DIGITAL SIMULATION LAB - II

(R22D7684)

LAB MANUAL

I M. Tech II Semester (2023-2024)

Department of Aeronautical Engineering

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(Autonomous Institution – UGC, Govt. of India)

Affiliated to JNTU, Hyderabad, Approved by AICTE - Accredited by NBA & NAAC – 'A' Grade - ISO 9001:2015 Certified)

Maisammaguda, Dhulapally (Post Via. Kompally), Secunderabad – 500100, Telangana State, India

MRCET VISION

To become a model institution in the fields of Engineering, Technology and Management. To have a perfect synchronization of the ideologies of MRCET with challenging demands of International Pioneering Organizations.

MRCET MISSION

To establish a pedestal for the integral innovation, team spirit, originality and competence in the students, expose them to face the global challenges and become pioneers of Indian vision of modern society.

MRCET QUALITY POLICY.

To pursue continual improvement of teaching learning process of Undergraduate and Post Graduate programs in Engineering & Management vigorously. To provide state of art infrastructure and expertise to impart the quality education.

PROGRAM OUTCOMES
(PO's)

Engineering Graduates will be able to:

1. **Engineering knowledge**: Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
2. **Problem analysis**: Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
3. **Design / development of solutions**: Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
4. **Conduct investigations of complex problems**: Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
5. **Modern tool usage**: Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
6. **The engineer and society**: Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
7. **Environment and sustainability**: Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
8. **Ethics**: Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
9. **Individual and team work**: Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
10. **Communication**: Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
11. **Project management and finance**: Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multi disciplinary environments.
12. **Life- long learning**: Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

DEPARTMENT OF AERONAUTICAL ENGINEERING

VISION

Department of Aeronautical Engineering aims to be indispensable source in Aeronautical Engineering which has a zeal to provide the value driven platform for the students to acquire knowledge and empower themselves to shoulder higher responsibility in building a strong nation.

MISSION

The primary mission of the department is to promote engineering education and research. To strive consistently to provide quality education, keeping in pace with time and technology. Department passions to integrate the intellectual, spiritual, ethical and social development of the students for shaping them into dynamic engineers

QUALITY POLICY STATEMENT

Impart up-to-date knowledge to the students in Aeronautical area to make them quality engineers. Make the students experience the applications on quality equipment and tools. Provide systems, resources and training opportunities to achieve continuous improvement. Maintain global standards in education, training and services.

PROGRAM EDUCATIONAL OBJECTIVES – Aeronautical Engineering

1. **PEO1 (PROFESSIONALISM & CITIZENSHIP):** To create and sustain a community of learning in which students acquire knowledge and learn to apply it professionally with due consideration for ethical, ecological and economic issues.
2. **PEO2 (TECHNICAL ACCOMPLISHMENTS):** To provide knowledge based services to satisfy the needs of society and the industry by providing hands on experience in various technologies in core field.
3. **PEO3 (INVENTION, INNOVATION AND CREATIVITY):** To make the students to design, experiment, analyze, and interpret in the core field with the help of other multi disciplinary concepts wherever applicable.
4. **PEO4 (PROFESSIONAL DEVELOPMENT):** To educate the students to disseminate research findings with good soft skills and become a successful entrepreneur.
5. **PEO5 (HUMAN RESOURCE DEVELOPMENT):** To graduate the students in building national capabilities in technology, education and research

PROGRAM SPECIFIC OUTCOMES – Aeronautical Engineering

1. To mould students to become a professional with all necessary skills, personality and sound knowledge in basic and advance technological areas.
2. To promote understanding of concepts and develop ability in design manufacture and maintenance of aircraft, aerospace vehicles and associated equipment and develop application capability of the concepts sciences to engineering design and processes.
3. Understanding the current scenario in the field of aeronautics and acquire ability to apply knowledge of engineering, science and mathematics to design and conduct experiments in the field of Aeronautical Engineering.
4. To develop leadership skills in our students necessary to shape the social, intellectual, business and technical worlds.

COURSE OUTCOMES

1.Basic knowledge on mathematical programming language.

2.Develop skills in programming language.

3.Ability to model aerospace problems through mathematical models.

4. Revise computational strategies for developing applications.

5.Ability to develop Simple to Complex applications using programming language.

Exp 1: STATIC ANALYSIS OF A COLUMN

Aim: To do static analysis on a given column

Procedure:

Preferences > structural

Pre-processor > element type > Add/Edit/Delete>Add>

Select **beam** *then* **2D Elastic 3,** Ok

Real constants > Add/Edit/Delete > Add >

Select BEAM 3, Ok

Give set No as **1.**

I =**83.33,** A=**100**, h=1**0** Ok

Material props > Material Models > Structural > Linear > Elastic > Isotropic >

EXY = **2e5,**

PRXY = **0.3**, Ok *then* close

Modelling > create > Keypoints > In Active **CS** >

0, 0, 0, Apply

0, 100, 0, ok

Modelling > Create > Lines > Lines >Straight Lines >

Join all the keypoints *in order*

Meshing > Mesh Attributes > Picked Lines >

Pick *the all line* Ok

Meshing > Size Cntrls > Manual Size > Lines >

Leave the No of Divisions *column blank (not zero or anything else)*

Give the Element Edge Length as **5**

Meshing > Mesh > Lines >

Pick *all the lines*, Ok

Main menu > Plot Ctrls > Style > Size and Shape >

Tick *in the box against* Display of Element (on)

Main menu > Plots > Multi Plots

Solution > Define Loads > Apply > Structural > Displacement > Keypoints >

Select *the first point*, Ok

Select **all degrees of freedom,** Ok

Solution > Define Loads > Apply > Structural > Force/Moment > on key points

Select the **2**nd key point, Ok

Select **FY** and give the value as **-1000**, **Ok**

Solution > solve > Current LS > Ok

General Postproc > Plot results > Deformed shape > Def + Undef edge

Click Ok

General Postproc > Element Table > define Elem Table > Add >

From the first list in the window select By Sequence No,

From the second list select SMISC 2, Apply

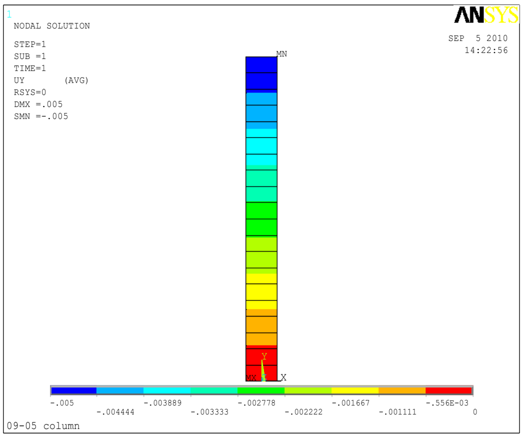
Similarly select SMISC 6, SMISC 8, SMISC 12, Ok, Close

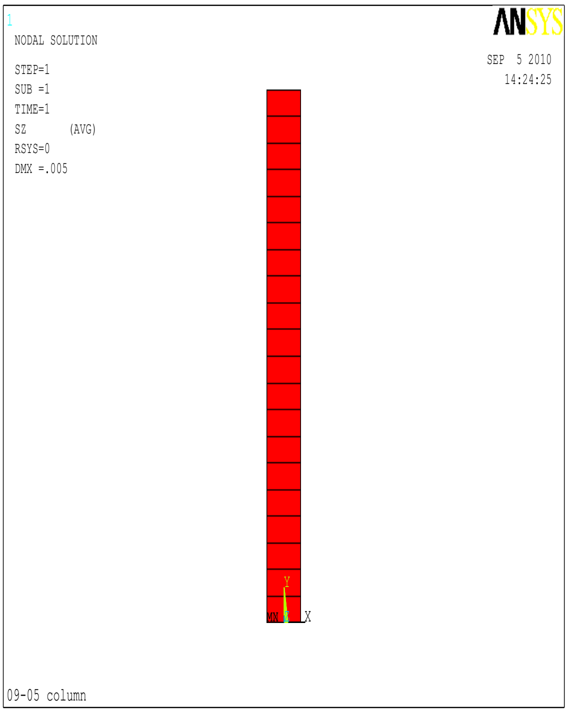
Plot results > Contour Plot > Line Elem Res >

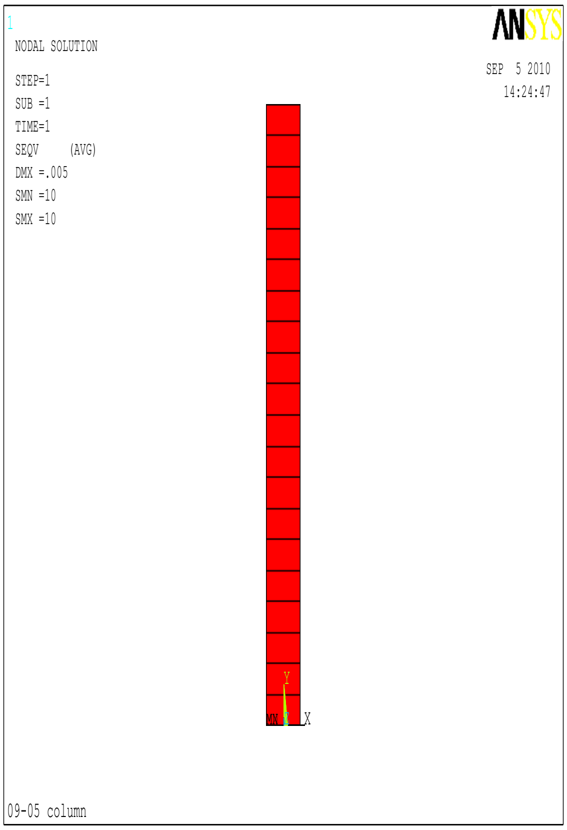
Select SMICS 2 & SMISC 8 *for* SFD, Ok

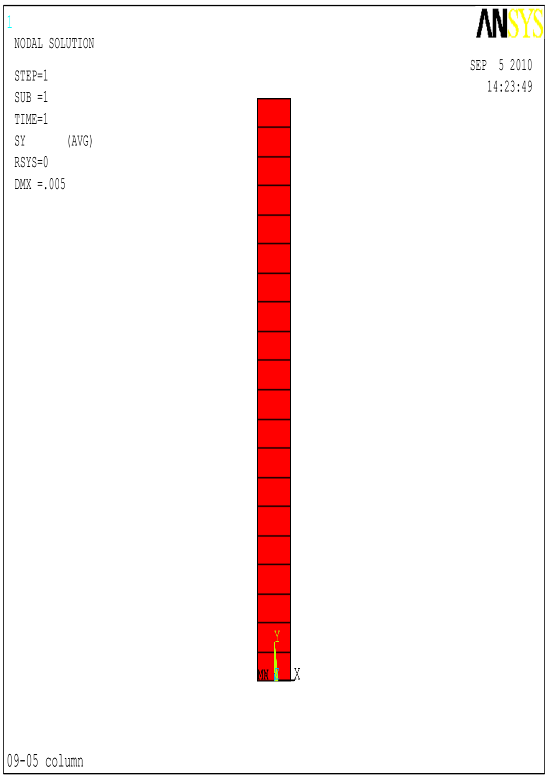
Select SMISC 6 & SMISC 12 *for BMD*, Ok

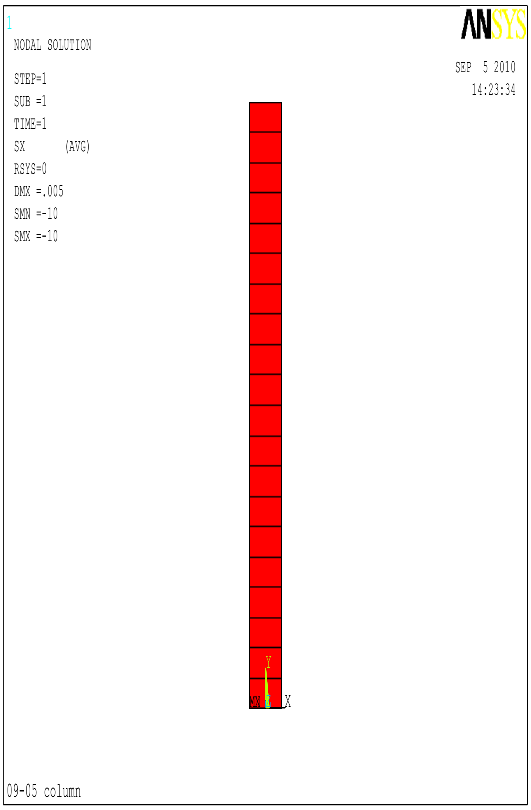
Result:-


NODAL SOLUTION
STEP=1
SUB =1
TIME=1
UY (AVG)
RSYS=0
DMX =.005
SMN =-.005
SEP 5 2010
14:22:56
MN
-.005
-.004444
-.003889
-.003333
-.002778
-.002222
-.001667
-.001111
-.556E-03
0
09-05 column


NODAL SOLUTION
STEP=1
SUB =1
TIME=1
SZ (AVG)
RSYS=0
DMX =.005
SEP 5 2010
14:24:25
09-05 column


1
NODAL SOLUTION
STEP=1
SUB =1
TIME=1
SEQV (AVG)
DMX =.005
SMN =10
SMX =10
ANSYS
SEP 5 2010
14:24:47
Y
MX
X
09-05 column


1
NODAL SOLUTION
STEP=1
SUB =1
TIME=1
SY (AVG)
RSYS=0
DMX =.005
SEP 5 2010
14:23:49
MX
Y
X
09-05 column


1
NODAL SOLUTION
STEP=1
SUB =1
TIME=1
SX (AVG)
RSYS=0
DMX =.005
SMN =-10
SMX =-10
SEP 5 2010
14:23:34
MN
Y
X
09-05 column

EXP 2: TWO DIMENSIONAL STATIC LINEAR ANALYSIS OF A CANTILEVER BEAM

AIM:- To Determine the stresses acting on a cantilever beam with a point load of -10000 N acting at one of its ends and perpendicular to the axis of the beam.

Length of the beam = 2m = 2000mm
Depth of the beam = 10 cm = 100mm
Width of the beam = 50mm
Cross sectional area = width * depth = 5000 mm^2
Moment of Inertia about Z axis = I_{zz} = (depth*width*3)/12

PROCEDURE

PRE PROCESSING

STEP 1: From the Main menu select preferences

Select structural and press OK

STEP 2: From the main menu select **Pre-processor**

Element type → Add / edit/Delete → Add → BEAM – 2D Elastic 3
Real constants → Add → Geometric Properties → Area = 5000, I_{zz} = 1250, Height = 50
Material properties → material models → Structural → Linear → Elastic → Isotropic
EX = 2e5; PRXY = 0.3

STEP 3: From the main menu select Pre-processor → **Modelling**

- Create the key points in the Workspace

Create → Key points → In active CS

X	0	2000
Y	0	0

Click APPLY to all the points and for the last point click OK

- Create LINES using the Key points

Create → Lines → Straight Line → Click on Key points to generate lines

STEP 4: Meshing the Geometry

From the main menu select **Meshing**
Meshing → Size controls → Manual size → Lines → All lines – Number of element divisions = **20** → Click OK

Meshing → Mesh → Lines – pick all

SOLUTION PHASE: ASSIGNING LOADS AND SOLVING

STEP 5: From the ANSYS main menu open **Solution**

Solution → Analysis type → New analysis – Static

STEP 6: Defining loads at the Key points

Solution → Define Loads → Apply → Structural → Displacement → On key points
Left end – ALL DOF arrested

Solution → Define loads → Apply → Structural → Force/moment → On key Points

Right end – Apply a load of Fy = -1000N

STEP 7: Solving the system

Solution →Solve → Current LS

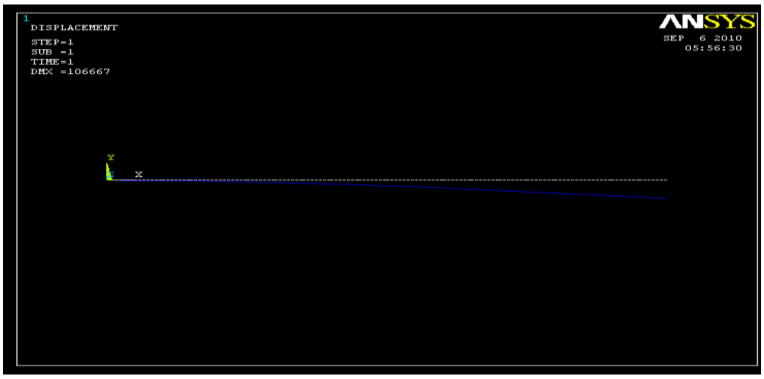
POSTPROCESSING: VIEWING THE RESULTS

1. **Deformation**

   From the main menu select **General post processing**

   General post processing → Plot Results → Deformed Shape

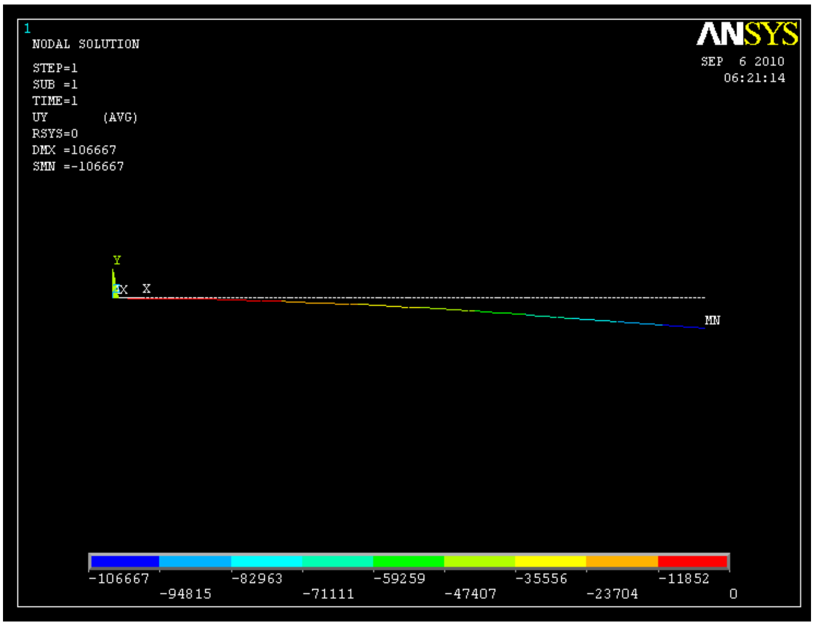
   Select 'Def + undef edge' and click 'OK' to view both the deformed and the undeformed object



1. **Nodal solution**

From the Utility menu select PLOT

PLOT →Results →Contour plot → Nodal solution – DOF solution – Y component of displacement – OK



<u>EXP 3: BUCKLING OF A PLATE</u>

AIM:- To Determine the stresses acting on a plate which has a thickness of 10 mm and a width of 100 mm

PREPROCESSING

STEP 1: From the Main menu select preferences

Select structural and press OK

STEP 2: From the main menu select **Pre-processor**

Element type → Add / edit/Delete → Add → Link – 2D spar 1
Real constants → Add → Geometric Properties → Area = 3250
Material properties → material models → Structural → Linear → Elastic → Isotropic
EX = 2e5; PRXY = 0.3; Density = 2700

STEP 3: : Modelling - From the main menu select **Pre-processor**

Modelling → Create Areas → Rectangle by center and corner → width = 100 and Height = 10

Modelling → operate→ extrude→ area→ By XYZ offset

Pick all, **x=0, y=0, z=100,** Ok

STEP 4: Meshing the geometry - From the main menu select **Preprocessor**

Meshing > Size Controls > smart size > Basics >

Change the size level to **10(coarse), OK**

Meshing > Mesh > Volume > free

Pick all

STEP 5: From the ANSYS main menu open **Solution**

Solution → Analysis type → New analysis – Modal

Solution → Analysis type → Analysis option → No of nodes to extract =5 **,** Ok

Solution → Define Loads → Apply →displacement →on line →
Select the **front and last line**, Ok

Select **UY**, **Ok**

STEP 6: Solving the system

Solution →Solve → Current LS

POSTPROCESSING: VIEWING THE RESULTS

1. Deformation

General Postproc > Read results > first set

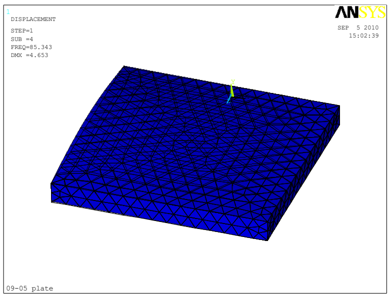
General Postproc > Plot results > Deformed shape > Def + Undef edge

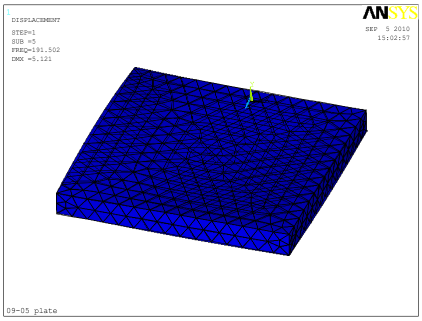
Click Ok

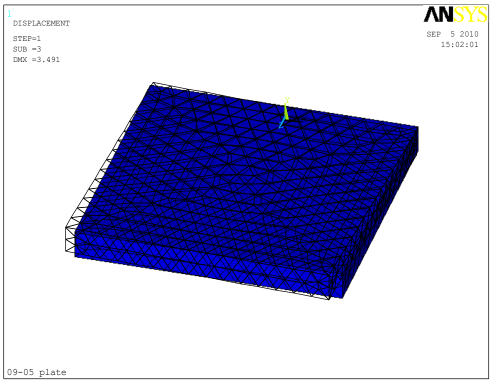
Repeat it 4 more times for all nodes.

Results: The natural frequency at nodes

- Node1: 166.82
- Node2: 173.031
- Node3: 235.562
- Node4: 85.343
- Node5: 191.502







EXP 4: 2-D STATIC LINEAR ANALYSIS OF A TRUSS STRUCTURE

AIM:- To Determine the nodal deflections, reaction forces, and stress for the truss system shown below (E = 200GPa, A = 3250mm^2).

PREPROCESSING

STEP 1: From the Main menu select preferences

Select structural and press OK

STEP 2: From the main menu select **Preprocessor**

Element type → Add / edit/Delete → Add → Link – 2D spar 1
Real constants → Add → Geometric Properties → Area = 3250
Material properties → material models → Structural → Linear → Elastic → Isotropic
EX = 2e5; PRXY = 0.3

STEP 3: From the main menu select Pre-processor → **Modelling**

- Create the key points in the Workspace

Pre-processor → Modelling → Create → Key points → In active CS

X	0	1800	3600	5400	7200	9000	10800	3600	7200
y	0	3118	0	3118	0	0	3118	6236	6236

Click APPLY to all the points and for the last point click OK

- Create LINES using the Key points

Pre-processor → Modelling → Create → Lines → Straight Line → Click on Key points to generate lines

STEP 4: Meshing the Geometry

From the main menu select **Meshing**
Meshing → Size controls → Manual size → Lines → All lines – Number of element divisions = 1 → Click OK
Meshing → Mesh → Lines – pick all

STEP 5: From the ANSYS main menu open **Solution**

Solution → Analysis type → New analysis – Static

STEP 6: Defining loads at the Key points

Solution → Define Loads → Apply → Structural → Displacement → On key points
Left end – ALL DOF arrested
Right end – deselect ALL DOF and select UY

Solution → Define loads → Apply → Structural → Force/moment → On key Points

Key point 1 – Fy = -28000
Key point 2 – Fy = -21000
Key point 3 – Fy = -28000
Key point 4 – Fy = -36000

STEP 6: Solving the system

Solution →Solve → Current LS

POSTPROCESSING

1. Reaction forces

From the main menu select **General post processing**

General post processing → List results → Reaction Solution

Select ‘ALL STRUC FORCE F’ and click OK

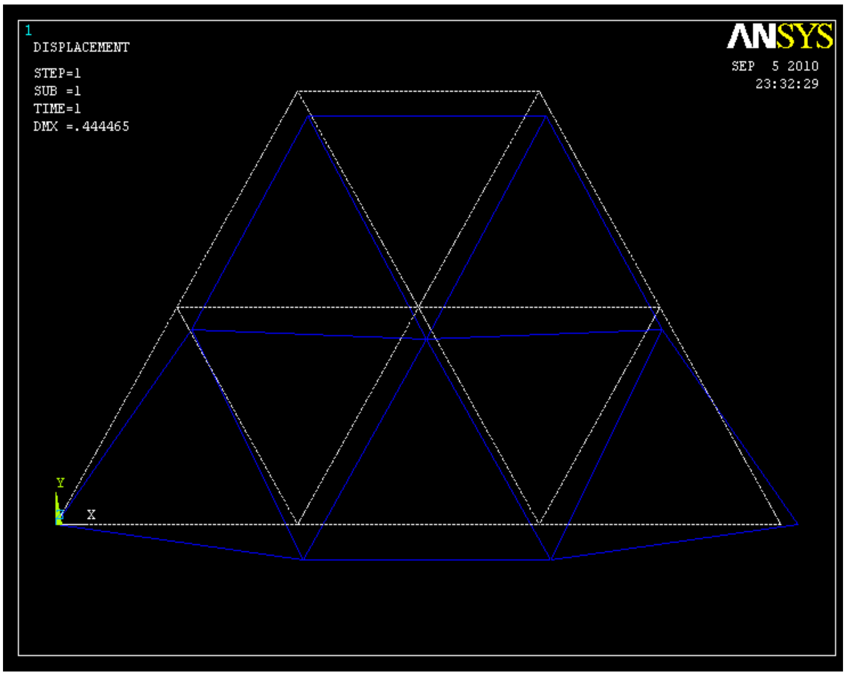
```
PRINT F    REACTION SOLUTIONS PER NODE

 ***** POST1 TOTAL REACTION SOLUTION LISTING *****

 LOAD STEP=     1  SUBSTEP=     1
  TIME=    1.0000      LOAD CASE=   0

 THE FOLLOWING X,Y,Z SOLUTIONS ARE IN THE GLOBAL COORDINATE SYSTEM

   NODE       FX          FY
      1  0.27285E-10  49000.
      8               57000.

TOTAL VALUES
VALUE   0.27285E-10 0.10600E+06
```

2. Deformation

From the main menu select **General post processing**

General post processing → Plot Results → Deformed Shape

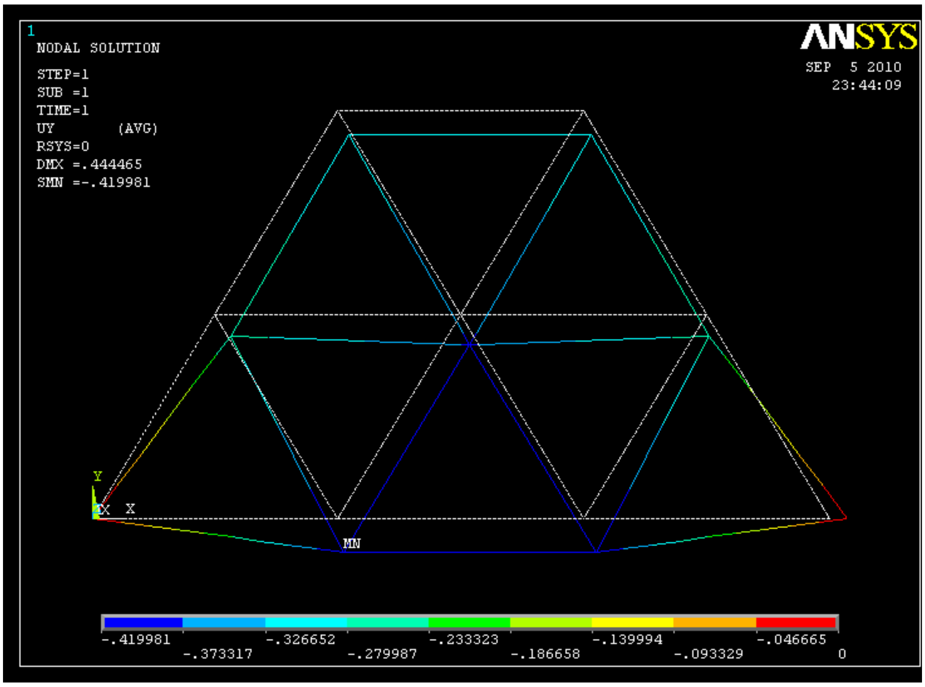
Select 'Def + undef edge' and click 'OK' to view both the deformed and the undeformed object.



3. DEFLECTION

From the 'General Postproc' menu select **Plot results →Contour Plot → Nodal Solution**.

- Select 'DOF solution' and 'USUM' in the window. Leave the other selections as the default values. Click 'OK'.
- From the **Utility Menu** select **Plot Controls → Style → Contours → Uniform Contours...**



EXP 5: ANALYSIS OF A THIN WALLED BEAM

AIM:- To Calculate the reaction frequencies in the thin walled beam as shown in the figure.

PREPROCESSING

STEP 1: From the Main menu select preferences

Select structural and press OK

STEP 2: From the main menu select **Pre-processor**

Element type → Add / edit/Delete → Add → Solid – 10 node 92
Material properties → material models → Structural → Linear → Elastic → Isotropic
EX = 2e5; PRXY = 0.3; Density = 7650

STEP 3: From the main menu select **Pre-processor**

- Create the key points in the Workspace

Pre-processor → Modelling → Create → Key points → In active CS

X	0	10	10	1	1	10	10	0
y	0	0	1	1	9	9	10	10

Click APPLY to all the points and for the last point click OK

- Create LINES using the Key points

Pre-processor → Modelling → Create → Lines → Straight Line → Click on Key points to generate lines

- Pre-processor → Modelling → Create → Areas →Arbitrary → By lines – Pick all
- Pre-processor → Modelling →Operate → Extrude →Areas → Along Normal – Z = 50 Ok

STEP 4: From the main menu select **Pre-processor**

Pre-processor → Meshing → Size Control → Smart Size

Preprocessor → Meshing → Volume → Free mesh

STEP 5: From the ANSYS main menu open **Solution**

Solution → Analysis type → New analysis – Modal Analysis

Solution → Analysis type →Analysis Options → In the window that appeared

Number of modes to extract = 5

STEP 6: Solution →Define Loads

Apply constraints → arrest all DOF on one side

STEP 7: Solving the system

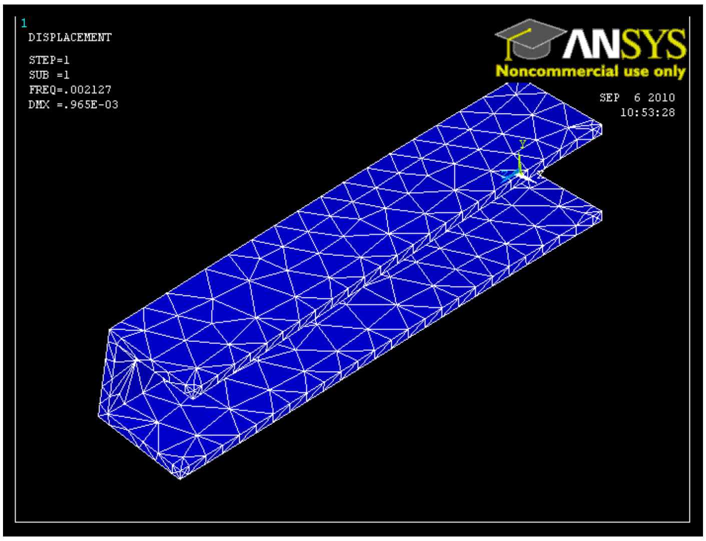
Solution →Solve → Current LS

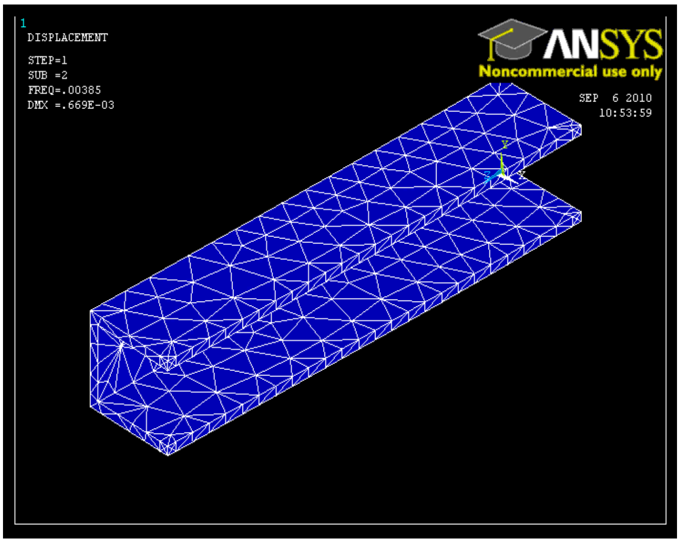
POSTPROCESSING:

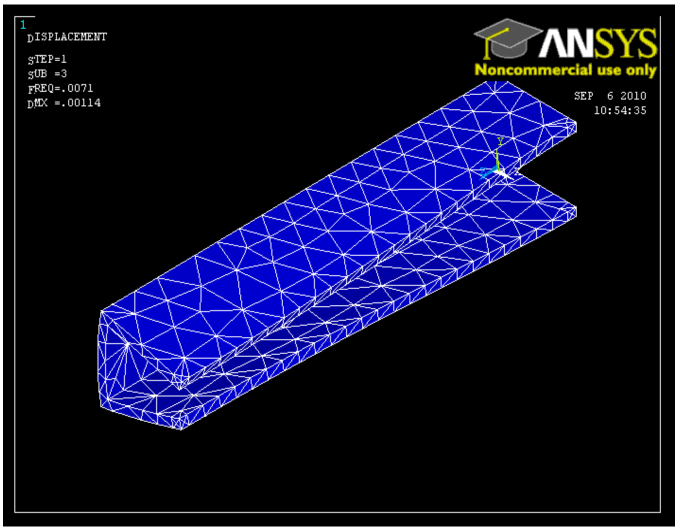
Maximum deformation and frequency obtained in five modes

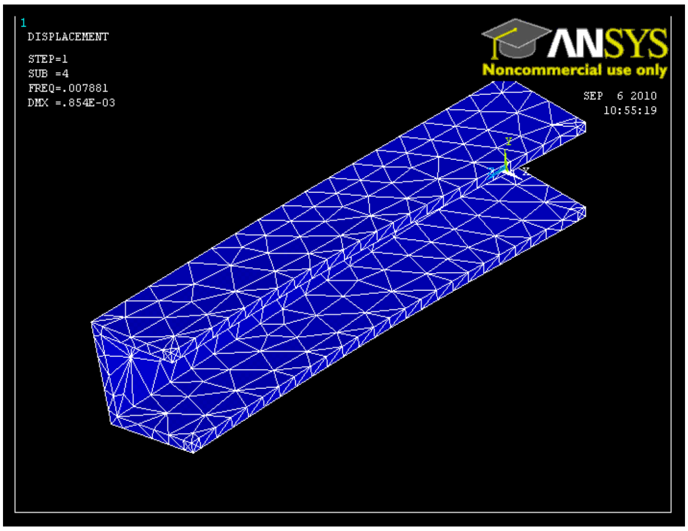
Mode 1:	**Frequency obtained :**	**0.002127**	
	Deflection Maximum	**:**	**0.9565e-03**
Mode 2:	**Frequency obtained :**	**0.002127**	
	Deflection Maximum	**:**	**0.9565e-03**
Mode 3:	**Frequency obtained :**	**0.0071**	
	Deflection Maximum	**:**	**0.00114**
Mode 4:	**Frequency obtained :**	**0.007881**	
	Deflection Maximum	**:**	**0.0854e-03**
Mode5:	**Frequency obtained :**	**0.008833**	
	Deflection Maximum	**:**	**0.001085**

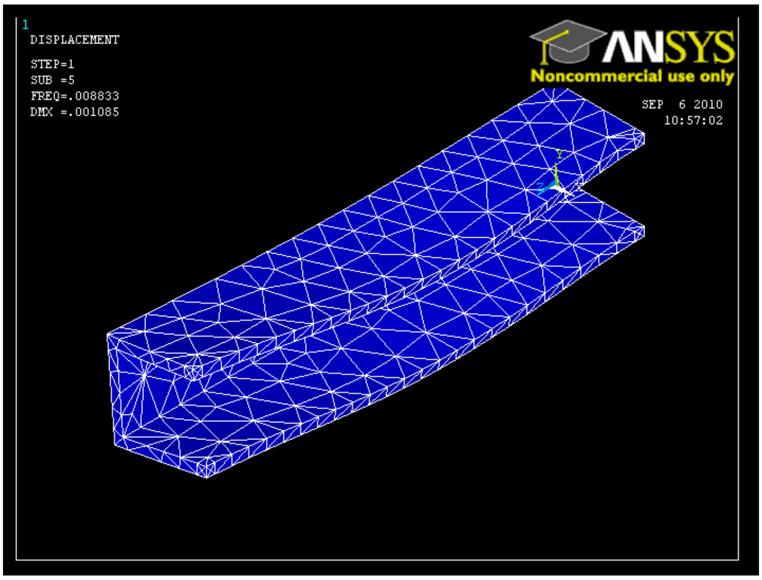
From this modal analysis we conclude that in mode 5 the frequency is high which results sinusoidal wave which indicates failure so the design change is preferred.


DISPLACEMENT
STEP=1
SUB =1
FREQ=.002127
DMX =.965E-03
ANSYS
Noncommercial use only
SEP 6 2010
10:53:28


DISPLACEMENT
STEP=1
SUB =2
FREQ=.00385
DMX =.669E-03
ANSYS
Noncommercial use only
SEP 6 2010
10:53:59


DISPLACEMENT
STEP=1
SUB =3
FREQ=.0071
DMX =.00114
ANSYS
Noncommercial use only
SEP 6 2010
10:54:35


DISPLACEMENT
STEP=1
SUB =4
FREQ=.007881
DMX =.854E-03
ANSYS
Noncommercial use only
SEP 6 2010
10:55:19


DISPLACEMENT
STEP=1
SUB =5
FREQ=.008833
DMX =.001085
ANSYS
Noncommercial use only
SEP 6 2010
10:57:02

EXP 5: ANALYSIS OF A NOSECONE

AIM:- To calculate the external loads acting on a nose cone.

PREPROCESSING -

STEP 1: From the Main menu select **Preferences**

Select structural and press OK

STEP 2: From the main menu select **Pre-processor**

Element type → Add / edit/Delete → Add → Solid – 10 node 92
Material properties → material models → Structural → Linear → Elastic → Isotropic
EX = 2e5; PRXY = 0.3; Density = 2700

STEP 3: From the main menu select **Pre-processor**

Modelling → Create Keypoints → In Active CS

Create the following key points in the work plane

X	0	0.1	0.2	0.3	0.4	0.5	0.6	0.7	0.8
y	0.1	0.11	0.14	0.19	0.26	0.35	0.46	0.59	0.74

X	0.9
y	0.91

Modelling → Create → Lines → Splines → Through KP's

Select all the key points and OK

Modelling → Copy → Lines

Select the spline and give the Y direction increment as 0.02

Modelling → Create → Lines → Straight line → Select the corners of the 2 splines and create a region bounded by four lines

Modelling → Create → Areas → Arbitrary → By lines → Select the 4 lines and Create the Area

STEP 4: Create the axis line and the volume

Modelling → Create → Keypoints →

X	0	0.9
y	0.93	0.93

Modelling → Create → Lines → Straight line →Select the above two keypoints and OK

Modelling → Operate → Extrude →Areas →Along lines → Select the area First and the keypoints of the Axis line later and OK

The Volume is created

STEP 5: Meshing the Geometry

Preprocessor →Meshing → Size controls → Basic → 10 coarse

Preprocessor → Meshing → Mesh → Areas – Free mesh – the external area

SOLUTION PHASE: ASSIGNING LOADS AND SOLVING

STEP 6: From the ANSYS main menu open **Solution**

Solution → Analysis type → New analysis – Static Analysis

STEP 7: Solution →Define Loads

Apply → Structural → Displacement → On areas → Pick areas on the end which has larger diameter and arrest all DOF

Apply → Structural → Pressure → Apply some external pressure on the Outer surface of the model == 10000

STEP 8: Solving the system

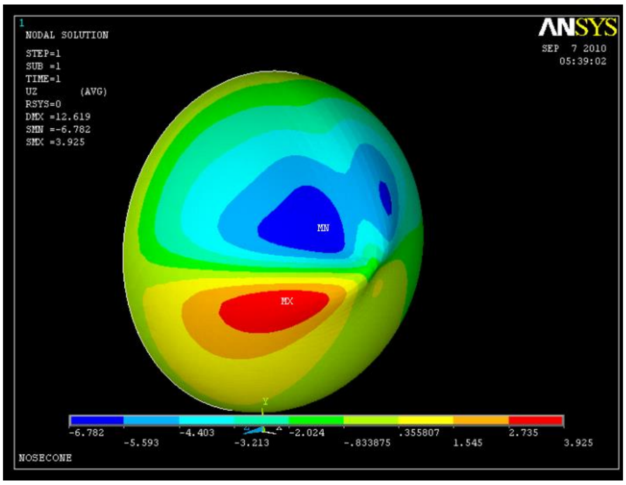
Solution →Solve → Current LS

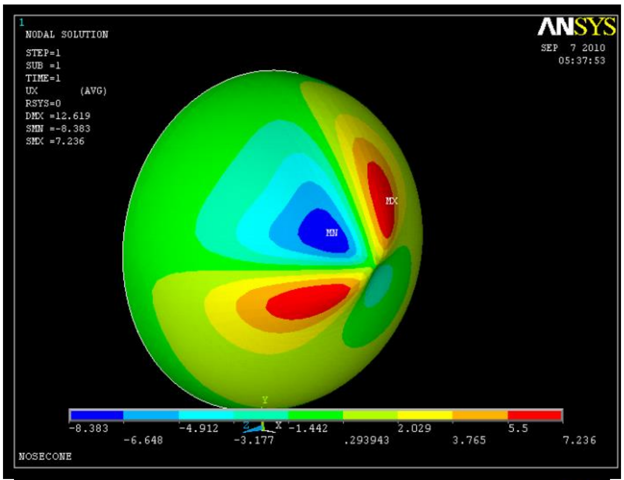
POSTPROCESSING: VIEWING THE RESULTS

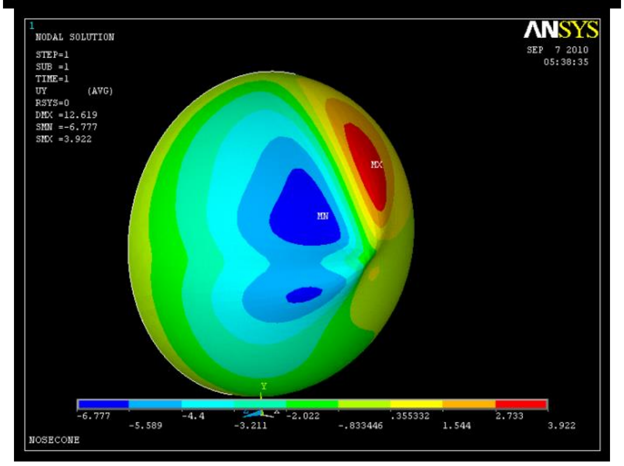
From the main menu select the general post processing

General post processing → Plot results → Deformed shape

General post processing → Contour plot →Nodal solution → DOF Solution → X displacement


NODAL SOLUTION
STEP=1
SUB =1
TIME=1
UZ (AVG)
RSYS=0
DMX =12.619
SMN =-6.782
SMX =3.925
ANSYS
SEP 7 2010
05:39:02
MN
MX
-6.782
-5.593
-4.403
-3.213
-2.024
-.833875
.355807
1.545
2.735
3.925
NOSECONE


NODAL SOLUTION
STEP=1
SUB =1
TIME=1
UX (AVG)
RSYS=0
DMX =12.619
SMN =-8.383
SMX =7.236
ANSYS
SEP 7 2010
05:37:53
MX
MN
-8.383
-6.648
-4.912
-3.177
-1.442
.293943
2.029
3.765
5.5
7.236
NOSECONE


NODAL SOLUTION
STEP=1
SUB =1
TIME=1
UY (AVG)
RSYS=0
DMX =12.619
SMN =-6.777
SMX =3.922
ANSYS
SEP 7 2010
05:38:35
MX
MN
-6.777
-5.589
-4.4
-3.211
-2.022
-.833446
.355332
1.544
2.733
3.922
NOSECONE

EXP 6: ANALYSIS OF A FUSELAGE

AIM:- To Calculate the deformation of the aluminum fuselage section under the application of internal loads.

PREPROCESSING

STEP 1: From the Main menu select preferences

Select structural and press OK

STEP 2: From the main menu select **Pre-processor**

Element type → Add / edit/Delete → Add → Solid – 10 node 92
Material properties → material models → Structural → Linear → Elastic → Isotropic
EX = 2e5; PRXY = 0.3; Density = 2700

STEP 3: From the main menu select **Pre-processor**

Pre-processor → modelling → Create → Areas → Circle → Annulus

WP x = 0 ; WP y = 0; Rad – 1 = 25; Rad -2 = 23 OK

Pre-processor → Modelling → Create → Circle → Solid –

WP x = 0; X = 22.5; Y = 0 Radius = 1.5 Apply
WP x = 0; X = -22.5; Y = 0 Radius = 1.5 Apply
WP x = 0; X =0; Y = 22.5; Radius = 1.5 Apply
WP x = 0; X = 0; Y = -22.5 Radius = 1.5 OK

Pre-processor → Modelling → Operate → Booleans → Add → Areas – Pick all OK

Pre-processor → Modelling → Operate → Extrude → Areas → By XYZ offset

X= 0; Y=0; Z = 50

STEP 4: Meshing the Geometry

Pre-processor →Meshing → Size controls → Basic → 10 coarse

Pre-processor → Meshing → Mesh → Volumes – Free mesh

SOLUTION PHASE:
STEP 5: From the ANSYS main menu open **Solution**

Solution → Analysis type → New analysis – Modal Analysis

STEP 6: Solution →Define Loads

Apply constraints → arrest all DOF on one side

STEP 7: Solving the system

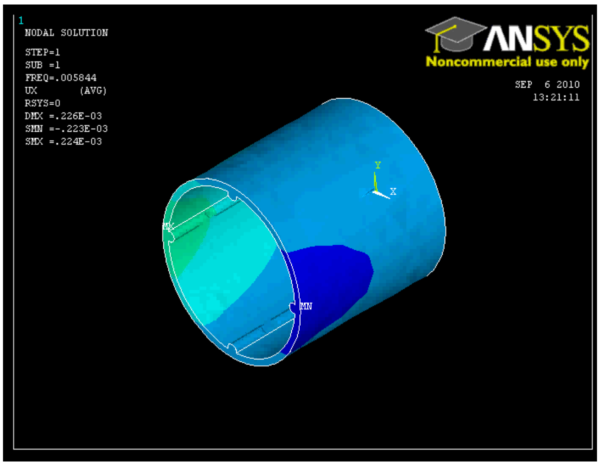
Solution →Solve → Current LS

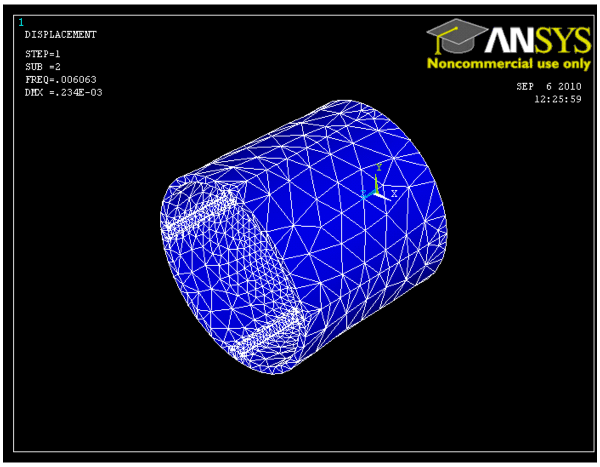
POSTPROCESSING: VIEWING THE RESULTS

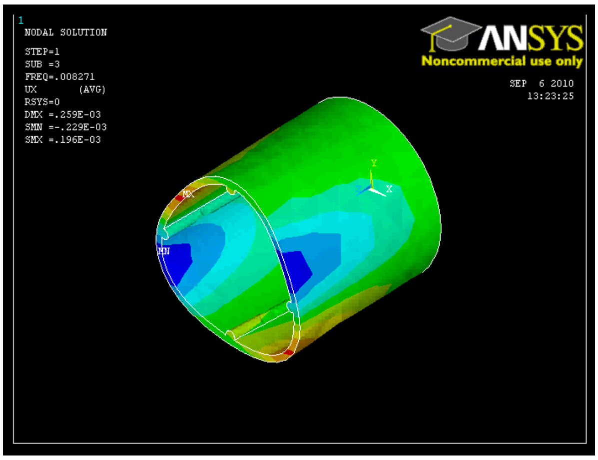
Maximum deformation and frequency obtained in five modes

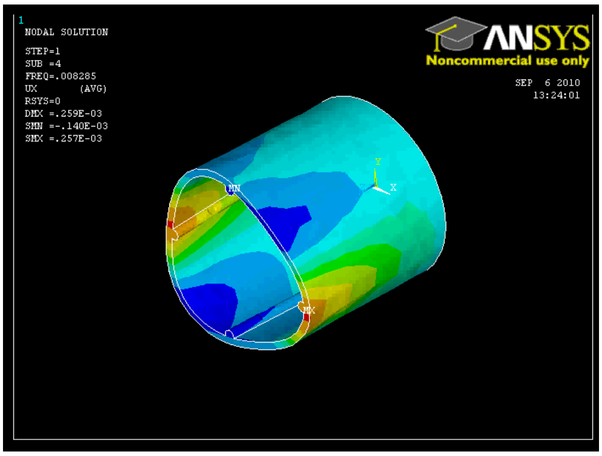
Mode 1:	**Frequency obtained :**	**0.005844**
	Deflection Maximum :	**0.226e-03**
Mode 2:	**Frequency obtained :**	**0.006063**
	Deflection Maximum :	**0.234e-03**
Mode 3:	**Frequency obtained :**	**0.008271**
	Deflection Maximum :	**0.259e-03**
Mode 4:	**Frequency obtained :**	**0.008285**
	Deflection Maximum :	**0.259e-03**
Mode5:	**Frequency obtained :**	**0.010321**
	Deflection Maximum :	**0.187e-03**

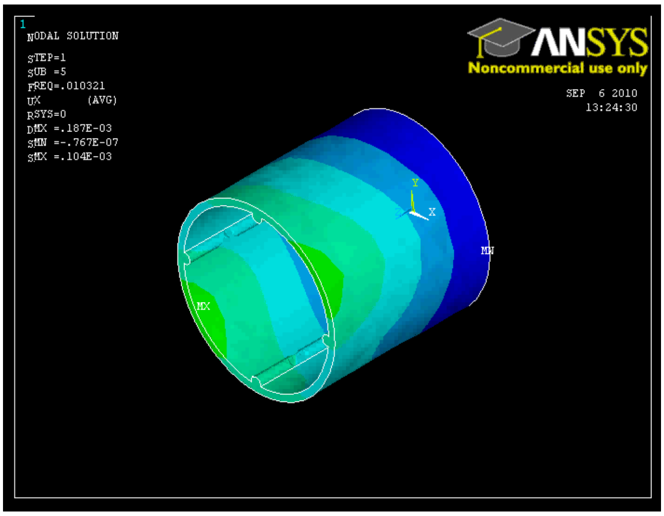
From this modal analysis we conclude that in mode 5 the frequency is high.


1
NODAL SOLUTION
STEP=1
SUB =1
FREQ=.005844
UX (AVG)
RSYS=0
DMX =.226E-03
SMN =-.223E-03
SMX =.224E-03
Noncommercial use only
SEP 6 2010
13:21:11
MX
MN
Y
X


1
DISPLACEMENT
STEP=1
SUB =2
FREQ=.006063
DMX =.234E-03
Noncommercial use only
SEP 6 2010
12:25:59


1
NODAL SOLUTION
STEP=1
SUB =3
FREQ=.008271
UX (AVG)
RSYS=0
DMX =.259E-03
SMN =-.229E-03
SMX =.196E-03
Noncommercial use only
SEP 6 2010
13:23:25
MX
MN


1
NODAL SOLUTION
STEP=1
SUB =4
FREQ=.008285
UX (AVG)
RSYS=0
DMX =.259E-03
SMN =-.140E-03
SMX =.257E-03
Noncommercial use only
SEP 6 2010
13:24:01
MN
MX


1
NODAL SOLUTION
STEP=1
SUB =5
FREQ=.010321
UX (AVG)
RSYS=0
DMX =.187E-03
SMN =-.767E-07
SMX =.104E-03
Noncommercial use only
SEP 6 2010
13:24:30
MN
MX

EXP 7: ANALYSIS OF NOZZLE

AIM:- To Calculate the deformation of the Nozzle section under the application of internal loads.

PREPROCESSING - DEFINING THE PROBLEM

STEP 1: From the Main menu select preferences

Select structural and press OK

STEP 2: From the main menu select **Pre-processor**

Element type → Add / edit/Delete → Add → Solid – 10 node 92
Material properties → material models → Structural → Linear → Elastic → Isotropic
EX = 2e5; PRXY = 0.3; Density = 7500

STEP 3: From the main menu select **Pre-processor**

Pre-processor → modelling → Create → keypoints

With formula $0.1+X^2$ we get locus points of the contour.

Pre-processor → Modelling → Create → line →spline

Select all the points in a order and click ok to get smooth contour of the nozzle.
Pre-processor → Operate → Boolean's → Copy
Select the spline and enter the 'y' distance to copy the spline.
Join two splines at the end by joining the end keypoints.
Pre-processor → Modelling → Create → areas → by lines.
Select all the lines to create the areas.
Pre-processor → modelling → Create → keypoints
Wp- 0,1 and Wp – 1,1
Pre-processor → Modelling → Create → line →St line
Select two key points.
Pre-processor → Modelling → Operate → Extrude → areas → about axis
Select the line which is created by (0,1) and (1,1) key points
Area will be revolved about 180^0.
Pre-processor → Modelling → Operate → reflect → Volumes.
Select the volume created and select the line to reflect.
Nozzle will be created as single entity.

STEP 4: Meshing the Geometry

Pre-processor →Meshing → Size controls → Basic → 10 coarse

Pre-processor → Meshing → Mesh → Volumes – Free mesh

STEP 5: From the ANSYS main menu open **Solution**

Solution → Analysis type → New analysis – Static Analysis

STEP 6: Solution →Define Loads

Apply constraints → arrest all DOF on one side of the nozzle.

STEP 7: Solution →Define Loads →Apply →Structural →Pressure

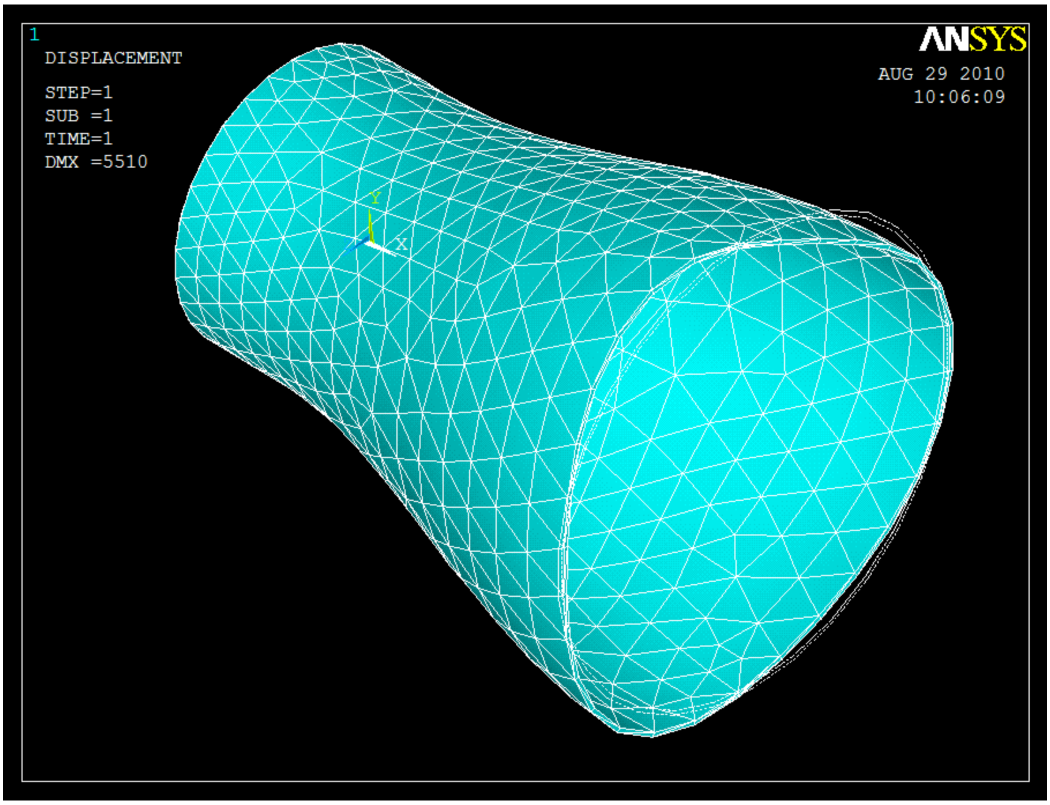
Select internal areas of nozzle.

STEP 8: Solving the system

Solution →Solve → Current LS

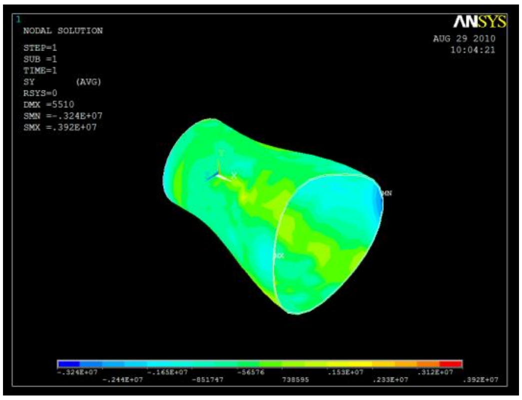
STEP 9: General post processing → Plot Results → Deformed Shape

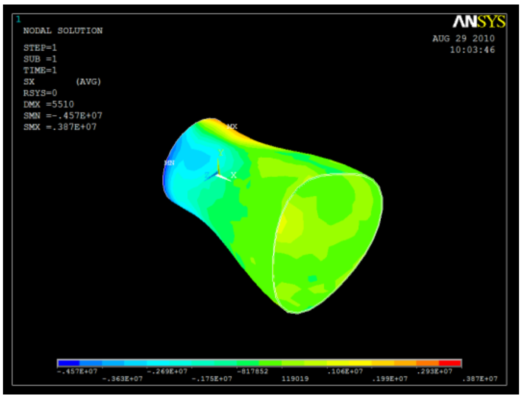
Select 'Def + undef edge' and click 'OK' to view both the deformed and the undeformed

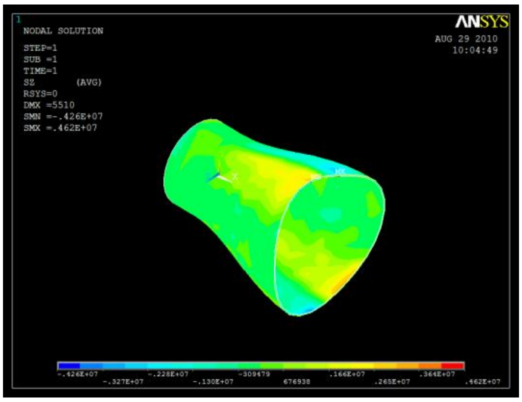


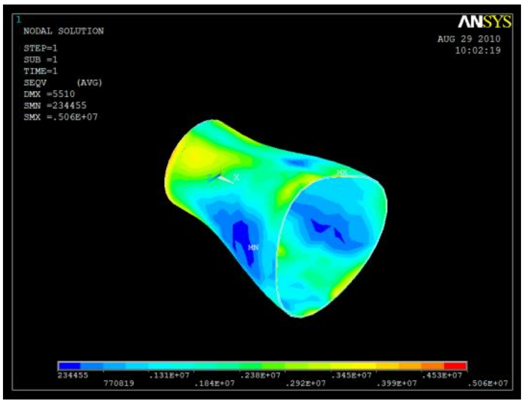
object.

Stresses in ‘X’, ‘Y’. ‘Z’ and Von misses stresses




1
NODAL SOLUTION
STEP=1
SUB =1
TIME=1
SX (AVG)
RSYS=0
DMX =5510
SMN =-.457E+07
SMX =.387E+07
AUG 29 2010
10:03:46
MN
MX
Y
X
-.457E+07
-.363E+07
-.269E+07
-.175E+07
-817852
119019
.106E+07
.199E+07
.293E+07
.387E+07


1
NODAL SOLUTION
STEP=1
SUB =1
TIME=1
SZ (AVG)
RSYS=0
DMX =5510
SMN =-.426E+07
SMX =.462E+07
AUG 29 2010
10:04:49
MX
X
-.426E+07
-.327E+07
-.228E+07
-.130E+07
-309479
676938
.166E+07
.265E+07
.364E+07
.462E+07

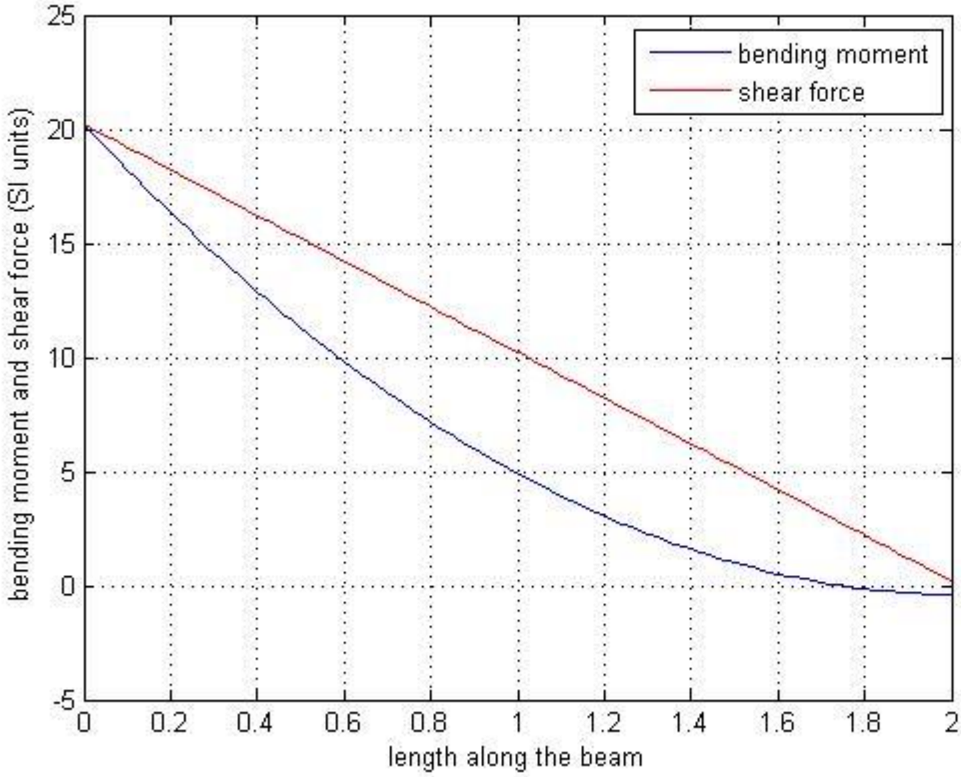

1
NODAL SOLUTION
STEP=1
SUB =1
TIME=1
SEQV (AVG)
DMX =5510
SMN =234455
SMX =.506E+07
AUG 29 2010
10:02:19
MN
MX
X
234455
770819
.131E+07
.184E+07
.238E+07
.292E+07
.345E+07
.399E+07
.453E+07
.506E+07

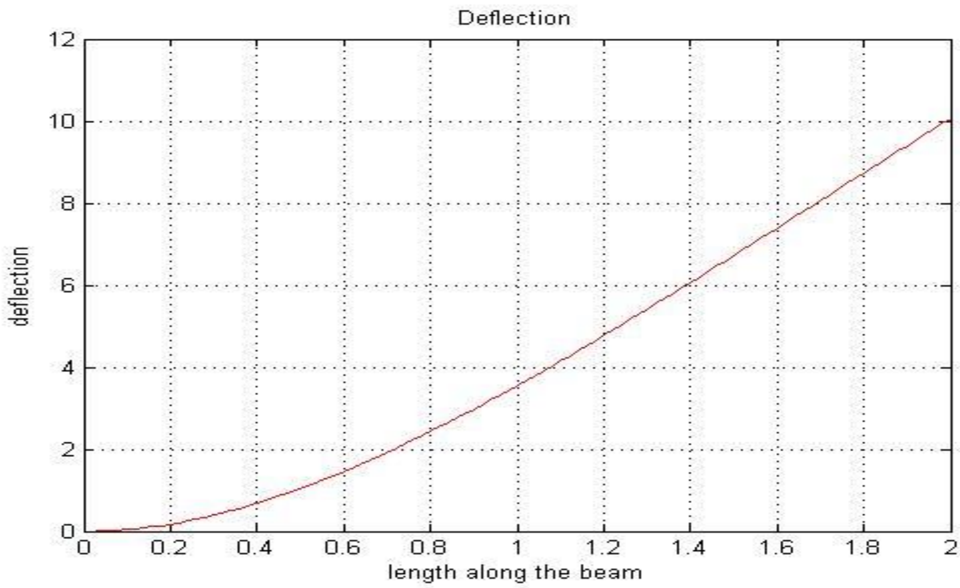
EXP 8: MATLAB CODE FOR BEAM ANALYSIS

```
% Simple algorithm to calculate the deflection, bending
moment, shear force in a cantilever beam.
% load = loading distribution

% Example,
% 1.Uniform loading of 10 N/m: load=repmat(10,1,1001), l=20,
dl=0.02
% 2.Point loading with P=400 N, load=[repmat(0,1,999) 10000
10000], l=20, dl=0.02

% length(load)=(l/dl)

function deflection=beam_deflection(load,ei,l,dl)
load=input('enter load values');
ei=input('enter ei');
l=input('enter length of beam');
dl=input('enter deflection of beam');
if length(load)~=(l/dl+1)
    error('Check inputs')
end
y=0:dl:l;
m=sum((y.*load))*dl;
v=sum(load)*dl;
u_4=load/ei;
u_3=v/ei;
for i=2:length(load)
    u_3(i)=u_3(i-1)-u_4(i-1)*dl;
end
u_2=m/ei;
for i=2:length(load)
    u_2(i)=u_2(i-1)-u_3(i-1)*dl;
end
u_1=0;
for i=2:length(load)
    u_1(i)=u_1(i-1)+u_2(i-1)*dl;
end
u=0;
for i=2:length(load)
    u(i)=u(i-1)+u_1(i-1)*dl;
end
deflection=u;
plot(y,u_2*ei)
hold on
plot(y,u_3*ei,'r')
legend('bending moment','shear force')
xlabel('length along the beam')
ylabel(' bending moment and shear force (SI units)')
grid
hold off
figure,plot(y,u,'r')
```

```
xlabel('length along the beam')
ylabel('deflection')
grid
title('Deflection')

enter load valuesrepmat(10,1,101)
enter ei2
enter length of beam2
enter deflection of beam0.02

ans =

  Columns 1 through 11

         0         0    0.0040    0.0120    0.0239    0.0396
0.0590    0.0820    0.1087    0.1388    0.1723

  Columns 12 through 22

    0.2091    0.2493    0.2926    0.3390    0.3885    0.4410
0.4964    0.5546    0.6156    0.6794    0.7457

  Columns 23 through 33

    0.8147    0.8861    0.9600    1.0363    1.1149    1.1957
1.2788    1.3640    1.4513    1.5406    1.6318

  Columns 34 through 44

    1.7250    1.8200    1.9169    2.0154    2.1157    2.2175
2.3210    2.4260    2.5325    2.6404    2.7497

  Columns 45 through 55

    2.8603    2.9722    3.0854    3.1998    3.3153    3.4319
3.5496    3.6683    3.7879    3.9086    4.0301

  Columns 56 through 66

    4.1525    4.2757    4.3996    4.5244    4.6498    4.7759
4.9027    5.0301    5.1580    5.2865    5.4155

  Columns 67 through 77

    5.5450    5.6749    5.8053    5.9360    6.0671    6.1986
6.3304    6.4624    6.5948    6.7273    6.8601

  Columns 78 through 88

    6.9931    7.1263    7.2597    7.3931    7.5267    7.6604
7.7942    7.9281    8.0620    8.1960    8.3300
```

```
  Columns 89 through 99

    8.4640    8.5980    8.7320    8.8659    8.9999    9.1338
9.2677    9.4015    9.5353    9.6690    9.8026

  Columns 100 through 101

    9.9361   10.0696
```





EXP 9: MATLAB CODE FOR PLATE BENDING ANALYSIS

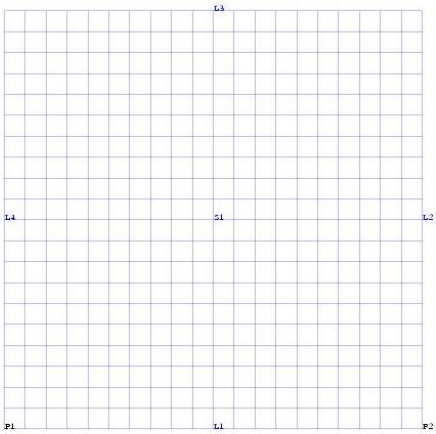
```
% Static Analysis of  plate
% Problem : To find the maximum bedning of plate when uniform
transverse
% pressure is applied.
% Two Boundary conditions are used, simply supported and
clamped
% Variable descriptions
%   ke = element stiffness matrix
%   kb = element stiffness matrix for bending
%   ks = element stiffness matrix for shear
%   f = element force vector
%   stiffness = system stiffness matrix
%   force = system vector
%   displacement = system nodal displacement vector
%   coordinates = coordinate values of each node
%   nodes = nodal connectivity of each element
%   index = a vector containing system dofs associated with
each element
%   pointb = matrix containing sampling points for bending
term
%   weightb = matrix containing weighting coefficients for
bending term
%   points = matrix containing sampling points for shear term
%   weights = matrix containing weighting coefficients for
shear term
%   bcdof = a vector containing dofs associated with boundary
conditions
%   bcval = a vector containing boundary condition values
associated with
%           the dofs in 'bcdof'
%   B_pb = matrix for kinematic equation for bending
%   D_pb = matrix for material property for bending
%   B_ps = matrix for kinematic equation for shear
%   D_ps = matrix for material property for shear
clear
clc
%
disp('Please wait Programme is under Run')
%  Input data
load coordinates.dat ;
% Input data for nodal connectivity for each element
load nodes.dat ;
nel = length(nodes) ;                  % number of elements
nnel=4;                          % number of nodes per element
ndof=3;                          % number of dofs per node
nnode = length(coordinates) ;% total number of nodes in system
sdof=nnode*ndof;                       % total system dofs
edof=nnel*ndof;               % degrees of freedom per element
```

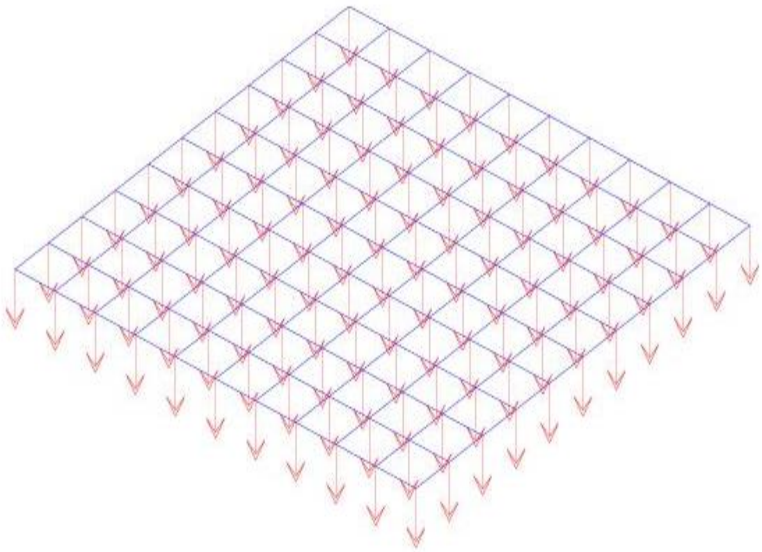
```
% Geometrical and material properties of plate
a = 1 ;                          % Length of the plate (along
X-axes)
b = 1 ;                          % Length of the plate (along
Y-axes)
E = 10920;                       % elastic modulus
nu = 0.3;                        % Poisson's ratio
t = 0.1 ;                        % plate thickness
I = t^3/12 ;
%
PlotMesh(coordinates,nodes)
% Order of Gauss Quadrature
nglb=2;                    % 2x2 Gauss-Legendre quadrature
for bending
ngls=1;                    % 1x1 Gauss-Legendre quadrature
for shear
% Initialization of matrices and vectors
force = zeros(sdof,1) ;            % System Force Vector
stiffness=zeros(sdof,sdof);        % system stiffness matrix
index=zeros(edof,1);               % index vector
B_pb=zeros(3,edof);            % kinematic matrix for bending
B_ps=zeros(2,edof);             % kinematic matrix for shear
% Transverse uniform pressure on plate
P = -1.*10^0 ;
%Computation of element matrices and vectors and their
%  For bending stiffness
[pointb,weightb]=GaussQuadrature('second');     % sampling
points & weights
D_pb= I*E/(1-nu*nu)*[1  nu 0; nu  1  0; 0  0  (1-nu)/2];
                                  % bending material property
%
%  For shear stiffness
%
[points,weights] = GaussQuadrature('first');    % sampling
points & weights
G = 0.5*E/(1.0+nu);                                  % shear modulus
shcof = 5/6;                        % shear correction factor
D_ps=G*shcof*t*[1 0; 0 1];          % shear material property
for iel=1:nel          % loop for the total number of elements
for i=1:nnel
node(i)=nodes(iel,i); %extract connected node for (iel)-th
element
xx(i)=coordinates(node(i),1);   % extract x value of the node
yy(i)=coordinates(node(i),2);   % extract y value of the node
end
ke = zeros(edof,edof);             % initialization of
element stiffness matrix
kb = zeros(edof,edof);             % initialization of
bending matrix
ks = zeros(edof,edof);       % initialization of shear matrix
f = zeros(edof,1) ;          % initialization of force vector
```

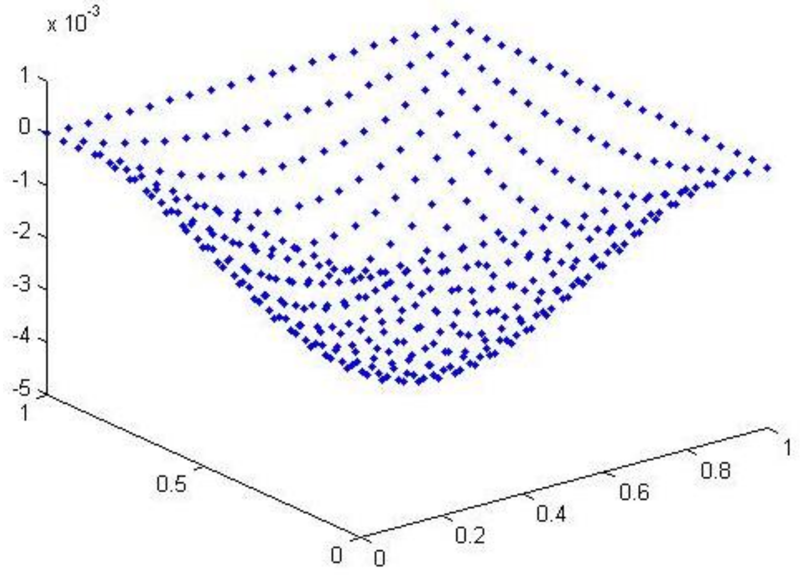
```
%  Numerical integration for bending term
for intx=1:nglb
xi=pointb(intx,1);                    % sampling point in x-axis
wtx=weightb(intx,1);                  % weight in x-axis
for inty=1:nglb
eta=pointb(inty,2);                   % sampling point in y-axis
wty=weightb(inty,2) ;                 % weight in y-axis
[shape,dhdr,dhds]=Shapefunctions(xi,eta);
% compute shape functions and derivatives at sampling point
[detjacobian,invjacobian]=Jacobian(nnel,dhdr,dhds,xx,yy);  %
compute Jacobian
[dhdx,dhdy]=ShapefunctionDerivatives(nnel,dhdr,dhds,invjacobia
n);
                     % derivatives w.r.t. physical coordinate
B_pb=PlateBending(nnel,dhdx,dhdy);    % bending kinematic
matrix
%  compute bending element matrix

kb=kb+B_pb'*D_pb*B_pb*wtx*wty*detjacobian;

end
end                         % end of numerical integration loop
for bending term
%  numerical integration for shear term
for intx=1:ngls
xi=points(intx,1);                     % sampling point in x-axis
wtx=weights(intx,1);                  % weight in x-axis
for inty=1:ngls
eta=points(inty,2);                     % sampling point in y-
axis
wty=weights(inty,2) ;                 % weight in y-axis

[shape,dhdr,dhds]=Shapefunctions(xi,eta);
        % compute shape functions and derivatives at sampling
point

[detjacobian,invjacobian]=Jacobian(nnel,dhdr,dhds,xx,yy);  %
compute Jacobian

[dhdx,dhdy]=ShapefunctionDerivatives(nnel,dhdr,dhds,invjacobia
n);
            % derivatives w.r.t. physical coordinate

fe = Force(nnel,shape,P) ;              % Force vector
B_ps=PlateShear(nnel,dhdx,dhdy,shape);        % shear
kinematic matrix
%  compute shear element matrix

ks=ks+B_ps'*D_ps*B_ps*wtx*wty*detjacobian;
f = f+fe*wtx*wty*detjacobian ;
```

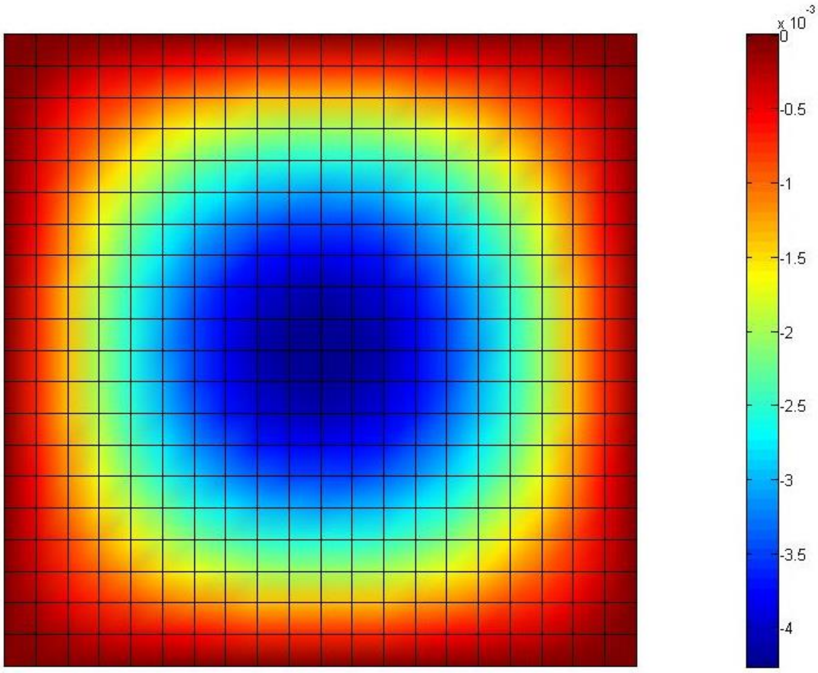
```
end
end                         % end of numerical integration loop
for shear term
%  compute element matrix
ke = kb+ks ;

index=elementdof(node,nnel,ndof);% extract system dofs
associated with element

[stiffness,force]=assemble(stiffness,force,ke,f,index);
                           % assemble element stiffness and
force matrices
end
% Boundary conditions

typeBC = 'ss-ss-ss-ss' ;        % Boundary Condition type
% typeBC = 'c-c-c-c'   ;
bcdof = BoundaryCondition(typeBC,coordinates) ;
bcval = zeros(1,length(bcdof)) ;

[stiffness,force] = constraints(stiffness,force,bcdof,bcval);
% Solution

displacement = stiffness\force ;
% Output of displacements
[w,titax,titay] = mytable(nnode,displacement,sdof) ;
% Deformed Shape
x = coordinates(:,1) ;
y = coordinates(:,2) ;
f3 = figure ;
set(f3,'name','Postprocessing','numbertitle','off') ;
plot3(x,y,w,'.') ;
title('plate deformation') ;

% Maximum transverse displacement
format long
D1 = E*t^3/12/(1-nu^2) ;
minw = min(w)*D1/(P*a^4)
% Contour Plots
PlotFieldonMesh(coordinates,nodes,w)
title('Profile of UZ/w on plate')
PlotFieldonDefoMesh(coordinates,nodes,w,w)
title('Profile of UZ on deformed Mesh') ;
```

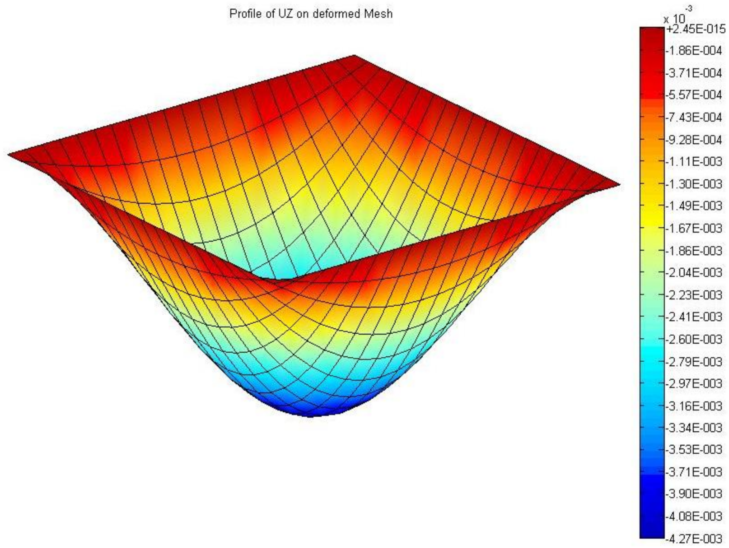
RESULTS:






x 10^-3
1
0
-1
-2
-3
-4
-5
1
0.5
0
0
0.2
0.4
0.6
0.8
1


x 10^-3
0
-0.5
-1
-1.5
-2
-2.5
-3
-3.5
-4


Profile of UZ on deformed Mesh
x 10^-3
+2.45E-015
-1.86E-004
-3.71E-004
-5.57E-004
-7.43E-004
-9.28E-004
-1.11E-003
-1.30E-003
-1.49E-003
-1.67E-003
-1.86E-003
-2.04E-003
-2.23E-003
-2.41E-003
-2.60E-003
-2.79E-003
-2.97E-003
-3.16E-003
-3.34E-003
-3.53E-003
-3.71E-003
-3.90E-003
-4.08E-003
-4.27E-003

EXP 10: SIMULATION OF AIRCRAFT MOTION – LONGITUDINAL AND LATERAL DYNAMICS

AIM:-To simulate the motion of an aircraft in longitudinal motion.

THEORY:-The acceleration due to gravity of a non -spherical, axisymmetric planet can be obtained according tothe equation:

$$\mathrm{g}^T = \frac{\partial \Phi_2}{\partial \mathbf{r}_{12}} = -\frac{\partial \Phi_1}{\partial \mathbf{r}_{12}}$$

by taking the gradient of the gravitational potential,

$$\Phi(r,\phi) = \frac{GM}{r}\left\{1 - \sum_{n=2}^{\infty}\left(\frac{R_e}{r}\right)^n J_n P_n(\cos\phi)\right\} \ ,$$

Where,

$$J_n \doteq -\frac{A_n}{GM{R_e}^n}$$

with respect to the position vector, $\mathbf{r} = r\mathbf{i_r} + r\varphi\boldsymbol{i\varphi}$as follows:

$$\mathrm{g} = -\left(\frac{\partial \Phi}{\partial \mathbf{r}}\right)^T = -\frac{\partial \Phi}{\partial r}\mathbf{i}_\mathrm{r} - \frac{\partial \Phi}{r\partial \phi}\mathbf{i}_\phi \ ,$$

Where,

$$\mathrm{g} = g_r \mathbf{i}_\mathrm{r} + g_\phi \mathbf{i}_\phi \ ,$$

$$g_r = -\frac{GM}{r^2}\left[1 - 3J_2\left(\frac{R_e}{r}\right)^2 P_2(\cos\phi) - 4J_3\left(\frac{R_e}{r}\right)^3 P_3(\cos\phi) - 5J_4\left(\frac{R_e}{r}\right)^4 P_4(\cos\phi)\right] \ ,$$

$$g_\phi = \frac{3GM}{r^2}\left(\frac{R_e}{r}\right)^2 \sin\phi\cos\phi\left[J_2 + \frac{1}{2}J_3\left(\frac{R_e}{r}\right)\sec\phi(5\cos^2\phi - 1) + \frac{5}{6}J_4\left(\frac{R_e}{r}\right)^2(7\cos^2\phi - 1)\right] \ .$$

The unit vectors **ir**and **i*φ***denote the radial and southward directions in the local horizon frameattached to the test mass. Due to a nonzero transverse gravity component, *gφ*, the direction of **g** differs from the radial direction, while its radial component, *gr*, is smaller in magnitude compared to that predicted by a spherical gravity model. These deviations are quite important in applications such as the flight of an atmospheric entry vehicle, and the long-range navigation of airplanes and missiles. For example, by ignoring the non - spherical gravity, one may commit an error of several hundred kilometres in an entry trajectory from a low earth orbit.

ALGORITHM:-

- For simulating the longitudinal motion of an aircraft, firstly a gravity .mfile must be executed which calculates the acceleration due to gravity .
- The m.file is executed in matlab.
- After the execution, inputs are to be given in the command window, like the values of radius of earth, latitude, g_c, gnorth.
- Based on the above values given matlab gives the six degrees of freedom of motion values which are required for simulating the aircrafts longitudinal motion.

PROGRAM:-

```
function [gc,gnorth]=gravity(r,lat)
phi=pi/2-lat;
mu=3.986004e14;%mu=GMe
Re=6378.135e3;
J2=1.08263e-3;
J3=2.532153e-7;
J4=1.6109876e-7;
gc=mu*(1-1.5*J2*(3*cos(phi)^2-1)*(Re/r)^2-2*J3*cos(phi)*(5*cos(phi)^2- 3)*(Re/r)^3-
(5/8)*J4*(35*cos(phi)^4-30*cos(phi)^2+3)*(Re/r)^4)/r^2;
gnorth=-3*mu*sin(phi)*cos(phi)*(Re/r)*(Re/r)*(J2+0.5*J3*(5*cos(phi)^2-
1)*(Re/r)/cos(phi)+(5/6)*J4*(7*cos(phi)^2-1)*(Re/r)^2)/r^2;
```

RESULTS:-

```
>>r= 6378135+2000
>>r = 6380135
>>lat=45*pi/180
>>0.7854
>>[gc, gnorth] = gravity (r, lat)
>>gc = 9.7842
>>gnorth = -0.0159
```

ATMOSPHERE *.m* FILE:-

```
function Y = atmosphere(h, vel, CL)
%(c) 2005 AshishTewari
R = 287; %sea-level gas constant for air (J/kg.K)
go = 9.806; %sea level acceleration due to gravity (m/s^2)
Na = 6.0220978e23; %Avogadro‘s number
sigma = 3.65e-10; %collision diameter (m) for air
S = 110.4; %Sutherland‘s temperature (K)
Mo = 28.964; %sea level molecular weight (g/mole)
To = 288.15; %sea level temperature (K)
Po = 1.01325e5; %sea level pressure (N/m^2)
```

```
re = 6378.14e3; %earth‘s mean radius (m)
Beta = 1.458e-6; %Sutherland‘s constant (kg/m.s.K^0.5)
gamma = 1.405; %sea level specific-heat ratio
B = 2/re; layers = 21; Z = 1e3*[0.00; 11.0191; 20.0631; 32.1619; 47.3501; 51.4125;
71.8020; 86.00; 100.00; 110.00; 120.00; 150.00; 160.00; 170.00; 190.00; 230.00; 300.00;
400.00; 500.00; 600.00; 700.00; 2000.00];
T = [To; 216.65; 216.65; 228.65; 270.65; 270.65; 214.65; 186.946; 210.65; 260.65; 360.65;
960.65; 1110.60; 1210.65; 1350.65; 1550.65; 1830.65; 2160.65; 2420.65; 2590.65; 2700.00;
2700.0];
M = [Mo; 28.964; 28.964; 28.964; 28.964; 28.964; 28.962; 28.962; 28.880; 28.560; 28.070;
26.920; 26.660; 26.500; 25.850; 24.690; 22.660; 19.940; 17.940; 16.840; 16.170; 16.17];
LR = [-6.5e-3; 0; 1e-3; 2.8e-3; 0; -2.8e-3; -2e-3; 1.693e-3; 5.00e-3; 1e-2; 2e-2; 1.5e-2; 1e-2;
7e-3; 5e-3; 4e-3; 3.3e-3; 2.6e-3; 1.7e-3; 1.1e-3; 0];
rho0 = Po/(R*To); P(1) = Po; T(1) = To; rho(1) = rho0; for i = 1:layers
if ~(LR(i) == 0)
C1 = 1 + B*( T(i)/LR(i) - Z(i) );
C2 = C1*go/(R*LR(i));
C3 = T(i+1)/T(i);
C4 = C3^(-C2);
C5 = exp( go*B*(Z(i+1)-Z(i))/(R*LR(i)) );
P(i + 1) = P(i)*C4*C5;
C7 = C2 + 1;
rho(i + 1) = rho(i)*C5*C3^(-C7);
else
C8 = -go*(Z(i+1)-Z(i))*(1 - B*(Z(i + 1) + Z(i))/2)/(R*T(i));
P(i+1) = P(i)*exp(C8); rho(i+1) = rho(i)*exp(C8);
end
end for i = 1:21
if h < Z(i+1)
if ~(LR(i)== 0)
C1 = 1 + B*( T(i)/LR(i) - Z(i) );
TM = T(i) + LR(i)*(h - Z(i));
C2 = C1*go/(R*LR(i));
C3 = TM/T(i);
C4 = C3^(-C2);
C5 = exp( B*go*(h - Z(i))/(R*LR(i)) );
PR = P(i)*C4*C5; %Static Pressure (N/m^2)
C7 = C2 + 1;
rhoE = C5*rho(i)*C3^(-C7); %Density (kg/m^3)
else
TM = T(i);
C8 = -go*(h - Z(i))*(1 - (h + Z(i))*B/2)/(R*T(i));
PR = P(i)*exp(C8); %Static Pressure (N/m^2)
rhoE = rho(i)*exp(C8); %Density (kg/m^3)
end
MOL = M(i) + ( M(i+1)-M(i) )*( h - Z(i) )/( Z(i+1) - Z(i) );
TM=MOL*TM/Mo;%KineticTemperatureasound = sqrt(gamma*R*TM); % Speed of Sound
(m/s)
MU = Beta*TM^1.5/(TM + S); % Dynamic Viscosity Coeff. (N.s/m^2)
KT = 2.64638e-3*TM^1.5/(TM + 245.4*10^(-12/TM));
```

```
Vm = sqrt(8*R*TM/pi); m = MOL*1e-3/Na; n = rhoE/m;
F = sqrt(2)*pi*n*sigma^2*Vm;
L = Vm/F; % Mean free-path (m)
Mach = vel/asound; % Mach Number
T0 = TM*(1 + (gamma - 1)*Mach^2/2);
MU0 = Beta*T0^1.5/(T0 + S);
RE0 = rhoE*vel*CL/MU0;
RE = rhoE*vel*CL/MU; % Reynold‘s Number
Kn = L/CL; % Knudsen Number
Kno = 1.25*sqrt(gamma)*Mach/RE0;
%flow regime parameter
ifKn>= 10
d = 1; % free-molecule flow
elseifKn<= 0.01
d = 2; % continuum flow
else
d = 3; % transition flow
end
Y = [TM; rhoE; Mach; Kn; asound; d; PR; MU; RE; KT];
return;
end
end
```

<u>RESULTS:-</u>

```
>> h=2000
>>vel=330
>> CL=0.001
>> y=atmosphere (h, vel, CL)
>>y = 1.0e+004 *
0.0275
0.0001
0.0001
0.0000
0.0333
0.0002
7.9499
0.0000
1.9248
0.0000
```

<u>PITCH UP.*m*FILE:-</u>

```
>> x=[2000 0 0.01 0 0 0.7854]
>>x = 1.0e+003 *
2.0000 0 0.0000 0 0 0.0008
>> t = [0 5 10 15 20 25]
>>t = 0 5 10 15 20 25
>> y = pitchup (t, x)
>> y = 1.0e+007 * -0.0000
-0.0000
-1.7397
0
0.0000
```

EXP 11: SATELLITE ATTITUDE DYNAMICS

AIM:-To simulate the torque free rotation values of axisymmetric and asymmetric spacecraft.

THEORY:-

ASYMETRIC SPACECRAFT:

$$J_{xx}\dot{\omega}_x + n\omega_y(J_{zz} - J_{yy}) \approx 0,$$
$$J_{yy}\dot{\omega}_y + n\omega_x(J_{xx} - J_{zz}) \approx 0,$$
$$J_{zz}\dot{\epsilon} \approx 0,$$

when a small disturbance, ωx(0), ωy(0), is applied. At a subsequent time, the angular velocity components can be expressed as ωz = n +_, and ωx, ωy. Since a small disturbance has been applied, we can treat _, ωx, ωy as small quantities and solve Euler‘s equations. If the solution indicates that _, ωx, ωy grow with time in an unbounded fashion, it will be evident that our assumption of small deviations remaining small is false, and we are dealing with an unstable equilibrium. Otherwise, we have a stable equilibrium. Hence, with the assumption of small deviation from equilibrium, we can write the approximate, linearized Euler equations.

$$e^{Kt} = \mathcal{L}^{-1}(s\mathbf{I} - \mathbf{K})^{-1} = \begin{pmatrix} \cos(\sqrt{k_1k_2}t) & -\sqrt{\frac{k_1}{k_2}}\sin(\sqrt{k_1k_2}t) \\ \sqrt{\frac{k_2}{k_1}}\sin(\sqrt{k_1k_2}t) & \cos(\sqrt{k_1k_2}t) \end{pmatrix}$$

Attitude thrusters can be used for controlling the attitude of a spin-stabilized, axisymmetric spacecraft, which involvesmulti-axis rotation (precession). If the spin rate is constant (ωz = n), the governing differential equations describing precession, Eq., are linear, thus enabling the use of time-optimal, bangbang, open-loop control in the same manner as the single-axis rotation.

PROGRAM:-

***.m* File**

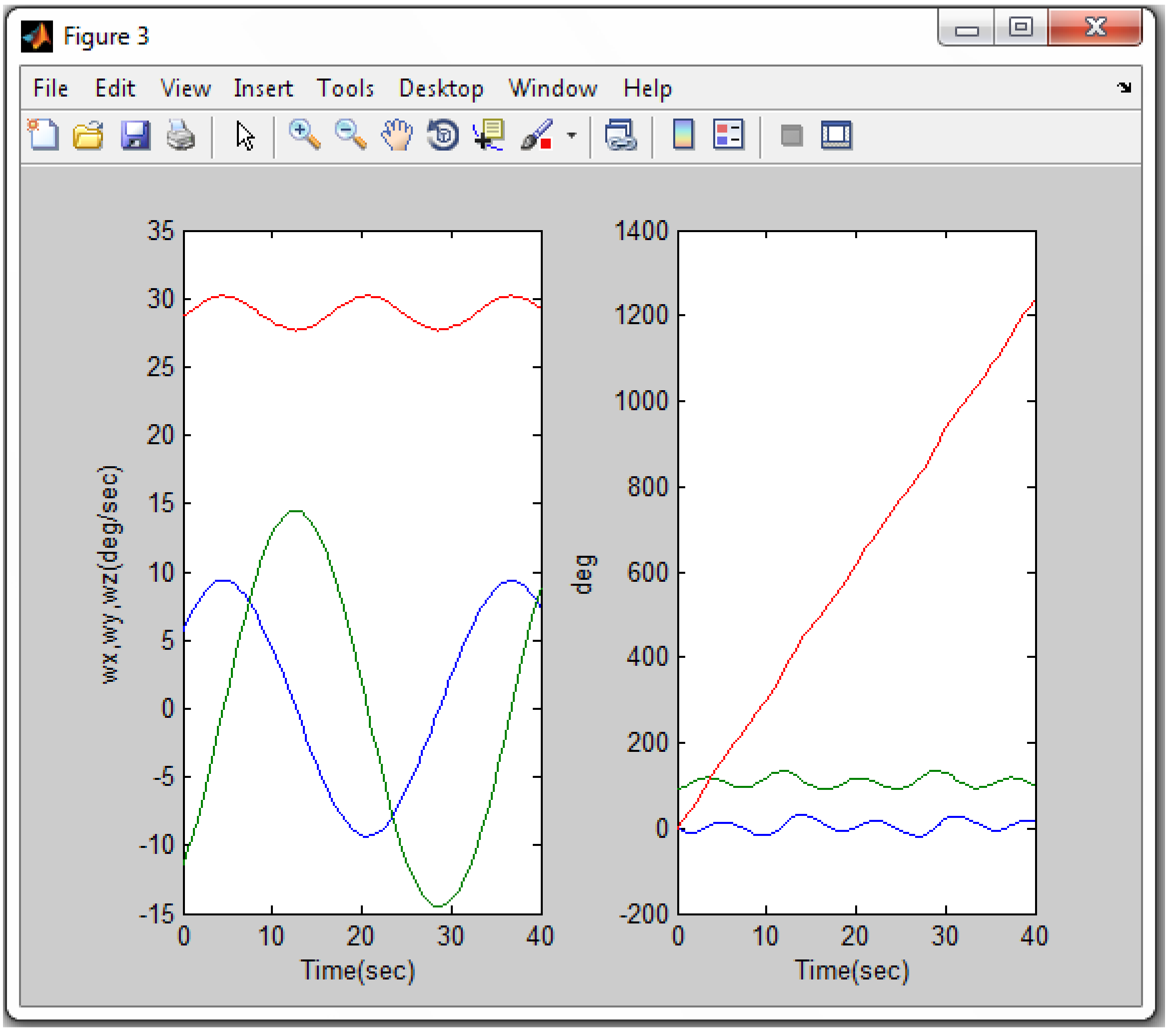
```
functionxdot=spacerotation(t,x)
%program for torque-free rotational dynamics and Euler 3-1-3 kinematics
%of rigid spacecraft
%x(1)=omega_x, x(2)=omega_y, x(3)=omega_z (angular velocity in rad/s)
%x(4)=psi, x(5)=theta, x(6)=phi (rad)
%(c) 2006 AshishTewari
J1=4000; J2=7500; J3=8500; %principal moments of inertia (kg.m^2)
xdot(1,1)=x(2)*x(3)*(J2-J3)/J1;
xdot(2,1)=x(1)*x(3)*(J3-J1)/J2;
xdot(3,1)=x(1)*x(2)*(J1-J2)/J3;
```

```
xdot(4,1)=(sin(x(6))*x(1)+cos(x(6))*x(2))/sin(x(5));
xdot(5,1)=cos(x(6))*x(1)-sin(x(6))*x(2);
        xdot(6,1)=x(3)-(sin(x(6))*cos(x(5))*x(1)+cos(x(6))*cos(x(5))*x(2))/sin(x(5));
```

RESULT:-

```
>> [t,x]=ode45(@spacerotation,[0 40],[0.1 -0.2 0.5 0 0.5*pi 0]‘);
>> subplot(121),plot(t,x(:,1:3)*180/pi),hold on,.
        >>subplot(122),plot(t,x(:,4:6)*180/pi)%time evolution of motion variables
```

PLOTS:-



EXP 12: SPACE THRUST

```
%program for rotational dynamics and Euler 3-1-3 kinematics
%of rigid, axisymmetric, spin-stabilized spacecraft
%due to torque pulses about ‘oy‘ principal axis
%x(1)=omega_x, x(2)=omega_y (angular velocity in rad/s)
%x(3)=psi, x(4)=phi (rad)
%u = impulsive torque about ‘oy‘ axis (N-m)
%(c) 2006 AshishTewari
J1=1500; J3=500; %principal moments of inertia (kg.m^2)
thd2=acos(J3/(J1-J3))
T=0.01;
n=1; %rad/s
%thd2=atan(umax*T/(n*J3))
Ts=pi/abs(n*(1-J3/J1))
x=[];
x(1,1)=0;
x(2,1)=J3*n*tan(thd2)/J1;
x(3,1)=0;x(4,1)=0;
[t1,x1]=ode45(@spacesymm,[0 Ts],x);
N=size(t1,1);
x(1,1)=0;
x(2,1)=0;
x(3,1)=x1(N,3);x(4,1)=x1(N,4);
[t2,x2]=ode45(@spacesymm,[Ts+T Ts+T+1.5],x);
t=[t1;t2];x=[x1;x2];
dtr=pi/180;
plot(t,x(:,1:2)/dtr,t,sqrt(x(:,1).*x(:,1)+x(:,2).*x(:,2))/dtr),...
xlabel('Time (s)'),ylabel('Precession angular velocity (deg./s)')
figure
plot(t,x(:,3)/dtr,t,x(:,4)/dtr),xlabel('Time (s)'),...
        ylabel('Precession angle, \psi, inertial spin angle, \phi (deg.)')
```

SPACESYMMETRIC.m File:-

```
functionxdot=spacesymm(t,x)
%program for rotational dynamics and Euler 3-1-3 kinematics
%of rigid, axisymmetric, spin-stabilized spacecraft
%x(1)=omega_x, x(2)=omega_y (angular velocity in rad/s)
%x(3)=psi, x(4)=phi (rad)
%(c) 2006 AshishTewari
J1=1500; J3=500; %principal moments of inertia (kg.m^2)
n=1; %rad/s
%umax=1000;%torque magnitude
%T=0.01;%impulse duration
%thd2=atan(umax*T/(n*J3));%nutation angle
thd2=acos(J3/(J1-J3));
xdot(1,1)=x(2)*n*(J1-J3)/J1;%Euler‘s eqn.(1)
xdot(2,1)=x(1)*n*(J3-J1)/J1;%Euler‘s eqn.(2)
xdot(3,1)=(sin(x(4))*x(1)+cos(x(4))*
```

xdot(4,1)=n*(1-J3/J1); %inertial spin rate