & U_6 \end{array} & \\
\begin{bmatrix}
k_{11}^{(1)}+0 & k_{12}^{(1)}+0 & 0+0 & 0+0 & k_{13}^{(1)}+0 & k_{14}^{(1)}+0 \\
k_{21}^{(1)}+0 & k_{22}^{(1)}+0 & 0+0 & 0+0 & k_{23}^{(1)}+0 & k_{24}^{(1)}+0 \\
0+0 & 0+0 & 0+k_{11}^{(2)} & 0+k_{12}^{(2)} & 0+k_{13}^{(2)} & 0+k_{14}^{(2)} \\
0+0 & 0+0 & 0+k_{12}^{(2)} & 0+k_{22}^{(2)} & 0+k_{23}^{(2)} & 0+k_{24}^{(2)} \\
k_{13}^{(1)}+0 & k_{23}^{(1)}+0 & 0+k_{13}^{(2)} & 0+k_{23}^{(2)} & k_{33}^{(1)}+k_{33}^{(2)} & k_{34}^{(1)}+k_{34}^{(2)} \\
k_{14}^{(1)}+0 & k_{24}^{(1)}+0 & 0+k_{14}^{(2)} & 0+k_{24}^{(2)} & k_{34}^{(1)}+k_{34}^{(2)} & k_{44}^{(1)}+k_{44}^{(2)}
\end{bmatrix}
& \begin{array}{c} U_1 \\ U_2 \\ U_3 \\ U_4 \\ U_5 \\ U_6 \end{array}
\end{array}
$$

Algorithm of Program

1. Define all the variable data in the given Excel file. Save the File as 'Truss.xls' in the MATLAB directory.

2. The MATLAB file saved as 'Truss.m' will read all the variable data from Excel File.

3. Define the Element Stiffness Matrices for each element

4. Define Structural Stiffness Matrix {S} by using matrix assembly procedure

5. Define Nodal Forces {F} in column matrix form

6. Eliminate rows and columns of Structural Stiffness matrix with respect to Supports.

7. Solve {F} ={S} {D} to find displacement matrix {D}

8. Back-substitute displacement values {D} to obtain secondary variables, including strain, stress, and reaction forces at constrained locations.

9. Display the Results and Finish.

Variables of Program

The following variables are required to be defined in Excel file:

1. Number of nodes and elements
2. The X,Y & Z coordinate of all nodes
3. Properties of all elements such as Modulus of Elasticity, Area of Cross Section; and the geometry of the problem (element connectivity)
4. Number of constrained nodes and direction of constrain (X,Y,Z).
5. Number of nodes loaded and direction of loading (X,Y,Z)

Snippets of Program

1. Read MS Excel Spreadsheet file

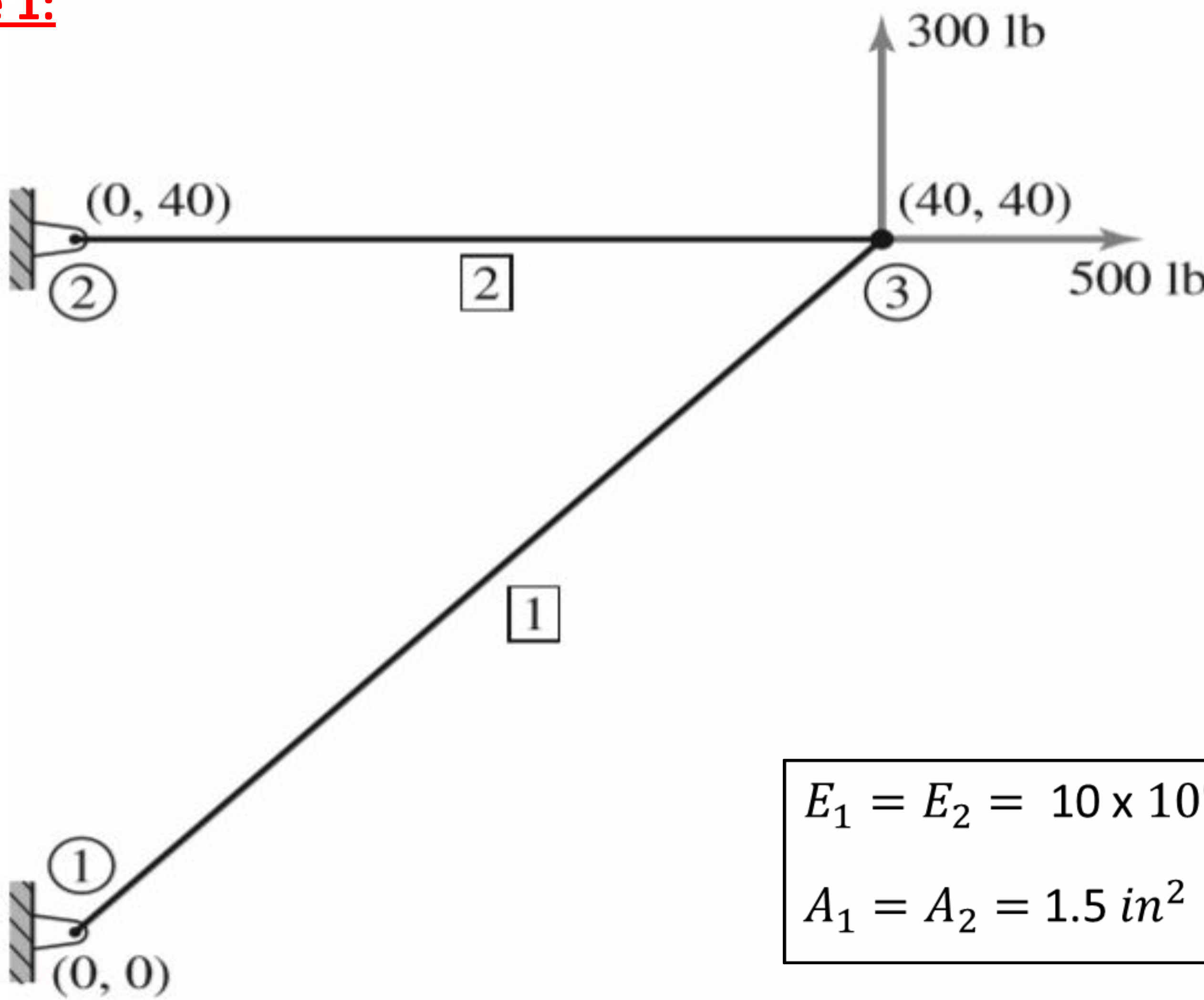
```
Num = xlsread(filename)

Eg.
truss = xlsread(truss.xls)
```

2. Create an array of Zeros:

```
M = zeros(4)

M =      0    0    0    0
     0    0    0    0
     0    0    0    0
     0    0    0    0
```

Verification of Program

Example 1:



$E_1 = E_2 = 10 \times 10^6$ lb$/in^2$

$A_1 = A_2 = 1.5\ in^2$

Verification with ANSYS® Results

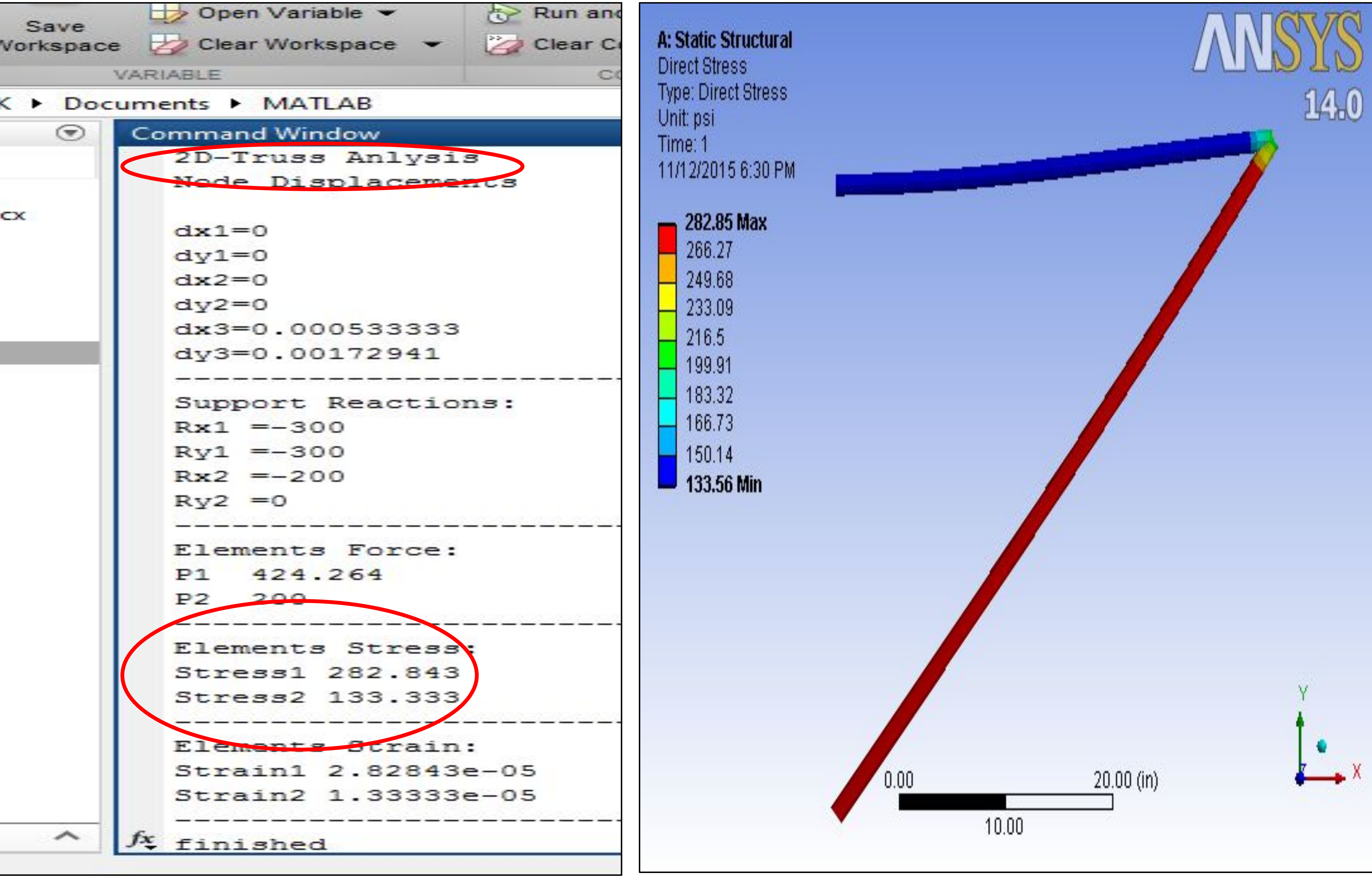
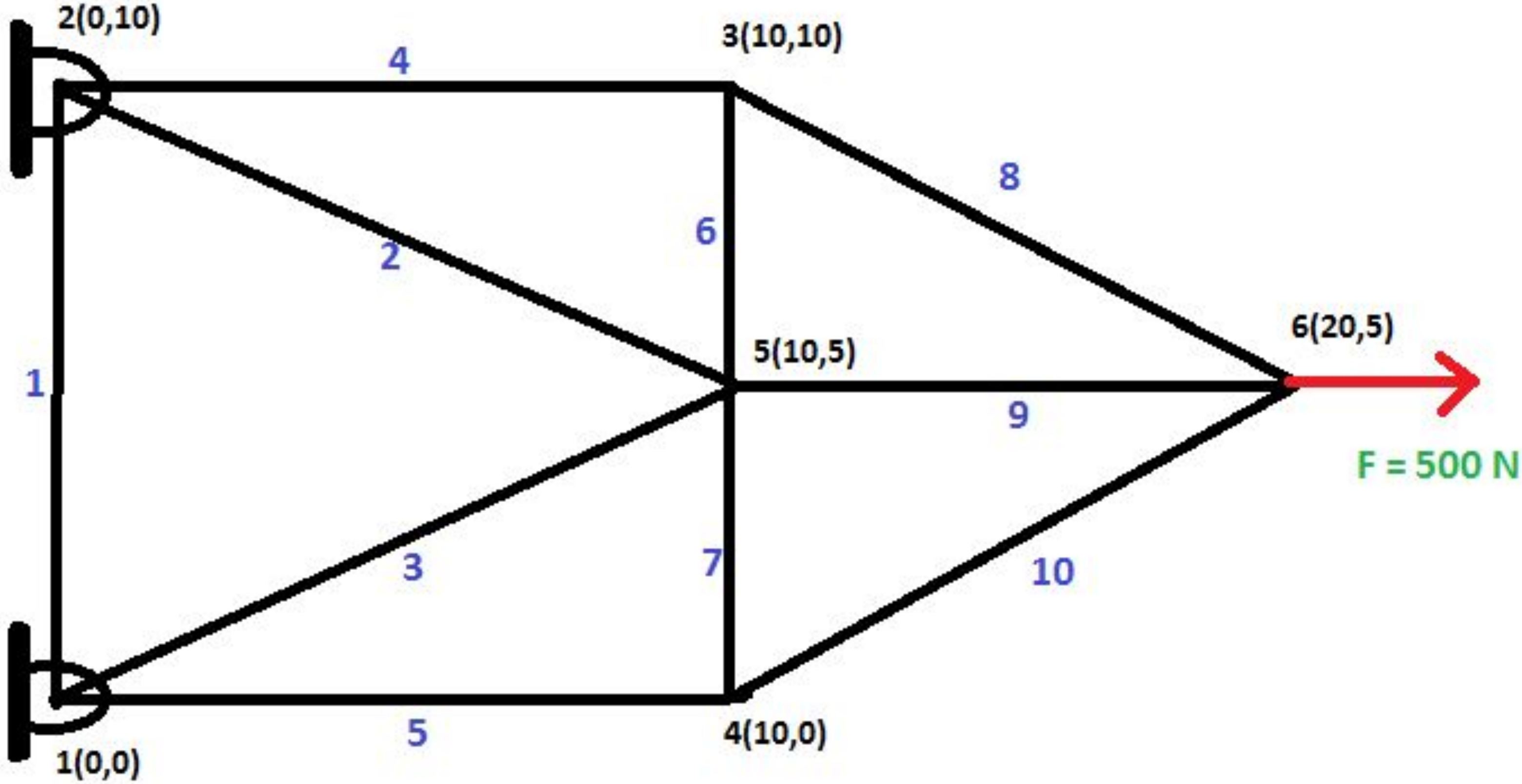
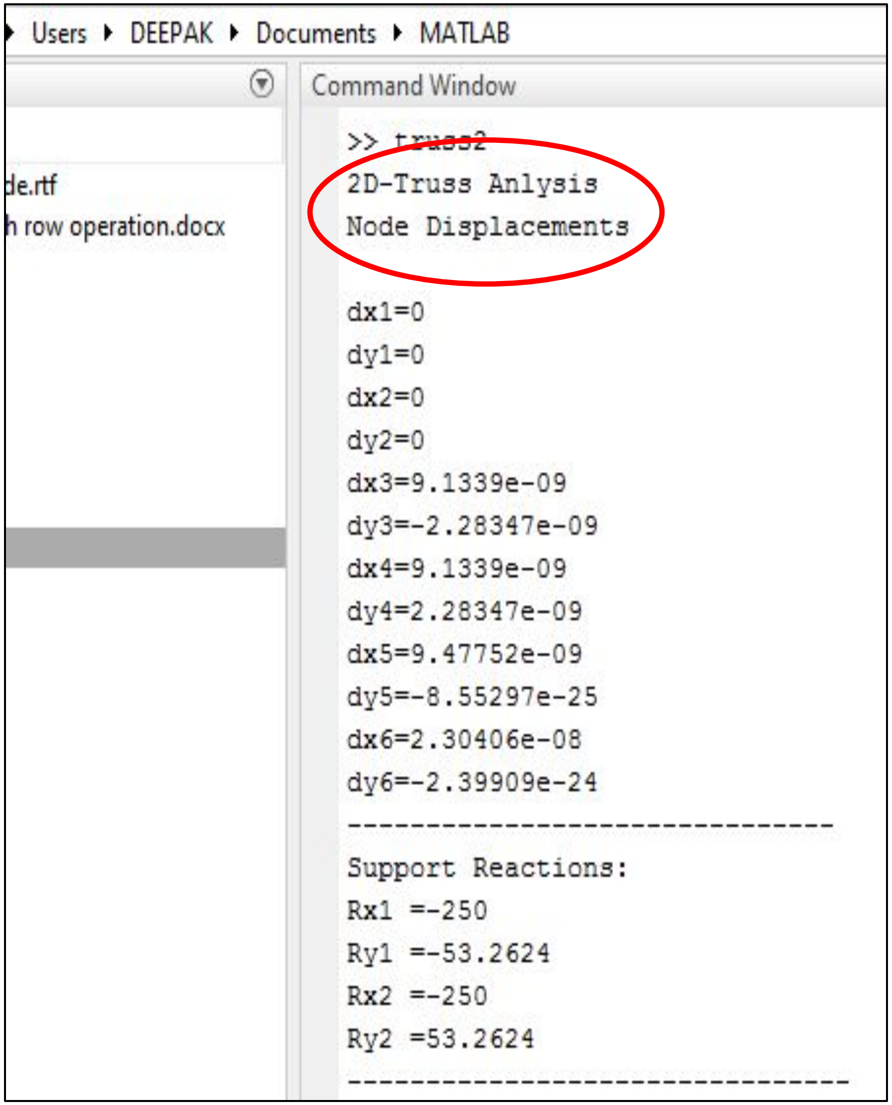


Fig 10: Stress values in Truss Structure

Example 2:



Verification with ANSYS® Results



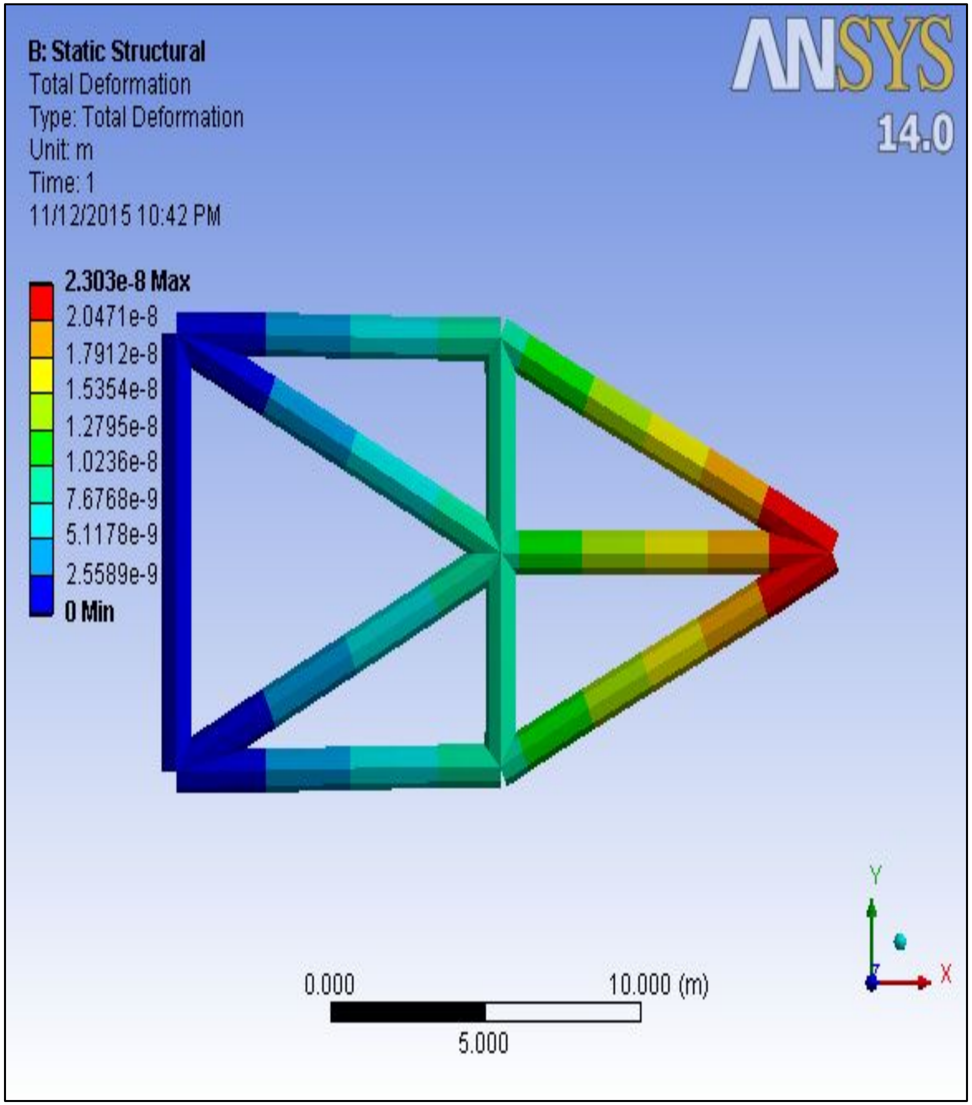
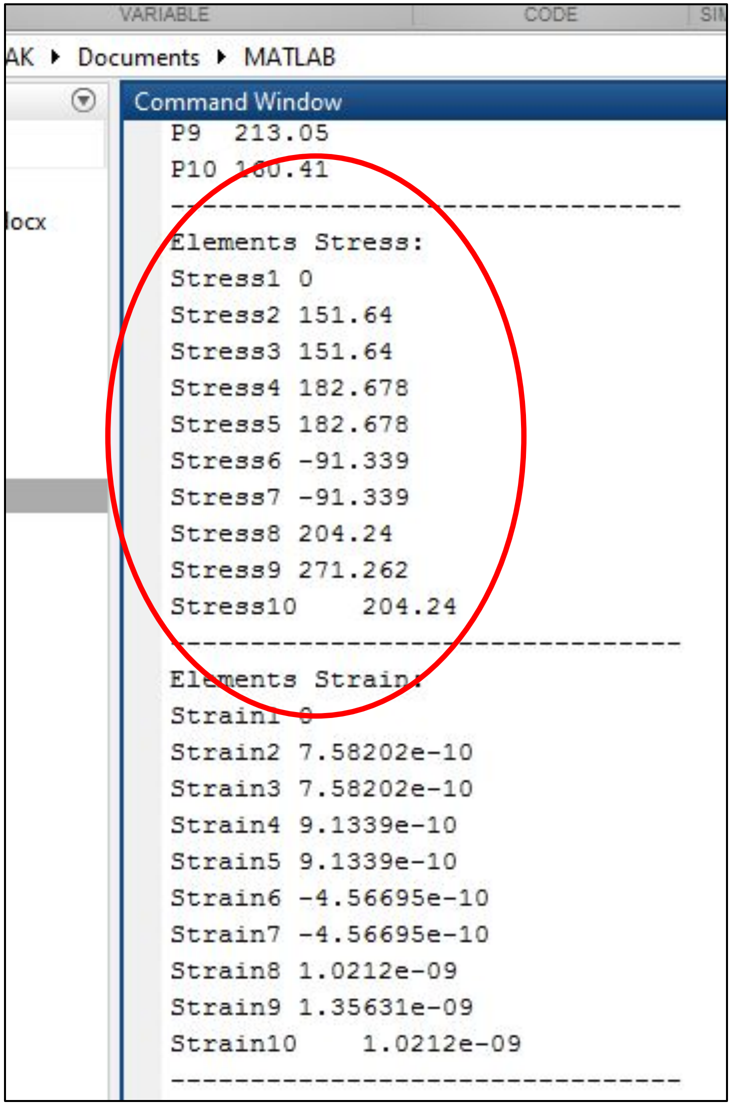


Fig 11: Deformation in Truss Structure

Verification with ANSYS® Results



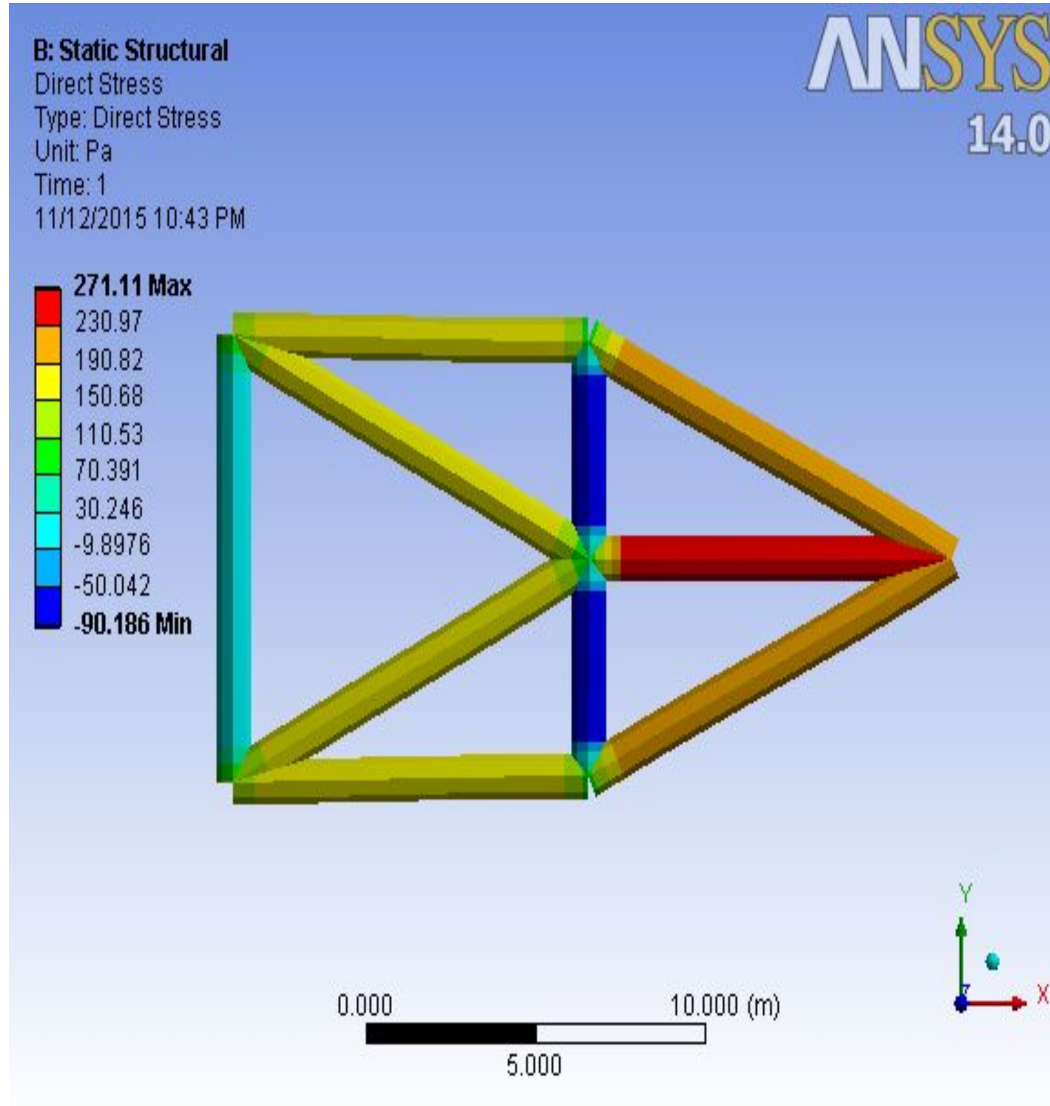
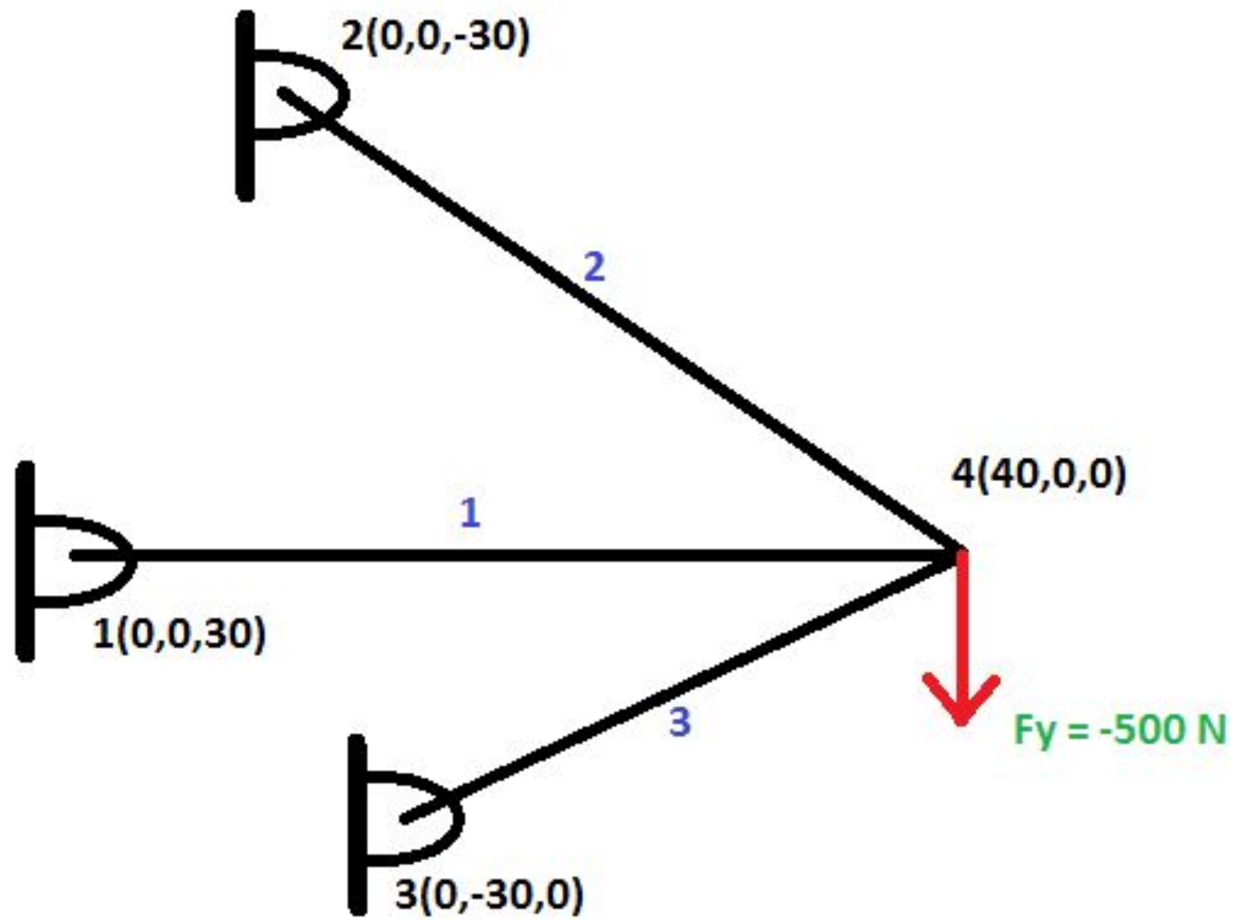
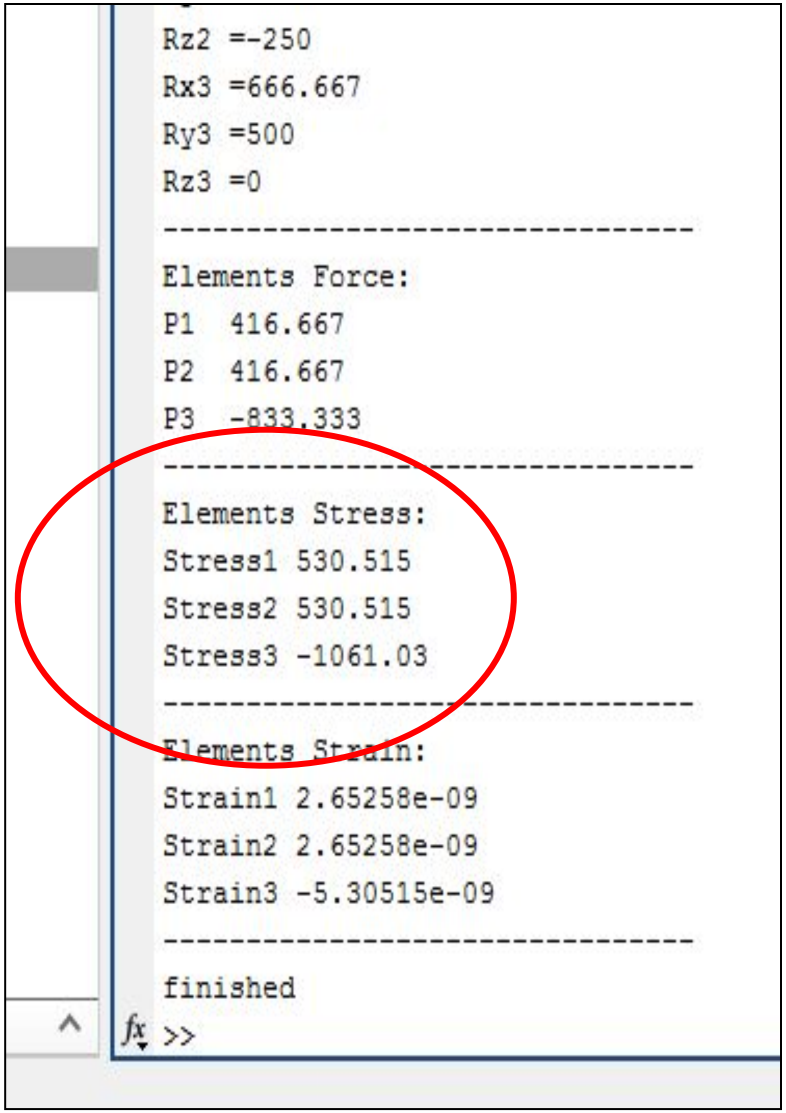


Fig 12: Stress values in Truss Structure

Example 3: 3D Truss Structure



Verification with ANSYS® Results



```
Rz2 =-250
Rx3 =666.667
Ry3 =500
Rz3 =0
--------------------------------
Elements Force:
P1  416.667
P2  416.667
P3  -833.333
--------------------------------
Elements Stress:
Stress1 530.515
Stress2 530.515
Stress3 -1061.03
--------------------------------
Elements Strain:
Strain1 2.65258e-09
Strain2 2.65258e-09
Strain3 -5.30515e-09
--------------------------------
finished
>>
```

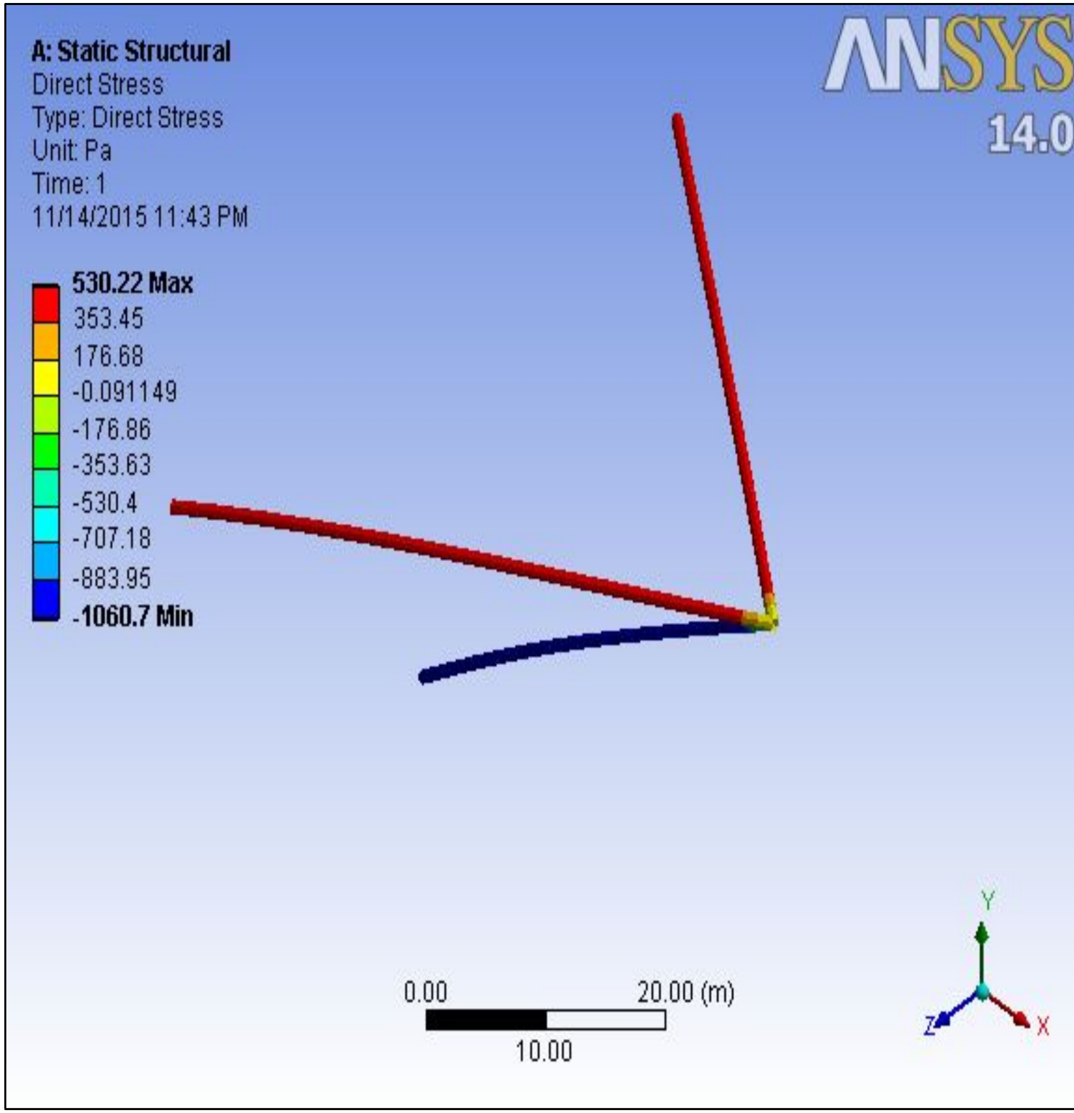
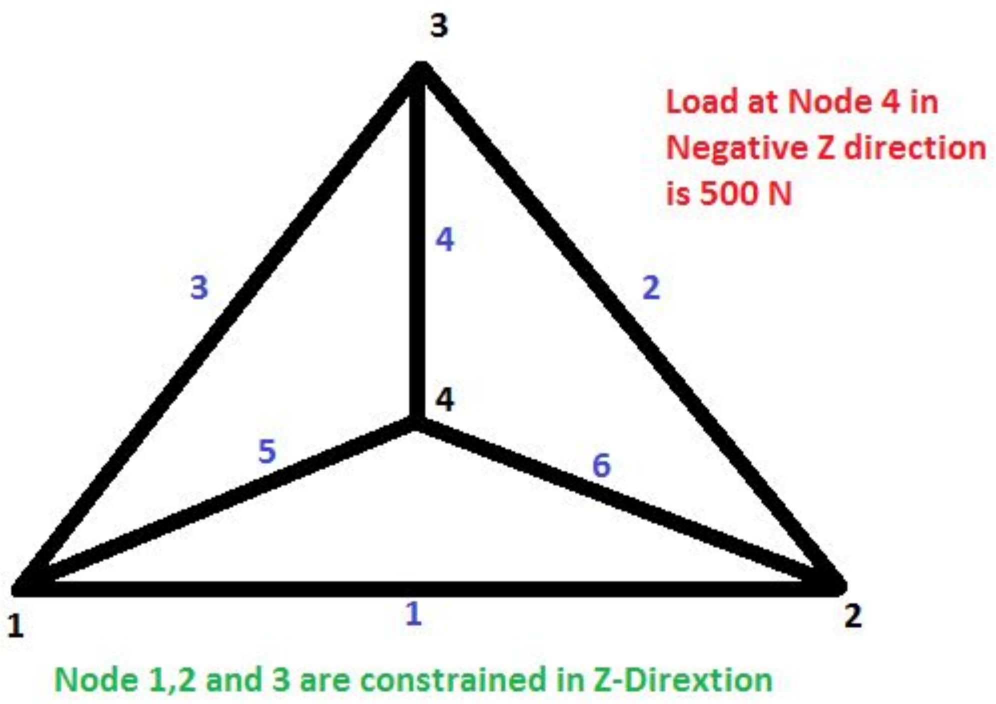
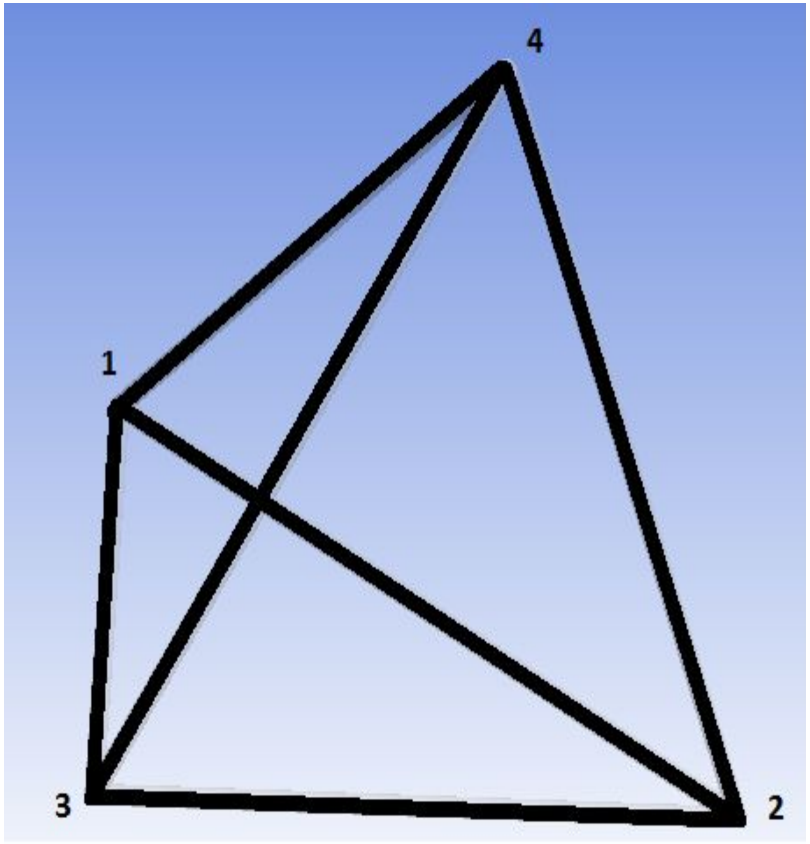


Fig 13: Stress values in Truss Structure

Example 3: Regular Tetrahedron





Verification with ANSYS® Results

DEEPAK ▸ Documents ▸ MATLAB

eration.docx

Command Window

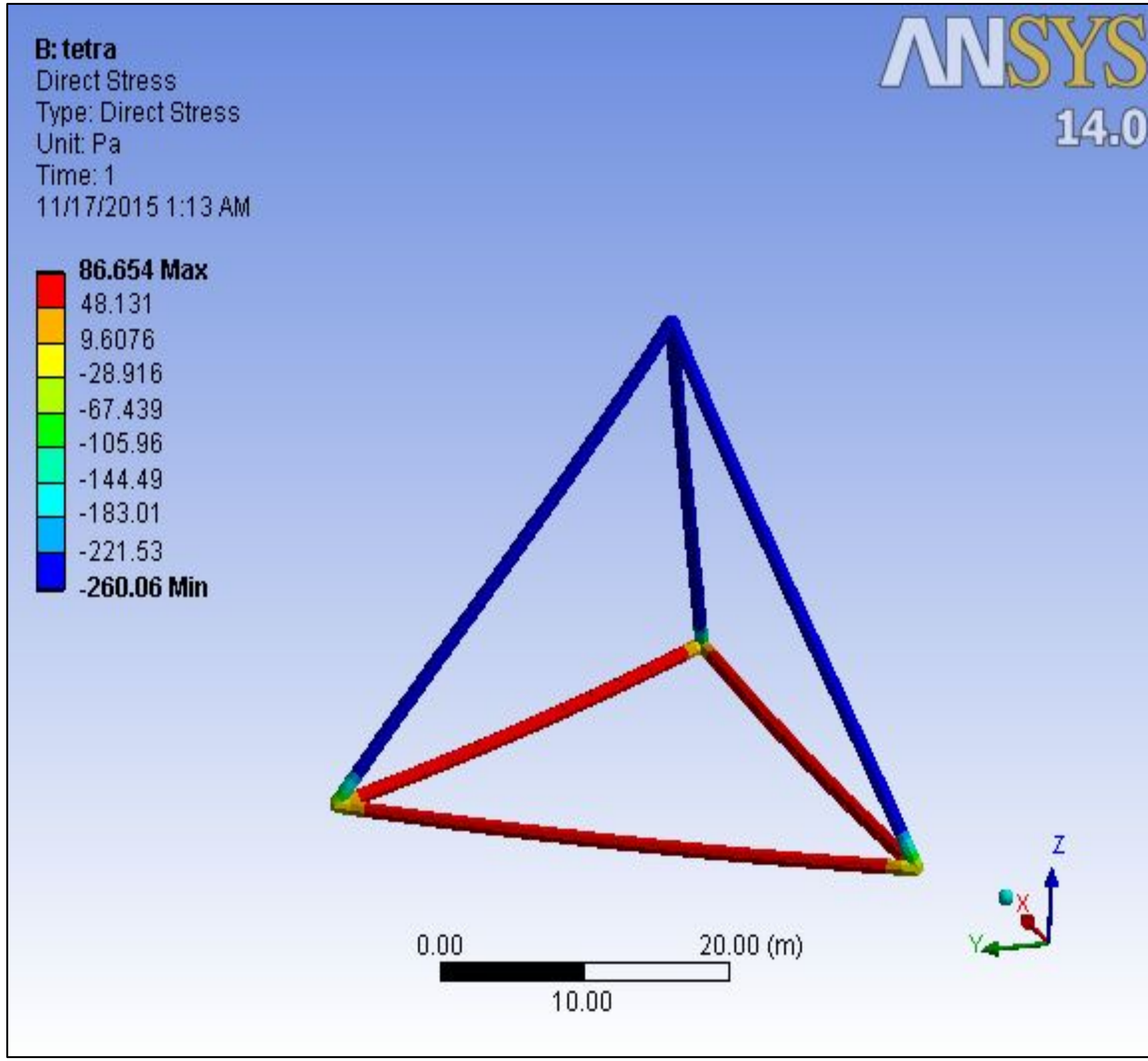
```
>> truss3
3D-Truss Anlysis
Node Displacements

dx1=-8.66328e-09
dy1=-5.00175e-09
dz1=0
dx2=8.66328e-09
dy2=-5.00175e-09
dz2=0
dx3=-7.76249e-22
dy3=1.00035e-08
dz3=0
dx4=-8.70211e-22
dy4=-3.53465e-21
dz4=-7.07354e-08
---------------------------------
```

Model (B4) > Static Structural (B5) > Solution (B6) > Probes

Object Name	*Deformation 1x*	*Deformation 1y*	*Deformation 2x*	*Deformation 2y*
State	Solved			
Definition				
Type	Deformation			
Location Method	Geometry Selection			
Geometry	1 Vertex			
Orientation	Global Coordinate System			
Suppressed	No			
Options				
Result Selection	X Axis	Y Axis	X Axis	Y Axis
Display Time	End Time			
Spatial Resolution	Use Maximum			
Results				
X Axis	-8.6654e-009 m		8.6654e-009 m	
Y Axis		-5.0824e-009 m		-5.0824e-009 m

Verification with ANSYS® Results

```
-----------------------------------
Elements Force:
P1  68.0414
P2  68.0414
P3  68.0414
P4  -204.124
P5  -204.124
P6  -204.124
-----------------------------------
Elements Stress:
Stress1 86.6328
Stress2 86.6328
Stress3 86.6328
Stress4 -259.898
Stress5 -259.898
Stress6 -259.898
-----------------------------------
Elements Strain:
Strain1 4.33164e-10
Strain2 4.33164e-10
Strain3 4.33164e-10
Strain4 -1.29949e-09
Strain5 -1.29949e-09
Strain6 -1.29949e-09
-----------------------------------
```



Real-Life Application

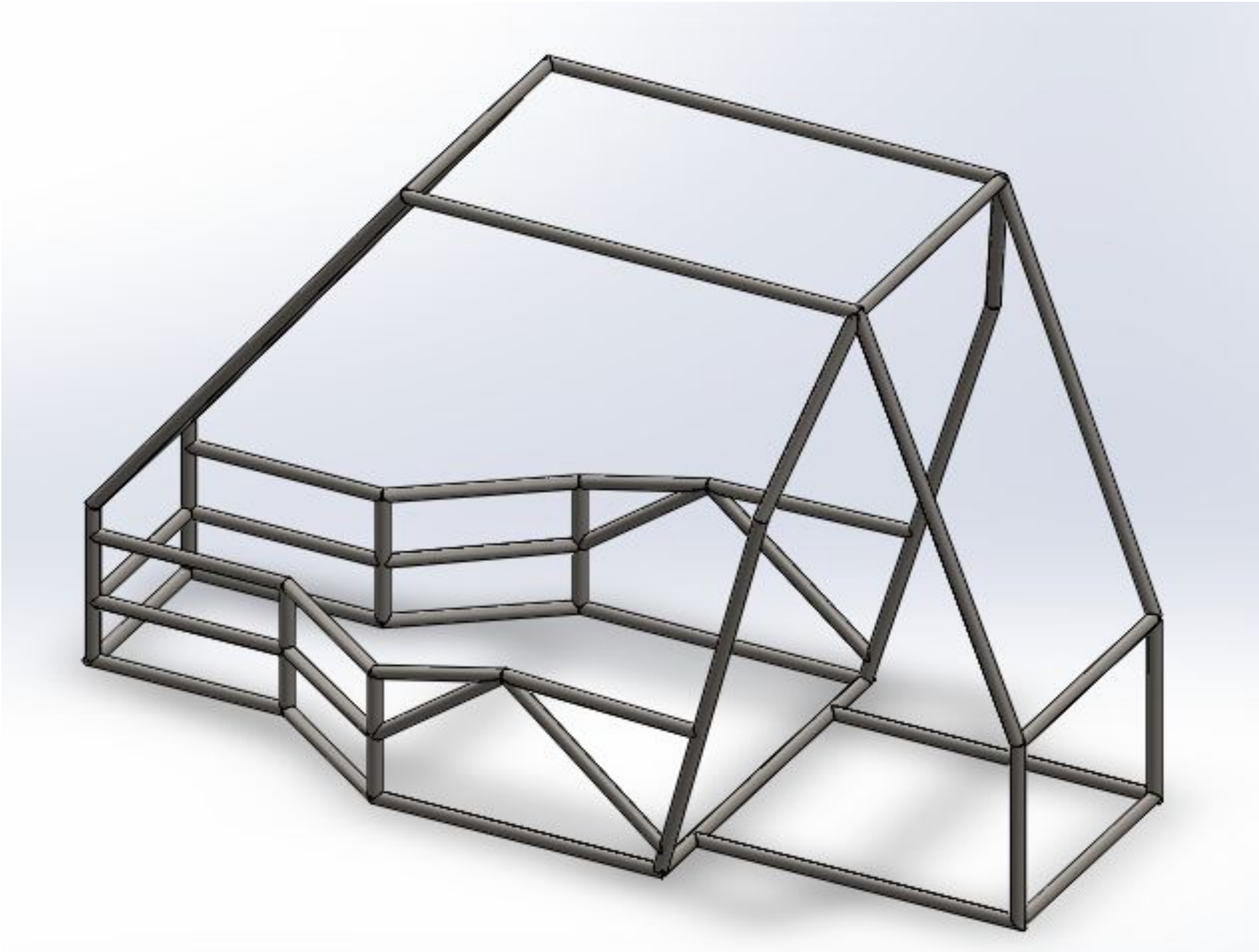


Fig: All-Terrain Vehicle Roll Cage

Further Improvements

1. Use this program to analyze Frame (Roll Cage) of All-Terrain Vehicle.

2. Develop a program to analyze the solid structures

3. The additional feature of this program would be automatic detection of geometry of truss structure.

4. **Commercialization of Product…!!!**