ACS College of Engineering
Approved by AICTE New Delhi, Affiliated to VTU, Belagavi
(A Unit of RajaRajeswari Group of Institutions)

NAAC 'A' Accredited

DEPARTMENT OF AEROSPACE ENGINEERING

18ASL66

LAB MANUAL

LIST OF EXPERIMENTS

Getting Started with ANSYS

Performing a Typical ANSYS Analysis

The ANSYS program has many finite element analysis capabilities, ranging from a simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis. The analysis guide manuals in the ANSYS documentation set describe specific procedures for performing analyses for different engineering disciplines.

A typical ANSYS analysis has three distinct steps:

- **PREPROCESSING**
- **SOLUTION**
- **POSTPROCESSING**

PREPROCESSING

Building a finite element model requires more of an ANSYS user's time than any other part of the analysis. First, you specify a Jobname and analysis title. Then, you use the PREP7 Pre-processor to define the element types, element real constants, material properties, and the model geometry.

Specifying a Jobname and Analysis Title

This task is not required for an analysis but is *recommended*.

Defining the Jobname

The *jobname* is a name that identifies the ANSYS job. When you define a jobname for an

analysis, the jobname becomes the first part of the name of all files the analysis creates. (The extension or suffix for these files' names is a file identifier such as .DB.) By using a jobname for each analysis, you ensure that no files are overwritten.

If you do not specify a jobname, all files receive the name *FILE* or *file*, depending on the operating system.

Command(s): **/FILNAME**

GUI: **Utility Menu>File>Change Jobname**

Defining Element Types

The ANSYS element library contains more than 100 different element types. Each element type has a unique number and a prefix that identifies the element category: BEAM4, PLANE77, SOLID96, etc. The following element categories are available.

BEAM	PLANE
COMBINation	SHELL
CONTACt	SOLID
FLUID	SOURCe
HYPERelastic	SURFace
INFINite	TARGEt
LINK	USER
MASS	INTERface
MATRIX	VISCOelastic (or viscoplastic)
PIPE	

The element type determines, among other things:

The degree-of-freedom set (which in turn implies the discipline-structural, thermal,

magnetic, electric, quadrilateral, brick, etc.) Whether the element lies in two-dimensional or three-dimensional space.

For example, BEAM4, has six structural degrees of freedom (UX, UY, UZ, ROTX, ROTY, ROTZ), is a line element, and can be modelled in 3-D space. PLANE77 has a thermal degree of freedom (TEMP), is an eight-node quadrilateral element, and can be modelled only in 2-D space.

Defining Element Real Constants:

Element real constants are properties that depend on the element type, such as cross-sectional properties of a beam element. For example, real constants for BEAM3, the 2-D beam element, are area (AREA), moment of inertia (IZZ), height (HEIGHT), shear deflection constant.

(SHEARZ), initial strain (ISTRN), and added mass per unit length (ADDMAS). Not all element

types require real constants, and different elements of the same type may have different real constant values.

As with element types, each set of real constants has a reference number, and the table of

reference number versus real constant set is called the *real constant table*. While defining the elements, you point to the appropriate real constant reference number using the **REAL** command.

(**Main Menu> Pre-processor>Create>Elements>Elem Attributes**).

Defining Material Properties:

Most element types require material properties. Depending on the application, material

Properties may be:

Linear or nonlinear

Isotropic, Orthotropic, or Anisotropic

Constant temperature or temperature dependent.

As with element types and real constants, each set of material properties has a material reference number. The table of material reference numbers versus material property sets is called the *material table*. Within one analysis, you may have multiple material property sets (to correspond with multiple materials used in the model). ANSYS identifies each set with a unique reference number.

Main Menu> Preprocessor> Material Props> Material Models.

Creating the Model Geometry

Once you have defined material properties, the next step in an analysis is generating a finite element model-nodes and elements-that adequately describes the model geometry.

There are two methods to create the finite element model: solid modelling and direct generation.

With *solid Modeling*, you describe the geometric shape of your model, then instruct the ANSYS program to automatically *mesh* the geometry with nodes and elements. You can control the size and shape of the elements that the program creates. With *direct generation*, you "manually" define the location of each node and the connectivity of each element. Several convenience operations, such as copying patterns of existing nodes and elements, symmetry reflection, etc. are available.

Apply Loads and Obtain the Solution

In this step, you use the SOLUTION processor to define the analysis type and analysis options, apply loads, specify load step options, and initiate the finite element solution. You also can apply loads using the PREP7 Preprocessor.

Applying Loads

The word *loads* as used in this manual includes boundary conditions (constraints, supports, or boundary field specifications) as well as other externally and internally applied loads. Loads in the ANSYS program are divided into six categories:

DOF Constraints

Forces

Surface Loads

Body Loads

Inertia Loads

Coupled field Loads.

You can apply most of these loads either on the solid model (key points, lines, and areas) or the finite element model (nodes and elements).

Two important load-related terms you need to know are load step and sub step. A *load step* is simply a configuration of loads for which you obtain a solution. In a structural analysis, for example, you may apply wind loads in one load step and gravity in a second load step. Load steps are also useful in dividing a transient load history curve into several segments. *Sub steps* are incremental steps taken within a load step. You use them mainly for accuracy and convergence purposes in transient and nonlinear analyses. Sub steps are also known as *time steps* steps taken over a period.

SOLUTION

To initiate solution calculations, use either of the following:

Command(s): **SOLVE.**

GUI: **Main Menu>Solution>Current LS**

When you issue this command, the ANSYS program takes model and loading information from the database and calculates the results. Results are written to the results file (*Jobname*.RST, *Jobname*.RTH, *Jobname*.RMG, or *Jobname*.RFL) and to the database.

The only difference is that only one set of results can reside in the database at one time, while you can write all sets of results (for all sub steps) to the results file.

POSTPROCESSING

Once the solution has been calculated, you can use the ANSYS postprocessors to review the results.

EXPERIMENT-1

MODELING OF SYMMETRIC AEROFOIL GEOMETRY AND GENERATION OF BODY FITTING ADAPTIVE MESH

Aim: Modeling of Symmetric/Cambered Airfoil geometry, and generation of body fitting mesh.

Apparatus: A computer hardware, software (ANSYS) with a graphical user interface.

PROCEDURE

The three main steps to be involved are

1. Pre-Processing

2. Solution

3. Post Processing

Start - All Programs – ANSYS - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

PREPROCESSING

1. Preference – Flotran CFD- h-Method - Ok.

2. Preprocessor - Element type - Add/Edit/Delete – Add – 2D FLOTRAN 141, FLOTRAN 141 – Ok.

3. Material props - Material Models – CFD – Density – 1.23 - Ok.

4. Select the symmetrical Aerofoil Coordinates placed in server. Paste it in the notepad as per the appropriate command – K, NPT,X,Y,Z (Eg- K,1,1,0.00126,0)

5. Write the next command **FLST**, NFIELD, NARG, TYPE, Otype, LENG

6. Write the next command **FITEM**, NFIELD, ITEM, ITEMY, ITEMZ

7. Write the next command

BSPLIN, P1, P2, P3, P4, P5, P6, XV1, YV1, ZV1, XV6, YV6, ZV6

8. Save the File as Aerofoil_Coordinates.dat or (refer APPENDIX)

9. File – Read Input From – Select the Aerofoil_Coordinates.dat File – Open.

10. Modelling – Create – Keypoints – In Active Cs – X – 6 – Y – 6 – Apply, X – (6) – Y (-6) – Apply, X – (6) – Y – (0) – Apply, X – (1.01) – Y (6) – Apply, X – (1.01) – Y (-6) – Apply, X – (-4.99) – Y (0) – OK.

10. Modelling – Create – Lines – (Create Rectangular Grid behind the aerofoil)

11. Modelling – Create – Arc – By End KPs and Rad – Pick end point – Ok – Pick Centre Point – Ok – Radius – (6) – Ok.

12. Modelling – Create – Areas – By Lines – Pick Lines (Create Rectangular Area behind the Aerofoil and C-Shaped area Before the Aerofoil) – Ok.

Note: Do not Create Area In the Aerofoil.

13. Meshing – Size Ctrls – Manual Size – Lines – Picked Lines – Pick (Upper Curved portion of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

14. Meshing – Size Ctrls – Manual Size – Lines – Picked Lines – Pick (Upper Curved portion of Aerofoil) – Ok – No. Of Element Divisions – 20 – Ok.

15. Meshing – Size Ctrls – Manual Size – Lines – Picked Lines – Pick (Vertical Upper Line in Grid) – Ok – No. Of Element Divisions – 20 – Ok.

16. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Horizontal Line in Grid) – Ok – No. Of Element Divisions – 20 – Ok.

17. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Upper Vertical line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

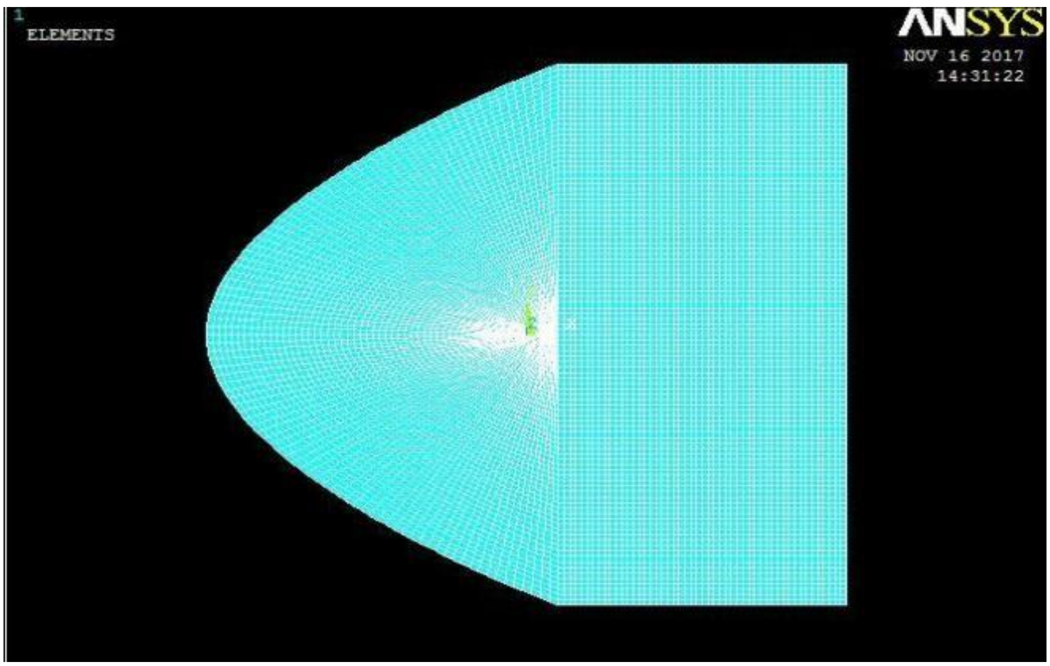
18. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Lower Vertical line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

19. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Upper Horizontal line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Lower Horizontal line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

20. Meshing – Mesh – Areas – Mapped – 3 Or 4 Sides – Pick (All Areas) – Ok.

21. PlotCtrls-Write Metafile-InvertWhite/Black



Result:

Body fitting mesh of Symmetric/Cambered Airfoil has been executed in ANSYS.

VIVA QUESTIONS

1. Define Aerofoil.
2. Explain all NACA aerofoil series.
3. Explain the terms used in the NACA series.
4. What is h-refinement?
5. What is p-refinement?
6. What is Preprocessor?
7. Explain the Element FLOTRAN141.
8. What is meant by Element type?
9. What is Body fitting Mesh?
10. Explain the procedure used to generate mesh over the aerofoil.
11. Define cambered aerofoil.
12. What is the use of cambered aerofoil?
13. What is the difference between structured and unstructured mesh?
14. What is the use of Body Fitting Mesh?
15. Define the tool ‘Solution’.
16. What are Isoperimetric elements?
17. What is the use of Isoperimetric Elements?
18. What is the shape function?
19. Write the uses of Shape function.
20. What is the difference between Key points and nodes?

EXPERIMENT-2

MODELING OF CAMBERED AEROFOIL GEOMETRY AND GENERATION OF BODY FITTING ADAPTIVE MESH

Aim: Modeling of Symmetric/Cambered Airfoil geometry, and generation of body fitting mesh.

Apparatus: A computer hardware, software (ANSYS) with a graphical user interface.

PROCEDURE

The three main steps to be involved are

1. Pre-Processing

2. Solution

3. Post Processing

Start - All Programs – ANSYS - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

PREPROCESSING

1. Preference – Flotran CFD- h-Method - Ok.

2. Preprocessor - Element type - Add/Edit/Delete – Add – 2D FLOTRAN 141, FLOTRAN 141 – Ok.

3. Material props - Material Models – CFD – Density – 1.23 - Ok.

4. Select the CAMBERED Aerofoil Coordinates placed in server. Paste it in the notepad as per the appropriate command – K, NPT,X,Y,Z (Eg- K,1,1,0.00126,0)

5. Write the next command **FLST**, NFIELD, NARG, TYPE, Otype, LENG

6. Write the next command **FITEM**, NFIELD, ITEM, ITEMY, ITEMZ

7. Write the next command

BSPLIN, P1, P2, P3, P4, P5, P6, XV1, YV1, ZV1, XV6, YV6, ZV6

8. Save the File as Aerofoil_Coordinates.dat or (refer APPENDIX)

9. File – Read Input From – Select the Aerofoil_Coordinates.dat File – Open.

10. Modelling – Create – Keypoints – In Active Cs – X – 6 – Y – 6 – Apply, X – (6) – Y (-6) – Apply, X – (6) – Y – (0) – Apply, X – (1.01) – Y (6) – Apply, X – (1.01) – Y (-6) – Apply, X – (-4.99) – Y (0) – OK.

10. Modelling – Create – Lines – (Create Rectangular Grid behind the aerofoil)

11. Modelling – Create – Arc – By End KPs and Rad – Pick end point – Ok – Pick Centre Point – Ok – Radius – (6) – Ok.

12. Modelling – Create – Areas – By Lines – Pick Lines (Create Rectangular Area behind the Aerofoil and C-Shaped area Before the Aerofoil) – Ok.

Note: Do not Create Area In the Aerofoil.

13. Meshing – Size Ctrls – Manual Size – Lines – Picked Lines – Pick (Upper Curved portion of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

14. Meshing – Size Ctrls – Manual Size – Lines – Picked Lines – Pick (Upper Curved portion of Aerofoil) – Ok – No. Of Element Divisions – 20 – Ok.

15. Meshing – Size Ctrls – Manual Size – Lines – Picked Lines – Pick (Vertical Upper Line in Grid) – Ok – No. Of Element Divisions – 20 – Ok.

16. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Horizontal Line in Grid) – Ok – No. Of Element Divisions – 20 – Ok.

17. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Upper Vertical line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

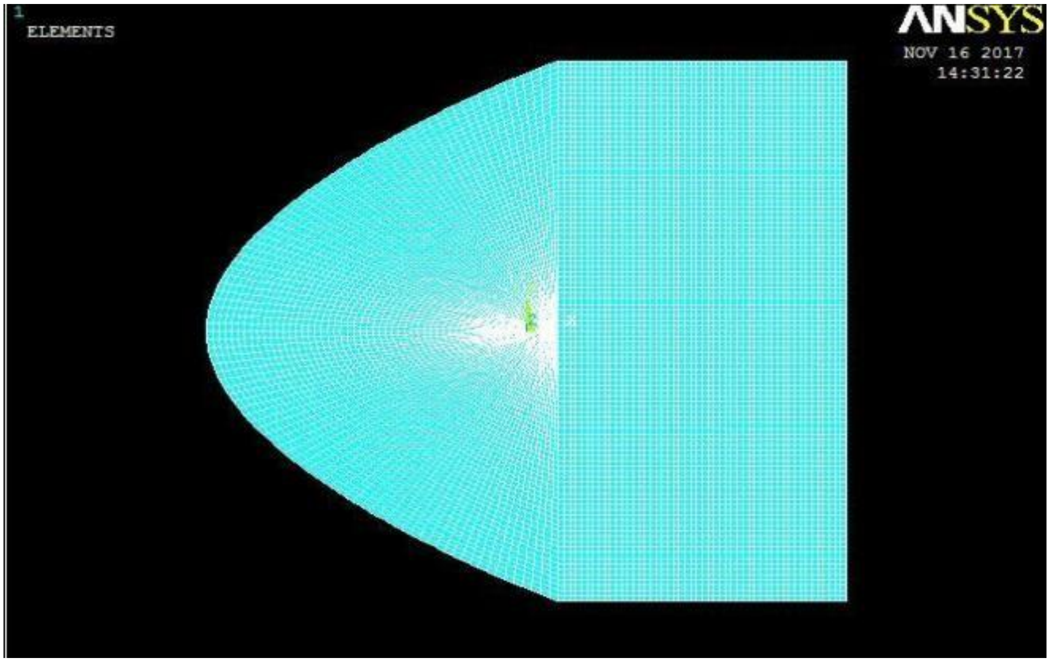
18. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Lower Vertical line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

19. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Upper Horizontal line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Lower Horizontal line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

20. Meshing – Mesh – Areas – Mapped – 3 Or 4 Sides – Pick (All Areas) – Ok.

21. PlotCtrls-Write Metafile-InvertWhite/Black



Result:

Body fitting mesh of Symmetric/Cambered Airfoil has been executed in ANSYS.

VIVA QUESTIONS

1. Define Aerofoil.
2. Explain all NACA aerofoil series.
3. Explain the terms used in the NACA series.
4. What is h-refinement?
5. What is p-refinement?
6. What is Preprocessor?
7. Explain the Element FLOTRAN141.
8. What is meant by Element type?
9. What is Body fitting Mesh?
10. Explain the procedure used to generate mesh over the aerofoil.
11. Define cambered aerofoil.
12. What is the use of cambered aerofoil?
13. What is the difference between structured and unstructured mesh?
14. What is the use of Body Fitting Mesh?
15. Define the tool ‘Solution’.
16. What are Isoperimetric elements?
17. What is the use of Isoperimetric Elements?
18. What is the shape function?
19. Write the uses of Shape function.
20. What is the difference between Key points and nodes?

EXPERIMENT -3

MODELING OF 2-D INCOMPRESSIBLE AND INVISCID FLOW OVER SYMMETRIC/ CAMBERED AEROFOIL AND PLOTTING OF PRESSURE DISTRIBUTION AND VELOCITY VECTORS FOR SUBSONIC/ SUPERSONIC MACHNUMBERS

Aim: Modeling of 2-D Incompressible and Inviscid flow over an aerofoil. Computations and analysis for velocity vectors and pressures distributions.

Apparatus: A computer hardware, software (ANSYS) with a graphical user interface.

PROCEDURE

The three main steps to be involved are

1. Pre Processing

2. Solution

3. Post Processing

Start - All Programs – ANSYS - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

PREPROCESSING

1. Preference – Flotran CFD- h-Method - Ok.

2. Preprocessor - Element type - Add/Edit/Delete – Add – 2D FLOTRAN 141, FLOTRAN 141 – Ok.

3. Material props - Material Models – CFD – Density – 1.23 - Ok.

4. Download Desired Aerofoil Coordinates from Internet. Paste it in the notepad as per the appropriate command – K,NPT,X,Y,Z (Eg- K,1,1,0.00126,0)

5. Write the next command **FLST**, `NFIELD`, `NARG`, `TYPE`, `Otype`, `LENG`

6. Write the next command **FITEM**, `NFIELD`, `ITEM`, `ITEMY`, `ITEMZ`

7.Write the next command

BSPLIN, P1, P2, P3, P4, P5, P6, XV1, YV1, ZV1, XV6, YV6, ZV6

8. Save the File as Aerofoil_Coordinates.dat

9. File – Read Input From – Select the Aerofoil_Coordinates.dat File – Open.

10. Modelling – Create – Keypoints – In Active Cs – X – 6 – Y – 6 – Apply, X – (6) – Y (-6) – Apply, X – (6) – Y – (0) – Apply, X – (1.01) – Y (6) – Apply, X – (1.01) – Y (-6) – Apply, X – (-4.99) – Y (0) – OK.

10. Modelling – Create – Lines – (Create Rectangular Grid behind the aerofoil)

11. Modelling – Create – Arc – By End KPs and Rad – Pick end point – Ok – Pick Centre Point – Ok – Radius – (6) – Ok.

12. Modelling – Create – Areas – By Lines – Pick Lines (Create Rectangular Area behind the Aerofoil and C-Shaped area Before the Aerofoil) – Ok.

Note: Do not Create Area In the Aerofoil.

13. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Upper Curved portion of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

14. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Upper Curved portion of Aerofoil) – Ok – No. Of Element Divisions – 20 – Ok.

15. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Vertical Upper Line in Grid) – Ok – No. Of Element Divisions – 20 – Ok.

16. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Horizontal Line in Grid) – Ok – No. Of Element Divisions – 20 – Ok.

17. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Upper Vertical line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

18. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Lower Vertical line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

19. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Upper Horizontal line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Lower Horizontal line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

20. Meshing – Mesh – Areas – Mapped – 3 Or 4 Sides – Pick (All Areas) – Ok.

21. Preprocessor - Loads – Define Loads – Apply – Fliud CFD – Velocity – On Lines – Pick (C-Section of the Grid) – Ok – Vx – 30 – Vy – 0 – Vz – 0 – Ok.

22. Preprocessor - Loads – Define Loads – Apply – Fliud CFD – Velocity – On Lines – Loop – Pick (Upper and lower Surface of Aerofoil) – Ok – Vx – 0 – Vy – 0 – Vz – 0 – Ok.

23. Preprocessor - Loads – Define Loads – Apply – Fliud CFD – Pressure DOF – On Lines – Pick (All the Three Sides of Rectangular Grid) – Ok – PRES Pressure Value – 0 – Ok.

24. FLOTRAN Set Up – Solution Options - (Leave the Default Settings) – Ok.

25. FLOTRAN Set Up – Execution Ctrl – EXEC Global Iterations – 1000 – Ok.

26. FLOTRAN Set Up – Additional Derived – RFL Out Derived – PTOT, TTOT, PCOE, MACH, RDFL – (Check) Yes – Ok.

27. FLOTRAN Set Up – Fluid Properties – Density – Constant – 1.23 – Velocity – Constant – Ok – Ok.

28. FLOTRAN Set Up – Flow Environment – Ref Conditions – Ok.

SOLUTION

29. Solution – RUN FLOTRAN – (It Takes Some time to Converge) – Solution is done – Close.

POSTPROCESSOR

30. General Postproc – Read Results – Last Set.

31. General Postproc – Plot Results – Control Plot – Nodal Solution – DOF Solution – Pressure – Ok.

32. General Postproc – Plot Results – Control Plot – Nodal Solution – DOF Solution – Fluid Velocity. 15.Read Results – last set

33.Plot results – vector plot – predefined – DOF Soln – Velocity V – ok.

34.Plot results – contour plot – nodal soln – other FLOTRAN quantities – total stagnation pressure –ok.

35.Plot Results – Flow Trace – Defi Trace Pt – Pick three or four points around the inlet region and two or three points in the recirculation region (along the upper wall of the transition region) – ok.

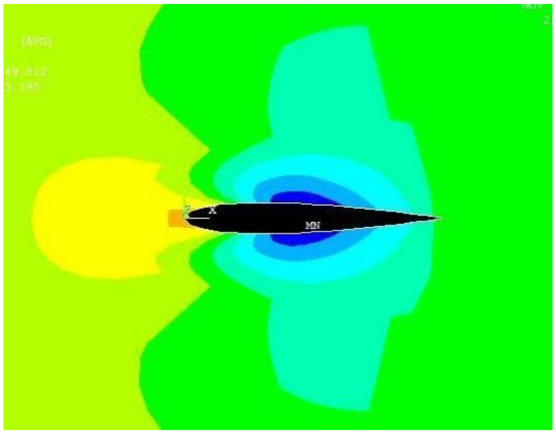
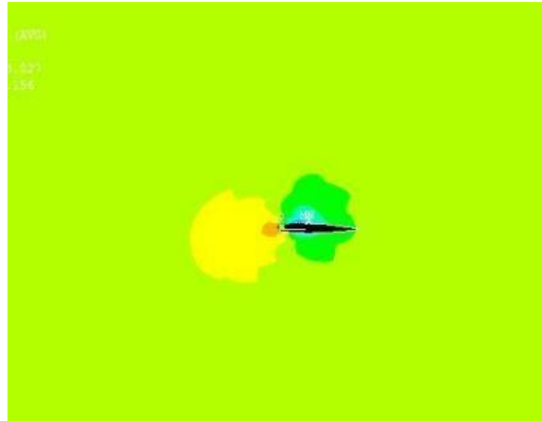
36.Utility Menu – PlotCtrls – Animate – Particle Flow – DOF Solution – Velocity VX – ok.

37.Path Operations – Define Path – By Nodes – Pick the lowest and then the highest point of the outlet – ok – enter Path Name – V – ok – close.

38.Path Operations – Map onto Path – user label for Item – VELOCITY – DOF Solution – VelocityVX – ok.

39.Path Operations – Plot path Item – on Graph – VELOCITY – ok.

40.PlotCntrls-Write Metafile-InvertWhite/Black

PRESSURE DISTRIBUTION OVER SYMMETRIC AEROFOIL

PRESSURE DISTRIBUTION OVER CAMBERED AEROFOIL

Result:

Incompressible and Inviscid flow over Airfoil has been executed. The Pressure Distribution and Velocity vectors over a Symmetric/Cambered Airfoil has been executed using Ansys

VIVA QUESTIONS

1. Define Loads.
2. Define Compressible flow.
3. What is the difference between compressible and incompressible flow.
4. Define Inviscid flow.
5. Define boundary conditions,
6. Write the necessary boundary conditions used for 2D incompressible Inviscid flow over an aerofoil.
7. Define velocity vector.
8. Define the tool FLOTRAN Set Up.
9. Define ANSYS.
10. Define ANSYS FLOTRAN CFD.

EXPERIMENT -4

MODELING OF 2-D COMPRESSIBLE AND VISCOUS FLOW OVER SYMMETRIC/ CAMBERED AEROFOIL AND PLOTTING OF PRESSURE DISTRIBUTION AND VELOCITY VECTORS FOR SUBSONIC/ SUPERSONIC MACHNUMBERS

AIM: Modeling of 2-D compressible and viscous flow over an aerofoil. Computations and analysis for velocity vectors and pressures distributions.

APPARATUS: A computer hardware, software (ANSYS) with a graphical user interface.

PROCEDURE

The three main steps to be involved are

1. Pre Processing

2. Solution

3. Post Processing

Start - All Programs – ANSYS - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

PREPROCESSING

1. Preference – Flotran CFD- h-Method - Ok.

2. Preprocessor - Element type - Add/Edit/Delete – Add – 2D FLOTRAN 141, FLOTRAN 141 – Ok.

3. Material props - Material Models – CFD – Density – 1.23 - Ok.

4. Download Desired Aerofoil Coordinates from Internet. Paste it in the notepad as per the appropriate command – K,NPT,X,Y,Z (Eg- K,1,1,0.00126,0)

5. Write the next command **FLST**, `NFIELD`, `NARG`, `TYPE`, `Otype`, `LENG`

6. Write the next command **FITEM**, NFIELD, ITEM, ITEMY, ITEMZ

7. Write the next command

BSPLIN, P1, P2, P3, P4, P5, P6, XV1, YV1, ZV1, XV6, YV6, ZV6

8. Save the File as Aerofoil_Coordinates.dat

9. File – Read Input From – Select the Aerofoil_Coordinates.dat File – Open.

10. Modelling – Create – Keypoints – In Active Cs – X – 6 – Y – 6 – Apply, X – (6) – Y (-6) – Apply, X – (6) – Y – (0) – Apply, X – (1.01) – Y (6) – Apply, X – (1.01) – Y (-6) – Apply, X – (-4.99) – Y (0) – OK.

10. Modelling – Create – Lines – (Create Rectangular Grid behind the aerofoil)

11. Modelling – Create – Arc – By End KPs and Rad – Pick end point – Ok – Pick Centre Point – Ok – Radius – (6) – Ok.

12. Modelling – Create – Areas – By Lines – Pick Lines (Create Rectangular Area behind the Aerofoil and C-Shaped area Before the Aerofoil) – Ok.

Note: Do not Create Area In the Aerofoil.

13. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Upper Curved portion of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

14. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Upper Curved portion of Aerofoil) – Ok – No. Of Element Divisions – 20 – Ok.

15. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Vertical Upper Line in Grid) – Ok – No. Of Element Divisions – 20 – Ok.

16. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Horizontal Line in Grid) – Ok – No. Of Element Divisions – 20 – Ok.

17. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Upper Vertical line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

18. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Lower Vertical line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

19. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Upper Horizontal line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Lower Horizontal line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

20. Meshing – Mesh – Areas – Mapped – 3 Or 4 Sides – Pick (All Areas) – Ok.

21. Preprocessor - Loads – Define Loads – Apply – Fliud CFD – Velocity – On Lines – Pick (C-Section of the Grid) – Ok – Vx – 30 – Vy – 0 – Vz – 0 – Ok.

22. Preprocessor - Loads – Define Loads – Apply – Fliud CFD – Velocity – On Lines – Loop – Pick (Upper and lower Surface of Aerofoil) – Ok – Vx – 0 – Vy – 0 – Vz – 0 – Ok.

23. Preprocessor - Loads – Define Loads – Apply – Fliud CFD – Pressure DOF – On Lines – Pick (All the Three Sides of Rectangular Grid) – Ok – PRES Pressure Value – 0 – Ok.

24. FLOTRAN Set Up – Solution Options – change the flow type as Compressible – Ok.

25. FLOTRAN Set Up – Execution Ctrl – EXEC Global Iterations – 1000 – Ok.

26. FLOTRAN Set Up – Additional Derived – RFL Out Derived – PTOT, TTOT, PCOE, MACH, RDFL – (Check) Yes – Ok.

27. FLOTRAN Set Up – Fluid Properties – Density – Constant – 1.23 – Velocity – Constant – Ok – Ok.

28. FLOTRAN Set Up – Flow Environment – Ref Conditions – Ok.

SOLUTION

29. Solution – RUN FLOTRAN – (It Takes Some time to Converge) – Solution is done – Close.

POSTPROCESSOR

30. General Postproc – Read Results – Last Set.

31. General Postproc – Plot Results – Control Plot – Nodal Solution – DOF Solution – Pressure – Ok.

32. General Postproc – Plot Results – Control Plot – Nodal Solution – DOF Solution – Fluid Velocity. 15.Read Results – last set

33.Plot results – vector plot – predefined – DOF Soln – Velocity V – ok.

34.Plot results – contour plot – nodal soln – other FLOTRAN quantities – total stagnation pressure –ok.

35.Plot Results – Flow Trace – Defi Trace Pt – Pick three or four points around the inlet region and two or three points in the recirculation region (along the upper wall of the transition region) – ok.

36.Utility Menu – PlotCtrls – Animate – Particle Flow – DOF Solution – Velocity VX – ok.

37.Path Operations – Define Path – By Nodes – Pick the lowest and then the highest point of the outlet – ok – enter Path Name – V – ok – close.

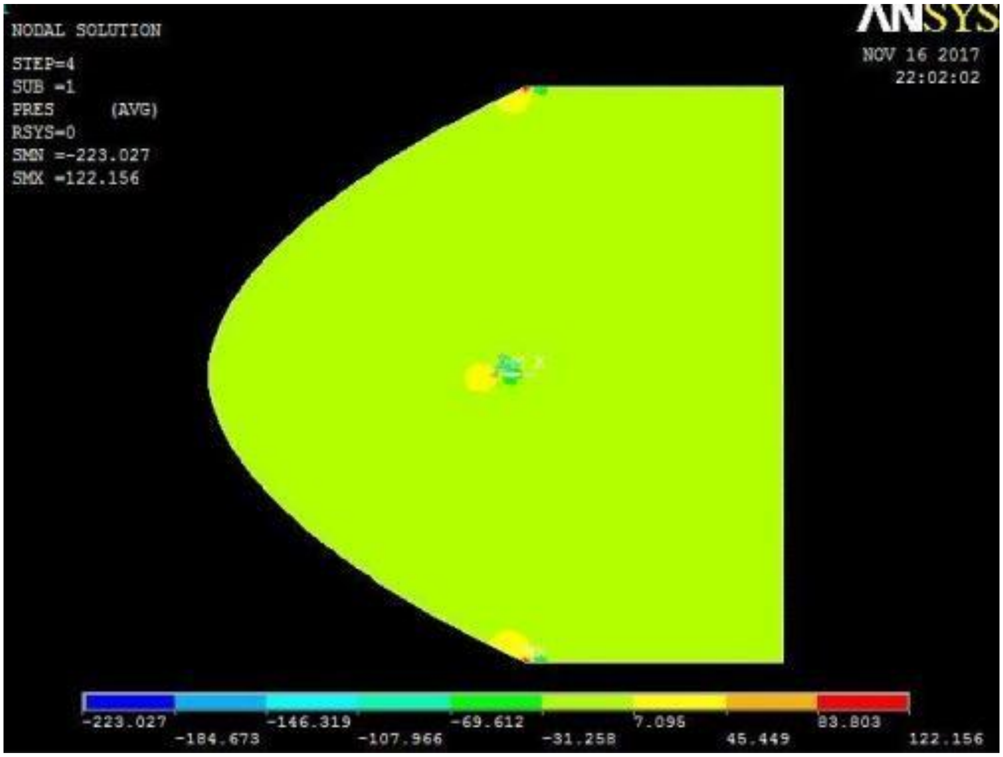
38.Path Operations – Map onto Path – user label for Item – VELOCITY – DOF Solution – VelocityVX – ok.

39.Path Operations – Plot path Item – on Graph – VELOCITY – ok.

40.PlotCntrls-Write Metafile-InvertWhite/Black

Result:

The compressible, viscous flow over an Airfoil is executed. The velocity vectors and pressure distribution has been obtained.



NODAL SOLUTION
STEP=4
SUB =1
PRES (AVG)
RSYS=0
SMN =-223.027
SMX =122.156
ANSYS
NOV 16 2017
22:02:02
-223.027
-184.673
-146.319
-107.966
-69.612
-31.258
7.096
45.449
83.803
122.156

VIVA QUESTIONS

1. Define Viscous Flow.
2. Define Boundary layer.
3. Define Boundary separation.
4. Draw Boundary layer diagram.
5. When can be the flow will be called as incompressible and compressible?
6. What is the governing equation for incompressible inviscid flow?
7. What is the governing equation for compressible viscous flow?
8. How to fix the dimension of the flow field.
9. What is shooting method?
10. Write down the Navier stokes equation for viscous flow

EXPERIMENT NO: 5

GEOMETRIC MODELING AND MESH GENERATION OF A 2-D CONVERGENT-DIVERGENT NOZZLE AND ANALYIS OF FLOW FOR ADIABATIC CONDITIONS

AIM:

To model and generate mesh for a 2-D Convergent-Divergent Nozzle and perform flow analysis for adiabatic conditions.

APPARATUS:

A Computer hardware, Ansys (Software) with a Graphical User Interface.

PROCEDURE:

There are three major steps involved in Ansys, they are

1. Pre Processing
2. Solution
3. Post Processing

Start- All Programs- Ansys- Mechanical APDL Product Launcher- Set the Working Directory as E Drive- Job Name as Roll NO., Ex.No- Click Run.

PREPROCESSING

1. Ansys Main Menu – Preferences-select – FLOTRAN CFD – ok.

2. Element type – Add/Edit/Delete – Add – FLOTRAN CFD – 2D FLOTRAN 141 – ok – close.

3.Modeling – Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 4, 0, 1 – apply –Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 6, 10, 0, 2.5 – ok.

Create – Lines – Lines – Tan to 2 lines – Pick upper line of left rectangle – ok – Pick the tangency end of the first line (upper right corner) – ok – Pick upper line of right rectangle – ok –Pick the tangency end of the first line (upper left corner) – ok – cancel.

4. Create – Area – Arbitrary – Through KPs – Pick 4 corners in counterclockwise order – ok.

5. Utility Menu – Plot – Lines

6. Meshing – Mesh Tool – Lines – set – (pick lines in flow direction along the inlet) – apply – enter 15 as No. of element divisions – enter -2 as Spacing ratio – apply – Pick the top and bottom lines of center area – apply – enter 12 as No. of element divisions – enter 1 as Spacing ratio – apply –Pick the top and bottom lines of outer region – apply – enter 15 as No. of element divisions – enter 3 as Spacing ratio – ok.

7.Meshing – Mesh Tool – Lines – Flip – pick the upper line of Outer region – ok.Meshing – Mesh Tool – Lines – set – Pick the 4 transverse direction lines (vertical lines) – ok –

Enter 10 as No. of element divisions – enter -2 as Spacing ratio – ok.

8. Meshing – Mesh Tool – Mesh Areas – Quad – Mapped – Mesh – pick all – ok.

9. Utility Menu – Plot – Lines

10.Main Menu – Preprocessor – Loads – Define Loads – Apply – Fluid/CFD – Velocity – On Lines– Pick the inlet line (the vertical line at the far left) – ok – VX – 1(value) – VY – 0 – ok.

Main Menu – Preprocessor – Loads – Define Loads – Apply – Fluid/CFD – Velocity – On Lines– Pick the six lines on the top and bottom – ok – VX – 0(value) – VY – 0 – ok.

Main Menu – Preprocessor – Loads – Define Loads – Apply – Fluid/CFD – Pressure DOF – OnLines – Pick the outlet line (vertical line on the far right) – ok – Pressure value – 0 – ok.

SOLUTION

11. FLOTRAN Set Up – Fluid Properties – Density – AIR IN – Viscosity AIR IN – ok – (Usedefault values) – ok.

12. FLOTRAN Set Up – Execution Ctrl – Global Iterations – 40 – ok.

13.FLOTRAN Set Up – Flow Environment – Ref Conditions – reference pressure – 14.7 –nominal, stagnation, and reference temperatures – 70 – temperature offset from absolute – 460 –ok.

14. RUN FLOTRAN – solution done – close.

GENERAL POSTPROCESSING

15. Read Results – last set

16. Plot results – Vector plot – predefined – DOF Solution – Velocity V – ok.

17. Plot results – contour plot – Nodal solution – other FLOTRAN quantities – total stagnation pressure –ok.

18.Plot Results – Flow Trace – Define Trace Pt. – Pick three or four points around the inlet region and two or three points in the recirculation region (along the upper wall of the transition region) – ok.

19.Utility Menu – PlotCtrls – Animate – Particle Flow – DOF Solution – Velocity VX – ok.

20.Path Operations – Define Path – By Nodes – Pick the lowest and then the highest point of the outlet – ok – enter Path Name – V – ok – close.

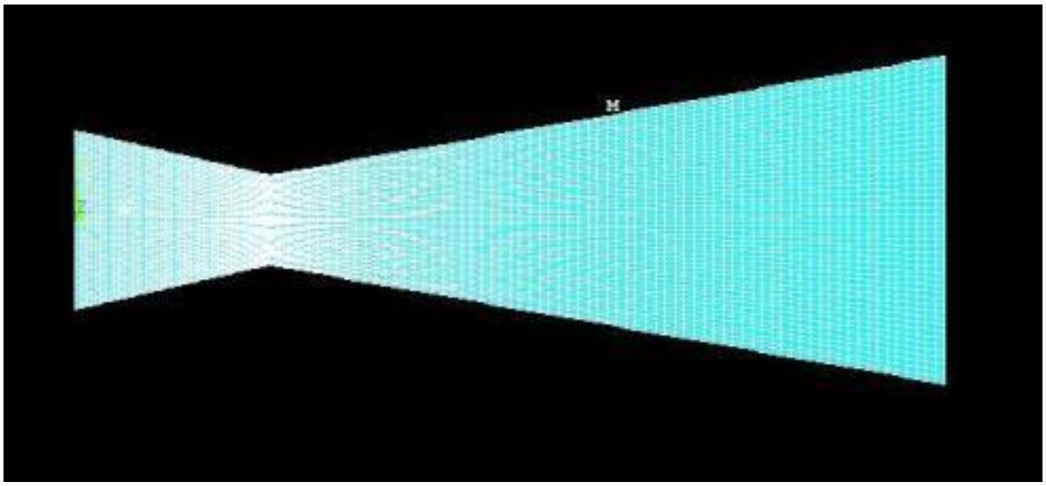
21.Path Operations – Map onto Path – user label for Item – VELOCITY – DOF Solution – Velocity VX – ok.

22. Path Operations – Plot path Item – on Graph – VELOCITY – ok.

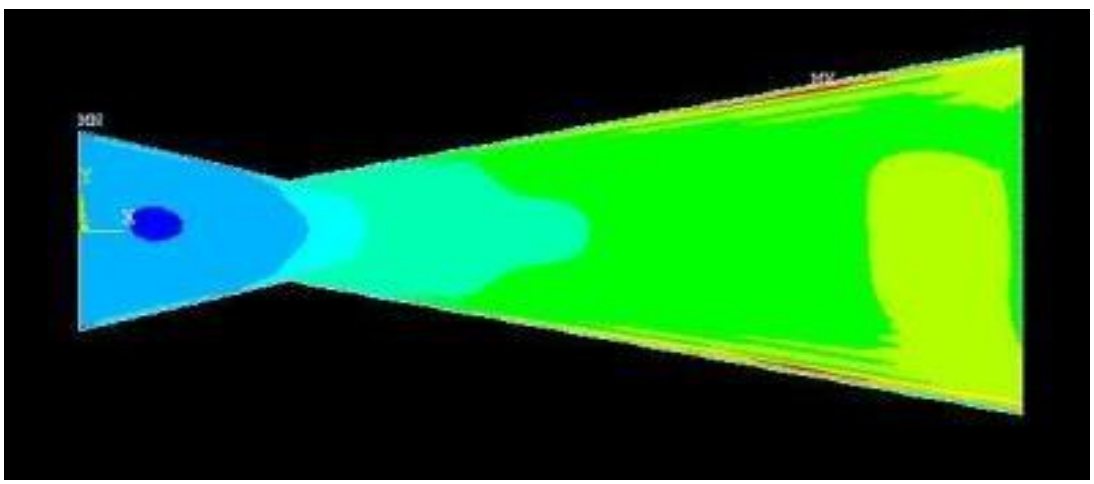
23. PlotCtrls-Write Metafile-Invert White/Black

RESULT:

Geometric Modeling and Mesh Generation of 2-D Convergent Divergent Nozzle has been done and flow analysis under adiabatic conditions has been executed.



Geometric Modelling and Mesh Generation of a 2-D Convergent Divergent Nozzle



Mach Number Distribution throught out the Convergent Divergent Nozzle

VIVA QUESTIONS

1. Define CD Nozzle?
2. What is the significance of CD Nozzle?
3. What are the necessary Boundary Conditions Required for CD Nozzle Analysis.?
4. Explain under expansion, over expansion and optimum expansion?
5. Define the point or region at which the flow velocity reaches Mach one?
6. What is method of characteristics?
7. Define area ratio and explain its significance in CD nozzle?
8. Explain the behavior of flow in a CD nozzle during subsonic and supersonic flow conditions?
9. Draw the pressure variation along the length of CD nozzle for different Mach numbers?
10. Explain the variation of mass flow with exit pressure?

EXPERIMENT NO: 6

GRID GENERATION ON FORE PORTION OF A SPACECRAFT MODEL

AIM:

To model and generate mesh for a grid generation on fore portion of a spacecraft model.

Apparatus: A computer hardware, software (ANSYS) with a graphical user interface.

PROCEDURE

The three main steps to be involved are

1. Pre-Processing

2. Solution

3. Post Processing

Start - All Programs – ANSYS - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

PREPROCESSING

1. Preference – Flotran CFD- h-Method - Ok.

2. Preprocessor - Element type - Add/Edit/Delete – Add – 2D FLOTRAN 141, FLOTRAN 141 – Ok.

3. Material props - Material Models – CFD – Density – 1.23 - Ok.

4. Select the K, NPT,X,Y,Z (Eg- K,1,1,0.00126,0)

5. Write the next command **FLST**, NFIELD, NARG, TYPE, Otype, LENG

6. Write the next command **FITEM**, NFIELD, ITEM, ITEMY, ITEMZ

7. Write the next command
BSPLIN, P1, P2, P3, P4, P5, P6, XV1, YV1, ZV1, XV6, YV6, ZV6

8. Save the File as Aerofoil_Coordinates.dat or (refer APPENDIX)

9. File – Read Input From – Select the Aerofoil_Coordinates.dat File – Open.

10. Modelling – Create – Keypoints – In Active Cs – X – 6 – Y – 6 – Apply, X – (6) – Y (-6) – Apply, X – (6) – Y – (0) – Apply, X – (1.01) – Y (6) – Apply, X – (1.01) – Y (-6) – Apply, X – (-4.99) – Y (0) – OK.

10. Modelling – Create – Lines – (Create Rectangular Grid behind the aerofoil)

11. Modelling – Create – Arc – By End KPs and Rad – Pick end point – Ok – Pick Centre Point – Ok – Radius – (6) – Ok.

12. Modelling – Create – Areas – By Lines – Pick Lines (Create Rectangular Area behind the Aerofoil and C-Shaped area Before the Aerofoil) – Ok.

Note: Do not Create Area In the Aerofoil.

13. Meshing – Size Ctrls – Manual Size – Lines – Picked Lines – Pick (Upper Curved portion of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

14. Meshing – Size Ctrls – Manual Size – Lines – Picked Lines – Pick (Upper Curved portion of Aerofoil) – Ok – No. Of Element Divisions – 20 – Ok.

15. Meshing – Size Ctrls – Manual Size – Lines – Picked Lines – Pick (Vertical Upper Line in Grid) – Ok – No. Of Element Divisions – 20 – Ok.

16. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Horizontal Line in Grid) – Ok – No. Of Element Divisions – 20 – Ok.

17. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Upper Vertical line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

18. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Lower Vertical line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

19. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Upper Horizontal line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Lower Horizontal line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

20. Meshing – Mesh – Areas – Mapped – 3 Or 4 Sides – Pick (All Areas) – Ok.

21. PlotCtrls-Write Metafile-InvertWhite/Black

VIVA QUESTIONS

1. What are the modes of Heat Transfer?
2. What is Aerodynamic Heating?
3. What is Ablative Heating?
4. What are the element types which we are using for Thermal Analysis in Ansys APDL?
5. What is Heat Flux?
6. What are the use of FINS?
7. What is the difference between Fine Mesh and Coarse mesh?
8. Explain Fourier's Law of Heat Conduction?
9. What is Conjugate Heat Transfer?
10. Provide examples for heat transfer in Aerospace Applications?

EXPERIMENT NO: 7

HIGH SPEED FLOW ANALYSIS PAST BLUNT OBJECT IN PRESENCE OF A BOW SHOCK WAVE USING ANSYS

AIM:

To model and generate mesh for a grid generation on fore portion of a spacecraft model.

Apparatus: A computer hardware, software (ANSYS) with a graphical user interface.

PROCEDURE

The three main steps to be involved are

1. Pre-Processing

2. Solution

3. Post Processing

Start - All Programs – ANSYS - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

PREPROCESSING

1. Preference – Flotran CFD- h-Method - Ok.

2. Preprocessor - Element type - Add/Edit/Delete – Add – 2D FLOTRAN 141, FLOTRAN 141 – Ok.

3. Material props - Material Models – CFD – Density – 1.23 - Ok.

4. Select the K, NPT,X,Y,Z (Eg- K,1,1,0.00126,0)

5. Write the next command **FLST**, NFIELD, NARG, TYPE, Otype, LENG

6. Write the next command **FITEM**, NFIELD, ITEM, ITEMY, ITEMZ

7. Write the next command
BSPLIN, P1, P2, P3, P4, P5, P6, XV1, YV1, ZV1, XV6, YV6, ZV6

8. Save the File as Aerofoil_Coordinates.dat or (refer APPENDIX)

9. File – Read Input From – Select the Aerofoil_Coordinates.dat File – Open.

10. Modelling – Create – Keypoints – In Active Cs – X – 6 – Y – 6 – Apply, X – (6) – Y (-6) – Apply, X – (6) – Y – (0) – Apply, X – (1.01) – Y (6) – Apply, X – (1.01) – Y (-6) – Apply, X – (-4.99) – Y (0) – OK.

10. Modelling – Create – Lines – (Create Rectangular Grid behind the aerofoil)

11. Modelling – Create – Arc – By End KPs and Rad – Pick end point – Ok – Pick Centre Point – Ok – Radius – (6) – Ok.

12. Modelling – Create – Areas – By Lines – Pick Lines (Create Rectangular Area behind the Aerofoil and C-Shaped area Before the Aerofoil) – Ok.

Note: Do not Create Area In the Aerofoil.

13. Meshing – Size Ctrls – Manual Size – Lines – Picked Lines – Pick (Upper Curved portion of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

14. Meshing – Size Ctrls – Manual Size – Lines – Picked Lines – Pick (Upper Curved portion of Aerofoil) – Ok – No. Of Element Divisions – 20 – Ok.

15. Meshing – Size Ctrls – Manual Size – Lines – Picked Lines – Pick (Vertical Upper Line in Grid) – Ok – No. Of Element Divisions – 20 – Ok.

16. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Horizontal Line in Grid) – Ok – No. Of Element Divisions – 20 – Ok.

17. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Upper Vertical line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

18. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Lower Vertical line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

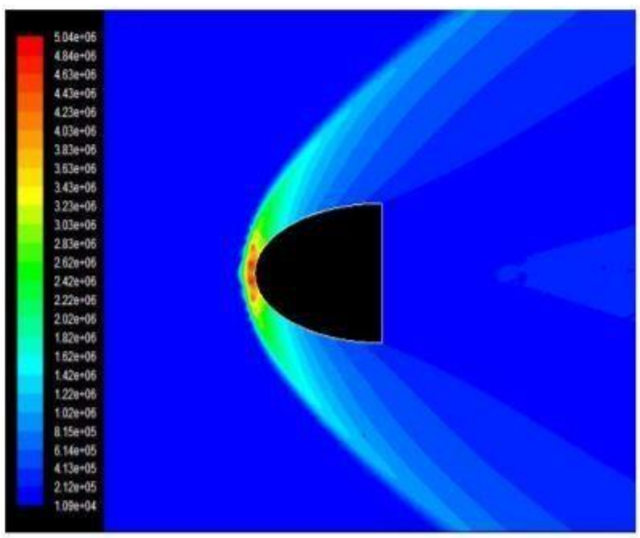
19. Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Upper Horizontal line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

Meshing – Size Cntrls – Manual Size – Lines – Picked Lines – Pick (Lower Horizontal line at the End of Grid) – Ok – No. Of Element Divisions – 20 – Ok.

20. Meshing – Mesh – Areas – Mapped – 3 Or 4 Sides – Pick (All Areas) – Ok.

21. PlotCtrls-Write Metafile-InvertWhite/Black

PRESSURE



VIVA QUESTIONS

1. What are the modes of Heat Transfer?
2. What is Aerodynamic Heating?
3. What is Ablative Heating?
4. What are the element types which we are using for Thermal Analysis in Ansys APDL?
5. What is Heat Flux?
6. What are the use of FINS?
7. What is the difference between Fine Mesh and Coarse mesh?

EXPERIMENT NO-8

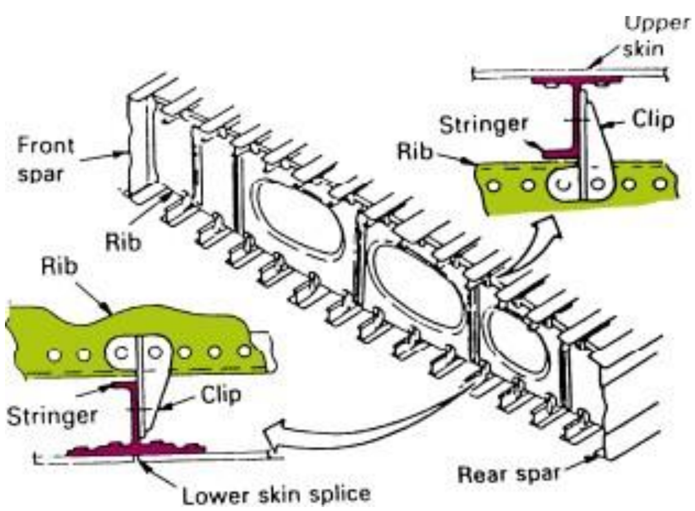
STRUCTURAL MODELING OF A 3-D WING

AIM:

Structural modelling of Wing torsion Box and Analysis of Stresses.

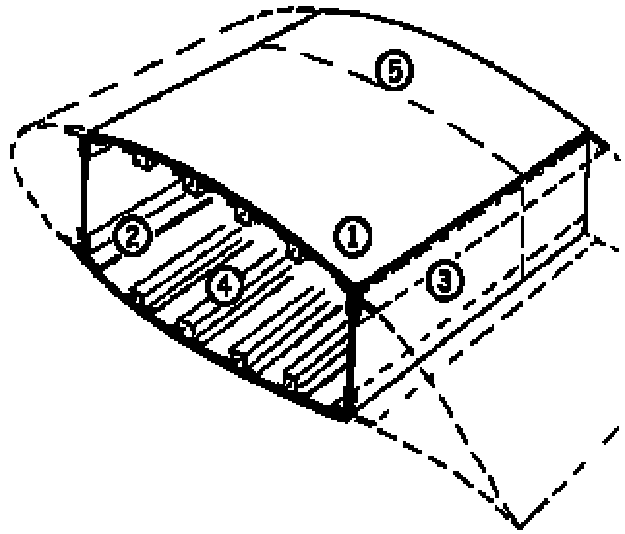
THEORY:

A torsion box consists of two thin layers of material (Skin) on either side of a lightweight core, usually a grid of beams. It is designed to resist torsion under an applied load. A hollow core door is probably the most common example of a torsion box. The aircraft wing torsion box consists of Skin, Stringers, Ribs, and Spars.



Cross section of Wing

Torsion box with a single spar is having low resistance against torsion. Torsion box which is having two spars will have differential bending, but spar will give better resistance.



1. **Thicker Skin:** Takes up Aerodynamic forces, Partially takes over the role of Spar Caps(bending function)
2. **Degenerated Spar Caps**
3. **Thicker web**
4. **Stringers:** Support the skin
5. **Ribs:** Provides Aerodynamic Shape

APPARATUS:

A Computer hardware, Ansys (Software) with a Graphical User Interface.

PROCEDURE:

There are three major steps involved in Ansys, they are

1. Pre Processing
2. Solution
3. Post Processing

Start- All Programs- Ansys- Mechanical APDL Product Launcher- Set the Working Directory as E Drive- Job Name as Roll NO., - Ex.No- Click Run.

PREPROCESSING:

1. Preference - Structural- h-Method - Ok.

2. Preprocessor - Element type - Add/Edit/Delete – Add – Shell, 3D 4node181 – Ok

3. Material props - Material Models – Structural – Linear – Elastic – Isotropic - EX 2e11, PRXY-0.3 - Ok.

4. Plots – Multi Plots.

5. Modelling – Create – Lines – Straight Lines – (Randomly create three Desired Lines by Picking Keypoints to Construct spars in the leading edges and trailing edge).

6. Modelling – Opearte – Boolean – Divide – Line By Line – Select Lines (to be divided - Upper and Lower surface of airfoil) – Apply – Select Lines (Used to Divide - Right and Left side lines in Both leading and Trailing edges) – Ok.

7. Modelling – Copy – Lines – Pick All – Z Axis – (-1).

8. Modelling – Create – Areas – By Skinning – Pick Lines (One by One Create Area) – Ok.

9. Sections – Shell – Lay-Up – Add/Edit – Thickness – 0.05 – Ok.

10. Meshing – Size Cntrls – Lines – Picked Lines – (Pick All lines in your Nose section) – No of Element Divisions – 15 – Ok.

11. Meshing – Mesh Tool – (Check) Mappped – 3 or 4 Sided – Ok. (Ignore Shape Violating Warnings).

12. Coupling/Ceqn – Coincident Nodes – Ok.

SOLUTION

7. Solution – Define Loads – Apply – Structural – Displacement - On Nodes – (Check) Box – (Select all the nodes in one side of the wing in order to make in Wing Root) – Ok - All DOF - Ok.

8. Solution – Define Loads – Apply – Structural – Pressure - Areas – (Select all Lower Surface of Wing) – 3e5 – Ok.

14. Solution – Define Loads – Apply – Structural – Pressure - Areas – (Select all Upper Surface of Wing) – -1e5 – Ok.

8. Solve – Current LS – Ok – Solution is done – Close.

POST PROCESSING

9. General post proc - Plot Result - Contour plot - Nodal Solution – Stress - Von Mises stress - Ok.

TO VIEW THE ANIMATION

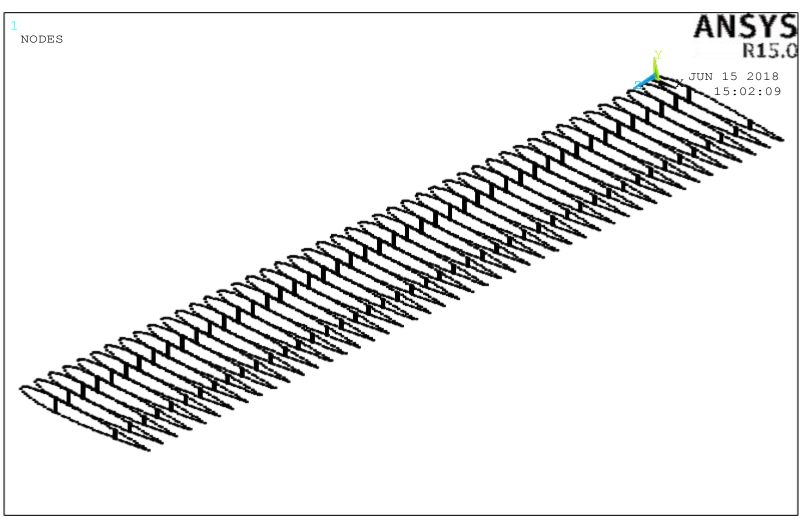
10. Plot control – Animates - Mode Shape – Stress - Von Mises - Ok.

11. Plot control – Animate - Save Animation - Select the proper location to save the file (E drive-user) - Ok.

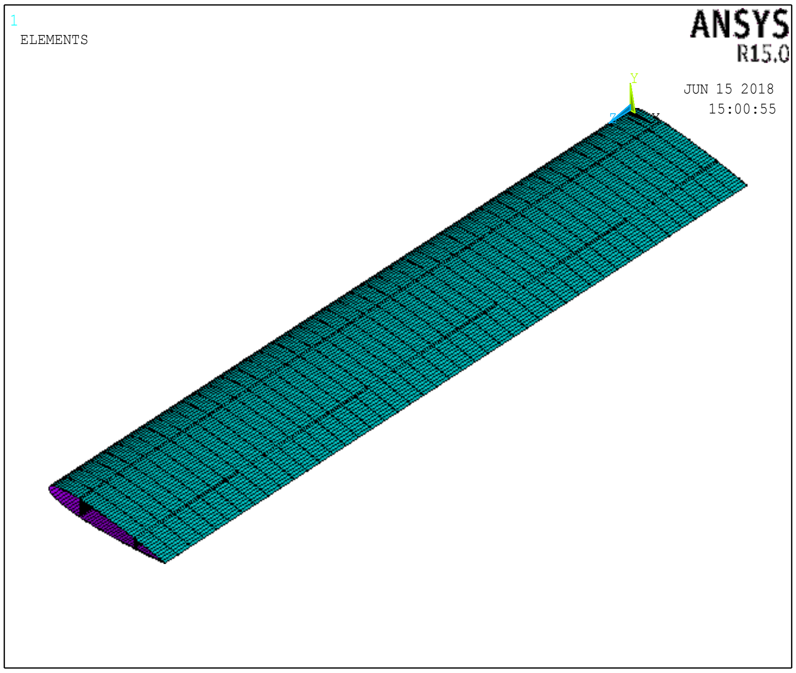
12. PlotCtrls-Write Metafile-Invert White/Black

RESULT:

Structural Modeling and Stress analysis of Torsion Box has been executed.



STRUCTURAL MODELING OF TORSION BOX IN ANSYS



MESHED VIEW OF WING TORSION BOX

VIVA QUESTION

1. Define Wing.
2. What are the Structural Components present in the Wing?
3. What are the different configuration of wing?
4. What are the different types of loads acting in the wing?
5. What are the different types of Wing Construction?
6. Define the tool ‘COPY’.
7. Define the the tool ‘Boolean’
8. Define the Stiffener.
9. Write the difference between stiffener and Stringer.
10. Which type of load is carried by the stiffener and stringer

EXPERIMENT NO: 9

STRUCTURAL MODELING OF A FUSELAGE BULKHEAD OF A SPACECRAFT

AIM:

To perform structural modeling and stress analysis of a Fuselage Frame.

APPARATUS:

A Computer hardware, Ansys (Software) with a Graphical User Interface.

PROCEDURE:

There are three major steps involved in Ansys, they are

1. Pre Processing
2. Solution
3. Post Processing

Start- All Programs- Ansys- Mechanical APDL Product Launcher- Set the Working Directory as E Drive- Job Name as Roll NO., - Ex.No- Click Run.

PREPROCESSING:

1. Ansys Utility Menu

File – clear and start new – do not read file – ok – yes.

2. Ansys Main Menu – Preferences select – STRUCTURAL – ok

3. Preprocessor

Element type – Add/Edit/Delete – Add – Shell – 3D 4node 181 – Ok – Add – Structural Mass – 3D mass 21 – Ok – Close.

Real constants – Add – MASS21 – ok – MASSX, MASSY, MASSZ – 1e-20 – Ok – Close.

Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 2e11

PRXY – 0.3 – ok – close.

4. Sections – Lay Up – Add/Edit – Thickness – 1e-5 – Ok.

5. Modelling – Create – Keypoints – In Active CS – x(0) – y(0) – z() – x(0.0381) – y(0) – z()– x(-0.0381) – y(0) – z() – x(-0.0381) – y(0.683) – z()– x(0.0381) – y(0) – z() – x(0.0381) – y(0.683) – z() – x(0.000412) – y(0) – z() – x(-0.000412) – y(0) – z() – x(-0.000412) – y(0.7846) – z() – x(0.000412) – y(0.7846) – z() – x(0.0381) – y(0) – z() – x(-0.0381) – y(0) – z() – x(0.0381) – y(0.78562) – z()

6. Modelling – Create – Lines – (Zoom In to Appropriate Level) – Create 'I' Section.

7.Modelling – operate – Extrude – Lines – About Axis – Pick All – Ok – Pick (Keypoints Near Axis) – Ok – ARC – 360 – NSEG – 5 – Ok.

8. Meshing – Size Cntrls – Lines – Picked Lines – Pick (Lines in 'I' Section of any Segment) – Ok – NDIV – 15 – Ok.

9. Meshing – Mesh Tool – Mapped (Check) – 3 Or 4 Sided – Pick All – Ok.

10. Meshing – Mesh Tool – Mesh – Keypoints – Mesh – Pick (Keupoint in the Axis (0,0,0)) – Ok.

11. Meshing – Mesh Attributes – Default Attributes – Element Type Number – 2 MASS21 – Ok.

12. Coupling/Ceqn – Rigid Region – Pick (Select the Node at the Centre of Axis Or You can type the particular Node Number in the box) – Apply – Top View – (Check)Box – Pick (Select the nodes in the right hand Side) – (Check) Single – Pick (Select the Node at the Axis Or You can type the particular Node Number in the box) – Ok – Ok.

13. Loads – Define Loads – Apply – Structural – Displacement – On Nodes – (Check) Box – Top View – Pick (Select the Left Hand Side Nodes) – DOFs to be Constrained – ALL DOF – Ok.

14. Loads – Define Loads – Apply – Structural – Force/Moment – On Nodes – Pick (Select the Node at the Centre of Axis) – Direction of Force/Mom – Fy – Value of Force/Mom – (-10000) – Ok.

15. Loads – Define Loads – Apply – Structural – Force/Moment – On Nodes – Direction of Force/Mom – Mx – Value of Force/Mom – (1000) – Ok.

SOLUTION

16. Solve – Current L.S. – Ok – Yes – Close.

POSTPROCESSOR

In General Postproc

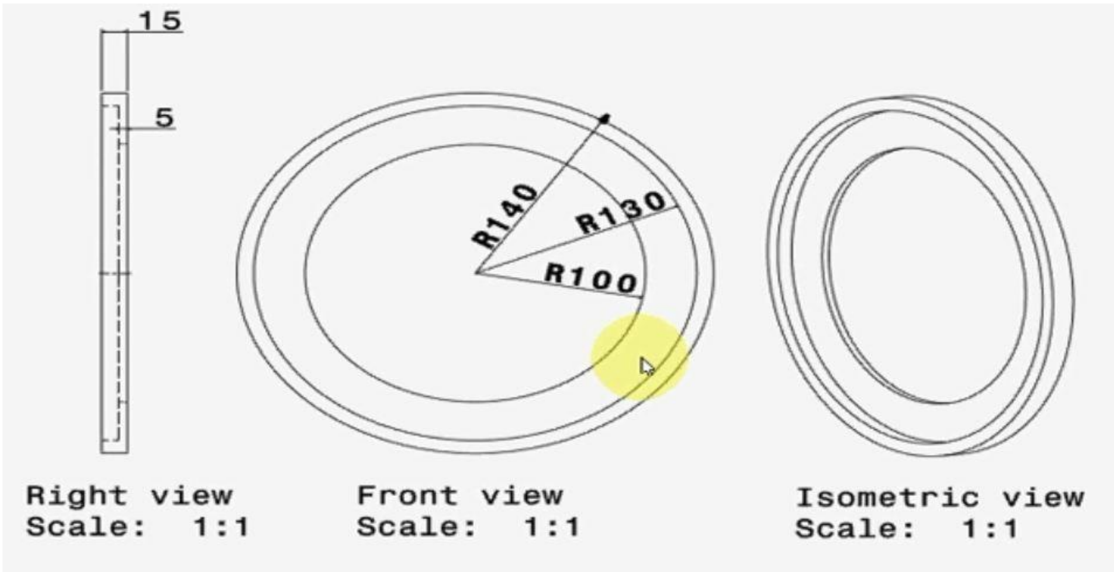
17. Read Results – Last Set.

18. Plot Results – Contour Plot – Nodal Solution – Displacement Vector Sum – Ok.

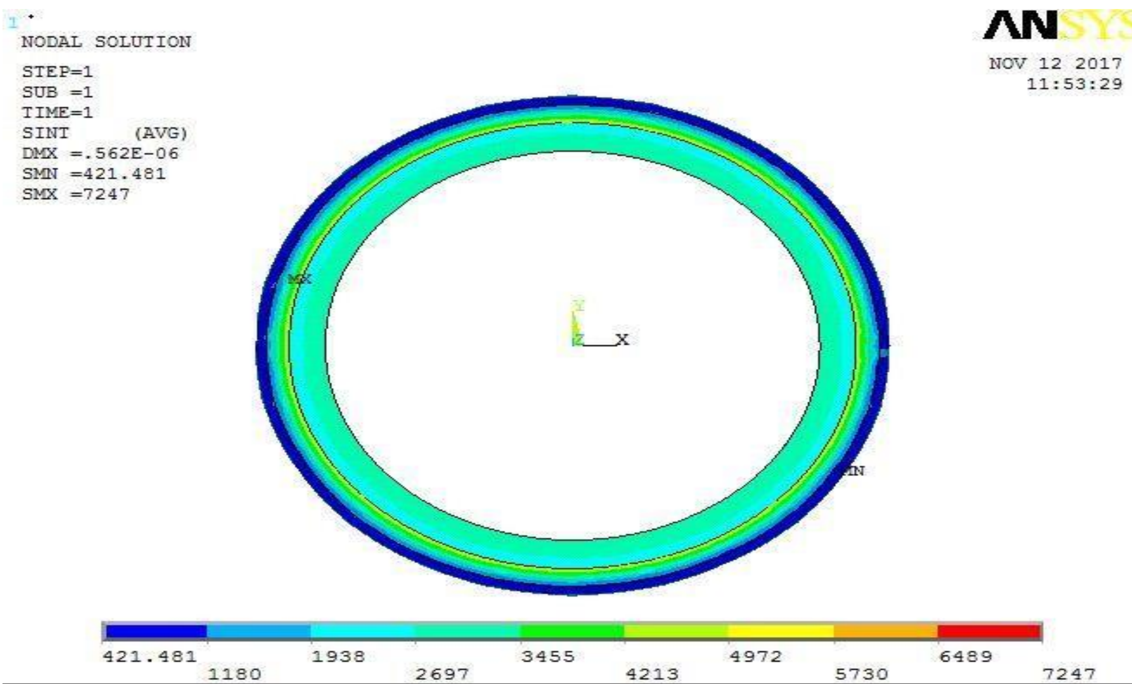
18. Plot Results – Contour Plot – Nodal Solution – Stress – Von-Mises Stress – Ok.

RESULT:

Structural Modeling and Stress analysis of a Fuselage Frame is executed in ANSYS.



DIMENSIONS OF FUSELAGE FRAME



STRESS DISTRIBUTION IN FUSELAGE FRAME

VIVA QUESTIONS

1. Define Fuselage.
2. Define Fuselage Bulk Head.
3. What is the uses of Bulk Head?
4. Define the tool ‘Extrude’
5. Define Shell Element.
6. Explain the Element type ‘Shell181’.
7. What is the difference between solid and shell element?
8. Define the Element type ‘Structural Mass’.
9. Which type of loads acting in the Bulk Head.
10. Define area rule

EXPERIMENT NO-10

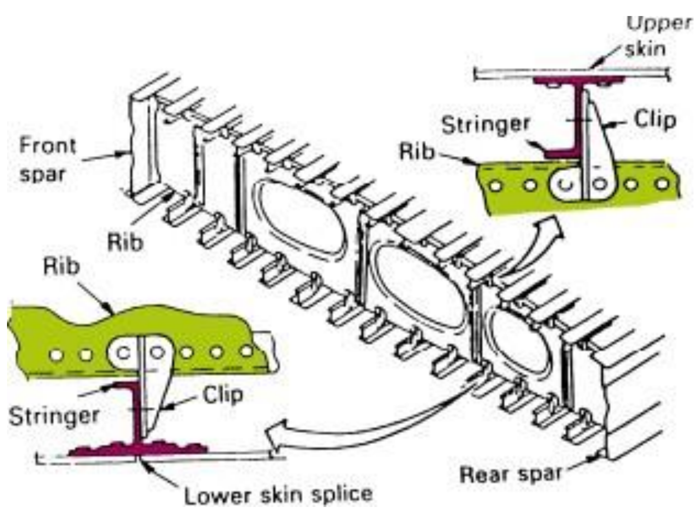
STRUCTURAL MODELING OF A 3-D WING

AIM:

Structural modelling of Wing torsion Box and Analysis of Stresses.

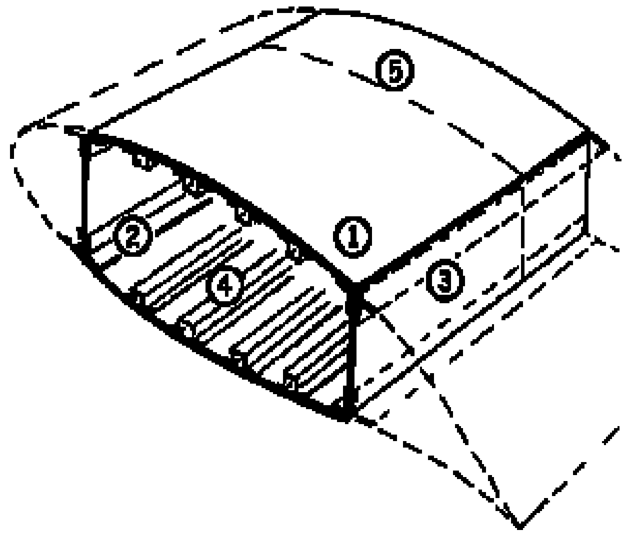
THEORY:

A torsion box consists of two thin layers of material (Skin) on either side of a lightweight core, usually a grid of beams. It is designed to resist torsion under an applied load. A hollow core door is probably the most common example of a torsion box. The aircraft wing torsion box consists of Skin, Stringers, Ribs, and Spars.



Cross section of Wing

Torsion box with a single spar is having low resistance against torsion. Torsion box which is having two spars will have differential bending, but spar will give better resistance.



6. **Thicker Skin:** Takes up Aerodynamic forces, Partially takes over the role of Spar Caps(bending function)
7. **Degenerated Spar Caps**
8. **Thicker web**
9. **Stringers:** Support the skin
10. **Ribs:** Provides Aerodynamic Shape

APPARATUS:

A Computer hardware, Ansys (Software) with a Graphical User Interface.

PROCEDURE:

There are three major steps involved in Ansys, they are

4. Pre Processing
5. Solution
6. Post Processing

Start- All Programs- Ansys- Mechanical APDL Product Launcher- Set the Working Directory as E Drive- Job Name as Roll NO., - Ex.No- Click Run.

PREPROCESSING:

1. Preference - Structural- h-Method - Ok.

2. Preprocessor - Element type - Add/Edit/Delete – Add – Shell, 3D 4node181 – Ok

3. Material props - Material Models – Structural – Linear – Elastic – Isotropic - EX 2e11,PRXY-0.3 - Ok.

4. Plots – Multi Plots.

5. Modelling – Create – Lines – Straight Lines – (Randomly create three Desired Lines by Picking Keypoints to Construct spars in the leading edges and trailing edge).

6. Modelling – Opearte – Boolean – Divide – Line By Line – Select Lines (to be divided - Upper and Lower surface of airfoil) – Apply – Select Lines (Used to Divide - Right and Left side lines in Both leading and Trailing edges) – Ok.

7. Modelling – Copy – Lines – Pick All – Z Axis – (-1).

8. Modelling – Create – Areas – By Skinning – Pick Lines (One by One Create Area) – Ok.

9. Sections – Shell – Lay-Up – Add/Edit – Thickness – 0.05 – Ok.

10. Meshing – Size Cntrls – Lines – Picked Lines – (Pick All lines in your Nose section) – No of Element Divisions – 15 – Ok.

11. Meshing – Mesh Tool – (Check) Mappped – 3 or 4 Sided – Ok. (Ignore Shape Violating Warnings).

12. Coupling/Ceqn – Coincident Nodes – Ok.

SOLUTION

7. Solution – Define Loads – Apply – Structural – Displacement - On Nodes – (Check) Box – (Select all the nodes in one side of the wing in order to make in Wing Root) – Ok - All DOF - Ok.

8. Solution – Define Loads – Apply – Structural – Pressure - Areas – (Select all Lower Surface of Wing) – 3e5 – Ok.

14. Solution – Define Loads – Apply – Structural – Pressure - Areas – (Select all Upper Surface of Wing) – -1e5 – Ok.

8. Solve – Current LS – Ok – Solution is done – Close.

POST PROCESSING

9. General post proc - Plot Result - Contour plot - Nodal Solution – Stress - Von Mises stress - Ok.

TO VIEW THE ANIMATION

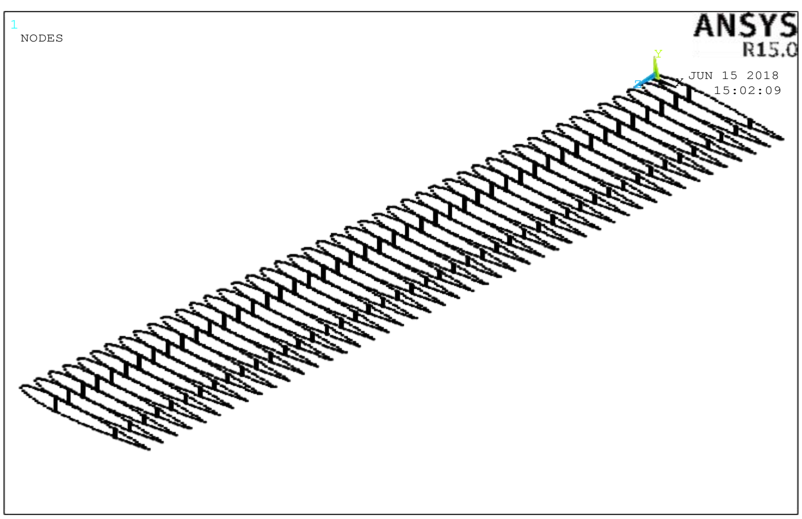
10. Plot control – Animates - Mode Shape – Stress - Von Mises - Ok.

11. Plot control – Animate - Save Animation - Select the proper location to save the file (E drive-user) - Ok.

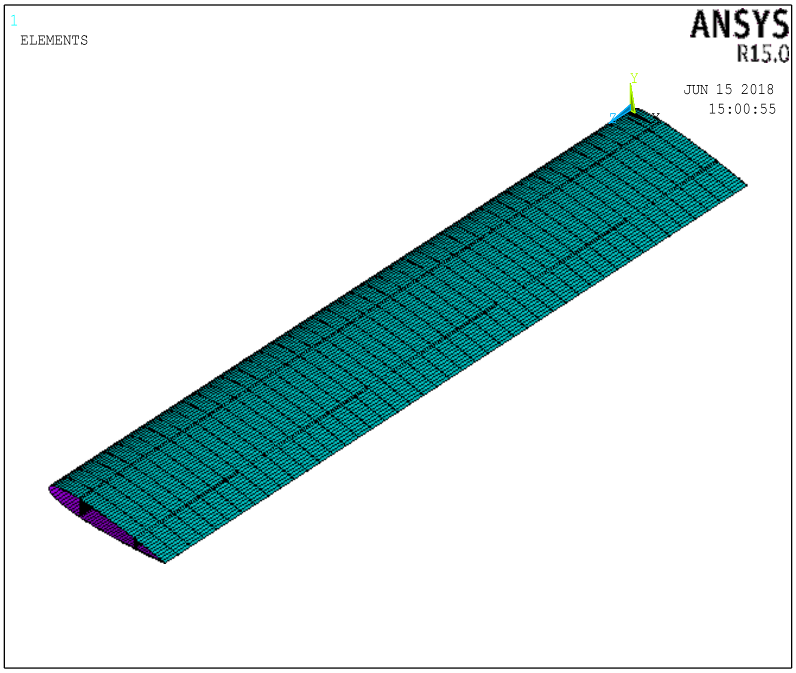
12. PlotCtrls-Write Metafile-Invert White/Black

RESULT:

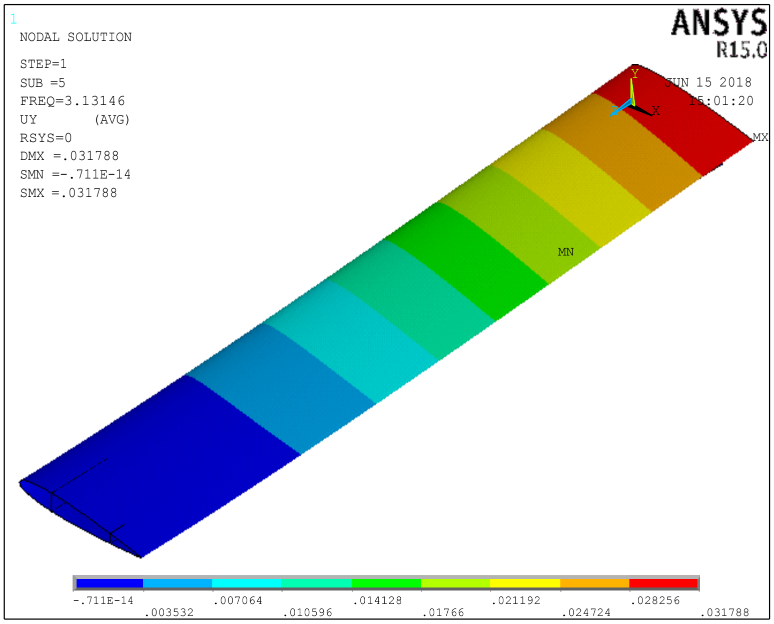
Structural Modeling and Stress analysis of Torsion Box has been executed.



STRUCTURAL MODELING OF TORSION BOX IN ANSYS



MESHED VIEW OF WING TORSION BOX



DISPLACEMENT DUE TO APPLIED LOAD

VIVA QUESTION

11. Define Wing.
12. What are the Structural Components present in the Wing?
13. What are the different configuration of wing?
14. What are the different types of loads acting in the wing?
15. What are the different types of Wing Construction?
16. Define the tool ‘COPY’.
17. Define the the tool ‘Boolean’
18. Define the Stiffener.
19. Write the difference between stiffener and Stringer.
20. Which type of load is carried by the stiffener and stringer

EXPERIMENT NO: 11

STRUCTURAL MODELING OF A FUSELAGE BULKHEAD OF A SPACECRAFT

AIM:

To perform structural modeling and stress analysis of a Fuselage Frame.

APPARATUS:

A Computer hardware, Ansys (Software) with a Graphical User Interface.

PROCEDURE:

There are three major steps involved in Ansys, they are

4. Pre Processing
5. Solution
6. Post Processing

Start- All Programs- Ansys- Mechanical APDL Product Launcher- Set the Working Directory as E Drive- Job Name as Roll NO., - Ex.No- Click Run.

PREPROCESSING:

1. Ansys Utility Menu

File – clear and start new – do not read file – ok – yes.

2. Ansys Main Menu – Preferences select – STRUCTURAL – ok

3. Preprocessor

Element type – Add/Edit/Delete – Add – Shell – 3D 4node 181 – Ok – Add – Structural Mass – 3D mass 21 – Ok – Close.

Real constants – Add – MASS21 – ok – MASSX, MASSY, MASSZ – 1e-20 – Ok – Close.

Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 2e11

PRXY – 0.3 – ok – close.

4. Sections – Lay Up – Add/Edit – Thickness – 1e-5 – Ok.

5. Modelling – Create – Keypoints – In Active CS – x(0) – y(0) – z() – x(0.0381) – y(0) – z()– x(-0.0381) – y(0) – z() – x(-0.0381) – y(0.683) – z()– x(0.0381) – y(0) – z() – x(0.0381) – y(0.683) – z() – x(0.000412) – y(0) – z() – x(-0.000412) – y(0) – z() – x(-0.000412) – y(0.7846) – z() – x(0.000412) – y(0.7846) – z() – x(0.0381) – y(0) – z() – x(-0.0381) – y(0) – z() – x(0.0381) – y(0.78562) – z()

6. Modelling – Create – Lines – (Zoom In to Appropriate Level) – Create ‘I’ Section.

7.Modelling – operate – Extrude – Lines – About Axis – Pick All – Ok – Pick (Keypoints Near Axis) – Ok – ARC – 360 – NSEG – 5 – Ok.

8. Meshing – Size Cntrls – Lines – Picked Lines – Pick (Lines in ‘I’ Section of any Segment) – Ok – NDIV – 15 – Ok.

9. Meshing – Mesh Tool – Mapped (Check) – 3 Or 4 Sided – Pick All – Ok.

10. Meshing – Mesh Tool – Mesh – Keypoints – Mesh – Pick (Keupoint in the Axis (0,0,0)) – Ok.

11. Meshing – Mesh Attributes – Default Attributes – Element Type Number – 2 MASS21 – Ok.

12. Coupling/Ceqn – Rigid Region – Pick (Select the Node at the Centre of Axis Or You can type the particular Node Number in the box) – Apply – Top View – (Check)Box – Pick (Select the nodes in the right hand Side) – (Check) Single – Pick (Select the Node at the Axis Or You can type the particular Node Number in the box) – Ok – Ok.

13. Loads – Define Loads – Apply – Structural – Displacement – On Nodes – (Check) Box – Top View – Pick (Select the Left Hand Side Nodes) – DOFs to be Constrained – ALL DOF – Ok.

14. Loads – Define Loads – Apply – Structural – Force/Moment – On Nodes – Pick (Select the Node at the Centre of Axis) – Direction of Force/Mom – Fy – Value of Force/Mom – (-10000) – Ok.

15. Loads – Define Loads – Apply – Structural – Force/Moment – On Nodes – Direction of Force/Mom – Mx – Value of Force/Mom – (1000) – Ok.

SOLUTION

16. Solve – Current L.S. – Ok – Yes – Close.

POSTPROCESSOR

In General Postproc

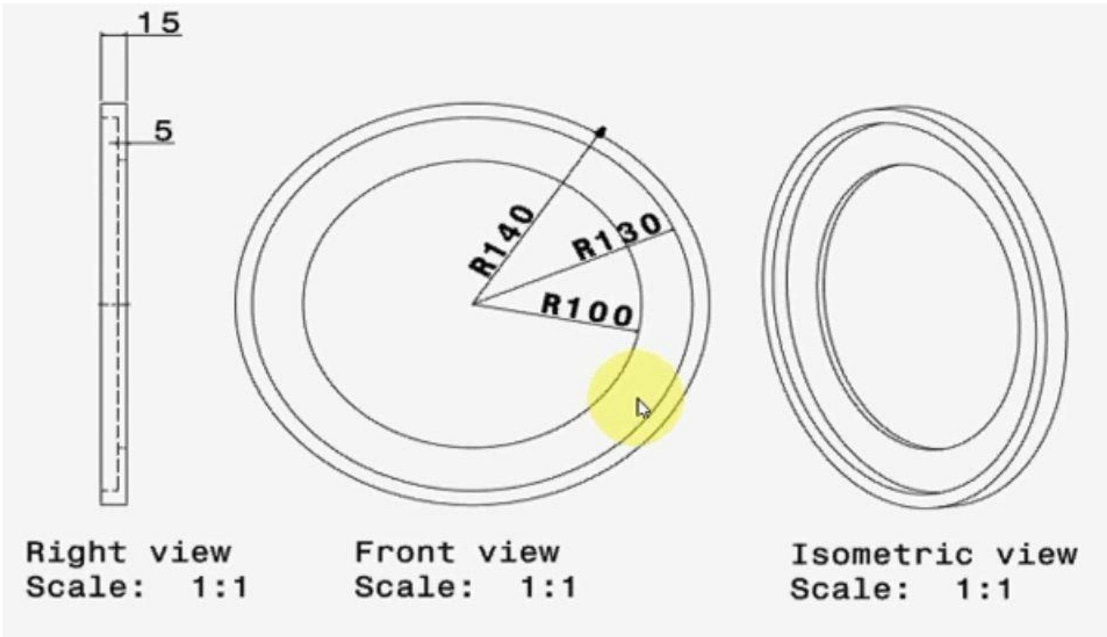
17. Read Results – Last Set.

18. Plot Results – Contour Plot – Nodal Solution – Displacement Vector Sum – Ok.

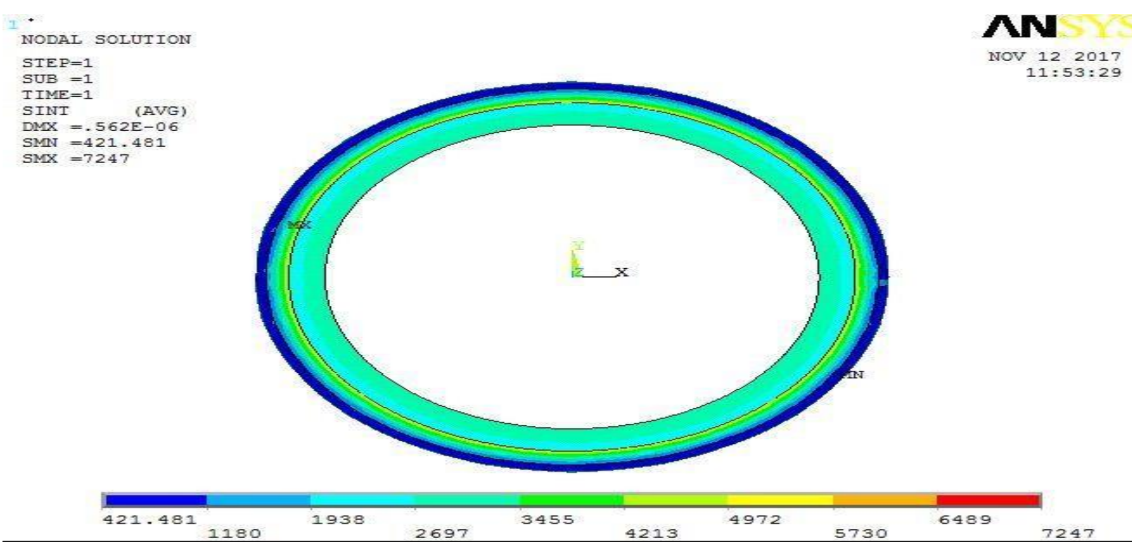
18. Plot Results – Contour Plot – Nodal Solution – Stress – Von-Mises Stress – Ok.

RESULT:

Structural Modeling and Stress analysis of a Fuselage Frame is executed in ANSYS.



DIMENSIONS OF FUSELAGE FRAME



STRESS DISTRIBUTION IN FUSELAGE FRAME

VIVA QUESTIONS

11. Define Fuselage.
12. Define Fuselage Bulk Head.
13. What is the uses of Bulk Head?
14. Define the tool ‘Extrude’
15. Define Shell Element.
16. Explain the Element type ‘Shell181’.
17. What is the difference between solid and shell element?
18. Define the Element type ‘Structural Mass’.
19. Which type of loads acting in the Bulk Head.
20. Define area rule

EXPERIMENT NO: 12

ESTIMATION OF SHEAR STRESS IN A PLATE OF VARYING STIFFNESS UNDER BENDING OR TORSION

AIM:

To determine the stress developed in a Tapered plate of varying thickness with a central hole when subjected to a static load in vertical direction.

APPARATUS:

A Computer hardware, Ansys (Software) with a Graphical User Interface.

PROCEDURE:

There are three major steps involved in Ansys, they are

1. Pre Processing
2. Solution
3. Post Processing

Start- All Programs- Ansys- Mechanical APDL Product Launcher- Set the Working Directory as E Drive- Job Name as Roll NO., Ex.No- Click Run.

PREPROCESSING

1. Preference - Structural- h-Method - Ok.

2. Preprocessor - Element type - Add/Edit/Delete – Add – Solid, 8 node 82 – Ok – Option – Choose Plane stress w/thk - Close.

3. Real constants - Add/Edit/Delete – Add – Ok – THK 0.5 – Ok - Close.

4. Material props - Material Models – Structural – Linear – Elastic – Isotropic - EX 2e5, PRXY 0.3 - Ok.

5. Modeling- Create- Keypoints- Inactive CS- (0,0),(0,50),(100,12..5),(100,37.5)- Ok- Close

6. Modeling- Create- Lines- By Keypoints- Pick all the Keypoints to form a Tapered plate.

7. Modeling- Create- Areas- By lines- Pick all the Lines- Ok

8. Meshing - Mesh Tool – Area – Set - Select the object – Ok - Element edge length 2/3/4/5 – Ok - Mesh Tool -Select TRI or QUAD - Free/Mapped – Mesh - Select the object - Ok.

SOLUTION

1. Solution – Define Loads – Apply – Structural – Displacement - On lines - Select the boundary where is going to be arrested – Ok - All DOF - Ok.

Pressure - On lines - Select the load applying area – Ok - Load PRES valve = 1 N/mm2- Ok.

2. Solve – Current LS – Ok – Solution is done – Close.

POST PROCESSING

1. General post proc - Plot Result - Contour plot - Nodal Solution – Stress - Von Mises stress - Ok.

TO VIEW THE ANIMATION

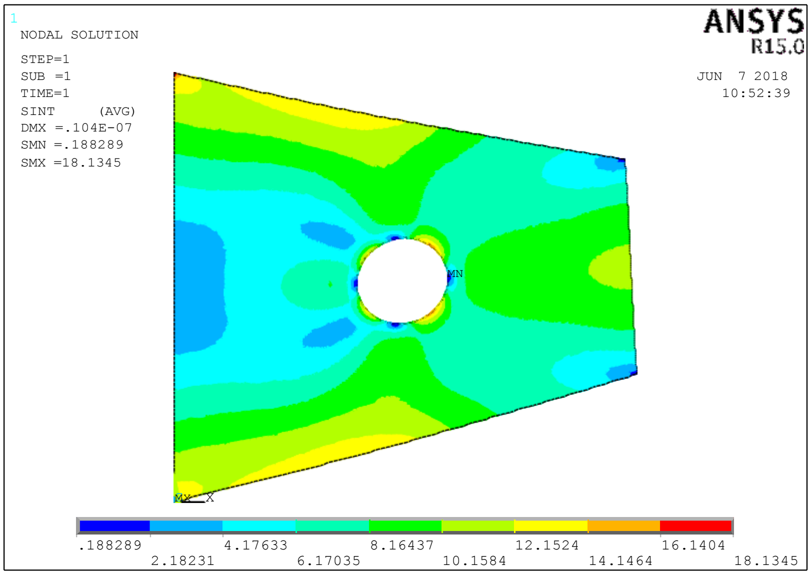
1. Plot control – Animates - Mode Shape – Stress - Von Mises - Ok.

2. Plot control – Animate - Save Animation - Select the proper location to save the file (E drive-user) - Ok.

3. PlotCtrls-Write Metafile-Invert White/Black

RESULT:

The Structural Modeling and Stress analysis of a Tapered plate with a central hole subjected to static vertical load has been executed in Ansys.



Stress Distribution of a Tapered Plate Subjected to a Vertical Load.

VIVA QUESTIONS

1. How we are modeling a tapered plate in Ansys?
2. Define plate.
3. How do you classify Plates?
4. Which theory is used to analyze thin plates and Thick Plates?
5. Provide the assumptions for Classical thin plate theory?
6. Define Anticlastic Bending?
7. How the buckling of Plate is varied from buckling of Columns?
8. Define ultimate compressive strength of a thin plate?
9. Define Effective Width.
10. Provide some applications of Plates in Aircraft industry?

EXPERIMENT NO: 13

FREE AND FORCED VIBRATION OF A STRUCTURAL FRAME

AIM:

To determine the Natural Frequency and Mode shapes of a Cantilever beam under Uniformly Distributed Load.

APPARATUS:

A Computer hardware, Ansys (Software) with a Graphical User Interface.

PROCEDURE:

There are three major steps involved in Ansys, they are

1. Pre Processing
2. Solution
3. Post Processing

Start- All Programs- Ansys- Mechanical APDL Product Launcher- Set the Working Directory as E Drive- Job Name as Roll NO., Ex.No- Click Run.

PREPROCESSING

1. Preference- Structural-h method- Ok

2. Preprocessor- Element type- Add/Edit/Delete-Add- Beam- 2 node 188- Ok

3. Material Props- Material Model- Structural- Linear- Elastic- Isotropic

   The material we are using here is *Steel*

   Provide the value of *EX= 210 GPa (Modulus of Elasticity)*

   Provide the value of *PRXY= 0.3 (Poisson's Ratio)*

   Provide the Value of *Density = 7800* Kg/m^3

4. Preprocessor- Sections- Beams- Common Section- Give ID as “1”- Name as “ Rect” – Subtype Select Rectangular Section- Give the value of B=0.01 m and H = 0.01m. Provide the Value of Nb and Nh as 0. This value defines how to mesh the Section.

5. Preprocessor- Modeling- Keypoints- In Active CS- XYZ Location-0, 0, 0- Apply- Again add one more Keypoint at 1.2,0,0. – Ok

6. Preprocessor- Modeling – Create – Lines- Straight Line- Pick the two key points which were created in the previous step- Line will be created.

7. Preprocessor- Meshing- Mesh Tool- Size Controls- Lines- Set- Pick the line- OK- a new dialogue box appears- put number of Element divisions as “100”- Ok

8. Preprocessor- Meshing- Mesh Tool- Mesh- Pick the line-OK

9. Preprocessor- Loads- Define Loads- Apply – Structural- Displacement- On Keypoints- Select the Keypoint at the Origin- OK- A new dialogue box appears- DOFs to be constrained- ALL DOF- VALUE as “0”-OK

10. Utility Menu- Plot- Nodes

11. Preprocessor- Loads- Define Loads- Apply – Structural- Pressure- On Beams- Select the Entire Nodes Except at the Fixed End- A new Dialogue box appears- Give the value of Pressure at Node I as well as Node J as 1000.

SOLUTION

1. Solution- Analysis Type- New Analysis- Modal

2. Solution- Analysis Type – Analysis Options – No: of Modes to Extract – 6

3. Solution – Analysis Type – Analysis Options – No: of Modes to Expand – 6

A new dialogue box appears provide the value of Start Frequency and End Frequency as “0”

4. Solution – Solve- Current LS

POSTPROCESSING

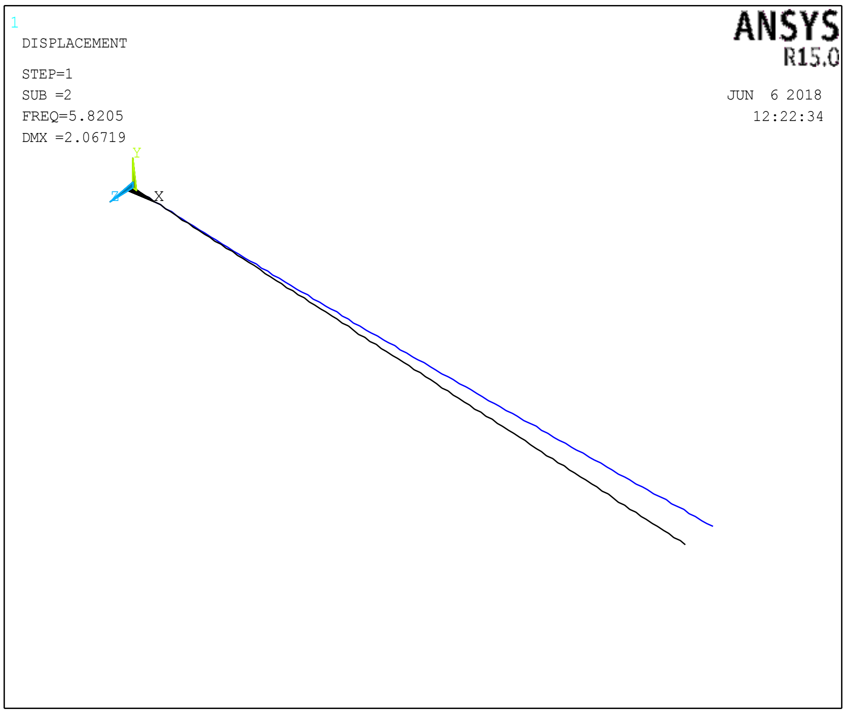
1. General Postproc- Read Results – By Pick- Select one particular Natural Frequency of at which the cantilever beam is vibrating.

2. General Postproc- Plot Results - Deformed Shape- A new dialogue Appears- Select Deformed + Undeformed

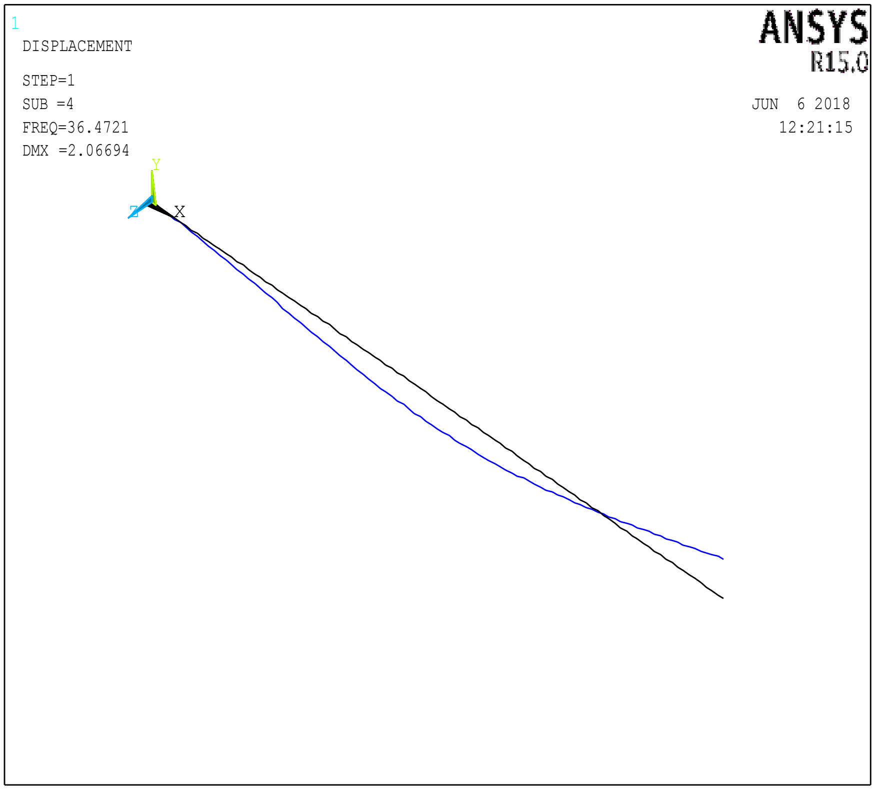
Now we can see the Way the cantilever beam has vibrated at the selected natural frequency. Similarly select each every modes to find out the shapes at which the cantilever beam is vibrating. In this case since the cross-section of the beam is a square there will be two natural frequency be the same.

RESULT:

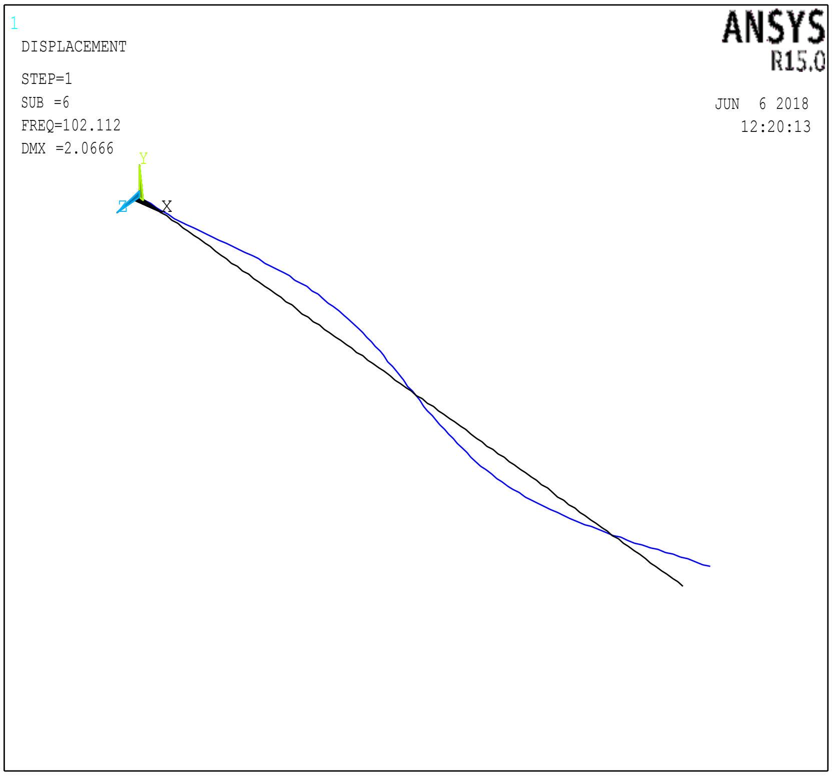
The Natural Frequency and Mode shapes of Cantilever beam has determined.



Mode -1


1
DISPLACEMENT
STEP=1
SUB =4
FREQ=36.4721
DMX =2.06694
ANSYS
R15.0
JUN 6 2018
12:21:15
Y
Z
X

Mode-2


1
DISPLACEMENT
STEP=1
SUB =6
FREQ=102.112
DMX =2.0666
ANSYS
R15.0
JUN 6 2018
12:20:13
Y
Z
X

Mode -3

VIVA QUESTIONS:

1. What is Vibration?
2. Define Mode and Mode Shapes?
3. Difference between Forced and Free Vibration?
4. What is damping and explain its Types?
5. Define Fundamental Mode?
6. What is Transmissibility?
7. What is Damping Ratio and its Significance?
8. What is Magnification Factor?
9. Define Phase Angle?
10. How vibrations can affect the Aircraft?

EXPERIMENT NO: 14

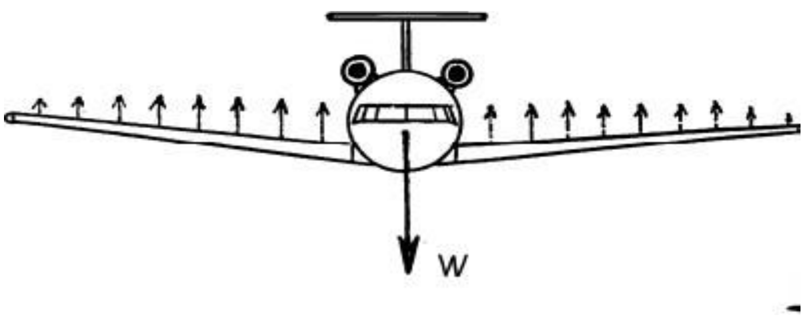
ANALYSIS OF ACTIVE VIBRATION CONTROL IN A SMART MATERIAL

AIM:

To model a tapered I-section spar in Ansys and to compute the stresses acting on the spar in a specific boundary conditions.

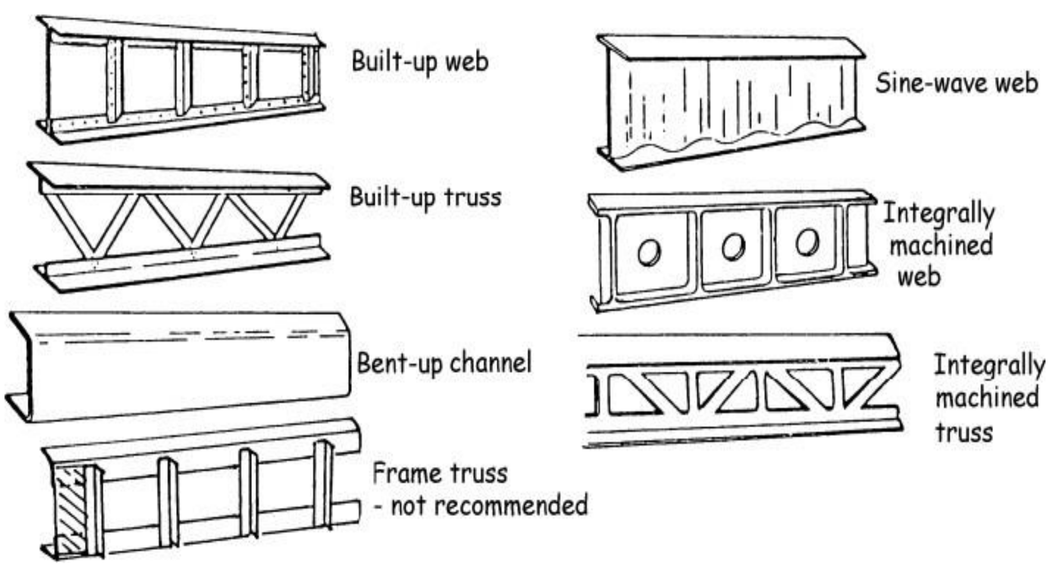
THEORY:

The function of spar is to carry the bending loads.



Aerodynamic Lift Distribution over Wing

The spar of a wing uses I-section, since I-section has better Section Modulus comparing to other sections. The spar consists of Spar Caps/Girders and Spar Web.



Different types of Spars Used in Aircraft Win

APPARATUS:

A Computer hardware, Ansys (Software) with a Graphical User Interface.

PROCEDURE:

There are three major steps involved in Ansys, they are

1. Pre Processing
2. Solution
3. Post Processing

Start- All Programs- Ansys- Mechanical APDL Product Launcher- Set the Working Directory as E Drive- Job Name as Roll NO., Ex.No- Click Run.

PREPROCESSING

1. Preference- Structural-h method- Ok

2. Preprocessor- Element type- Add/Edit/Delete-Add- Beam- 2 node 188- Ok

3. Material Props- Material Model- Structural- Linear- Elastic- Isotropic

The material we are using here is Aluminium 2024-T3,

Provide the value of **EX= 73.3 GPa (Modulus of Elasticity)**

Provide the value of **PRXY= 0.33 (Poisson's Ratio)**

4. Preprocessor- Sections- Beams- Common Section- Give ID as "1"- Name as " Start" – Subtype Select I-Section- Give the value of W1, W2, W3 as 200 and value of thickness as t1,t2,t3 as 6- Apply- Mesh view- Ok

5. Preprocessor- Sections- Beams- Common Section- Give ID as “2”- Name as “ END” – Subtype Select I-Section- Give the value of W1, W2, W3 as 100 and value of thickness as t1,t2,t3 as 2- Apply- Mesh view- Ok

6. Preprocessor- Sections- Beams- Taper Sections- By XYZ Location- In the dialogue box appeared put the Taper Section ID as 3, Section name as “Taper” - Give the Beginning Section ID as “ Start” which we already defined in previous step- Give XYZ for beginning section as 0,0,0- Give the Ending Section ID as “END” which we defined in the previous step-Give the XYZ for the ending section as 1000,0,0

7. Preprocessor- Modeling- Keypoints- In Active CS- XYZ Location-0, 0, 0- Apply- Again add one more Keypoint at 1000,0,0

8. Preprocessor- Modeling – Create – Lines- Straight Line- Pick the two key points which were created in the previous step- Line will be created.

9. Preprocessor- Meshing- Mesh Tool- Element Attributes-Set- Select Section Number- Taper- OK

10. Preprocessor- Meshing- Mesh Tool- Size Controls- Lines- Set- Pick the line- OK- a new dialogue box appears- put number of Element divisions as “100”- Ok

11. Preprocessor- Meshing- Mesh Tool- Mesh- Pick the line-OK

Now the line is meshed.

12. Utility Menu- Plotctrls-Style-Size and Shape- Display of Element should be “ON”

Now you can see the Tapered I- Section

13. Utility Menu- Plot- Keypoints

14. Preprocessor- Loads- Define Loads- Apply – Structural- Displacement- On Keypoints- Select the Keypoint at the Origin- OK- A new dialogue box appears- DOFs to be constrained- ALL DOF- VALUE as “0”-OK

15.Utility Menu- Plot- Lines

16 Preprocessor- Loads- Define Loads- Apply- Pressure- On Beams- Select the Entire Nodes- Give the value of Pressure Node I as 1000 and Node J as 500

In this step we applying a Uniformly Varying Load

SOLUTION

1. Solution- Solve- Current LS

POSTPROCESSING

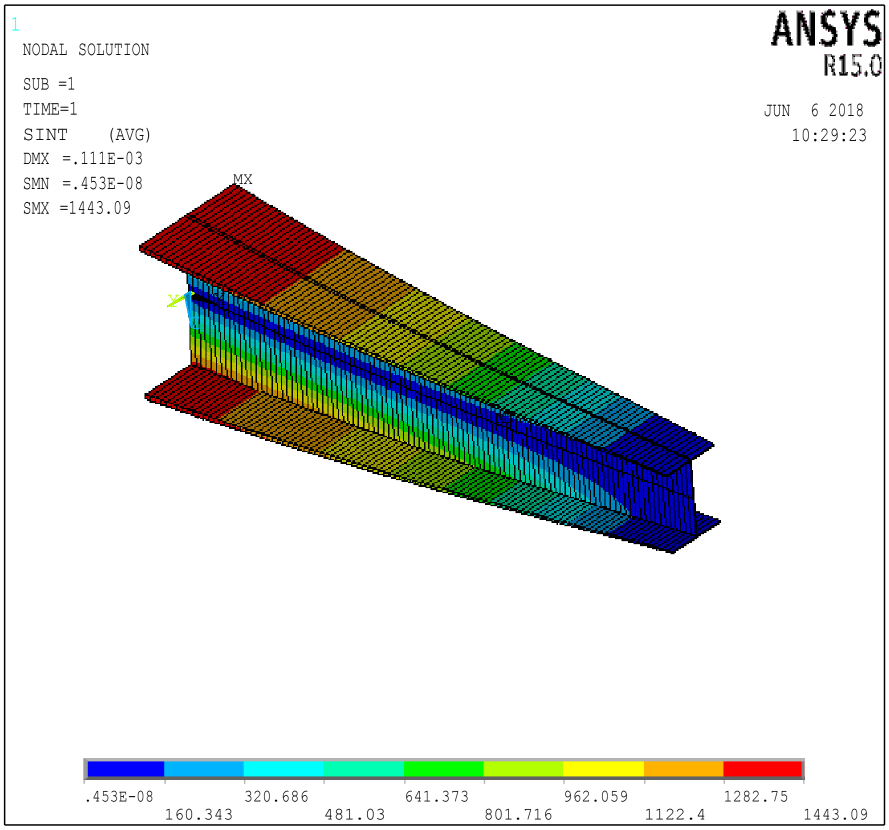
1. General Postproc- Read Results- Last Set

2. General Postproc- Contour Plot- Nodal Solutions- Stress- Stress Intensity- Ok

Now You can view the Stresses acting at different regions of the Spar.

RESULT:

Structural Modeling and Stress analysis of a Tapered I Section Spar has been completed.



Stress Distribution in a Tapered I-section Spar

VIVA QUESTIONS

1. What are the Structural Components of a Wing?
2. What are the function of a Spar?
3. Why I-section is preferably used for Spar Design?
4. What is the significance of Section Modulus?
5. What is bending?
6. Why we are using Beam 188 Elements?
7. What is the difference between Fine Mesh and Coarse mesh?
8. What is unsymmetrical Bending?
9. Define Shear Flow?
10. What is Shear Centre?