



DEPARTMENT OF AERONAUTICAL ENGINEERING

**AE3711-AERO ENGINE AND AIRFRAME
LABORATORY**

List of Experiments

Sl. No	Date	Name of the experiments	Signature
1		Dismantling of an aircraft piston engine.	
2		Assembling of an aircraft piston engine.	
3		Study of Camshaft operation, firing order and magneto, valve timing	
4		Study of lubrication and cooling system	
5		Study of auxiliary systems, pumps and carburetor	
6		Aircraft wood gluing-single & double scarf joints	
7		Preparation of Single/Double Riveted Lap joint	
8		Preparation of Single/Double Riveted butt joint	
9		Sheet metal forming	
10		Tube bending and flaring	

INTRODUCTION TO AIRCRAFT REPAIR

General:

The structure of an aircraft must be strong enough to carry all the loads to which it might be subjected to normal and external conditions.

The airframe structure parts are constructed from a variety of materials and are joined by rivets, bolts, screws and welding etc. They are composed for various parts are stringers, longerons, ribs and bulkheads etc.

The structure members are designed to perform a specific function or to save a definite purpose. The prime objective of an aircraft repair is to restore the damaged part to the original condition. Very often replacement is the only way in which this can be achieved effectively, when repair of the damaged part is possible. First study the part carefully so that the function is carefully understood.

Strength may be the principle requirements in the repair of certain structure while other may need entirely different qualities.

There appearance between streamline shape and accessibility the function of any damaged part member that there are other kind of damage which caused by impact of damage which from foreign object or condition. A rough landing may be over load one of the landing gear causing it to the object.

Inspection of damage:

In visual inspection damage there may be other kinds of damage caused by impact from the foreign object or collision during inspection and sizing up of the repair job. Consider how far the damage caused by pod or collection. They would be classified as no damage. A shock occurring at the one end of the member will be transmitted throughout its run in spite all rivets that are partially and attaching structure along the member. Any evidence of damage, a float examination that is partially fails and for holes which have been elongated. Other kind of damage caused by corrosion or weathering this is known as corrosion damage.

Classification of Damages:

Damages may be classified in to general classes,

1. Negligible damage
2. Damage repairable by patching
3. Damage repaired by insertion
4. Damage necessitating replacement of parts

Negligible Damage:

The damage which does not affect the structural integrity of the member involved or damage which can be corrected by simple procedure without placing flight restriction on the aircraft is called as negligible damage.

Small dents, scratches, cracks or holes that can be repaired by smoothing, stop drilling or hammering out on otherwise repaired without the rise of additional material in classification.

Damage repairable by patching:

Damage repairable by patching is any damage exceeding negligible with or which can be repaired by bridging the damaged parts and components with an essential the spline as patch material used in internal riveted and bolted repair is normally some type of the material but the one is behavior. In patch filler plates of the some gauge and type of material as that in the damaged components may be used for bearings purpose or to (remain) return to the damaged part to its original contour.

Damage Repairable by Insertion:

Damage which can be repaired by cutting away the damaged section and replacing it with a line section, securing an insertion in a spiller at each and is repaired insertion.

Damage Necessitating Replacement of parts:

Replacement of an entire part is considered when one or more of the following condition exists.

- When complicated part has been externally damaged
- When surrounding structure or in accessibility makes repair impractical.
- When damaged part is relatively easy to replace
- When perfect and fittings are damaged beyond the limits is negligible.

Basic Principle of repair by Sheet metal:

Sheet metal works deals with working on the metal of 16-30 gauge, with hand tools and simple Machines. It is one of the major applications in engineering industry.

It has its own significance as useful trade in engineering work. Sheet works such as making a tray, box, funnel, chimney, air duct, fabricate boiler shells and pipe joints from thin or thick plates, etc. are few examples of its application.

For successful working in the trade, one should have a thorough knowledge of projective geometry and development of surfaces.

The most important and first step of repairing a structure changes to the size of the job, making an accurate estimate of member's material strength, thickness and length which have to be regained.

Maintaining Original Length:

If the original length of the structure is to be maintained the patch should have cross sectional area equal to or greater than that of original damaged area if the members subjected to compression or to be bending loads place the splice on the outside of the member to a structure a higher resistance to such if the splice can't be brought or places on the outside of the member use material that is stronger than the material used in original member.

To reduce the possibility of chocks starting from the corner of the member by the make cut outs either circular especially once in shape it is necessary to use rectangular cutouts make the radius of curvature.

Maintaining Original Contour:

From all repairs in such a manner that they will fit the original contour perfectly. A smooth contour is perfectly desirable when making patches on the smooth external parts of high speed aircraft.

Keep Weight to a Minimum:

Keep the weight of the repair to a minimum make the size up of the patch as small as practical and use no more results than as inaccessibility as cases repair disturb the original balance of the structure.

Caution:

If the center of gravity get shifted because of repair or modification of components and should be tested for its balancing.

AIRCRAFT WOOD GLUING – SINGLE SCARF JOINT**EXPT.NO: 1****DATE:****AIM:**

To make an aircraft wood gluing by using single scarf joint.

THEORY:

Scarf joints are often used in boat building. A scarf joint is a long angled joint. The long angle provides a lot of "glue surface" for the joint, making it very strong.

A scarf joint is made by joining two pieces of wood having tapered, beveled, or chamfered ends which over-lap together, as opposed to a butt joint where squared ends of the mating pieces simply butt together. Scarf joints are used to make longer members where single members of sufficient length are not available or are too costly. Both solid wood pieces and sheets of plywood can be scarf joined using epoxy. With the proper cutting and gluing methods, such joints will be amazingly strong, exceeding that of the joining wood members.

With scarf joints, it is preferable for the two pieces to fit as closely as possible for ultimate strength. However, minor irregularities of fit are acceptable since the epoxy will have the ability to bridge these without significant loss of strength.

APPARATUS REQUIRED:

1. Marking tools
2. Measuring tools
3. Bench vice
4. Cutting tools
5. Jack plane
6. Rasp file

MATERIAL REQUIRED:

Soft wood of size 300x50x50 mm

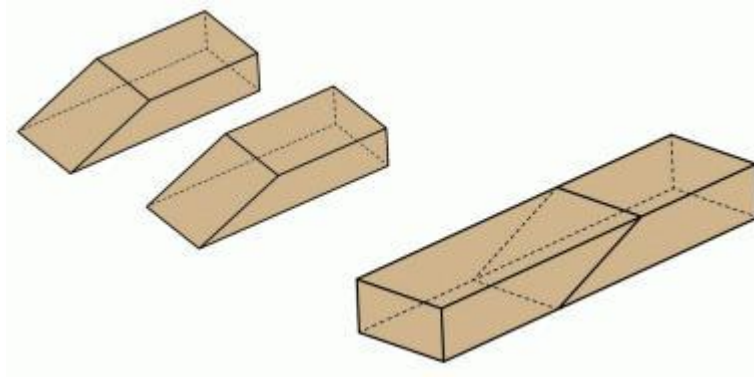
DIAGRAM:

Fig.1 Aircraft wood gluing-single scarf joint

PROCEDURE:

1. The given work piece is firmly clamped in the carpentry vice.
2. Using the try square, the right angles of the faces are checked.
3. Using the steel rule, make a mark at the centre of the work piece.
4. Using the try square, make a right angle mark at the centre of the work piece.
5. Using the steel rule, make a mark (20mm) from the centre of the work piece to both right and left end.
6. Using the try square, make a right angle mark 20mm from the centre of the work piece to both right and left end.
7. Repeat the above steps for the other side of the work piece and mark a diagonal for scarf joint.
8. Using the steel rule and marking gauge, marking is done for single scarf joint.
9. Using the cutting tool makes the cut and in one half, the unwanted portions of wood are removed by using the tenon saw and firmer chisel. The same procedure is done for the other half of work piece.
10. The finished two pieces are glued together to form the scarf joint.

RESULT:

Thus the required single scarf joint is obtained.

VIVA-VOCE QUESTIONS:

1. What is scarf joint?
2. Where the scarf joint is used?
3. What is the difference between butt joint and scarf joint?

UNIVERSITY QUESTIONS:

1. Make single scarf joint in the given work piece as per the given dimensions.

AIRCRAFT WOOD GLUING – DOUBLE SCARF JOINT**EXPT.NO: 4****DATE:****AIM:**

To make an aircraft wood gluing by using double scarf joint.

THEORY:

Scarf joints are often used in boat building. A scarf joint is a long angled joint. The long angle provides a lot of "glue surface" for the joint, making it very strong.

A scarf joint is made by joining two pieces of wood having tapered, beveled, or chamfered ends which over-lap together, as opposed to a butt joint where squared ends of the mating pieces simply butt together.

Scarf joints are used to make longer members where single members of sufficient length are not available or are too costly. Both solid wood pieces and sheets of plywood can be scarf joined using epoxy. With the proper cutting and gluing methods, such joints will be amazingly strong, exceeding that of the joining wood members.

With double scarf joints, it is preferable for the two pieces to fit as closely as possible for ultimate strength. However, minor irregularities of fit are acceptable since the epoxy will have the ability to bridge these without significant loss of strength.

APPARATUS REQUIRED:

1. Marking tools
2. Measuring tools
3. Bench vice
4. Cutting tools
5. Jack plane
6. Rasp file

MATERIAL REQUIRED:

Soft wood of size 300x50x50 mm

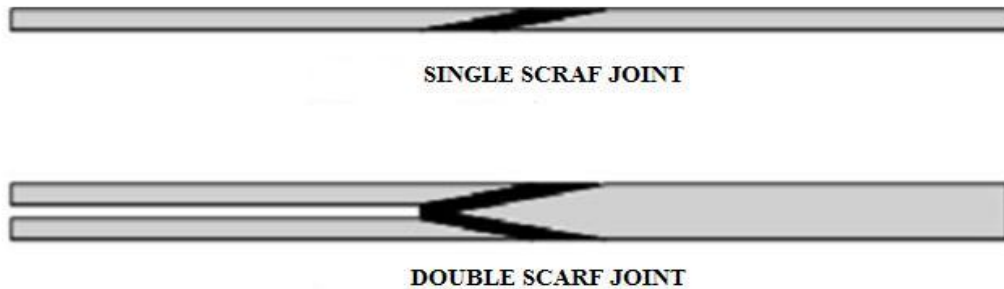
DIAGRAM:

Fig.2 Aircraft wood gluing-double scarf joint

PROCEDURE:

1. The given work piece is firmly clamped in the carpentry vice.
2. Using the try square, the right angles of the faces are checked.
3. Using the steel rule, make a mark at the centre of the work piece.
4. Using the try square, make a right angle mark at the centre of the work piece.
5. Using the steel rule, make a mark (20mm) from the centre of the work piece to both right and left end.
6. Using the try square, make a right angle mark 20mm from the centre of the work piece to both right and left end.
7. Repeat the above steps for the bottom surface of the work piece and mark a diagonal for scarf joint.
8. Using the steel rule and marking gauge, marking is done for double scarf joint.
9. Using the cutting tool makes the cut and in one half, the unwanted portions of wood are removed by using the tenon saw and firmer chisel. The same procedure is done for the other half of work piece.
10. The finished two pieces are glued together to form the scarf joint.

RESULT:

Thus the required double scarf joint is obtained.

VIVA-VOCE QUESTIONS:

1. What is double scarf joint?
2. What is the need of double scarf joint?
3. What is the difference between single scarf joint and double scarf joint?

UNIVERSITY QUESTIONS:

1. Make single scarf joint in the given work piece as per the given dimensions.

STUDY OF PATCH REPAIR BY USING TIG WELDING**EXPT.NO: 3a****DATE:****AIM:**

To make a patch repair by using TIG Welding.

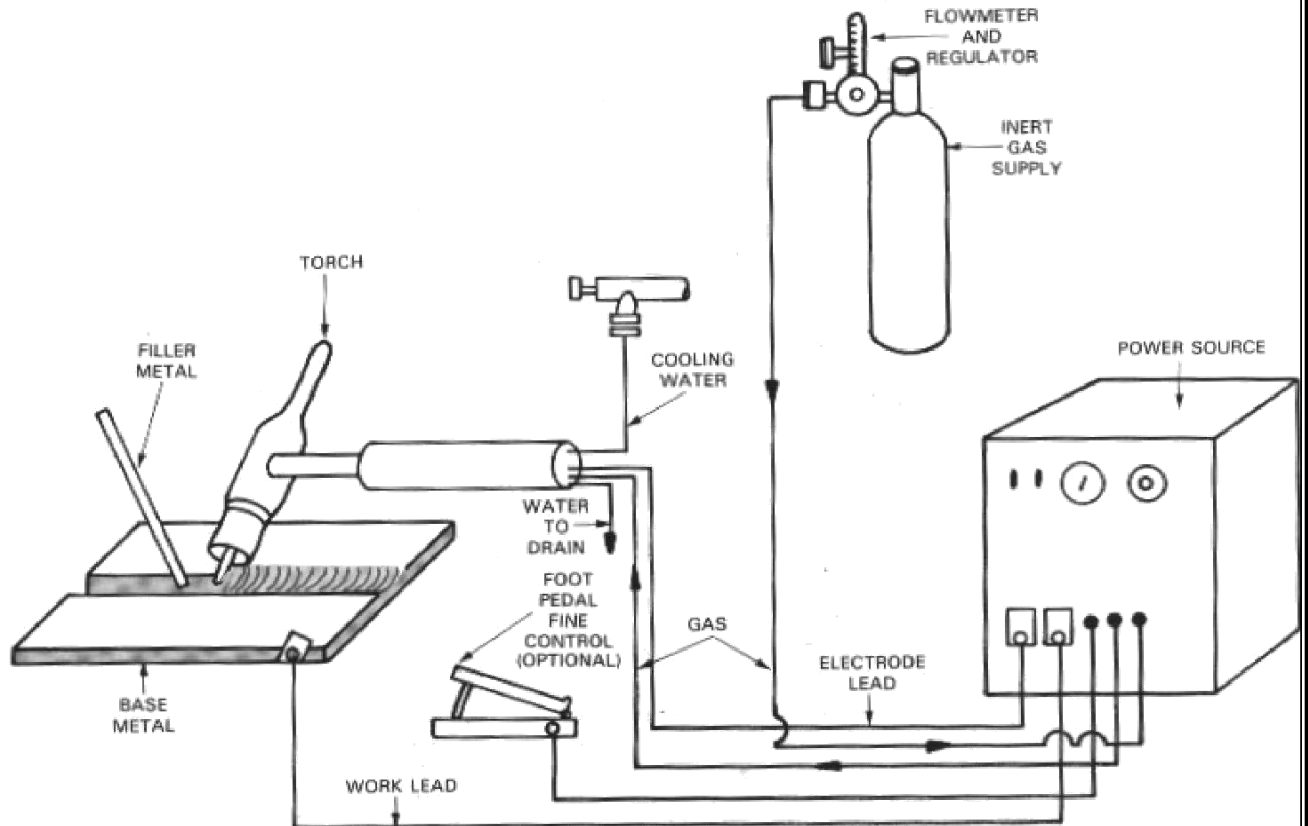
THEORY:**TUNGSTEN INERT GAS WELDING (TIG):**

Tungsten Inert Gas Welding classified as gas tungsten arc welding (GTAW) is accomplished with the mass of torch with a non consumable tungsten electrode. The electrode is used to sustain the arc and the molten pool of metal. Filler rod is added to the pool to develop the assigned thickness of bed inert gas. Usually argon is fed to the weld area through the gas cup on the torch. The gas cup surrounds the electrode and directs the gas in pattern to prevent the intrusion of oxygen and nitrogen from the gas. In some cases, it has been found beneficial to mix small amount of oxygen and other gases to gain the best result.

This process is used for welding steel, Aluminum, cast iron, Magnesium, Stainless steel etc. The inert gas used is Argon or Helium.

APPARATUS REQUIRED:

1. Inert gas cylinder
2. Inert gas regulator
3. Inert gas shutoff valve
4. Arc welding machine
5. Welding cables for electrode and ground connector
6. A welding bench
7. Water supply with inlet and outlet hose
8. Non consumable tungsten electrode

DIAGRAM:**Fig.1 TIG welding setup****OUTPUT SPECIFICATIONS:**

- Current Range : 3-200A
- Maximum OCV : 85V
- Cooling : F-Air
- Protection Glass : IP 23
- Insulation Gas : H

INPUT SPECIFICATIONS:

- Main fuse : 16A
- Power factor : 0.99
- Weight : 21kg

PROCEDURE:

1. Connect the welding cylinder with the Co₂ Cylinder.
2. Switch on 3 phase power supply.
3. Set the solid in the plant.
4. Mild steel plate strut lap over another.
5. Using TIG Welding machine first both lapping and butting would be done.

RESULT:

Thus the required patch repair is prepared by using TIG Welding.

VIVA-VOCE QUESTIONS:

1. Explain the term TIG?
2. Explain the working of TIG welding?
3. What is the use of filler rod?

UNIVERSITY QUESTIONS:

1. Make the patch repair by TIG welding in the given work piece.

STUDY OF PATCH REPAIR BY USING MIG WELDING

EXPT.NO:3b

DATE:

Aim:

To make a patch repair by using Metal Inert Gas Welding (MIG).

THEORY:

MIG (Metal Inert Gas Welding) is classified as Gas Metal Arc Welding (GTAW) by the AWS. In this welding the electric arc will be produced between a consumable metal electrode and the work metal. The wire electrode is fed continuously through the center of the torch or pre set controlled speeds, shielding gas is fed through the torch and the gas is completely covering the weld puddle with a shield of gas.

Thus tends to complete automation of the welding process power, gas flow, wire feed and over the work piece are present when using a welding machine. The electrode melted by the arc and deposited as welded joints.

APPARATUS REQUIRED:

- Inert gas cylinder
- Gas regulator and flow metals
- Gas hoses and connectors
- A power saver and welding leads
- MIG Welding gun
- A spool of electrode wire
- Water supply with water hoses

SPECIFICATIONS:

OUTPUT SPECIFICATIONS:

- Current voltage : 50-300A
- OCV : 17-38V

TECHNIQUES:

Two generate techniques or processes are employed when MIG Welding is aviation maintenance in most of the cases:

1. Spray-Arc Transfer
2. Short- Circuit Transfer

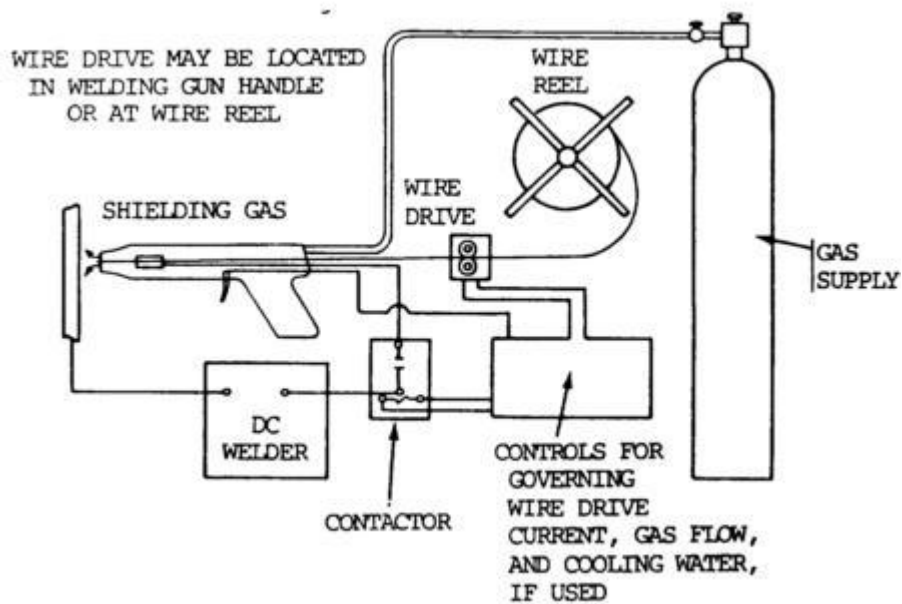
DIAGRAM:

Fig.1 MIG welding setup

PROCEDURE:

1. Connect the welding cylinder with the Argon Cylinder.
2. Switch on 3 phase power supply.
3. Set the solid in the plant.
4. Mild steel plate strut lap over another.
5. Using MIG Welding machine first both lapping and butting would be done.

RESULT:

Thus the required patch repair by using Metal Inert Gas Welding was done.

VIVA-VOCE QUESTIONS:

1. Explain the term MIG?
2. Explain the working of MIG welding?
3. What is the use of wire drive?

UNIVERSITY QUESTIONS:

1. Make the patch repair by TIG welding in the given work piece.

STUDY OF PATCH REPAIR WELDING USING PLASMA ARC WELDING**EXPT.NO:3c****DATE:****AIM:**

To make different types of welded joints like Lap, Butt and Corner joints by using Plasma Arc Welding.

THEORY:

Metal Arc welding is one type of electric arc welding. The heat required for joining metals is obtained from an electric arc. The electric motor or generators are used to supply high electric current between electrode and work piece. When the electrode and work piece are brought with small air gap (3mm) an arc is produced.

The electrodes serve as filler rod also and arc melts the welding surface so that the metals to be joined are fused together. The temperature of arc is approximately 4000 deg. Electrode used in arc welding is generally coated with the flux. Thus flux produces a shield around of the molten metal. This prevents the reaction of metal with oxygen and nitrogen.

APPARATUS REQUIRED:

1. A welding generator or Transformer or Rectifier
2. Two cable leads- one for work piece and another one for electrode
3. Electrode holder
4. Protection shield goggles
5. Gloves, Apron, Wire brush
6. Chipping Hammer

SPECIFICATIONS:**INPUT:**

- Main Fuse : 16A
- Insulation Glass : A
- Weight : 85kg

OUTPUT:

- Current Range : 50 -300A
- OCV : 17-38V
- Cooling : F-Air
- Protection Glass: IP-23

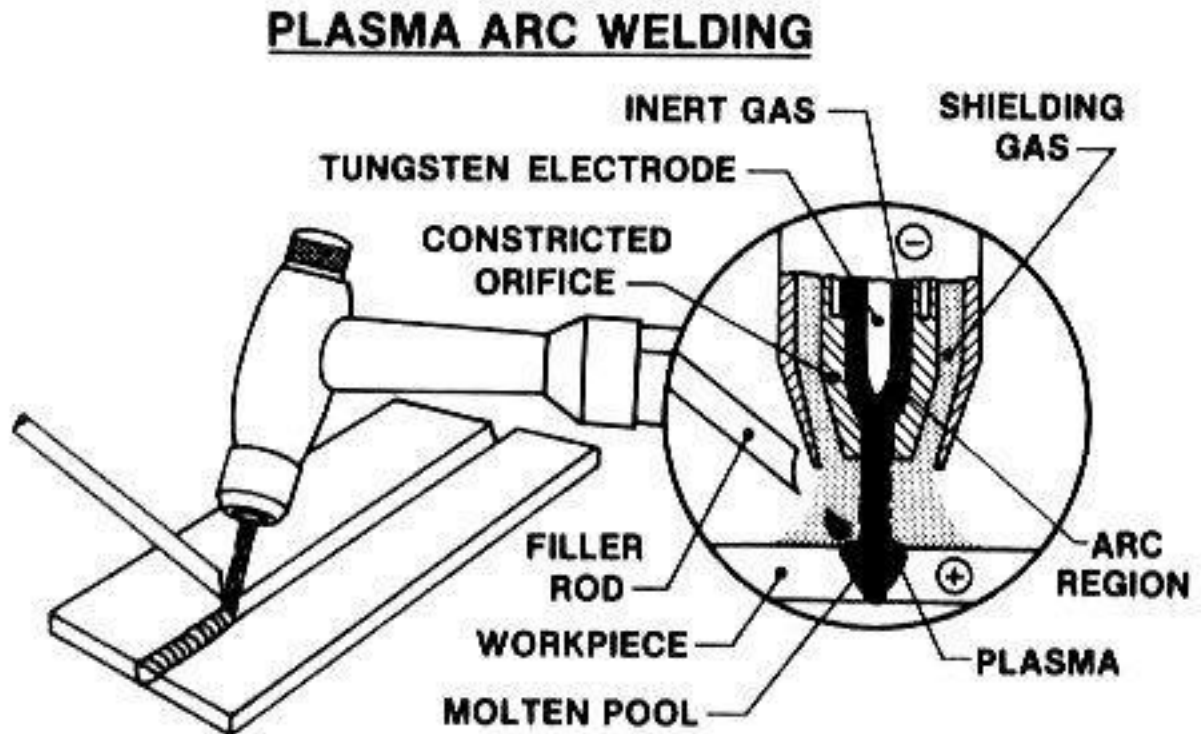
DIAGRAM:

Fig.1 Plasma arc welding setup

PROCEDURE:

1. Connect the welding machine with CO_2 cylinder
2. Switch on the 3 phase power supply.
3. Arrange the sheet metal one over another joint together as per three type of joint.
4. Hold the electrode with the help of electrode holder and bring it near to the work piece as shown in Fig.
5. The arc is produces between work piece and electrode.
6. Due to the immense heat the molten work piece and the molten electrode from the welded joint.
7. Excess material deposits on the work piece are removed by using chipping hammer.

RESULT:

Using arc welding machine, patch repair welding like Lap, Butt and other joints were made.

VIVA-VOCE QUESTIONS:

1. Explain the term PAC?
2. Explain the working of plasma arc welding?
3. What is the use of shielding gas?

UNIVERSITY QUESTIONS:

1. Make the patch repair by plasma arc welding in the given work piece.

WELDED SINGLE AND DOUBLE “V” JOINT

EXPT.NO:6

DATE:

AIM:

To join the given work pieces as a single and double ‘V’ butt joint by arc welding.

THEORY:

Welding is the process of joining similar metals by the application of heat. Welding can be done with or without application of pressure. While welding, the edge of metal piece are either melted or brought to a plastic condition. Welding can be done with the addition of filler material or without it. Welding is used for making permanent joints.

Types of welding

There are two types of welding process. They are

1. Plastic welding
2. Fusion welding

Plastic welding

In plastic welding the metal pieces are heated to a plastic state. Then they are pressed together to make the joints. The plastic welding is also known as pressure welding. During plastic welding there is no filler metal is required.

Example: spot welding, projection welding.

Fusion welding

In fusion welding the metal at the joint is heated to a molten state. Then it is allowed to solidify. Pressure is not applied in this welding process. Addition of filler metal is required for this type of welding. So it is known as non-pressure welding.

Example: gas welding, arc welding.

Arc welding

Arc welding is the process of joining two metal pieces by melding their edges by an electric arc. The electric arc is produced between two conductors. The electrode is one conductor and the work piece is another conductor. The electrode itself melts and supplies the necessary filler metal.

MATERIAL USED:

Mild Steel plates

TOOLS REQUIRED:

- Welding power supply
- Flat file
- Welding rod
- Chipping hammer
- Electrode holder
- Wire brush
- Gloves and apron
- Earthing clamps
- Shield and goggles

PROCEDURE:**For single V joint,**

1. Fix the first work piece in the bench vice, make chamfering to the bevel angle 30° on right side and maintain the root face of 0.3 m.
2. Fix the second work piece in the bench vice, make chamfering to the bevel angle 30° on left side and maintain the root face of 0.3 m.
3. First of all, the work pieces must be thoroughly cleaned to remove rust, scale and other foreign materials.
4. Then the given work pieces are placed on the table in such a way that work pieces are brought close to close to each other so that it forms a V shapes when the plates butt each other.
5. Appropriate power supply should be given to the electrode and the work pieces.
6. Now the welding current output may be adjusted.
7. When current is passed, arc is produced between electrode and work piece.
8. Now set the two work pieces in correct position and maintain the gap 3mm and tag at both ends of the work pieces as shown in figure.
9. Then the welding is carried out throughout the length.

10. As soon as the welding process is finished, switch off the current supply and allow the work piece to cool.

11. Slag is removed by chipping process with the help of chipping hammer.

12. Finally using wire brush, welded portions are cleaned.

For double V joint,

1. Fix the first work piece in the bench vice, make chamfering to the bevel angle 30° on right side to the depth of 1.5 m.

2. Fix the other side of first work piece in the bench vice, make chamfering to the bevel angle 30° on right side to the depth of 1.5 m.

3. Fix the second work piece in the bench vice, make chamfering to the bevel angle 30° on left side to the depth of 1.5 m.

4. Fix the other side of second work piece in the bench vice, make chamfering to the bevel angle 30° on left side to the depth of 1.5 m.

5. First of all, the work pieces must be thoroughly cleaned to remove rust, scale and other foreign materials.

6. Then the given work pieces are placed on the table in such a way that work pieces are brought close to close to each other so that it forms a V shapes when the plates butt each other.

7. Appropriate power supply should be given to the electrode and the work pieces.

8. Now the welding current output may be adjusted.

9. When current is passed, arc is produced between electrode and work piece.

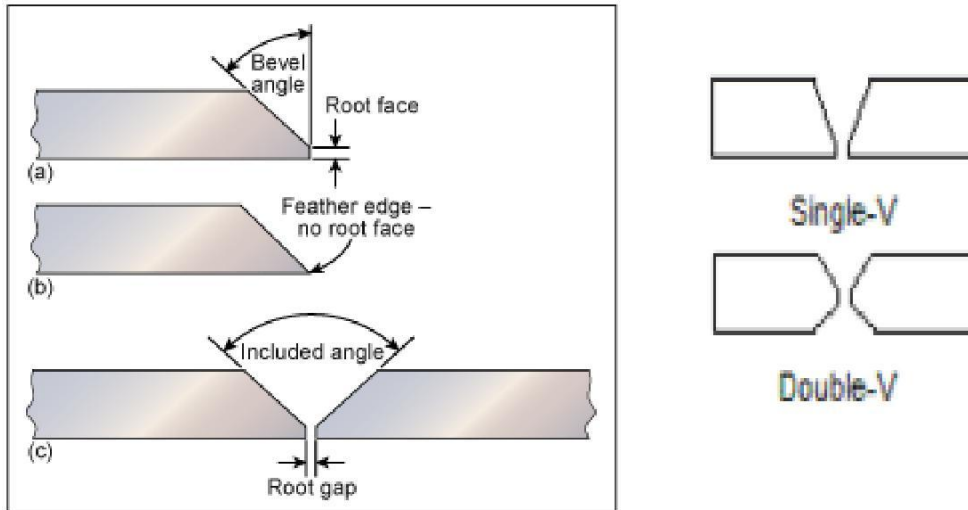
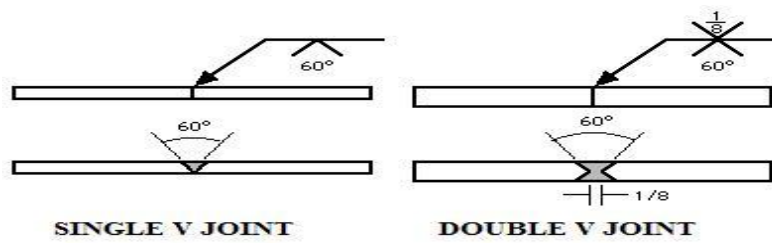
10. Now set the two work pieces in correct position and maintain the gap 3mm and tag at both ends of the work pieces as shown in figure.

11 Then the welding is carried out throughout the length.

12. As soon as the welding process is finished, switch off the current supply and allow the work piece to cool.

13. Slag is removed by chipping process with the help of chipping hammer.

14. Finally using wire brush, welded portions are cleaned.

DIAGRAM:**Fig.1 Root diagram for single V joint****Fig.2 Root diagram for double V joint****RESULT:**

Thus the required single and double 'V' butt joint is made by arc welding process.

VIVA-VOCE QUESTIONS:

1. What is a root face?
2. What is a root gap?
3. What is the difference between single and double V joint?

UNIVERSITY QUESTIONS:

1. Make the patch repair by single and double V joint in the given work piece.

FABRIC PATCH REPAIR

EXPT.NO: 7

DATE:

AIM:

To make a fabric patch repair in aircraft.

APPARATUS REQUIRED:

- Measuring tape
- Fabric
- Scissors
- MEK - aircraft paint remover
- Sponge and 2 cups
- Horse hair brush

THEORY:

Fabric over frame aircraft, like the Citabria, Cub, Aviat and Super Cub, sometimes develop rips, tears or holes in the fabric. The Federal Aviation Administration allows owners to complete small repairs to the fabric providing the rip or tear is not on a part of the airplane that directly affects the aircraft's stability. In addition, if the rip or tear is deemed unsafe upon an FAA inspection, the aircraft may be given an Airworthiness Directive -- also called an AD -- that must be repaired and adhered to for the life of the aircraft. For small fabric repairs use a tested method for repair. For safety's sake, have the repair inspected by an aircraft mechanic before flying.

PROCEDURE:

1. Pour some of the MEK aircraft fluid into a cup. Dip the sponge into the fluid and wipe around the hole or tear and add an extra 2 inches to the area. Wipe until all paint is removed from the fabric.
2. Pull at the edges of the hole or tear to fray the fabric material. Let the MEK dry. Scrub the hole and tear area with the horse hair brush.

3. Pour some of the aircraft glue into a separate cup. Dip the sponge brush into the glue and apply a generous coat of glue to the tear and the 2 inch area around the tear. Let the glue dry.
4. Measure the length and the width of the hole in the garment.
5. Add 3 inches to the length and width.
6. Cut a rectangle of fabric using the measurements using a pair of sharp scissors.
7. Fold down the raw edges towards the wrong side of the fabric.
8. Apply another coat of glue to the tear or hole and to the back of the fabric patch. Place the patch onto the hole or tear. Heat the iron to 300 degrees and iron the patch onto the fabric so the glue melts into the fabric. Do not use a heat gun as it is too difficult to regulate the temperature.
9. Apply a coat of the polybrush fluid and let it dry. Apply the acrylic aircraft fabric paint to the repair patch after the polybrush fluid is dry.

DIAGRAM:



Fig.1 Fabric patch repair

RESULT:

Thus the repair in aircraft skin is done by fabric patch.

VIVA-VOCE QUESTIONS:

1. What is a perpex?
2. When the fabric patch is used?
3. What are the types of resin?

UNIVERSITY QUESTIONS:

1. Make the fabric patch repair for the damaged transparent panel.

RIVETED PATCH REPAIR

EXPT.NO: 7 b

DATE:

AIM:

To learn about aircraft (skin) structural repairs by means of testing of riveted joint.

THEORY:

Rivets are a permanent mechanical fastener. Before being installed a rivet consists of a smooth cylindrical shaft with a head on one end. The end opposite the head is called the buck-tail. On installation the rivet is placed in a punched or drilled hole, and the tail is upset, or bucked (i.e., deformed), so that it expands to about 1.5 times the original shaft diameter, holding the rivet in place. To distinguish between the two ends of the rivet, the original head is called the factory head and the deformed end is called the shop head or buck-tail.

Because there is effectively a head on each end of an installed rivet, it can support tension loads (loads parallel to the axis of the shaft); however, it is much more capable of supporting shear loads (loads perpendicular to the axis of the shaft). Bolts and screws are better suited for tension applications.

Fastenings used in traditional wooden boat building, such as copper nails and clinch bolts, work on the same principle as the rivet but were in use long before the term rivet was introduced and, where they are remembered, are usually classified among nails and bolts respectively.

TOOLS REQUIRED:

- Steel rule
- Hammer
- Chisel
- Dolly
- Vernier Caliper
- Snap and file
- Hand vice

APPARATUS REQUIRED:

- Bench vice
- Electrical drilling machine
- Hand riveting tool
- Shear pedals

MATERIALS REQUIRED:

Document – Aircraft Maintenance Manual

Lap Joint - 80x80x1.5 = 1

Butt Joint - 70x80x1.5 – 2

Rivets - 1/8 dia.

Aluminum slip

DETERMINING RIVET LENGTH:

Rivet length = Allowance + Material Thickness

$$L = A + MT$$

Rivet Diameter	Allowance
1/8	3/16
5/32	1/4
3/16	5/16
1/4	3/8

SPECIFICATIONS:

Margin: 10 mm

Pitch: 10 mm

Hole diameter: 3.2 mm

Tolerance: 0.02 mm

Allowance: 4.5 mm

PROCEDURE:

1. Riveted holes for riveting must be clean, round and of proper size and clean of basis.
2. Drew out the damaged area of the skin into regular shape by common radius.
3. Size the fresh plate within tolerance limits, smoothen the edges of the plates by file. Also ensure the exact size by Vernier caliper.
4. Prepare a back plate to lap around the finish patch plate.
5. Grip the plates (for butt joints and three plates for lap joints) by hand vice.
6. Size the length of the rivets.
7. Align the plates and insert the proper rivet on hole.
8. Rivet the job by using snap and dolly or with rivet gun.
9. While riveting the head should be formed properly.
10. Clean the riveted area and check for the source problem.
11. Complete the work for job.

The minor damage to the outside skin of an aircraft can be repaired by applying a patch to the inside of the damage sheet.

A filler plug must be installed in the hole made by removal of damaged skin area forms a smooth outside surface. The patch may be of elongated patch or returned patch.

DIAGRAM:

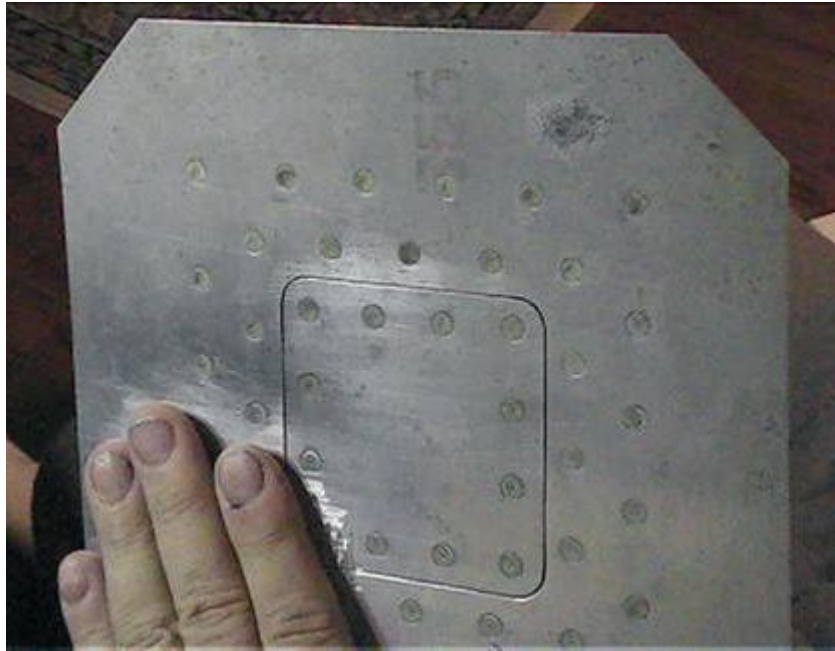
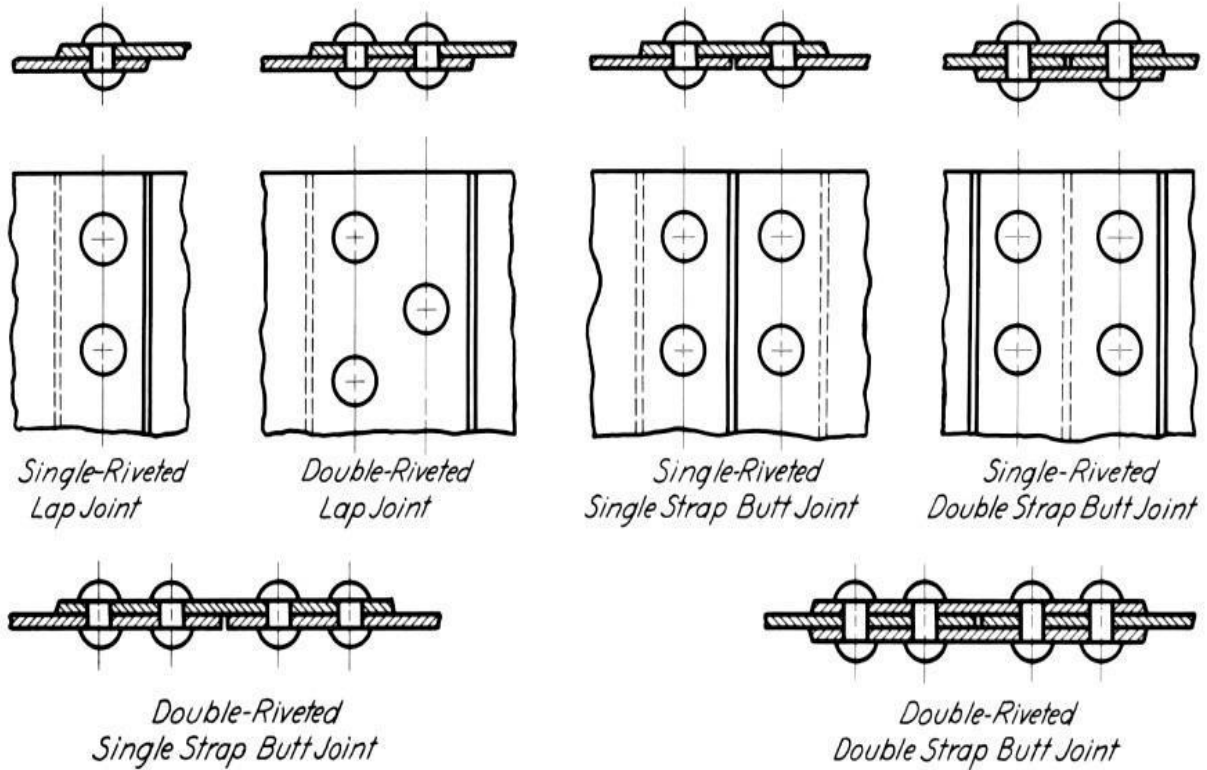


Fig.1 Riveted patch repair



Forms of riveted joints.

Fig.2 Types of riveted patch repair

RESULT:

Thus the repair in aircraft skin is done by Lap patch riveting.

VIVA-VOCE QUESTIONS:

1. What is a double riveted lap joint?
2. When the riveted patch is used?
3. What are the types of lap joint?

UNIVERSITY QUESTIONS:

1. Make the riveted patch repair for the damaged aircraft metal skin.

TUBE BENDING

EXPT. NO: 8a

DATE:

AIM:

To obtain a smooth bend without flattening the tube.

THEORY:

Tube bending can be done by using hand tool, hydraulically operated pipe bending machine and even with the help of heat also.

Bending of tube can also be done by means of tube bending machine.

It is mounted on a tripod stand and can swivel about its own axis at any angle so as to give a wide range for operation.

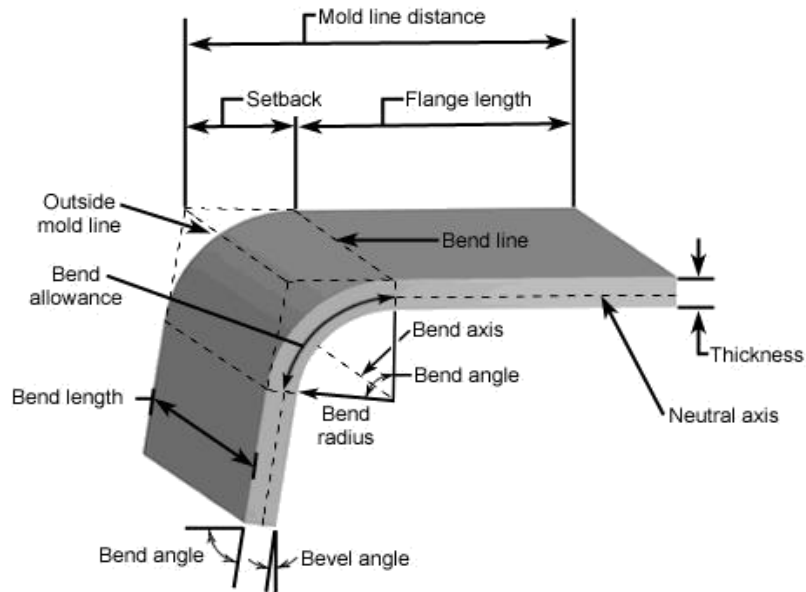
MARKING THE POINT OF BEND:

The bend point is marked as per table.

Tube Length	90°	60°	45°	30°
15mm tube	70mm	46mm	35mm	23mm
22mm tube	100mm	66mm	50mm	33mm

PROCEDURE:

1. First of all cut the pipe slightly larger than its requirement.
2. Mark the place on the tube where the bend is required.
3. Position the tube between two blocks in the tube bending machine.
4. Operate the hydraulic jack lever. It moves the radius bend bars forward the tube and while doing so the tube is bent to desired angle.
5. If the bend is not proper the same procedure is repeated two or three times.

DIAGRAM:**Fig.1 Tube bending****RESULT:**

Thus the desired degree of bend is made in the tube.

VIVA-VOCE QUESTIONS:

1. What are the types of bend?
2. What are the different methods of tube bending?
3. What is bend radius?

UNIVERSITY QUESTIONS:

1. Make the tube bending in the given work piece.

TUBE FLARING

EXPT. NO: 8b

DATE:

AIM:

To renew the damaged tube length with necessary end fittings and flaring to prevent fluid damage.

THEORY:

Flare fittings are a type of compression fitting used with metal tubing, usually soft steel and ductile (soft) copper, though other materials are also used. Tube flaring is considered to be a type of forging operation and is usually a cold working procedure. During assembly, a flare nut is used to secure the flared tubing's tapered end to the also tapered fitting, producing a pressure-resistant, leak-tight seal. Flared connections offer a high degree of long-term reliability and for this reason are often used in mission-critical and inaccessible locations.

The tool used to flare tubing consists of a die that grips the tube and a mandrel that is forced into the end of the tube to form the flare by cold working. The most common flare fitting standards in use today are the 45-degree SAE style, and the 37-degree AN style, also used with the JIC system. The AN/JIC style generally has a higher pressure rating for a given size tubing. SAE and AN/JIC fittings are completely incompatible due to the different flare angle. Further, AN fittings (or those complying with subsequent standards) and JIC fittings are not interchangeable for design-controlled applications due to differing quality standards.

MATERIALS REQUIRED:

Document – Aircraft Maintenance Manual

Aluminum tubes

TOOLS REQUIRED:

- Flaring tool
- Rough file

APPARATUS REQUIRED:

- Slaves
- Pipple nuts

DIAGRAM:

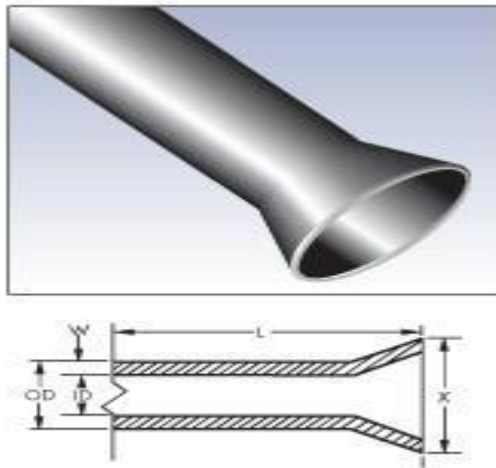
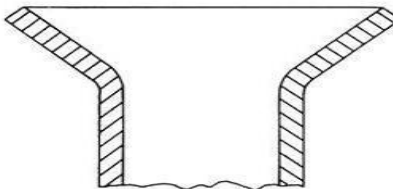
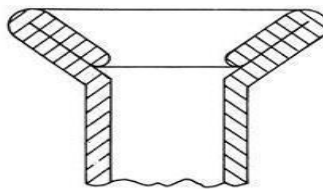


Fig.1 Tube flaring



A. Single-flared end



B. Double-flared end

Fig.2 Types of flaring

PROCEDURE:

1. Shape and size the tube squarely.
2. Slide in the sleeves and nuts as shown in drawings.
3. Open the damp screw of the flaring tool, insert the tube to flared clamping blocks tighten the clamp screw.
4. Use 37° flaring screws in the cone and fill the desired amount of flaring.
5. Remove the cone and undamped the pipe from the tool.
6. The portion of the tube on the sleeve indicates the maximum amount of flaring.

RESULT:

Thus the desired amount of flaring is made in the tube.

VIVA-VOCE QUESTIONS:

1. What is flaring?
2. What is need of flaring?
3. What is flared radius?

UNIVERSITY QUESTIONS:

1. Make the tube flaring in the given work piece.

SHEET METAL FORMING

EXPT. NO: 9

DATE :

AIM:

To fabricate channel section by sheet metal forming.

INTRODUCTION:

In metal working terminology forming means changing the shape by bending and deforming the solid metal.

Forming involves the use of extremely light gauge alloys of a dedicate nurture the wrong process of forming may be hasten fatigue or may sudden structure failure. In all the aircraft metals pure aluminum is the most easily formed.

THEORY:

The sheets are formed into shape, at the factory on large process or by drop hammers equipped with the dies of correct shape. Most parts can be formed without annealing the metal. For extensive forming or complex cases the metal should be in the dead soft or annealed condition.

The stainless steel tubes can be formed by the usual methods but require more skill. The methods used in forming operations include shrinking, stretching, bumping, crimping, roll forming and folding.

PROCEDURE FOR MAKING STRAIGHT LINE BENDS:

1. The required length of the sheet metal should be cue in proper size.
2. Using the sharp pencil mark the set back lines as shown in figure. Set back is the distance from the bend tangent line to the mold point.
3. When making a fold or bend in a sheet metal bend allowance should be calculated, it's the length of material required for the bend

$$= 2\pi(R + 0.5T)/4$$

Where, R- Radius of the bend

T- Thickness of the sheet.

4. The metal is positioned in the bench vice with two wooden blocks.
5. The 2 wooden forming blocks should have one edge rounded for the desired radius of bend.
6. The wooden forming block is bended to the desired angle by tapping lightly with a mallet bend metal.
7. Continue this process until the protruding metal is forced down to the desired angle.
8. Remove the formed metal from vice and check for its proper size.
9. Ensure that the perfection of the metals being kept curved with soft metal jaw cops.

DIAGRAM:



Fig.1 Sheet metal

RESULT:

Thus the required channel section is made.

VIVA-VOCE QUESTIONS:

1. What is sheet metal forming?
2. What is need of sheet metal forming?
3. What is bend radius?

UNIVERSITY QUESTIONS:

1. Make the channel section by sheet metal forming.

PREPARATION OF GLASS EPOXY COMPOSITE LAMINATES

EXPT.NO:10

DATE:

AIM:

To learn the preparation of glass epoxy composite laminates and specimens.

INTRODUCTION:

A composite material is a combination of two or more phases are joined permanently together so that strength of the combined material is greater than any of the component material. Component materials in Composites retain their identities.

Composite structures are used in aircraft components of fiber materials combined with matrix.

Kevlar, Carbon fiber (Graphite), Boron, Tungsten, Quartz, Silicon carbide, ceramics and SPECTRA are commonly used Composite materials.

Composite structure can either be a,

- Solid Laminate:
- Honey comb / rigid foam sandwich construction

SOLID LAMINATE:

It is used by bonding together several layers of resin fairing material impregnated with resin matrix.

SANDWICH:

It is made by taking high density laminate on solid surface and back plate and a low density case between them.

MATERIALS REQUIRED:

- Glass fibre
- Epoxy resin
- Hardener
- Thinner

APPARATUS REQUIRED:

- Ply
- Mila ply
- Roller
- Weighing machine
- Owen

PROCEDURE:

1. Cut the required length of glass fibre and placed it on the mila ply.
2. Mix the resin and hardener in the ratio of 10:1 and stir well using the stirrer.
3. Apply the resin on the fibre by using roller to make a lamina.
4. Keep another piece of glass fibre and repeat the above procedure.
5. Repeat the above procedure to make a solid laminate.
6. Keep two plies on both top and bottom surface of the laminate and keep it in owen.

DIAGRAM:

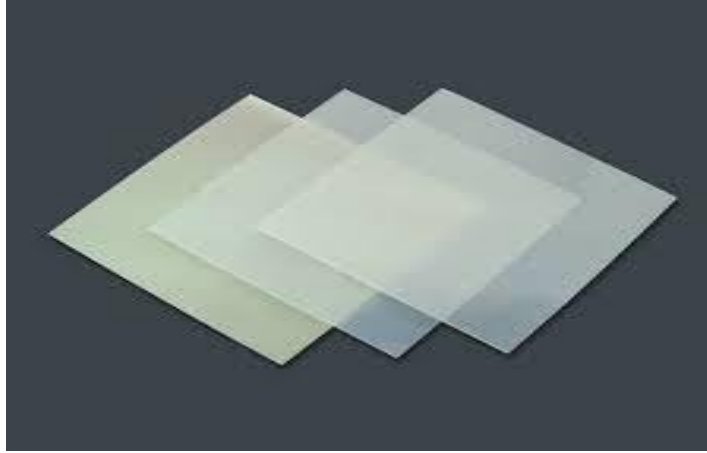


Fig.1 Glass epoxy laminate

RESULT:

Thus the required glass epoxy laminate is made.

VIVA-VOCE QUESTIONS:

1. What is composite material?
2. What are the advantages of composite?
3. What is specific strength?

UNIVERSITY QUESTIONS:

1. Make the glass epoxy laminate as per the ASTM standard.

DETERMINATION OF ELASTIC CONSTANTS OF COMPOSITE LAMINATE

EXPT.NO:10

DATE:

AIM:

To determine the elastic constants of glass epoxy composite laminate

THEORY:

Tensile testing, also known as tension testing is a fundamental materials science test in which a sample is subjected to a controlled tension until failure. The results from the test are commonly used to select a material for an application, for quality control, and to predict how a material will react under other types of forces. Properties that are directly measured via a tensile test are ultimate tensile strength, maximum elongation and reduction in area. From these measurements the following properties can also be determined: Young's modulus, Poisson's ratio, yield strength, and strain-hardening characteristics. Uniaxial tensile testing is the most commonly used for obtaining the mechanical characteristics of isotropic materials. For anisotropic materials, such as composite materials and textiles, biaxial tensile testing is required.

The principle behind a triaxial shear test is that the stress applied in the vertical direction (along the axis of the cylindrical sample) can be different from the stresses applied in the horizontal directions perpendicular to the sides of the cylinder, i.e. the confining pressure). In a homogeneous and isotropic material this produces a non-hydrostatic stress state, with shear stress that may lead to failure of the sample in shear. In homogeneous and anisotropic samples (e.g. bedded or jointed samples) failure may occur due to bending moments and, hence, failure may be tensile. Also combinations of bending and shear failure may happen in inhomogeneous and anisotropic material.

A solid is defined as a material that can support shear stress without moving. However, every solid has an upper limit to how much shear stress it can support. The triaxial test is designed to measure that limit. The stress on the platens is increased until the material in the cylinder fails and forms sliding regions within itself, known as **shear**

bands. A motion where a material is deformed under shear stress is known as **shearing**. The geometry of the shearing in a triaxial test typically causes the sample to become shorter while bulging out along the sides. The stress on the platen is then reduced and the water pressure pushes the sides back in, causing the sample to grow taller again. This cycle is usually repeated several times while collecting stress and strain data about the sample.

During the shearing, a granular material will typically have a net gain or loss of volume. If it had originally been in a dense state, then it typically gains volume, a characteristic known as Reynolds' dilatancy. If it had originally been in a very loose state, then contraction may occur before the shearing begins or in conjunction with the shearing.

From the triaxial test data, it is possible to extract fundamental material parameters about the sample, including its angle of shearing resistance, apparent cohesion, and dilatancy angle. These parameters are then used in computer models to predict how the material will behave in a larger-scale engineering application. An example would be to predict the stability of the soil on a slope, whether the slope will collapse or whether the soil will support the shear stresses of the slope and remain in place. Triaxial tests are used along with other tests to make such engineering predictions.

APPARATUS REQUIRED:

- Universal testing machine

FORMULA USED:

1. Young's modulus, $E = \text{Stress/Strain}$
2. Poisson's ratio, $\mu = \text{lateral strain/ longitudinal strain}$

PROCEDURE:

1. Cut the laminate as per the ASTM standard.
2. Make the clamp at both ends.
3. Fix the clamped ends at the UTM holder.
4. Fix the electrical strain gauges where the strain has to measure.
5. Connect the strain gauges to the strain indicator.
6. Switch on the UTM and do the tension test.
7. Note down the longitudinal and lateral strains.
8. Note down the stress.

DIAGRAM:



Fig.1 Tensile test in UTM

Standard tensile test specimen

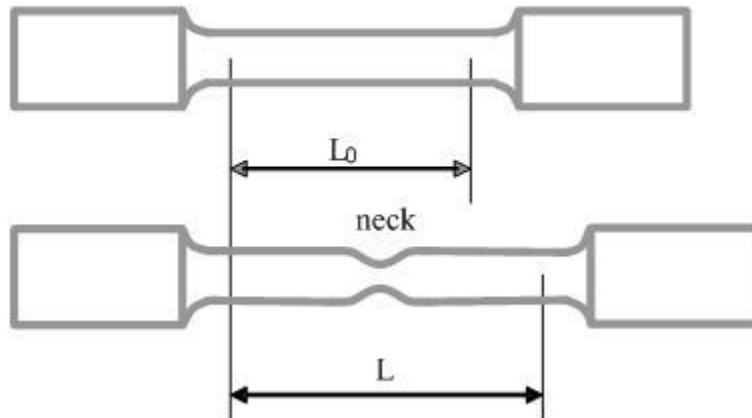


Fig.2 Tensile test specimen

RESULT:

Thus the elastic constants of glass epoxy composite laminate are determined.

VIVA-VOCE QUESTIONS:

1. What is tensile test?
2. What is shear test?
3. What is specific modulus?

UNIVERSITY QUESTIONS:

1. Determine the elastic constants of the given glass epoxy laminate in the ASTM standard.

THE END

**MANUAL PREPARED BY
K.YOKESVARAN. M.Tech.,
ASSISTANT PROFESSOR,
DEPARTMENT OF AERONAUTICAL ENGINEERING,
MADHA ENGINEERING COLLEGE, KUNDRATHUR.**



DEPARTMENT OF AERONAUTICAL ENGINEERING

LABORATORY MANUAL

**AE3781-COMPUTATIONAL ANALYSIS
LABORATORY**

AE3781-COMPUTATIONAL ANALYSIS LABORATORY

LIST OF EXPERIMENTS

1. Grid independence study and convergence test using any simple case like cylinder
2. Simulation of flow over an aero foil
3. Simulation of flow over backward facing step.
4. Simulation of Karman vortex trail (vortex shedding) using circular cylinder.
5. External flow simulation of subsonic and supersonic aero foils.
6. Internal flow simulation of subsonic, sonic and supersonic flow through a CD nozzle.
7. Structural analysis of bar and beam
8. Structural analysis of truss.
9. Structural analysis of tapered wing.
10. Structural analysis of fuselage structure.
11. Analysis of composite laminate structures.
12. Heat transfer analysis of structures.

Exp no:

Date:

SIMULATION OF FLOW THROUGH A CONVERGING-DIVERGING NOZZLE

AIM:

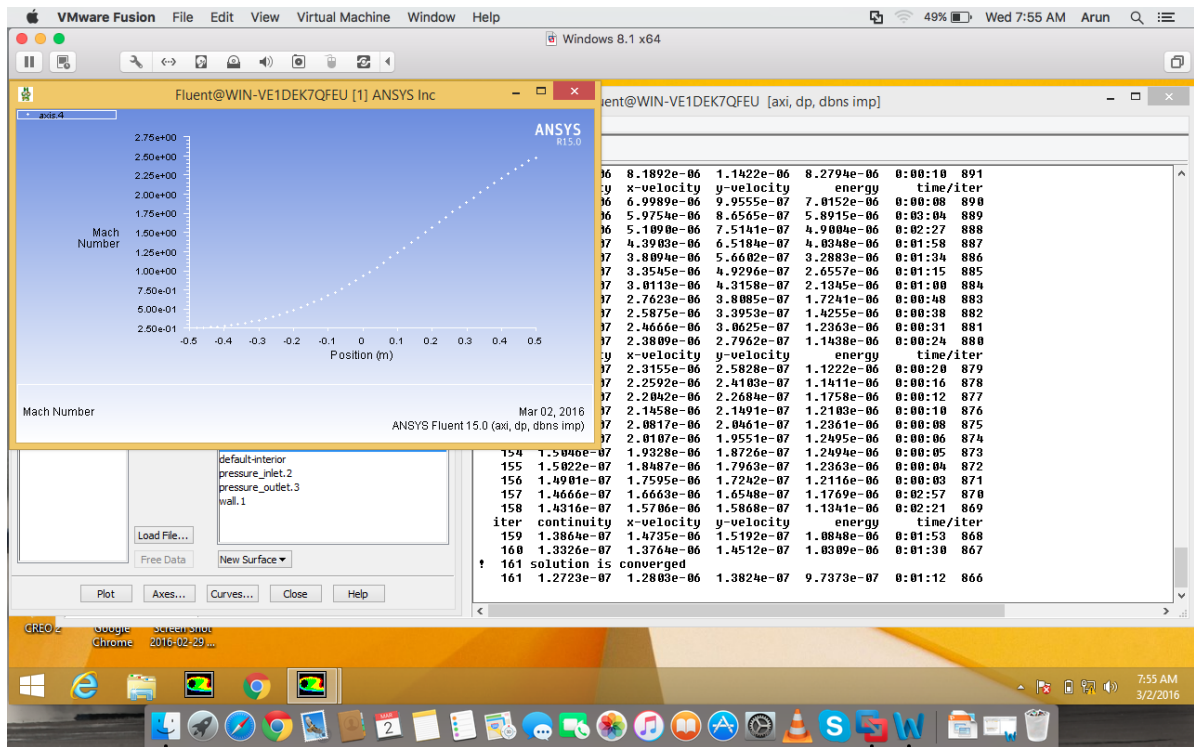
To simulate the flow through a converging-diverging nozzle using ANSYS FLUENT15.

SOFTWARE USED:

ANSYS FLUENT15

PROCEDURE:

1. Click start → click All Program → click ANSYS FLUENT 15.
2. Turn on radio button to 2d and click ok.
3. Turn on radio button to Density-Based and turn on radio button to Axisymmetric.
4. Click File → Read → Mesh and import the mesh file.
5. Click Mesh → Check and conform that the mesh is correct.
6. Click Display → Enable faces and click ok.
7. Click Models → Turn on energy equation and change laminar flow to in viscid.
8. Click Materials → click Fluid → change constant to ideal gas.
9. Click Define → Operating condition → Operating pressure=0.
10. Click Define → Boundary condition → Pressure_inlet.2 → Edit → GTP=101325 and IGP=99348.
11. Click Solve → Initialization → Hybrid initialization → Initialize.
12. Click Run Calculation → Number of iterations=1000 → Calculate.
13. Click Graphics and Animation → Contours → Enable Filled → Display.
14. Click Display → Views → Mirror plane → Axis_4.
15. Click Plots → XY Plots → Pressure vs Static pressure → Plot.
16. Click Plots → XY Plots → Velocity vs Mach Number → Plot.
17. Click Save case file.



Result:

Thus the simulation of flow through the Converging-Diverging nozzle has been done in ANSYS FLUENT15 and results have been found.

Exp no:

Date:

SIMULATION OF SUPERSONIC FLOW OVER A WING OF BICONVEX CROSS SECTION.

AIM:

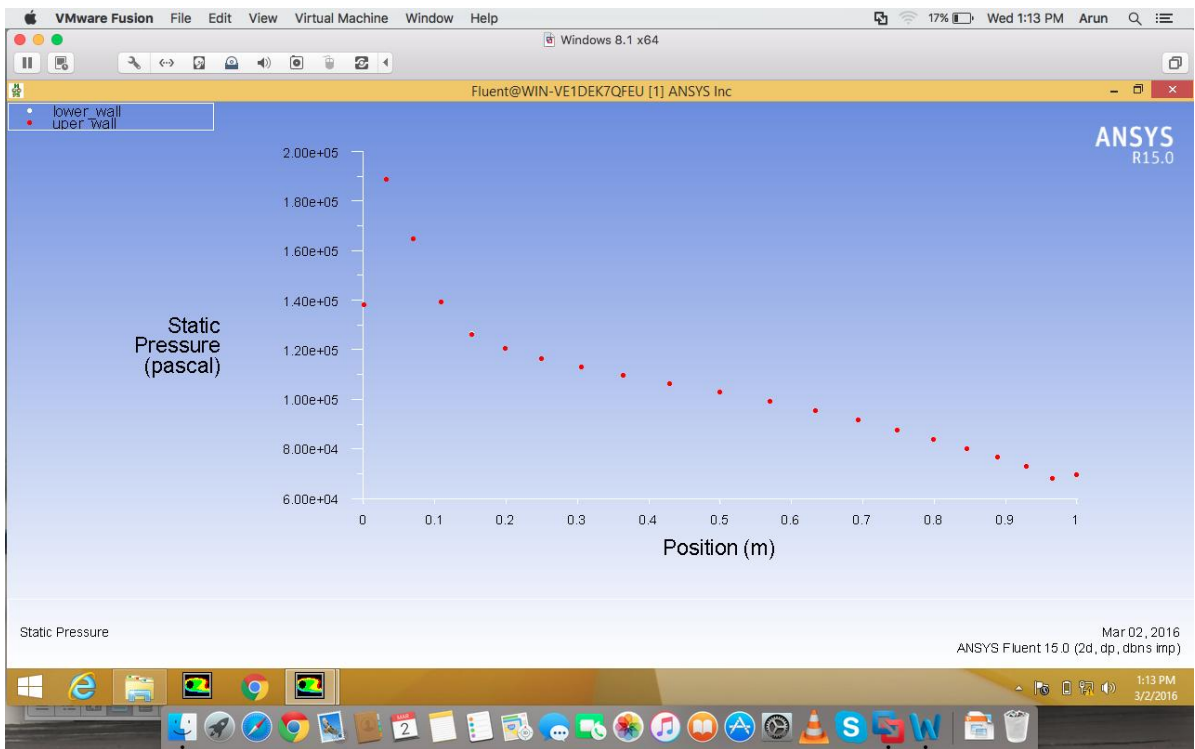
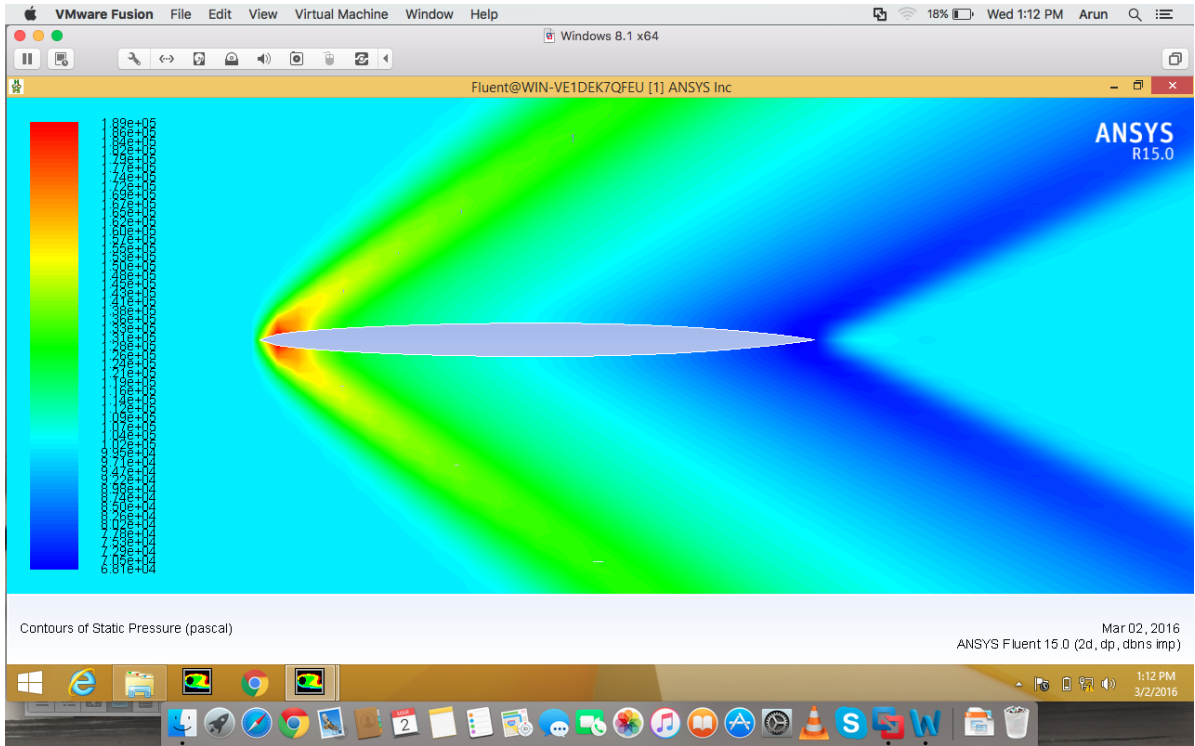
To simulate the simulate the supersonic flow over a wing of biconvex cross section using ANSYS FLUENT15.

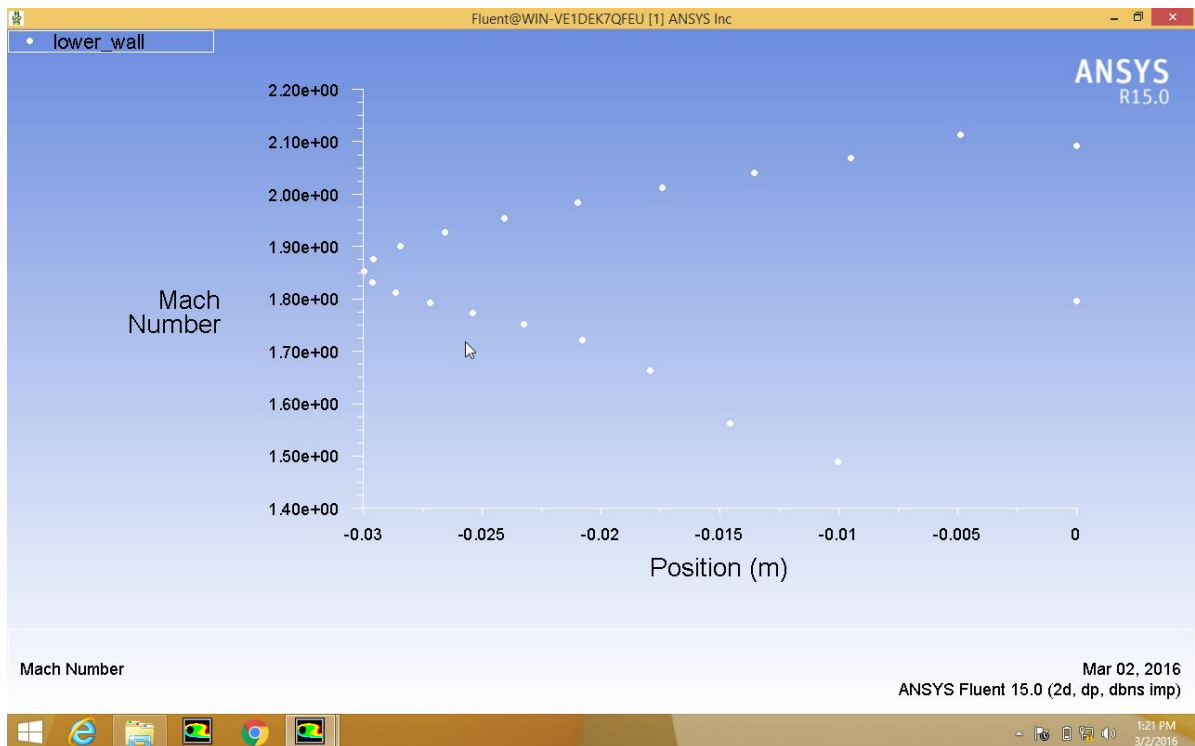
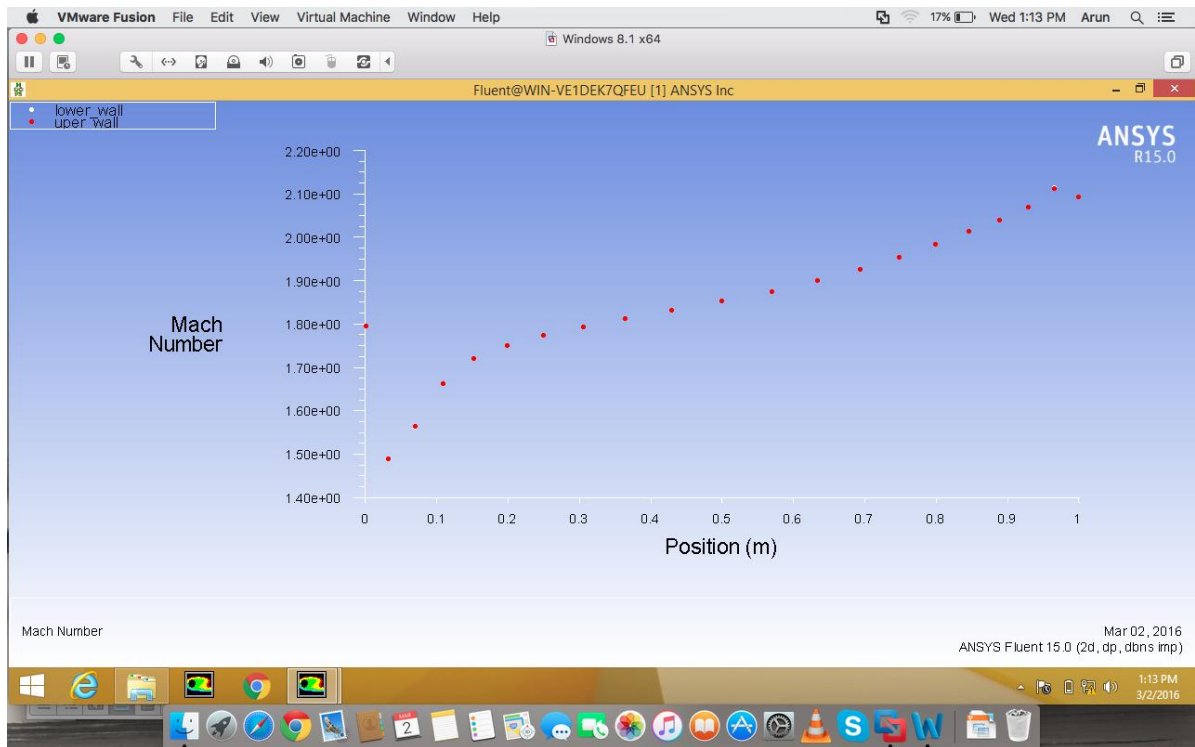
SOFTWARE USED:

ANSYS FLUENT15

PROCEDURE:

1. Click start → click All Program → click ANSYS FLUENT 15.
2. Turn on radio button to 2d and click ok.
3. Turn on radio button to Density-Based.
4. Click File → Read → Mesh and import the mesh file.
5. Click Mesh → Check and conform that the mesh is correct.
6. Click Display → Enable faces and click ok.
7. Click Models → Turn on energy equation and change laminar flow to in viscid.
8. Click Materials → click Fluid → change constant to ideal gas.
9. Click Define → Operating condition → Operating pressure=0.
10. Click Define → Boundary condition → Pressure_far_field.3 → Edit → GTP=101325 and Mach number=2.
11. Click Solve → Initialization → Standard Initialize → Initialize.
12. Click Run Calculation → Number of iterations=1000 → Calculate.
13. Click Graphics and Animation → Contours → Enable Filled → Display.
14. Click Plots → XY Plots → Pressure vs Static pressure → Plot.
15. Click Plots → XY Plots → Velocity vs Mach Number → Plot.
16. Plot C_l and C_d for the wing of biconvex cross section by changing $X=0$ and $Y=1$ → Plot.
17. Click Save case file.





Result:

Thus the simulation of supersonic flow over a wing of biconvex cross section has been done in ANSYS FLUENT15 and results have been found.

Exp no:

Date:

HOT FLOW SIMULATION THROUGH AN AXIAL FLOW TURBINE BLADE PASSAGE.

AIM:

To simulate the hot flow simulation through an axial flow turbine blade passage.

SOFTWARE USED:

ANSYS FLUENT15

PROCEDURE:

1. Click start → click All Program → click ANSYS FLUENT 15.
2. Turn on radio button to 3d and click ok.
3. Turn on radio button to Pressure Based.
4. Click File → Read → Mesh and import the mesh file.
5. Click Mesh → Check and conform that the mesh is correct.
6. Click Display → Enable faces and click ok.
7. Click Define → Units → Angular velocity → RPM → Close.
8. Click Define → Models → Viscous → K-epsilon → Near wall treatment → Standard wall function.
9. Click Define → Mixing plane → Upstream Zone = Pressure outlet rotor and Downstream Zone = Pressure inlet stator and click create.
10. Click Material → Air.
11. Click Cell zone condition → Fluid Rotor → Edit → Enable Frame motion → Rotation axis direction z(-1) → Rotational velocity = 1800 and click ok.
12. Click Cell zone condition → Fluid Stator → Edit → Rotation axis direction z(-1) and click ok.
13. Click Define → Boundary Condition → Periodic11 → Periodic Type → Rotation.
14. Click Define → Boundary Condition → Periodic22 → Periodic Type → Rotation.
15. Click Define → Boundary Condition → Pressure Inlet Rotor → Reference frame → Absolute → Rotation axis direction z(-1) and in Turbulence → Specification method → Intensity & Hydraulic diameter → Turbulence intensity = 1 and Hydraulic diameter = 0.074 and click ok.
16. Click Define → Boundary Condition → Pressure outlet stator → Turbulence → Specification method → Intensity & Viscosity ratio →

Backflow Turbulent Intensity% = 1 and Backflow Turbulent Viscosity Ratio =1.

17. Click Define→Boundary Condition → Rotor inlet hub→ Wall motion→ Moving wall→ Motion → Absolute and Rotational→ Rotation-Axis Direction z(-1).

18. Click Define→Boundary Condition → Rotor inlet shroud→ Wall motion→ Moving wall→ Motion → Absolute and Rotational→ Rotation-Axis Direction z(-1).

19. Click Define→Boundary Condition → Rotor shroud → Wall motion→ Moving wall→ Motion → Absolute and Rotational→ Rotation-Axis Direction z(-1).

20. Click Solution methods in partial discretization → Turbulence kinetic energy =Power law and Turbulence dissipation rate = power law.

21. Click Solution controls → Under-relaxation Factor→ Pressure =0.2, Density=1, Body force=1, Momentum=0.5, Turbulent kinetic energy=0.5, Turbulent dissipation rate=0.5, Turbulent viscosity=1.

22. Click Solution Initialization→Standard Initialization→Absolute→Initial Values z=(-1)→ Initialize.

23. Click Run calculation → Number of iteration =1000→ Calculate.

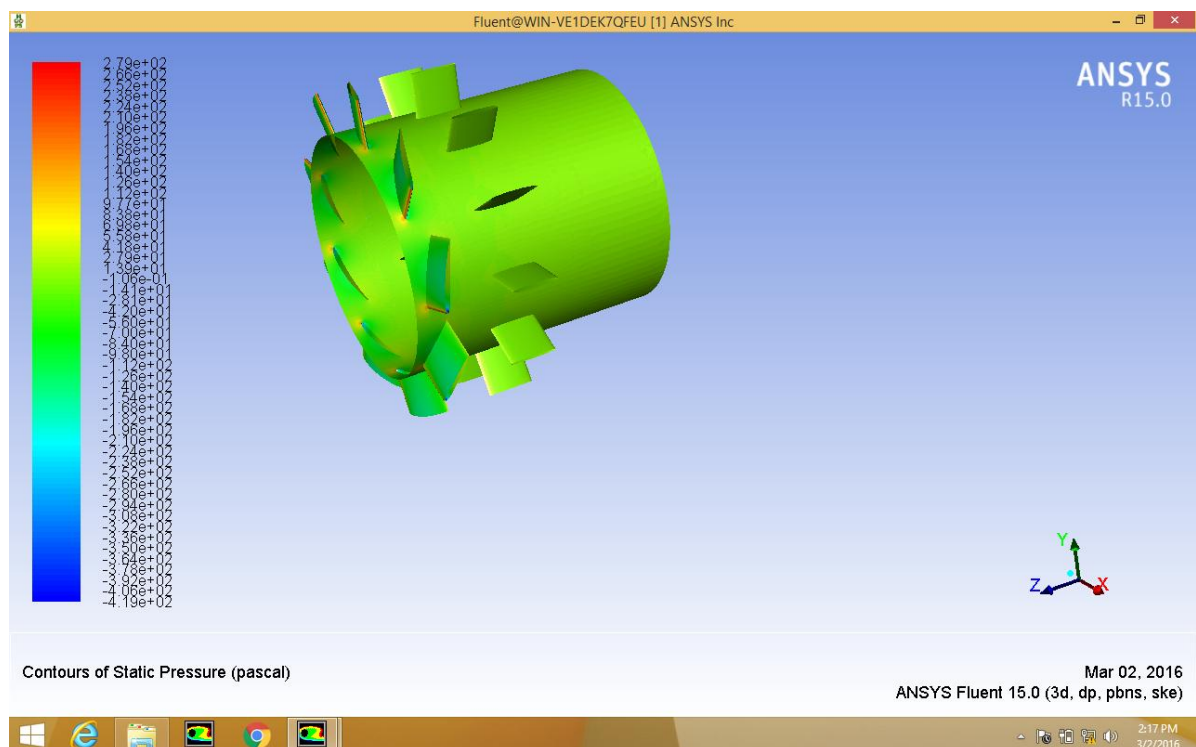
24. Click Graphics and Animations→Contours→Filled, click all surfaces and increase the level to 100 and click display.

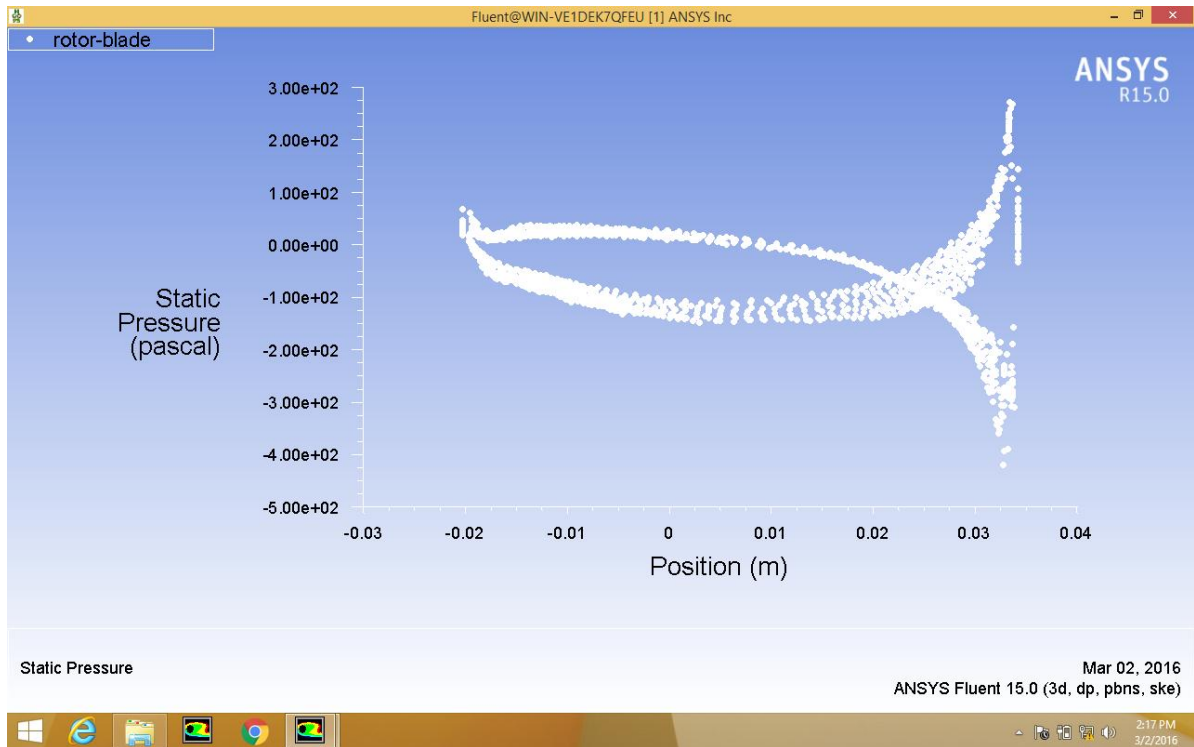
25. Click Display→Views→Periodic report→Define→Fluid stator→Set.

26. Click Display→Lights→Enable Head lights.

27. Click Plots→XY Plots→Select rotor blade→Plot.

28. Click Save and save the case file.





Result:

Thus the hot flow simulation through an axial flow turbine blade passage has been done in ANSYS FLUENT15 and results have been found.

Exp no:

Date:

SIMULATION OF FLOW THROUGH SUBSONIC AND SUPERSONIC DIFFUSERS.

AIM:

To simulate the flow through subsonic and supersonic diffuser.

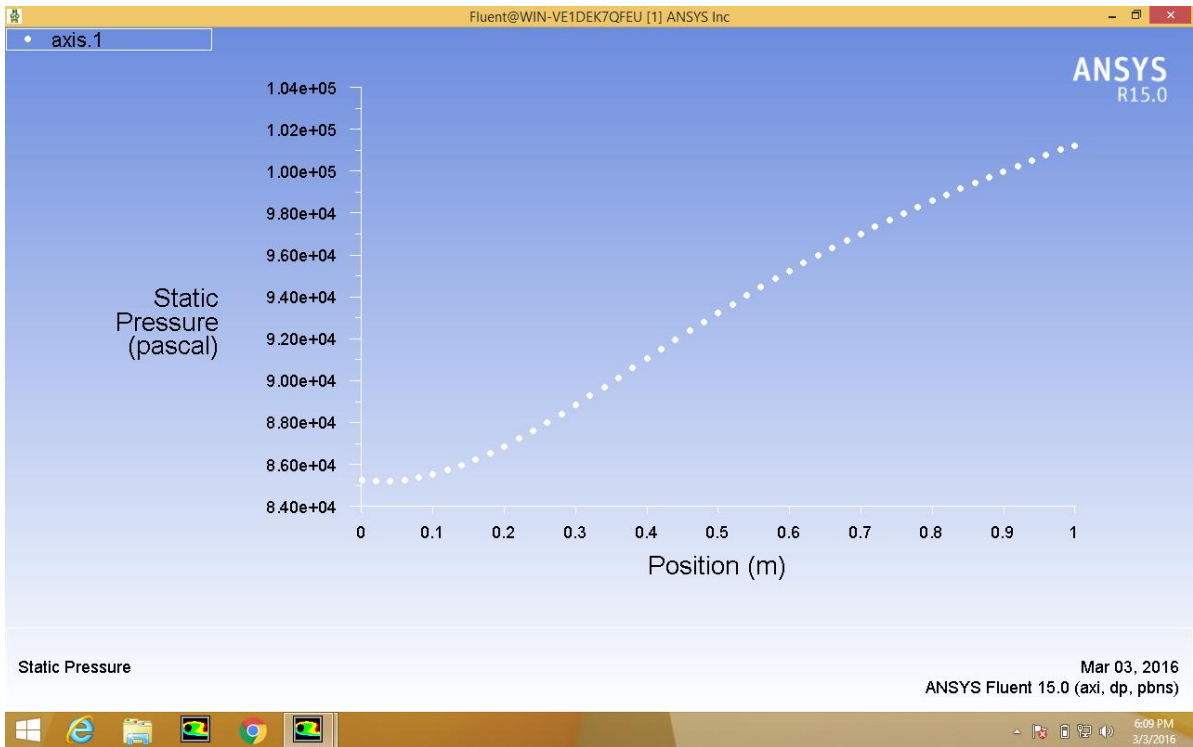
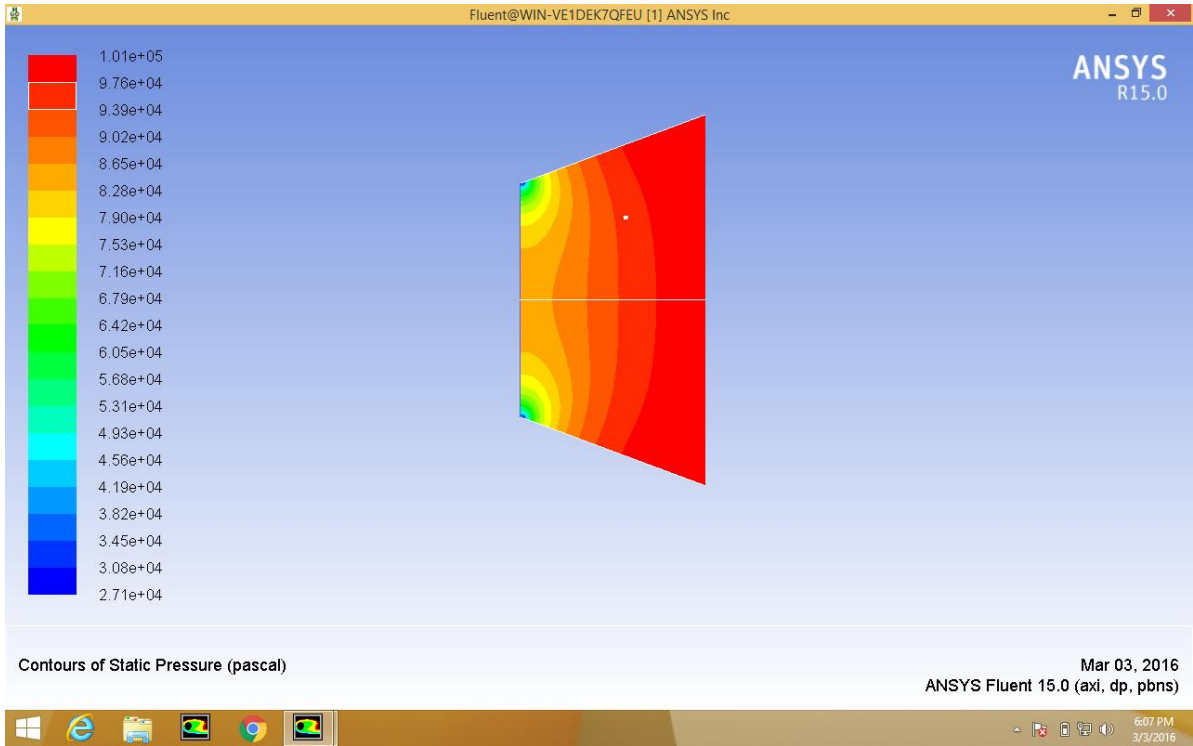
SOFTWARE USED:

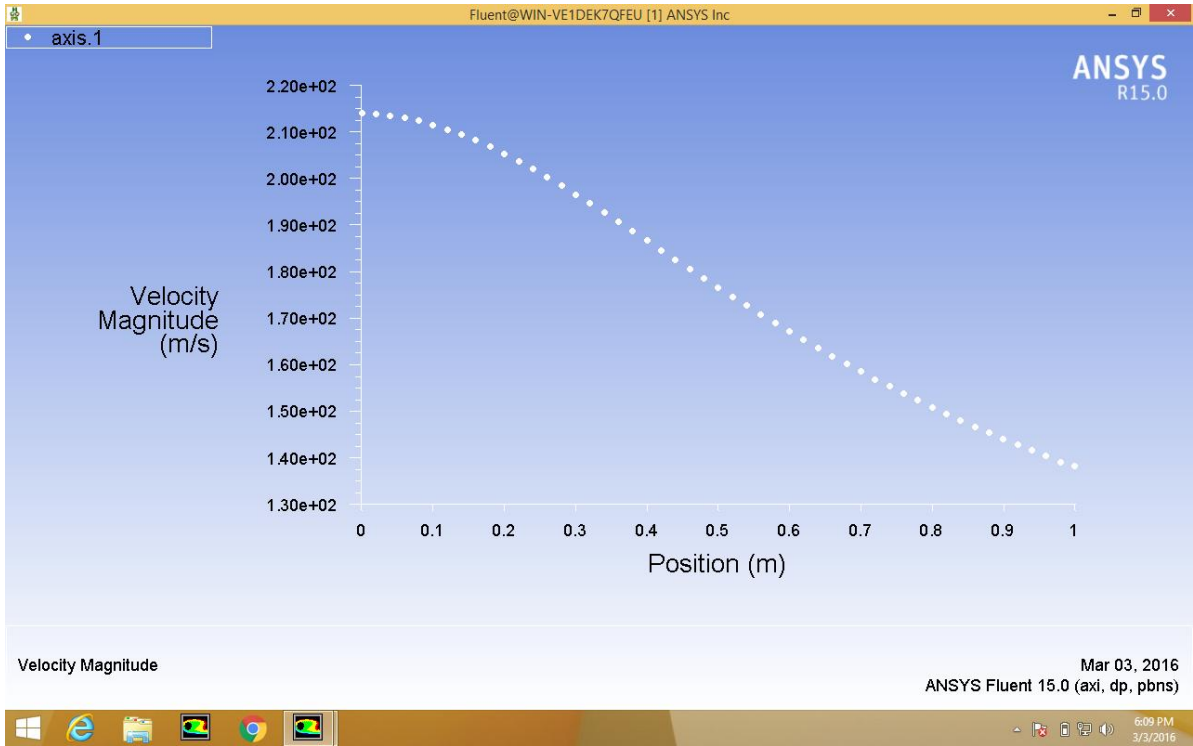
ANSYS FLUENT15

PROCEDURE:

(Subsonic Diffuser)

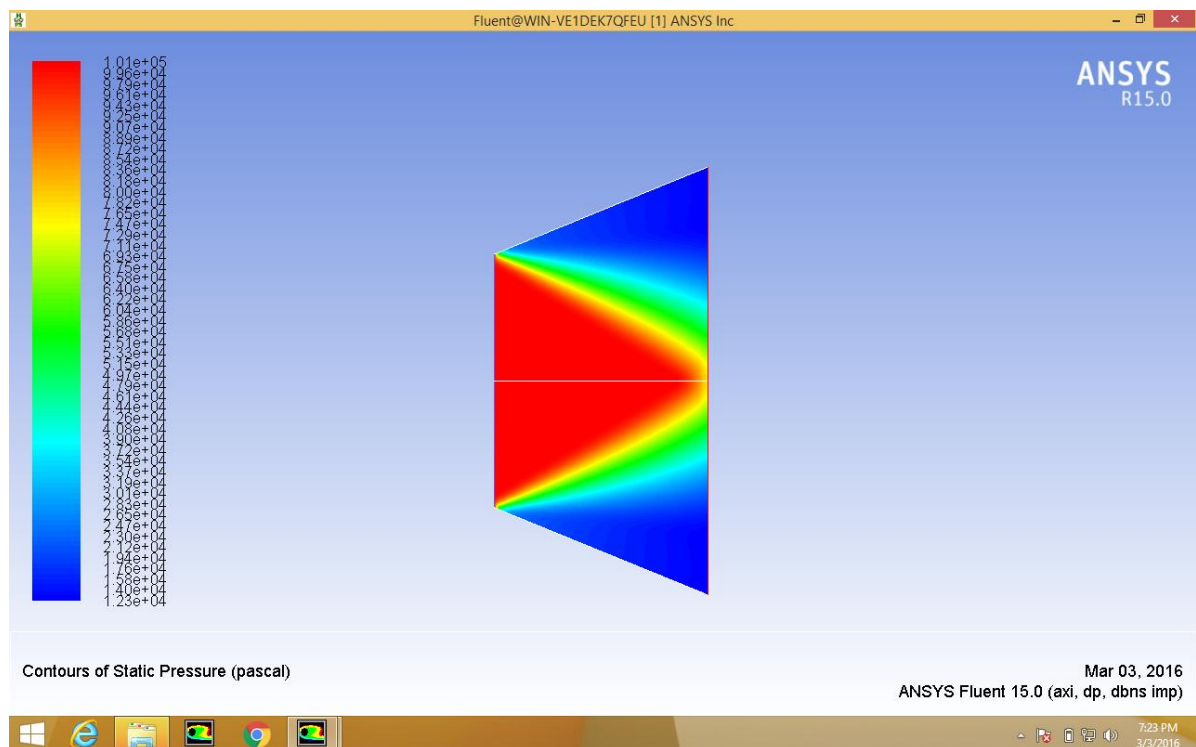
1. Click start → click All Program → click ANSYS FLUENT 15.
2. Turn on radio button to 2d and click ok.
3. Turn on radio button to Pressure-Based and turn on radio button to Axisymmetric.
4. Click File → Read → Mesh and import the mesh file.
5. Click Mesh → Check and conform that the mesh is correct.
6. Click Display → Enable faces and click ok.
7. Click Models → Change laminar flow to in viscid.
8. Click Materials → click Fluid.
9. Click Define → Operating condition → Operating pressure=101325.
10. Click Define → Boundary condition → Pressure_inlet.3 → Edit → GTP=113136.4 and IGP=0.
11. Click Solve → Initialization → Standard initialization → Initialize.
12. Click Run Calculation → Number of iterations=1000 → Calculate.
13. Click Graphics and Animation → Contours → Enable Filled → Display.
14. Click Display → Views → Mirror plane → Axis_4.
15. Click Plots → XY Plots → Pressure vs Static pressure → Plot.
16. Click Plots → XY Plots → Velocity vs Velocity Magnitude → Plot.

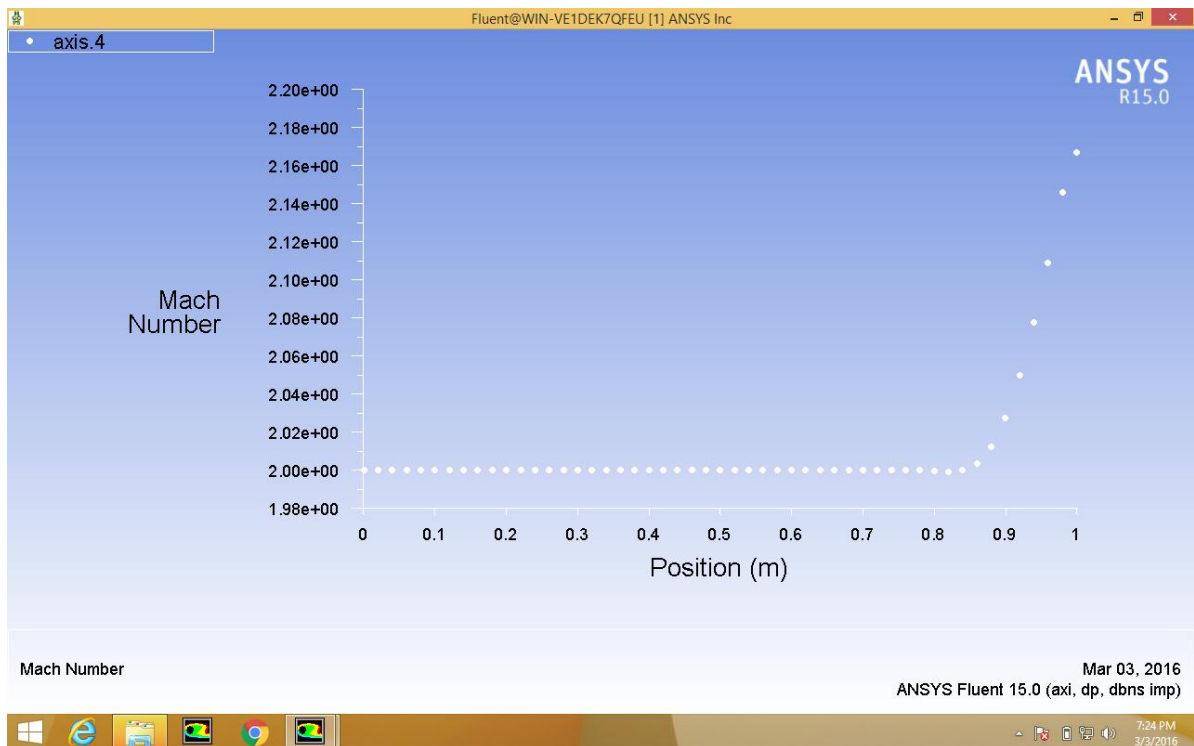
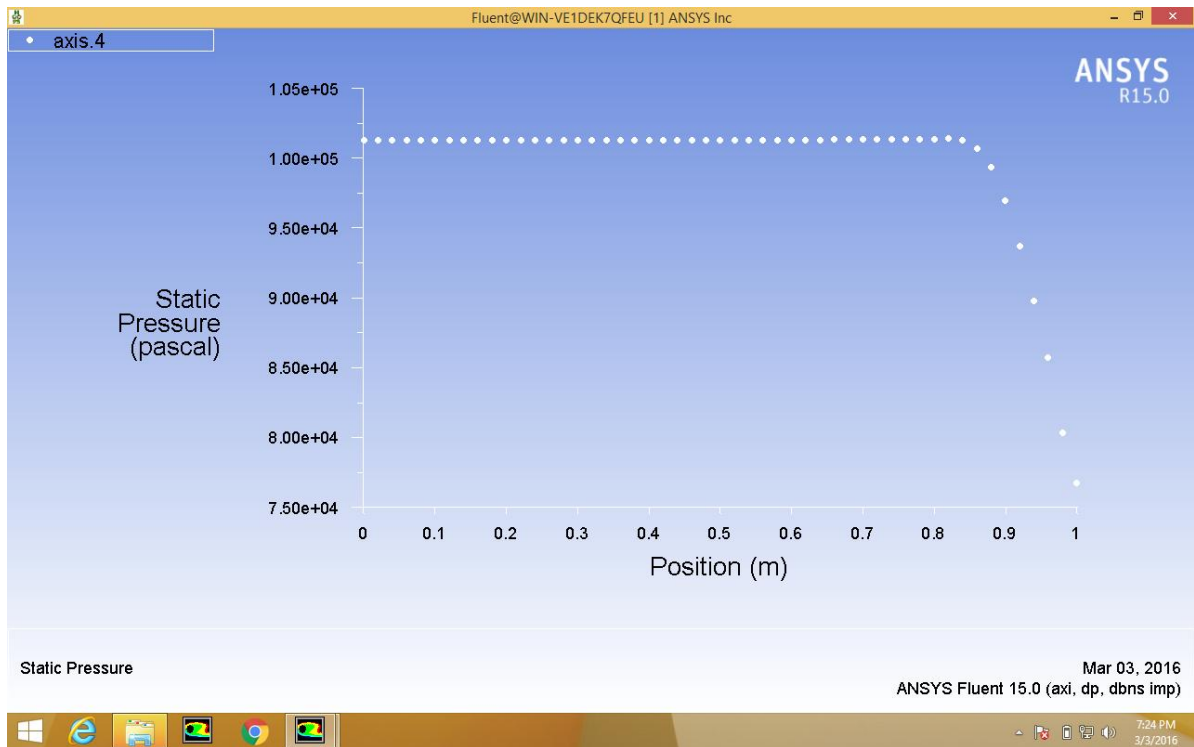




(SUPERSONIC DIFFUSERS)

1. Click start → click All Program → click ANSYS FLUENT 15.
2. Turn on radio button to 2d and click ok.
3. Turn on radio button to Density-Based and turn on radio button to Axisymmetric.
4. Click File → Read → Mesh and import the mesh file.
5. Click Mesh → Check and conform that the mesh is correct.
6. Click Display → Enable faces and click ok.
7. Click Models → Turn on energy equation and change laminar flow to in viscid.
8. Click Materials → click Fluid → change constant to ideal gas.
9. Click Define → Operating condition → Operating pressure=0.
10. Click Define → Boundary condition → Pressure_inlet.2 → Edit → GTP=792840.4 and IGP=101325.
11. Click Solve → Initialization → Standard initialization → Initialize.
12. Click Run Calculation → Number of iterations=1000 → Calculate.
13. Click Graphics and Animation → Contours → Enable Filled → Display.
14. Click Display → Views → Mirror plane → Axis_4.
15. Click Plots → XY Plots → Pressure vs Static pressure → Plot.
16. Click Plots → XY Plots → Velocity vs Mach Number → Plot.
17. Click Save case file.





Result:

Thus the simulation of flow through the Subsonic and supersonic diffusers has been done in ANSYS FLUENT15 and results have been found.

Exp no:

Date:

Analysis of composite laminated structure

Aim:

To design and analysis of composite laminated structure.

Commands used:

- Element Table
- Real constant
- Material properties
- Modeling
- Meshing
- Define loads
- Solution
- Post processing

Tools Required:

Ansys 14.5

Procedure:

1. Open the ansys mechanical APDL.
2. Create the new file and save it in the directory.
3. Select the element type in pre processor and select the required element. The element that can be used for composites are Shell 99, Shell 91, Shell 181, Solid 46 and Shell 191.
4. Select the material properties option in pre processor and give the material properties for various layups of composite materials.
5. Select the modeling option in pre processor and select shell and select layup for and create required layups and orientation of the layups.
6. Select the modeling option in pre processor and select areas and provide the dimension for the layup.
7. Select the meshing option in pre processor and select mesh tool and mesh the areas.
8. Select solution option and select define loads and select structural and provide the loads and constraints as required.
9. Select solve option and give ok.
10. Select Post processing and select plot results and select deformed shape and undeformed.

11. Select Post processing and select contour plot for viewing the various stress and deformation results.

12. Save the file in specified directory.

Commands:

1. Start→All programs→Ansys 15.0-→Mechanical ADPL 15.0
2. Preference→Structural→Ok
3. PreProcessor→Element Type→Add/Edit→Shell→3D 4-node 181→Ok→Close
4. Material properties→Material models→Structural →Linear→Elastic→Orthotropic →

EX	58e9
EY	10e9
EZ	10e9
PRXY	0.29
PRYZ	0.29
PRXZ	0.259
GXY	8e9
GYZ	5e9
GXZ	5e9

Press Ok

Density→2200

5.Material→New model→Define material :2

6.Material model 2→Isotropic

EX	200E9
PXY	0.3

Press Ok

Density→7800

7.Section→Shell→Layup→Add/Edit

Thickness	Material	Orientation
0.001	2	0
0.001	1	-45
0.002	1	45
0.002	2	0

Press Ok

8.Plot section

Range of layers	1	4
-----------------	---	---

Press Ok

9.Modelling→Create→Area→Rectangle→By dimension

X-Cordinates	-0.02	0.02
Y-Cordinates	-0.02	0.02

Press Ok

10.Meshing→Mesh Tool→Mesh→Pick All

11.Meshing→Refine→Nodes→pick all→level of refinement→4→ok

12.Plot control→style→size and shape→display of element→on

13.Solution→analysis type→new analysis→static→ok

14.Define loads→apply→structural→displacement→on lines→isometric view→pick 2 lines in the left and right corners of the plate→ok→all DOF→ok

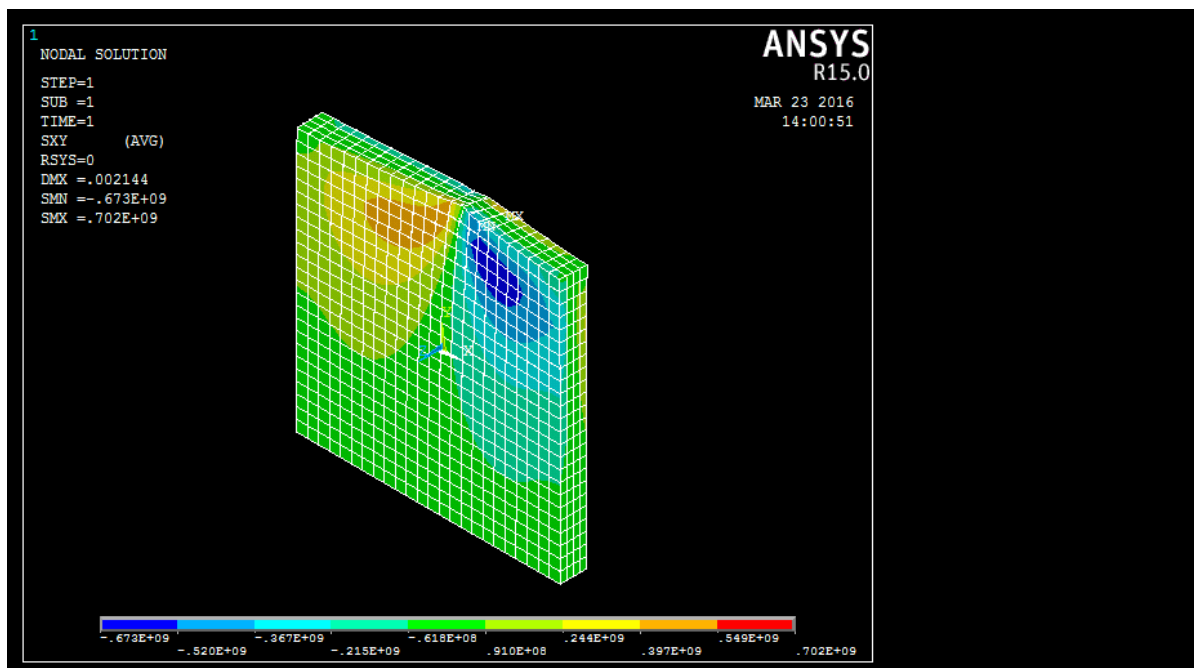
15.Force/moment→nodes→on nodes→select any one node in the upper surface→ok→along two direction FZ

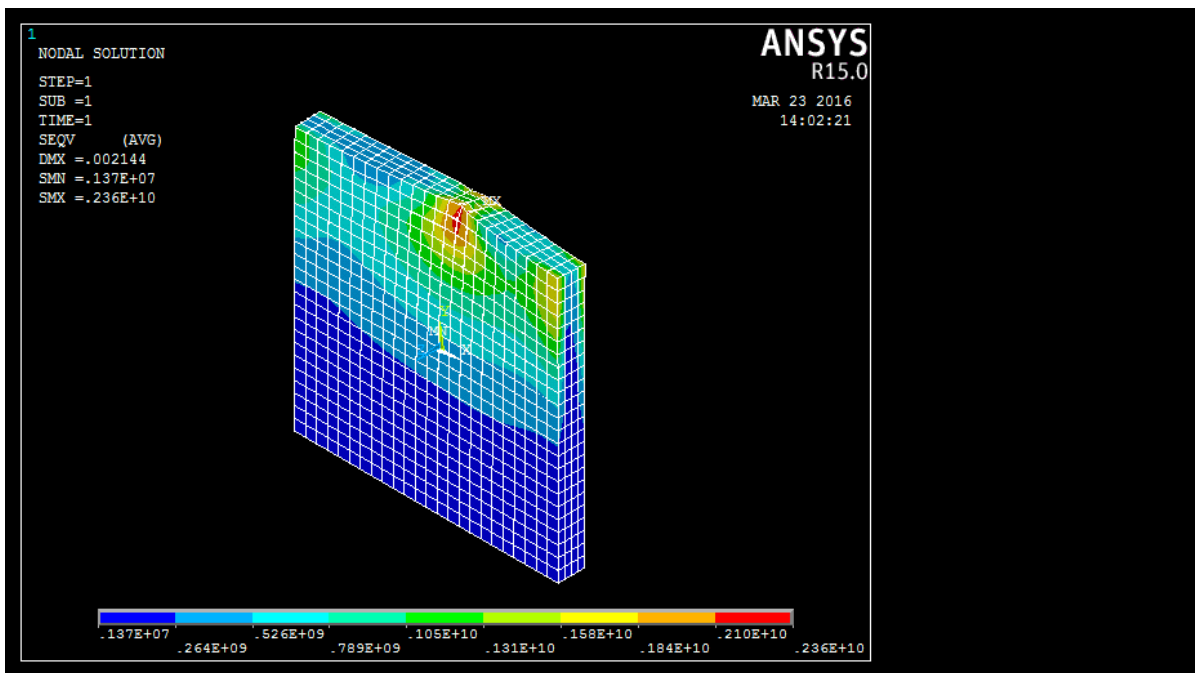
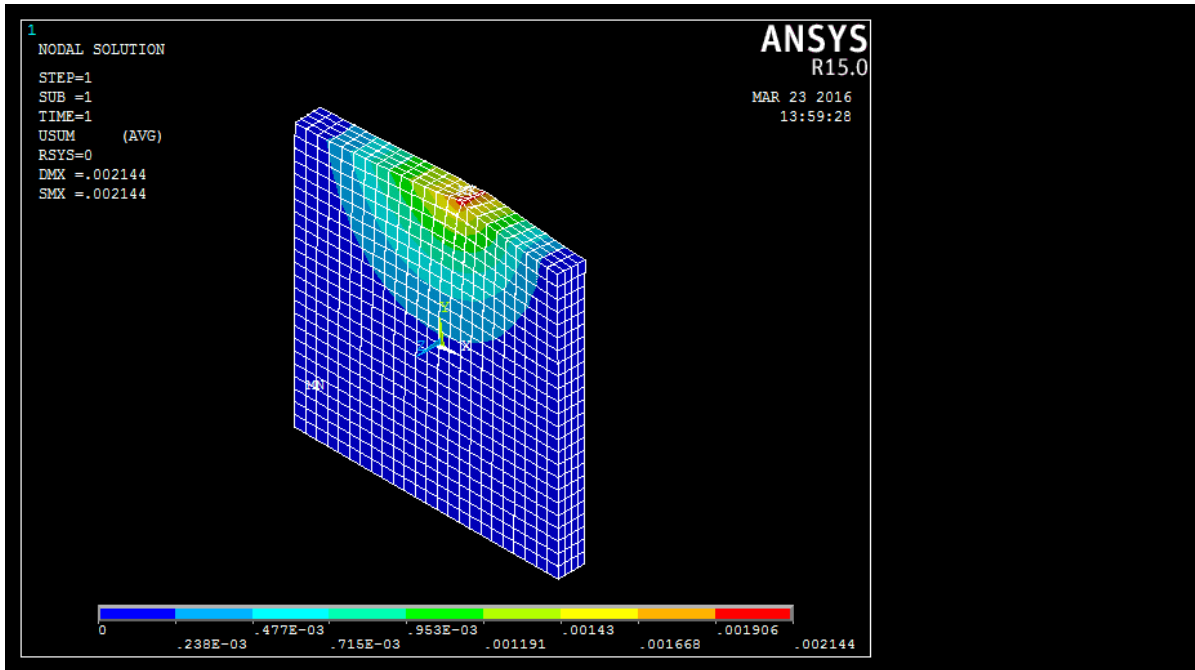
Value is -10000→ok

16.solve→current LS→solution is done

17.general post processor→plot result→contour plot→nodal solution→displacement vector sum/shearstress

18.Capture and save the image→close





Result:

Thus the composite laminate was designed and analysed using Ansys15 software.

Exp no:

Date:

Structural analysis of landing gear

Aim:

To design and analysis of Landing gear.

Commands used:

- Element Table
- Real constant
- Material properties
- Modeling
- Meshing
- Define loads
- Solution
- Post processing

Tools Required:

Ansys 15

Procedure:

1. Open the ansys mechanical APDL.
2. Create the new file and save it in the directory.
3. Select the element type in pre processor and select the required element.
4. Select the material properties option in pre processor and give the material properties .
5. Select the modeling option in pre processor and select keypoints and create keypoints.
6. Select the modeling option in pre processor and select lines and create lines.
7. Select section option and create a beam section of required dimension.
8. Select the meshing option in pre processor and select mesh tool and mesh the lines.
9. Select solution option and select define loads and select structural and provide the loads and constraints as required.
10. Select solve option and give ok.
11. Select Post processing and select plot results and select deformed shape and undeformed.
12. Select Post processing and select contour plot for viewing the various stress and deformation results.

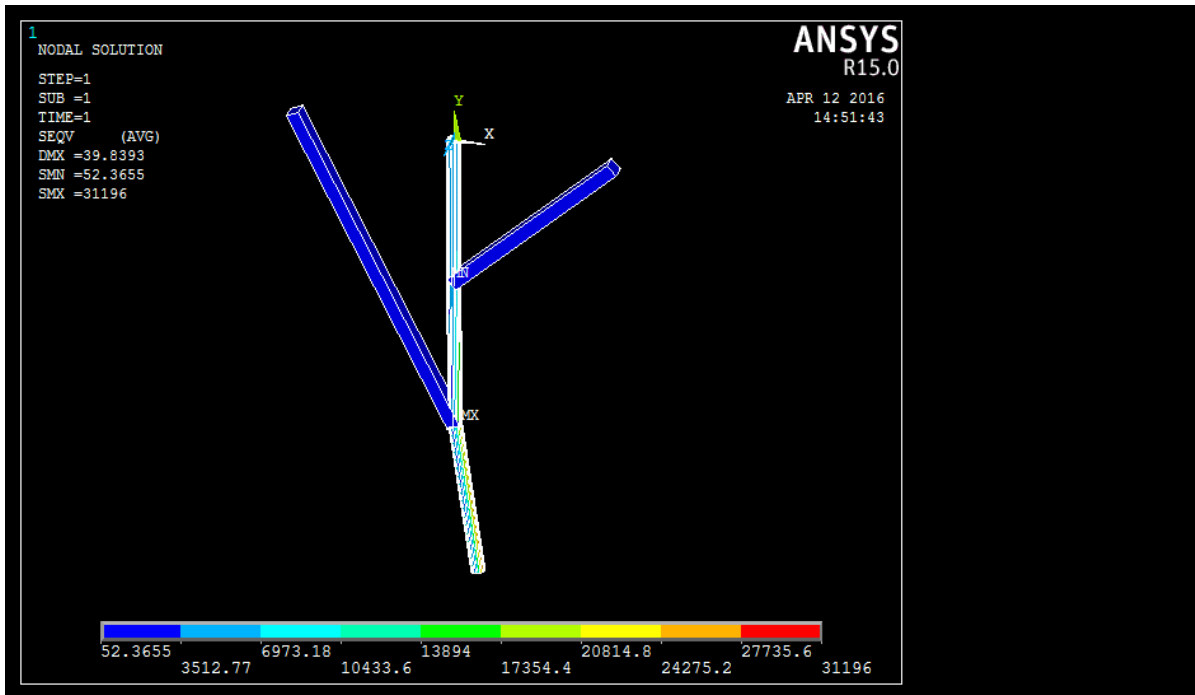
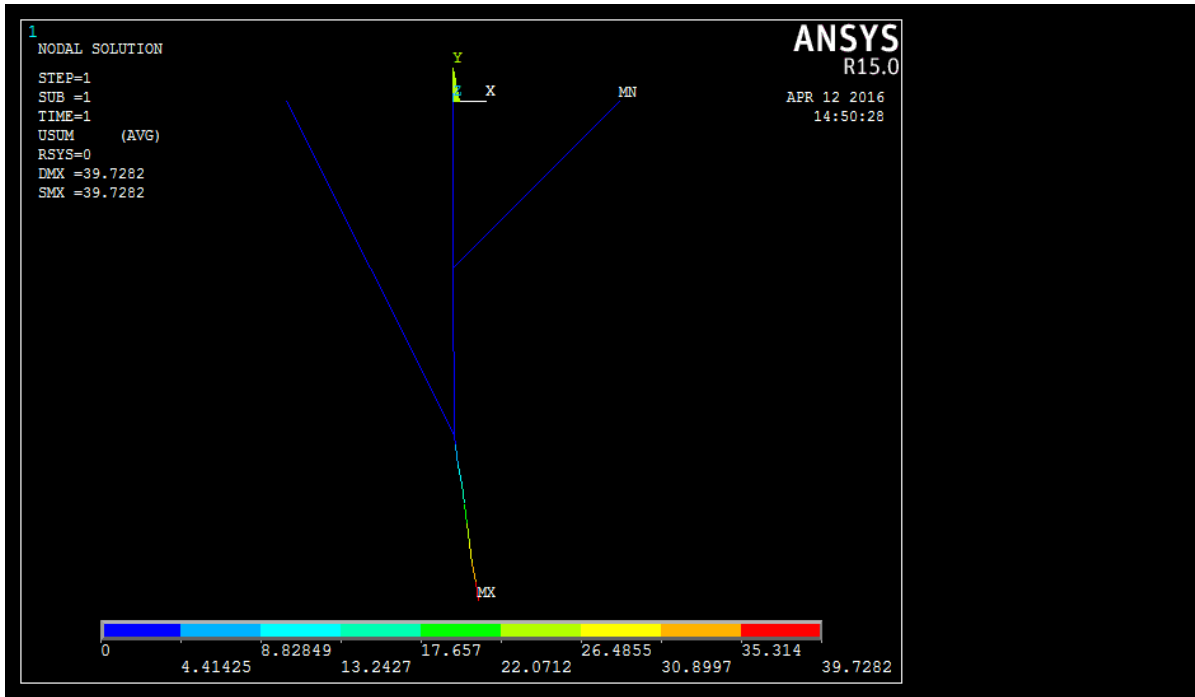
13. Save the file in specified directory.

Commands:

1. Open Ansys Mechanical APDL 15.
2. Click Preference → Enable Structure → ok.
3. Click Preprocessor → Element type → Add Element → Add → Link → 3d finite stn 180 → Apply → Beam → 2 node 188.
4. Click Real Constant → Add → Link 180 → Area =1 → Tension & Compression = Both → ok → close.
5. Click Material Props → Material models → EX = 2e5; PRXY = 0.3; DENS= 2E-9.
6. Click Section → Beam → Common Section → Sub type = rigid → R=0.5; N=20; T=10 → ok.
7. Click Modeling → Keypoints → In Active CS → ok.

X	Y	Z
0	0	0
12	0	0
-12	0	0
0	-12	0
0	-24	0
0	-36	0

8. Click Lines → Straight line → Select all keypoints and make it as a landing gear.
9. Click Meshing → Mesh tool → Set → Link 180 → ok → Line set → Click bar element and mesh the line → NDL = 1 → ok.
10. Click Plots → Multi plots.
11. Click Meshing → Mesh tool → Set → Beam 188 → ok → Line set → Click beam element and mesh the line → NDL =1 → ok.
12. Click Load → Analysis type → New → Static → ok.
13. Click Define load → Apply → Structural → Displacement → On Keypoints → UX,UY,UZ,ROTX,ROTY → Value =0.
14. Click Define load → Apply → Structural → Displacement → On Keypoints → All DOF → Value =0.
15. Click Define load → Apply → Structural → Force → FX=500; FY=500; → ok.
16. Click Solution → Solve → Current LS → OK.
17. Click General postproc → Plot results → Contour plot → Nodal Solution → Von Mises Stress → ok.
18. Click Plot ctrl → Capture image → Save file → ok.



Result:

Thus the landing gear was designed and analysed using Ansys software.

Exp no:

Date:

Structural analysis of aircraft wing

Aim:

To design and analysis of aircraft wing.

Commands used:

- Element Table
- Real constant
- Material properties
- Modeling
- Meshing
- Define loads
- Solution
- Post processing

Tools Required:

Ansys 15

Procedure:

1. Open the ansys workbench.
2. Create the new file and save it in the directory.
3. Select the structural analysis from tool box.
4. Select the engineering data select edit and provide the material properties.
5. Select the geometry and select import geometry and import the iges wing file.
6. Select the model option and select edit geometry
7. Select the meshing option and provide the necessary meshing parameters.
8. Select static structural and select analysis settings and provide the necessary constraints and loads.
9. Select solution and select solve.
10. Select solution and select required stress and deformation for viewing the results.
11. Save the file in specified directory.

Commands:

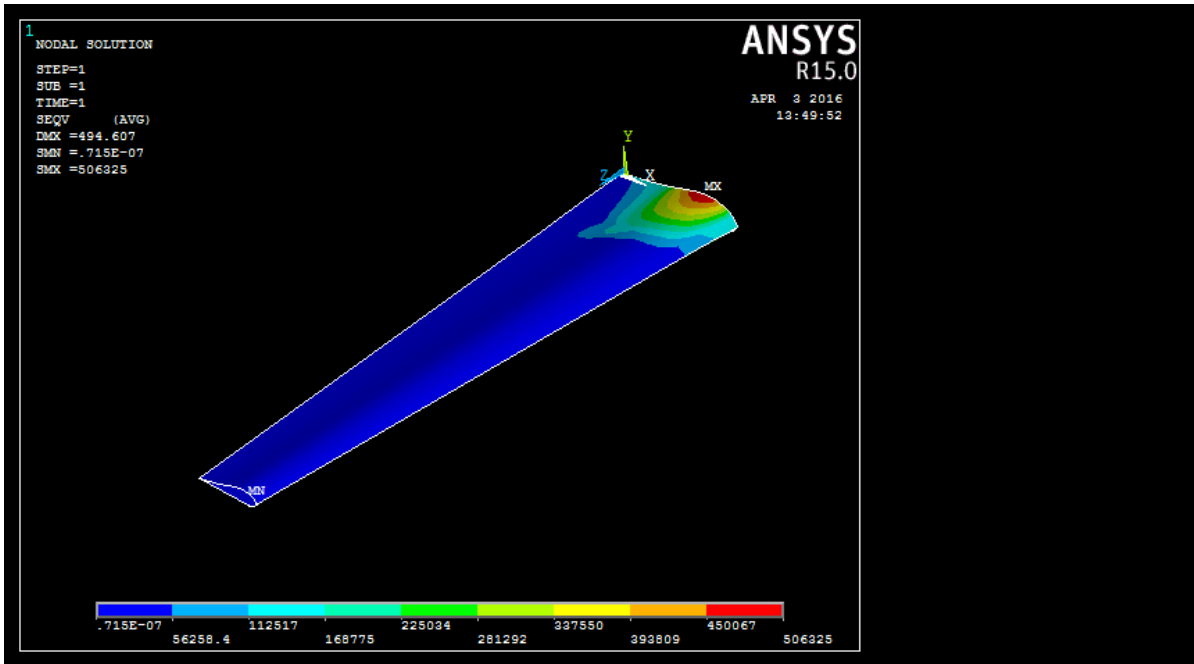
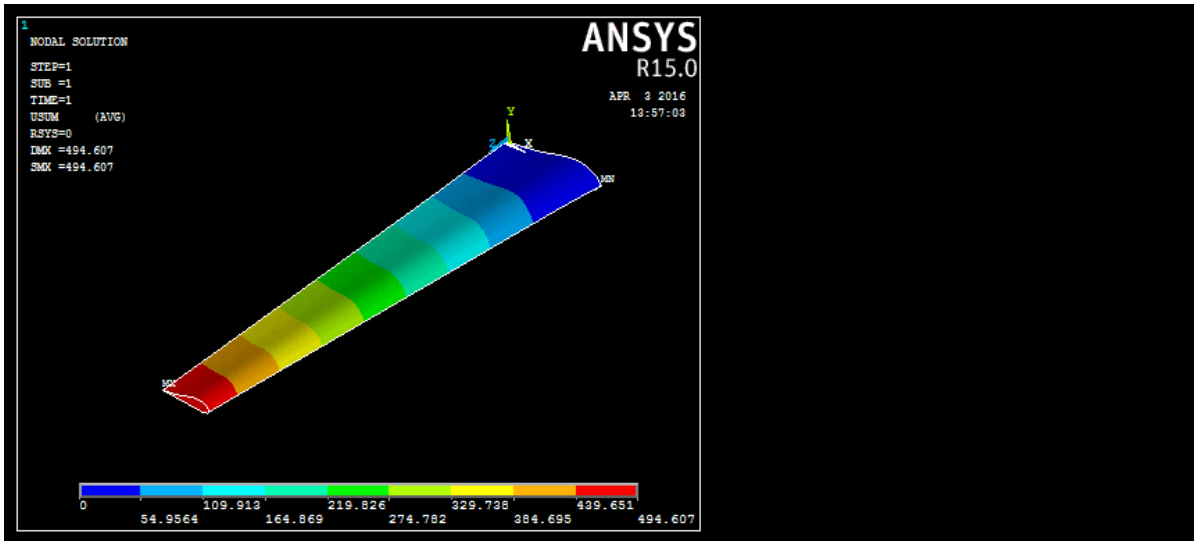
1. Open Ansys Mechanical APDL 15.
2. Click Preference → Enable Structure → ok.
3. Click Preprocessor → Element type → Add → Solid → Brick 8 node 185 → click Apply → click Not solved → Mesh facet 200 → ok.
4. Click Mesh → Option → k_1 → Quad 4 node → ok → close.
5. Materials Props → Materials Models → Structural → Linear → Elastic → Isotropic → $E_x=38e3$; $PRXY=0.3$ → ok → Density → $DENS= 8.3E-5$ → ok.
6. Click Modeling → Create → Keypoints → In Active CS → Enter the keypoints → ok.

NODE	X	Y	Z
1	0	0	0
2	2	0	0
3	2.3	.2	0
4	1.9	.45	0
5	1	.25	0

7. Click line → Straight line → Click keypoint 1 and 2 → ok.
8. Click line → Splines → Spline thru kps → Click key points 1,5,4,3,2 → ok.
9. Click Area → Arbitrary → By lines → Select lines → ok.
10. Click Operate → Extrude → Areas → By xyz offset → Select surface → ok.

DX	DY	DZ	RX	RY	RZ
1.5	1	10	.5	.5	.5

11. Click Mesh Tool → Lines set → Select all lines → ok → $EDL=.1$ → ok.
12. Click Mesh Tool → Mesh → Area → Mesh → Select aerofoil area → ok .
13. Click Plot → Volume.
14. Click Mesh Tool → Set → 2 Mesh 200 → ok → Volume → Click hex radio button → Click sweep radio button → Sweep → ok.
15. Click Loads → Analysis Type → New Analysis → Static → ok.
16. Click Define Loads → Apply → Structure → Displacement → on nodes → Top view → Pan zoom → Box zoom → All DOF → ok.
17. Click Force → On nodes → Bottom View → Pan zoom → Box zoom → Click node → ok → $FY=-10000$ → ok.
18. Click Solution → Solve → Current LS → OK.
19. Click General postproc → Plot results → Contour plot → Nodal Solution → Von Mises Stress → ok.
20. Click Plot ctrl → Capture image → Save file → ok.



RESULTS:

Thus the structural analysis of the tapered wing has been done using Ansys Mechanical APDL 15.

Exp no:

Date:

Structural analysis of fuselage structure

Aim:

To design and analysis of aircraft wing.

Commands used:

- Element Table
- Real constant
- Material properties
- Modeling
- Meshing
- Define loads
- Solution
- Post processing

Tools Required:

Ansys 15

Procedure:

1. Open the ansys workbench.
2. Create the new file and save it in the directory.
3. Select the structural analysis from tool box.
4. Select the engineering data select edit and provide the material properties.
5. Select the geometry and select import geometry and import the iges wing file.
6. Select the model option and select edit geometry
7. Select the meshing option and provide the necessary meshing parameters.
8. Select static structural and select analysis settings and provide the necessary constraints and loads.
9. Select solution and select solve.
10. Select solution and select required stress and deformation for viewing the results.
11. Save the file in specified directory.

Commands:

1. Open Ansys Mechanical APDL 15.

2. Click Preference → Enable Structure → ok.
3. Click Preprocessor → Element type → Add → Shell → 3d 4node 181 → Apply → Structural mass → 3d mass 21 → ok.
4. Click Real constant → Add → 3d mass 21 → ok.

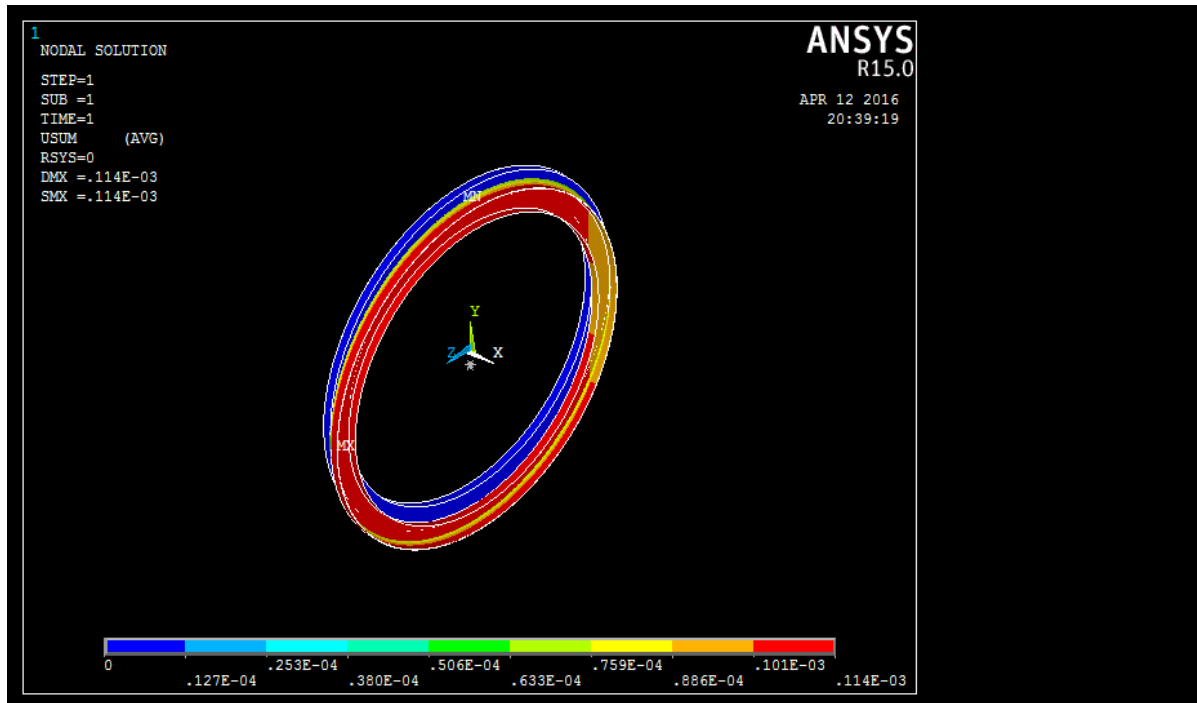
MASSX	1E-20
MASSY	1E-20
MASSZ	1E-20

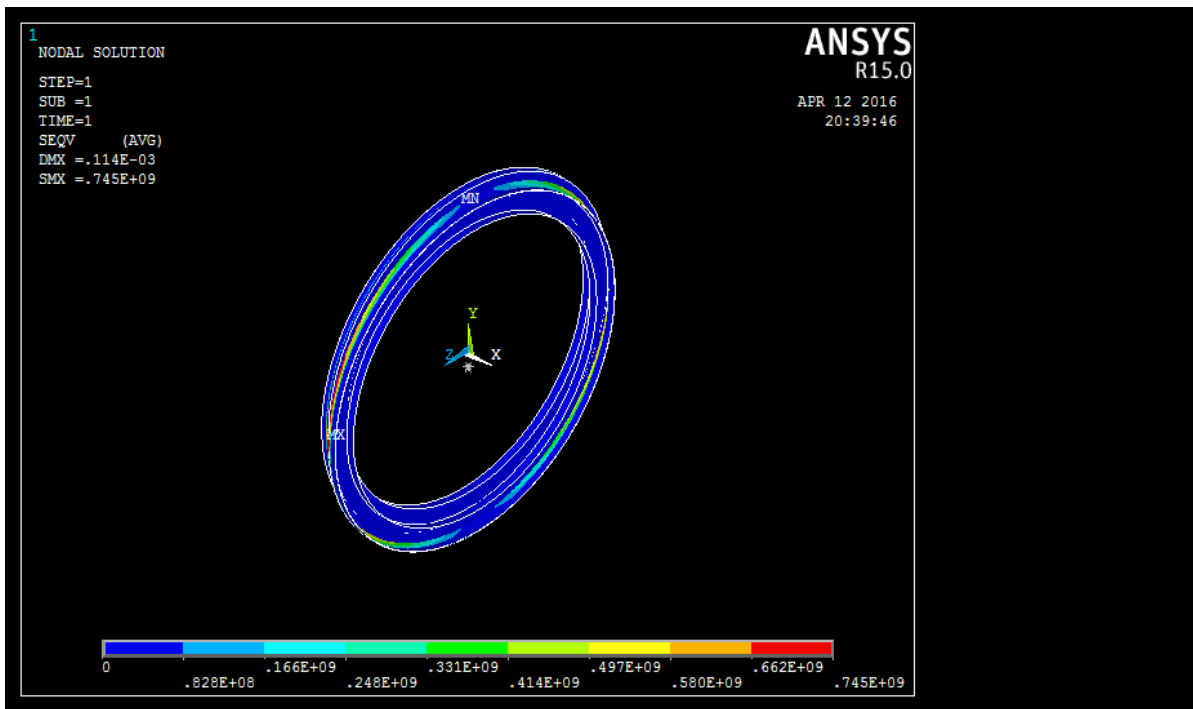
5. Click Material properties → Material models → Structural → Linear → Elastic → Isotropic → E=2e11, PRXY=0.3 → ok.
6. Click Section → Shell → Layup → add → Thickness =1e-5 → ok.
7. Click Modeling → Create → Keypoints → In Active Cs →

X	Y	Z
0	0	0
0.0381	0	0
-0.0381	0	0
-0.0381	0.683	0
0.0381	0.683	0
0.000812	0.683	0
-0.000812	0.683	0
-0.000812	0.7846	0
0.000812	0.7846	0
0.0381	0.7846	0
-0.0381	0.7846	0
-0.0381	0.7846+0.00102	0
0.0381	0.7846+0.00102	0

8. Click Lines → Straight lines → click the key points and make it like I section →
9. Click Plot → Multiplots.
10. Click operate → Extrude → Lines → about an axis → Click keypoints → No.of.area =5 → ok.
11. Click Meshing → Mesh tool → Mapped → Mesh → Pick all.
12. Click Mesh tool → Keypoints → Mesh → Origin → ok.
13. Click Mesh Attributes → Default Attribute → Element type no → Mass 21 → ok.
14. Click Coupling & Ceqn → Rigid region → Origin → ok → topview → Box → select right half → ok → DOF → All applicable → ok.
15. Click Loads → Define Loads → Apply → Structural → Displacement → on nodes → top view → Box → Select left half → ok → All DOF → ok.
16. Click Force → On nodes → Origin → ok → FY=-10000 → ok.
17. Click Moment → On nodes → Origin → ok → MX=1000 → ok.
18. Click Solution → Solve → Current LS.

19. Click General Postprocess → Plot Results → Deformed Shape → Def+Undeform → ok.
20. Click Contour plot → Nodal Solution → Stress → Von mises stress → ok.
21. Click Plot Ctrl → Capture image → Save.





Result:

Thus the structural analysis of the fuselage structure has been done using Ansys Mechanical APDL 15.