

MANUAL
For
CAD / CAM Laboratory

B.E Mechanical and Production



Mechanical Engineering Department

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Course Outcomes

At the end of the course, the student

1. Will be able to draw isometric and orthogonal projections and sectional views of various mechanical components.
2. Will be able to draw free hand sketches of various mechanical components
3. Will be able to understand the shape and structure of different types of joints, screws, keys and Couplings
4. Will be sufficiently knowledgeable to use both the software and drafter to produce assembly views of various mechanical components from part drawings.

List of Experiments:

I. Machine Drawing (AutoCAD):

1. Format of drawing sheet & title block,
2. Conventions of drawing lines and dimensions,
3. Convention for sectional views.
4. Simple machine elements.
5. Riveted and screwed fastenings.
6. Joints and coupling.

II. Assembly drawing (SOLIDWORKS/ CATIA/ PRO-E):

7. Connecting rod.
8. Eccentric.
9. Cross head.
10. Stuffing box.
11. Lathe Tool Post.
12. Revolving centre.
13. Pedestal bearing (Plummer block).
14. Screw Jack.

Note: The test is for the ability of the student to read and interpret drawing. The drawing should include part list in standard format.

INTRODUCTION TO CAD/CAM

CAD/CAM is a term which means computer-aided design and computer manufacturing. It is the technology concerned with the use of digital computers to perform certain functions in designs and production. Thus technology is moving in the direction of greater integration of design and manufacturing, two activities which have traditionally been treated as distinct and separate functions in a producing firm the computer systems consist of the hardware and software to perform the specialized design functions required by the particular user firm. The CAD hardware typically includes the computer, one or more graphics display terminals, keyboards, and other peripheral equipment. The CAD software consists of the computer programs to implement computer graphics on the system plus application programs to implement functions of the user company. Examples of these application programs include stress-strain analysis of components, dynamic response of mechanisms, heat transfer calculations, and numerical control part programming. These factors give rise to differences in CAD system requirements.

SOLID MODELLING (or **modelling**) is a consistent set of principles for mathematical and computer modelling of three dimensional solids. Solid modelling is distinguished from related areas of geometric modelling and computer graphics by its emphasis on physical fidelity. Together, the principles of geometric and solid modelling form the foundation of computer aided design and in general support the creation, exchange, visualization, interrogation, and annotation of digital models of physical objects.

The use of solid modelling techniques allows for the automation of several difficult engineering calculations that are carried out as a part of the design

process. Simulation, planning and verification of processes such as machining and assembly were one of the main catalysts for the development of solid modelling. More recently, the range of supported manufacturing applications has been greatly expanded to include sheet metal manufacturing, injection moulding, welding pipe routing etc. Beyond traditional manufacturing, solid modelling techniques serves as the foundation for rapid prototyping, digital data archival and reverse engineering by reconstructing solids from samples points on physical objects, mechanical analysis using finite elements, motion planning and NC path verification, kinematic and dynamic analysis of mechanisms, and so on. A central problem in all these applications is the ability to effectively represent and manipulate 3-D geometry in a fashion that is consistent with the physical behaviour of real artefacts.

SOLID MODELLING METHODS

There are two basic methods used to create solid models. They are Constructive Solid Geometry(CSG) methods, and Boundary Representation(b-rep) methods.

CSG uses solid primitives (rectangular prisms, spheres, cylinders, cones, etc) and Boolean operations (unions, subtractions, intersections) to create the solid model. B-rep methods start with one or more wire frame profiles, and create a solid model by extruding, sweeping, revolving or skinning these profiles. The Boolean operations can also be used on the profiles themselves and the solids generated from these profiles. Solids can also be created by combining surfaces, which often have complex shapes, through a sewing operation. This can be used for example, to create the body of an aerodynamic vehicle such as an airplane, with its carefully designed wing profiles. Further details on these two methods can be found in Zeid. These two methods can often be combined in order to create the desired parts.

Each of these methods has its limitations, and parts which are very difficult to create using just one or the other method can be created much more easily by using a combination of both methods. Thus most commercial solid modelling systems are hybrids using both CSG and B-rep methods.

CAD PACKAGE

AutoCad is a software application for computer aided design and drafting. The software supports both 2D and 3D formats. The software is developed and sold by Autodesk. It was first released in December 1982 by Autodesk in the year following the purchase of the first form of the software by Autodesk founder, John Walker. AutoCAD is Autodesk's flagship product and by March 1986 had become the most ubiquitous microcomputer design program in the world, utilizing functions such as polylines and curvefitting. Prior to the introduction of AutoCAD, most other CAD programs ran on mainframe computers or minicomputers, with each CAD operator (user) working at a graphical terminal or workstation. According to Autodesk company information, the AutoCad software is now used in a range of industries, employed by architects, project managers and engineers, amongst other professions, and as of 1994 there had been 650 training centres established across the world to educate users about the company's primary products

AutoCAD LT

AutoCAD LT is the lower cost version of AutoCAD with reduced capabilities, first released in November 1993. Autodesk developed AutoCAD LT to have an entry level CAD package to compete in the lower price level. AutoCAD is priced at \$495, became the first AutoCAD priced below \$1000. It is sold directly by Autodesk and can be also be purchased at computer stores

As of the 2011 release of the AutoCAD LTMSRP has risen to \$1200. While there are hundreds of small differences the full AutoCAD package and AutoCAD LT, currently there are a few major recognized differences in the software features

- 3D capabilities: AutoCADLT packs lack the ability to create visualise and render 3d models as well as 3D printing
- Network licensing: AutoCAD LT cannot be used on multiple machines over a network
- Customization: AutoCAD LT does not support customization with LIS,ARX and VBA
- Management and automation capabilities with sheet manager and action recorder
- CSD standards management tools

CATIA(computer AideCAD three dimensional interactive application)

It is a multi- platform CAD/CAM/CAE commercial software suite developed by the French company Dassault systems and marketed worldwide by IBM. Written in the C++ programming language, CATIA is the cornerstone of the Dassault Systems product lifecycle management software suite.

The software was created in the late 1970's and early 1980's to develop Dassaults mirage fighter jet, then was adopted in the aerospace automotive shipbuilding and other industries.

CATIA competes in the **CAD/CAM/CAE** market with **SiemensNX,Pro/ENGINEER, Autodesk Inventor and SolidEdge**

CATIA started as an in house development in 1977 by French aircraft manufacturer Avions Marcel Dassault, at that time customer of the CADAM CAD software

Initially named CATI (Conception Assistee Tridimensionnelle Interactive) it was renamed CATIA in 1981, when Dassault created a subsidiary to develop and sell the software, and signed a non-exclusive distribution agreement with IBM

In 1984, the Boeing Company chose CATIA as its main 3D CAD tool, becoming its largest customer

In 1988, CATIA version 33 was ported from mainframe computers to UNIX

In 1990, General Dynamics Electric Boat Corp chose CATIA as its main 3D CAD tool, to design the U.S Navy Virginia class submarine

In 1992, CADAM was purchased from IBM and the next year CATIA CADAM V4 was published. In 1996 it was ported from one to four UNIX operating systems, including IBM AIX, Silicon Graphics IRIX, Sun Microsystems SunOS, and Hewlett-Packard HP-UX

In 1998, an entirely rewritten version of CATIA, CATIA V5 was released, with support for UNIX, Windows NT and Windows Xp since 2001

In 2008, Dassault announced and released CATIA v6. While the server can run on Microsoft Windows, Linux or AIX, client support for any operating system other than Microsoft Windows is dropped

Commonly referred to as a 3D product Lifecycle Management software suite, CATIA supports multiple stages of product development (CAx), from conceptualization, design (CAD), manufacturing (CAM) and engineering (CAE)

CATIA can be customized via application programming interfaces (API). V4 can be adapted in the FORTRAN and C programming languages under an API called CAA. V5 can be adapted via the visual basic and C++

programming languages, an AI called CAA2 or CAA v5 that is a component model (COM) like interface

CAM PACKAGES

HYPERMESH

Altair Hyper Mesh is a high performance finite element pre and post-processor for major finite element solvers, allowing engineers to analyse design conditions in a highly interactive and visual environment. *HyperMesh's user interface* is easy to learn and supports the direct use of CAD geometry and existing finite element models, providing robust interoperability and efficiency. Advanced automation tools within Hyper Mesh allows the user to optimize meshes from a set of quality criteria, change existing meshes through morphing and generate mid surfaces from models of varying thickness.

BENEFITS OF HYPERMESH

- Reduce time and engineering analysis cost through high performance finite element modelling and post processing
- Reduce learning time and improve productivity with an intuitive user interface and best in class functionality
- Reduce redundancy and model development costs through the direct use of CAD geometry and existing file finite elements models
- Simplify the modelling process for complex geometry through high speed, high quality auto mesing
- Broadest support of commercial solvers by providing direct interfaces to a wide array of analysis codes ensuring the best code can be run for specific situations
- Cost-effective pricing to deliver maximum functionality for your software investment.

CNC Software/Master cam

Founded in Massachusetts in 1983, **CNC software Inc**, is one of the oldest developers of PC based computer aided design/computer aided manufacturing (CAD/CAM) software. They are one of the first to introduce CAD/CAM software designed for both machinists and engineers. **Master cam**, CNC softwares main product, started as a 2D CAM system with CAD tools that let machinists design virtual parts on a computer screen and also guided computer numerical controlled (CNC) machine tools in the manufacture of parts. Since then, Mastercam has grown into the most widely used CAD/CAM package in the world. CNC software, Inc. is now located in Tolland Connecticut.

Mastercams comprehensive set of predefined toolpath, including countour, drill, pocketing, face, peel, mill, engraving, surface high speed, advanced multiaxis, and many more- enable machinists to cut parts efficiently and accurately. Mastercam users can create and cut parts using one of the many supplied machine and control definitions, or they can use Mastercams advanced tools to create their own customised definitions. Mastercam also offers a level of flexibility that allows the integrations of 3rd party applications, called C-hooks, to address unique machine or process specific scenarios

Mastercam's name is a double entendre: it implies mastery of CAM, which involves todays latest machine tool control technology; and it simultaneously pays homage to yesterday's machine tools control technology by echoing the older term master cam, which referred to the main cam or model that a traces followed.

CAE PACKAGES

ANSYS

ANSYS is a general purpose software, used to simulate interactions of all disciplines of physics, structures, vibrations, fluid dynamics, heat transfer and electromagnetic for engineers

So ANSYS, which enables to simulate tests or working conditions, enables to test in virtual environment before manufacturing prototypes of products. Furthermore determining and improving weak points, computing life and foreseeing probable problems are possible by 3D simulations in virtual environment

ANSYS software with its modular structure as seen in the table below gives an opportunity of taking only needed features. ANSYS can work integrated with other used engineering software on desktop by adding CAD and FEA connection modules

ANSYS can import CAD data and also enables to build a geometry with its “preprocessing” abilities. Similarly in the same pre-processor, finite element model (a.k.a mesh) which is required for computation is generated. After defining loadings and carrying out analyses, results can be viewed as numerical and graphical

ANSYS can carry out advanced engineering analyses quickly, safely and practically by its variety of contact algorithms, time based loading features and non-linear material models

ANSYS Workbench is a platform which integrates simulation technologies and parametric CAD systems with unique automation and performance. The power of ANSYS workbench comes from ANSYS solver algorithms with

years of experience. Furthermore, the object of ANSYS workbench is verification and improving of the product in virtual environment

ANSYS workbench, which is written for high level compatibility especially with PC, is more than an interface and anybody who has an ANSYS license can work with ANSYS workbench. As same as ANSYS interface, capacities of ANSYS workbench are limited due to possessed license

LS DYNA

LSDYNA is an advanced general purpose multiphysics simulation software package developed by the Livermore software technology corporation (LSTC). While the package continues to contain or and more possibilities for the calculation of many complex, real world problems, its origins and core competency lie in highly nonlinear transient dynamic finite elements analysis (FEA) using explicit time integration. LS DYNA is being used by the automobile, aerospace, construction, military, manufacturing, and bioengineering industries.

LS DYNA originated from the 3D FEA program DYNA3D, developed by Dr. John O Hallquist at Lawrence Livermore National Laboratory (LLNL in 1976. DYNA3D was created in order to simulate the impact of full fusing option (FUFO) or “dial a yield” nuclear bomb for low altitude release (impact velocity of 40 m/s). At the time no 3D software was available to simulate the impact and 2D software was inadequate. While the FUFO bomb was cancelled, development of DYNA3D continued. DYNA3D used explicit time integration to study non linear dynamic problems, with the original applications being mostly stress analysis of structures undergoing various types of impacts. The program was initially very simple largely due to the lack of computational resources at that time. A 2 dimensional version of the same software was developed concurrent. In 1978, the DYNA3D

source code was released into the public domain without restrictions after a request from France

In 1979 a new version of DYNA3D was released which was programmed for optimal performance on the CRAY-1 supercomputers. This new release contained improved sliding interface treatment which was an order of magnitude faster than the previous contact treatment. This version also eliminated structural and higher order solid elements of the first version, while including element wise integration of the integral difference method developed in 1974

The 1982 release included nine additional material models which allowed for new simulations, such as explosive structures and soil structure interactions. The release also permitted the analysis of structural response due to penetrating projectiles. Improvements in 1982 further boosted the execution speed by about 10%. Hallquist was the sole developer of DYNA3D until 1994, when he was joined by Dr. David J Benson. In 1986, many capabilities were added. The added features included beams, shells, rigid bodies, single surface contact, interface friction, discrete springs and dampers, optional hourglass treatment, optional exact volume integration, and VAX/VMS, IBM, UNIX, COS operating system compatibility. At this point, DYNA3D became the first code to have a general single surface contact algorithm.

SOLIDWORKS

INTRODUCTION

Solid works is the state of the art in computer aided design (CAD). SolidWorks represents an object in a virtual environment just as it exists in reality, i.e., having volume as well as surfaces and edges. This, along with exceptional ease of use, makes SolidWorks a powerful design tool. Complex 3-D parts with contoured surfaces and detailed features can be modelled quickly and easily with SolidWorks. Then many parts can be assembled in a virtual environment to create a computer model of the finished product. In addition, traditional engineering drawings can be easily extracted from the solids models of both the parts and final assembly. This approach opens the door to innovative design concepts, speeds product development, and minimising design errors. The result is the ability to bring the high quality products to the market.

SolidWorks Corporation developed SolidWorks registered as a three dimensional, feature based, solidmodelling system for personal computers. Solid modelling represents objects in a computer as volumes, rather than just as collections of edges and surfaces. Features are three-dimensional geometries with direct analogies to shapes that can be machined or manufactured, such as holes or rounds. Feature based solid modelling creates and modifies the geometric shapes of an object in a way that represents common manufacturing processes. This makes SolidWorks a very powerful and effective tool for engineering design.

As with other computer programs, Solidworks organizes and stores data in files. Each file has a name followed by a period (dot) and an extension. There are several file types used in Solidworks, but the most common file types and their extension are

Part files .prt or .sldprt

Assembly files .asm or .sldasm

Drawing files .drw or .slddrw

Part files are the files of the individual parts that are modelled. Part files contain all of the pertinent information about the part. Because SolidWorks is a solid-modelling program, the virtual part on the screen will look very similar to the manufacturing part.

Assembly files are created from several individual part files that are virtually assembled (in the computer) to create the finished product

Drawing files are the two dimensional engineering representations of both the part and the assembly file. The drawings should contain all of the necessary information for the manufacturing of the part, including tolerances, dimensions, and so on.

The part file is the *driving* file for all other file types. The modelling procedure begins with part files. Subsequent assemblies and drawings are based on the original part files. One advantage of the SolidWorks files is the feature of dynamic links. Any change to a part file will automatically be updated in any corresponding assembly or drawing file.










Therefore, both drawing and assembly files must be able to find and access their corresponding part files in order to be opened. SolidWorks uses information embedded within the file and the filename to maintain these links automatically.

SOLID WORKS MODULES

Some of the modules listed

- Sheet metal Design
- Advanced Surface Modelling
- Weldments
- Mold Design using SolidWorks
- Simulation Training
- 3DVIA composer Training
- Driveworks Training
- SolidCAM Training
- SolidWorks AI Fundamentals

The 2D to 3D tools will help you to convert a 2D drawing to a 3D part. Some of the tools can be used in any sketches.

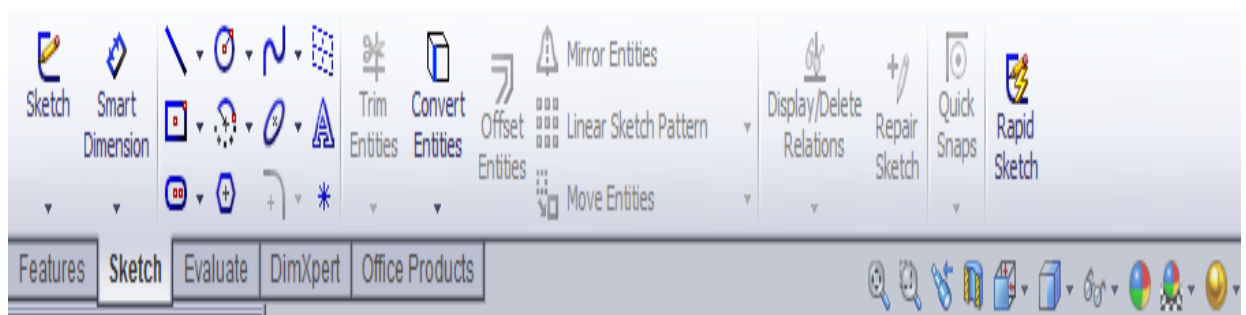
-  ➤ **FRONT**- the selected sketch entities become the front view in converting to a 3D part
-  ➤ **TOP**- the selected sketch entities become the top view in converting to a 3D part
-  ➤ **RIGHT**- the selected sketch entities become the right view in converting to a 3D part.
-  ➤ **LEFT**- the selected sketch entities become the left view in converting to a 3D part
-  ➤ **BOTTOM**- the selected sketch entities become the bottom view in converting to a 3D part
-  ➤ **BACK**- the selected sketch entities become the back view in converting to a 3D part
-  ➤ **Auxiliary**- the selected sketch entities become an auxiliary view in converting to a 3D part. You must select a line in another view to specify the angle of the auxiliary view
-  ➤ **Create Sketch from Selections**- the selected sketch entities become a new sketch. You can extract a sketch and then modify it before creating a feature
-  ➤ **Repair Sketch**- you can fix errors in a sketch so that the sketch can be used to extrude or cut a feature.

Typical errors can be overlapping geometry, small gaps, or many small segments that are collected into a single entity.

- **Align Sketch**- select an edge in one view to align to the edge selected in a second view. The order of selection is important.
- **Extrude**- extrude a feature from the selected sketch entities. You do not have to select a complete sketch
- **Cut**- cut a feature from the selected sketch entities. You do not have to select a complete sketch

TOOL BARS

To specify which toolbars are displayed on the screen, select *View Toolbars* (i.e., select *Toolbars* from the *View* menu). Be sure that the *Features*, *Sketch*, *Sketch Relations*, *Sketch Tools*, *Standard*, *Standard Views* and *View* toolbars are checked, as shown in the below picture. If they are not, click on each of these items until all are checked. If other toolbars are checked, click on them to uncheck them. It may be necessary to select *View*, then *toolbars* again to display the menu after checking (or un-checking) an item to confirm that the desired change was made. The *Standard Views* toolbar may appear as a dialog box in the Graphics Window instead of as a toolbar. If so, click the blue bar at the top of the dialog box with the left mouse button and drag it to the toolbar at the upper right of the graphics Window. Release the mouse button. The dialog box should change to a toolbar.




TOOLBAR MENU

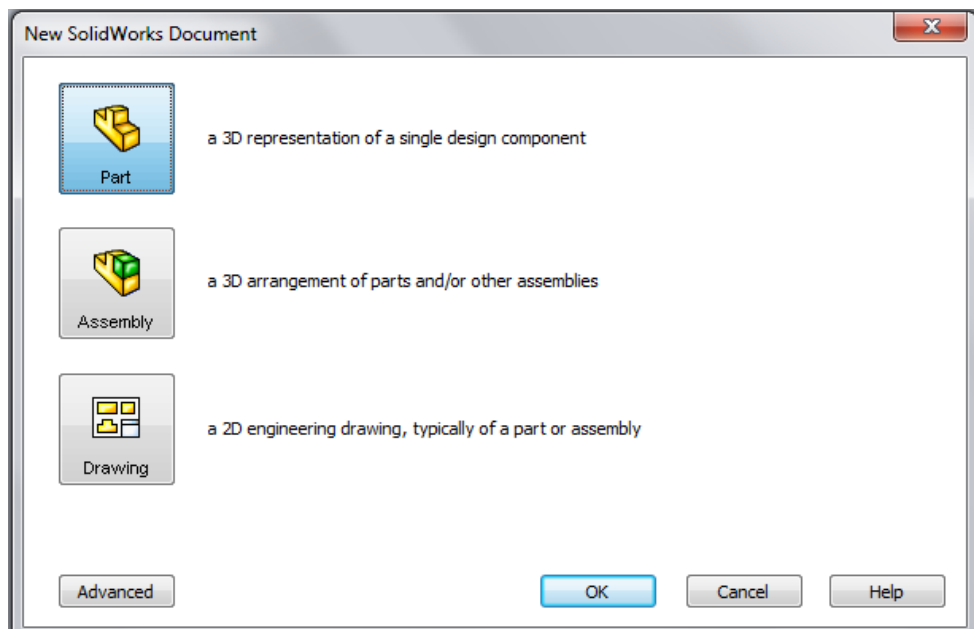
- The **Sketch** toolbar contains tools to set up and manipulate a sketch
- The **Sketch Tools** toolbar contains tools to draw lines, circles, rectangles, arcs and so on

- The **Sketch Relations** toolbar contains tools for contrasting elements of a sketch by using dimensions or relations
- The **Features** toolbar contains tools that modify sketches and existing features of a part
- The **Standard** toolbar contains the usual commands available for manipulating files (Open, Save, Print and so on), editing documents (Cut, Copy, and Paste) and accessing Help.
- The **Standard Views** toolbar contains common orientations for a model.
- The **View** toolbar contains tools to orient and rescale the view of a part.

You can find these toolbars around the Graphics Window by checking and un-checking tem in the *View→Toolbars* menu. The toolbars will appear as you check them and disappear as you uncheck them. Currently, most of the items in the toolbars are grayed out, since they are unusable. They will become active when they are available for use.

Creating a new Part

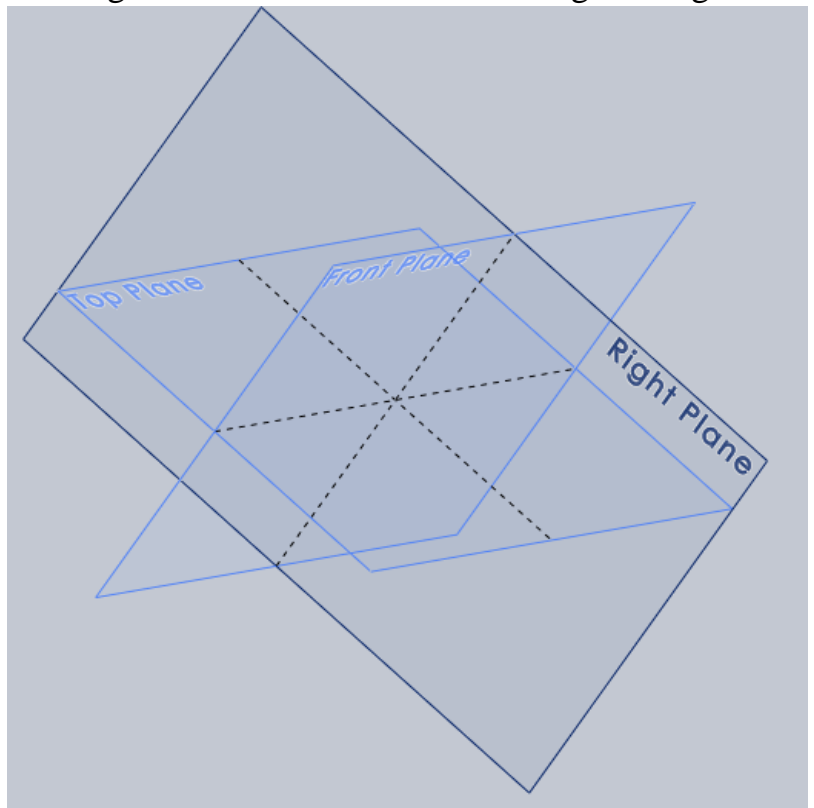
With the SolidWorks window open, select **File→New** in the menu bar, or click the New button (a blank sheet icon) in the *Standard* toolbar. 



SolidWorks Document dialog box appears as shown in the figure. You will be modelling a new part. If Part is already highlighted, click *OK*. If it is not highlighted click Part, then *OK*.

A new window appears with the name Part1, as shown in the figure below. On the left side of the window is the feature manager design tree (FMDT).

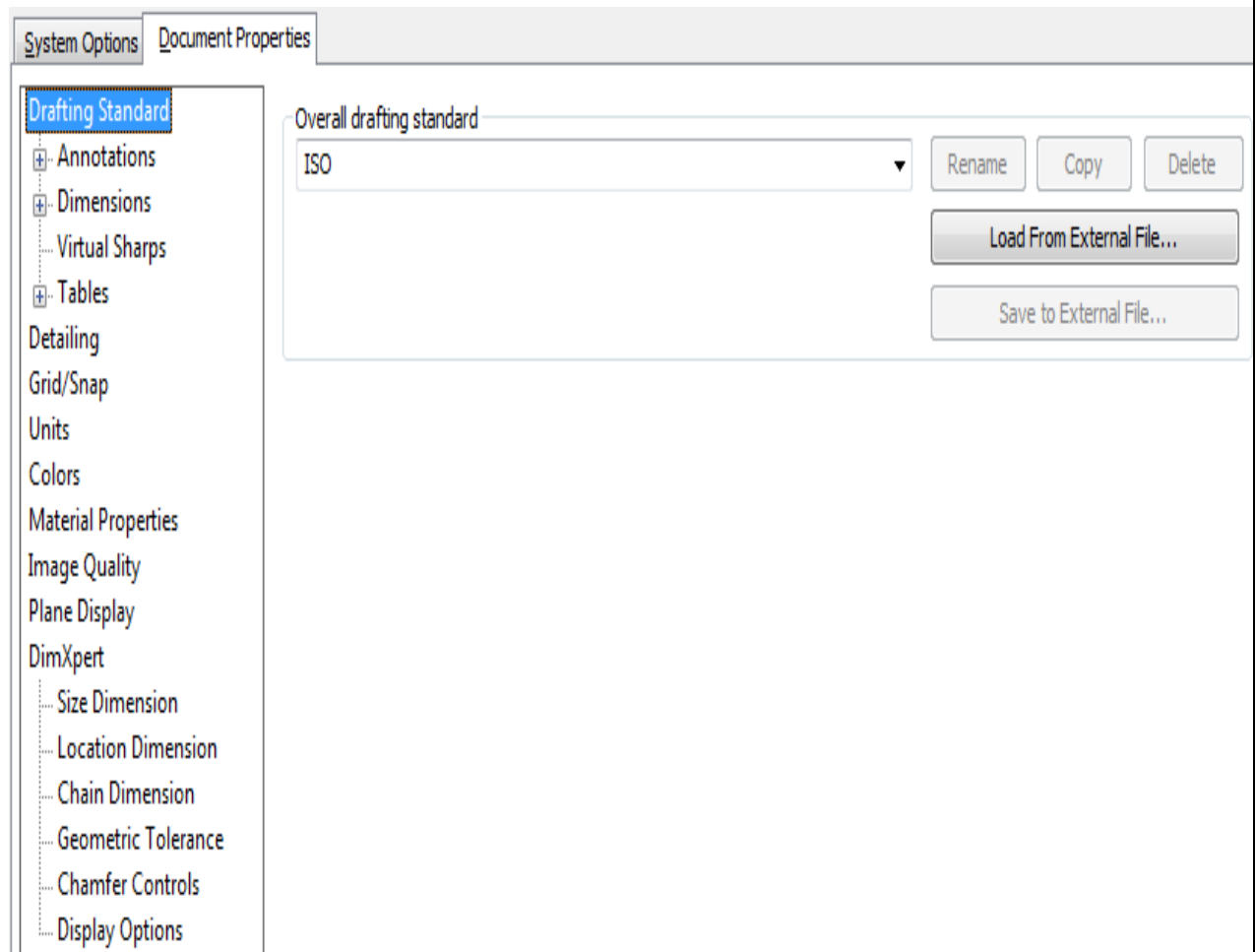
It contains a list of the features that have been created so far. Every new part starts with six features: annotations, lighting, three datum planes, and an origin. The planes are three mutually perpendicular planes that are created in space as references for constructing features of the part that you are modelling. The three planes intersect at the origin, which is in the centre of the Graphics Window. The arrows in the lower left corner of the Graphics Window show the coordinate directions. As the part is modelled, the features that are created will appear in the FMDT. These features can be highlighted or modified by clicking on them in the feature manager design tree. For example, click on a plane or the origin in the FMDT to highlight these items. **Front** is the plane of the screen, **Top** is the horizontal plane perpendicular to the screen, and **Right** is the vertical plane perpendicular to the screen. Finish by clicking on Part1 in FMDT, so that no plane is highlighted.



Sketching

Every part begins as a cross section sketched on a 2D plane. Once a sketch is made, it is extruded or revolved into the third dimension to create a three dimensional object. This is the base feature of the part

To set the units and grid size to be used, click the **Grid** toolbar button with the left mouse button. Documents properties dialog box will appear. Click **Units** on the left side of the dialog box to set the units. Setup appropriate units (inches or mm) with the desired number of decimal or fraction denominator.



Click **Grid/Snap** on the left side of the dialogue box to control the grid that will appear on the screen when a cross section is sketched. Be sure that all three of the boxes under Grid are checked. Adjust the grid spacing to the desired values.

Snap controls the way in which sketched lines are related to the grid. The points that are sketched should snap to the nearest intersection of grid lines when they are close. Be sure the snap point box is checked. If not click the box to activate it












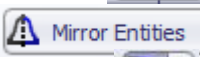




Click **Detailing** on the left side of the tile dialogue box. If necessary, change **Dimensioning Standard** to ANSI for inch units and ISO in case of mm units. Then click OK at the bottom of the dialogue box to accept the values

Open a new sketch by selecting **insert sketch** or by clicking the **sketch button** (a pencil drawing a line) in the sketch toolbar. (note that, for most commands in SolidWorks, it is possible to initiate the command from either the Menu bar or the toolbars). A grid should appear on the screen, indicating

that the sketch mode is now active. The window's name changes to **sketch of Part1**. In the bottom right corner of the screen, the status bar reads editing sketch. You are ready to sketch in the **front plane**.

SKETCH TOOLS

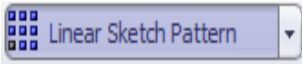
The **Sketch tools:** Toolbar contains tools to create and modify 2D features, called sketch entities. Sketch entities are items that can be drawn on the sketch. The following sketch entities and sketch tools are available:

-  ➤ **Line** creates a straight line.
-  ➤ **Centrepoint arc** creates a singular arc from a centre point, a start point and an end point.
-  ➤ **Tangent arc** creates a circular arc tangent to an existing sketch entity.
-  ➤ **3 pt arc** creates a circular arc through 3 points.
-  ➤ **Circle** creates a circle.
-  ➤ **Spline** creates a curved line that is not a circular arc.
-  ➤ **Polygon** creates a regular polygon.
-  ➤ **Rectangle** creates a rectangle.
-  ➤ **Points** creates a reference point that is used for constructing other sketch entities.
-  ➤ **Centreline** creates a reference line that is used for constructing other sketch.
-  ➤ **Convert entities** creates a sketch entity by projecting an edge, curve, or contour onto the sketch plane.
-  ➤ **Mirror** reflects entities about a centreline.
-  ➤ **Fillet** creates a tangent arc between two sketch entities by rounding an inside or an outside corner.
-  ➤ **Offset entities** creates a sketch curve that is offset from a selected sketch entity by a specified distance.
-  ➤ **Trim** removes a portion of a line or curve.
-  ➤ **Display/Delete Relations** it displays and deletes the geometrical relations between entities.
- **Repair Sketch** it repairs the selected sketch.

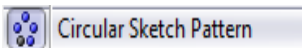


➤ **Rapid Sketch** it allows the 2D sketch plane to change dynamically.

➤ **Construction geometry** creates entities that aid in sketching.



➤ **Linear sketch step and repeat** creates a linear pattern of sketch entities.

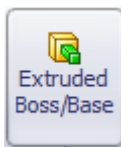


➤ **Circular sketch step and repeat** creates a circular pattern of sketch entities.

Move and Hold the cursor over each of the tools to display its function but do not click on the tool. Note the description of each tool in the status bar at the bottom of the SolidWorks window. Some of these tools may not be included in the toolbars or other tools may be available, depending on the way in which it was previously set up. All tools are available on the **tools** menu.

FEATURE TOOLS

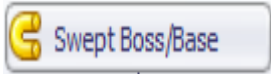
The **Feature tool** toolbar, contains tools to create and modify 3D features, called feature entities. Feature entities are items that can be drawn on the feature. Features include **multibody part** capability. You can include separate extrude, revolve, loft, or sweep features, within the same part document. The following feature entities and feature tools are available:



➤ **Extruded Boss/Base** add material by increasing length in either directions. You can create extruded boss/bases, extruded cuts. The extrude feature can be a solid, a thin feature, or a surface.

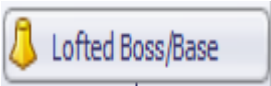


➤ **Revolved Boss/Base** add material by revolving one or more profiles around a centerline. You can create revolved boss/bases, revolved cuts, or **revolved surfaces**. The revolve feature can be a solid, a thin feature, or a surface.

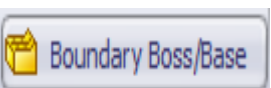


➤ **Swept Boss/Base** creates a base, boss, cut, or surface by moving a profile (section) along a path, according to these rules:

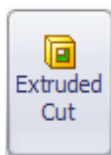
- The profile must be closed for a base or boss sweep feature; the profile may be open or closed for a surface sweep feature.
- The path may be open or closed.
- The path may be a set of sketched curves contained in one sketch, a curve, or a set of model edges.
- The path must intersect the plane of the profile.
- Neither the section, the path, nor the resulting solid can be self-intersecting.
- The guide curve must be coincident with the profile or with a point in the profile sketch.



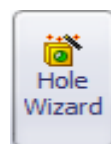
➤ **Lofted Boss/Base** creates a feature by making transitions between profiles. A loft can be a base, boss, cut, or surface. You create a loft using two or more profiles. Only the first, last, or first and last profiles can be points. All sketch entities, including guide curves and profiles, can be contained in a single 3D sketch.



➤ **Boundary Boss/Base** produces very high quality, accurate features useful for creating complex shapes for markets focused on consumer product design, medical, aerospace, and molds.



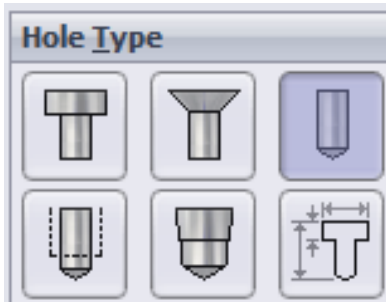
➤ **Extruded Cut** removes material by increasing length in either directions. You can create extruded boss/bases, extruded cuts. The extrude feature can be a solid, a thin feature, or a surface.



➤ **Hole Wizard** The hole type you select determines the capabilities, available selections, and graphic previews.

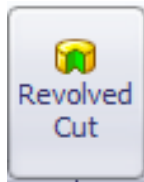
After you select a hole type, you determine the appropriate fastener. The fastener dynamically updates the appropriate parameters. Use the PropertyManager to set the hole type parameters and locate the holes. In addition to the dynamic graphic preview based on end condition and depth, graphics in the PropertyManager show specific details, as they apply to the type of hole you select.

Capabilities

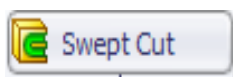


You can create these types of Hole Wizard holes:

1. Counterbore
2. Countersink
3. Hole
4. Straight Tap
5. Tapered Tap
6. Legacy



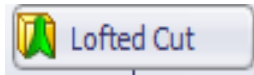
➤ **Revolved Cut** removes material by revolving one or more profiles around a centerline. You can create revolved boss/bases, revolved cuts, or **revolved surfaces**. The revolve feature can be a solid, a thin feature, or a surface.



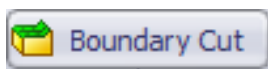
➤ **Swept Cut** Sweep creates a base, boss, cut, or surface by moving a profile (section) along a path, according to these rules:

- The profile must be closed for a base or boss sweep feature; the profile may be open or closed for a surface sweep feature.
- The path may be open or closed.
- The path may be a set of sketched curves contained in one sketch, a curve, or a set of model edges.
- The path must intersect the plane of the profile.

- Neither the section, the path, nor the resulting solid can be self-intersecting.
- The guide curve must be coincident with the profile or with a point in the profile sketch.



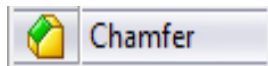
- **Lofted Cut** creates a feature by making transitions between profiles. A loft can be a base, boss, cut, or surface. You create a loft using two or more profiles. Only the first, last, or first and last profiles can be points. All sketch entities, including guide curves and profiles, can be contained in a single 3D sketch.



- **Boundary Cut** tools removes very high quality, accurate features useful for creating complex shapes for markets focused on consumer product design, medical, aerospace, and molds.



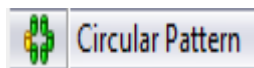
- **Fillet** creates a rounded internal or external face on the part. You can fillet all edges of a face, selected sets of faces, selected edges, or edge loops.



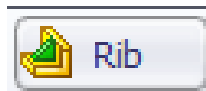
- **Chamfer** tool creates a beveled feature on selected edges, faces, or a vertex.



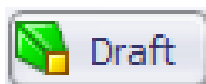
- **Linear Pattern** repeats the selected features in an array based on a **seed feature**. You can create a linear pattern, a circular pattern, a curve driven pattern, a fill pattern, or use sketch points or table coordinates to create the pattern.

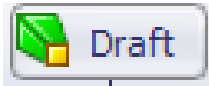


- **Circular Pattern** it patterns features, faces and bodies around the axis.

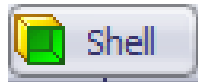


- **Rib** is a special type of extruded feature created from open or closed sketched contours. It adds material of a specified thickness in a specified direction between the contour and an existing part. You can create a rib using single or multiple sketches. You can also create rib features with draft, or select a reference contour to draft.





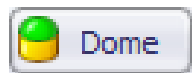
- **Draft** tapers faces using a specified angle to selected faces in the model. One application is to make a molded part easier to remove from the mold. You can insert a draft in an existing part or draft while extruding a feature. You can apply draft to solid or surface models.



- **Shell** tool hollows out a part, leaves open the faces you select, and creates thin-walled features on the remaining faces. If you do not select any face on the model, you can shell a solid part, creating a closed, **hollow model**. You can also shell a model using **multiple thicknesses**.



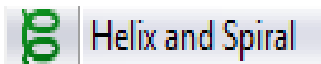
- **Wrap** it wraps the closed sketch contour(s) unto a face.



- **Dome** You can create one or more dome features simultaneously on the same model.

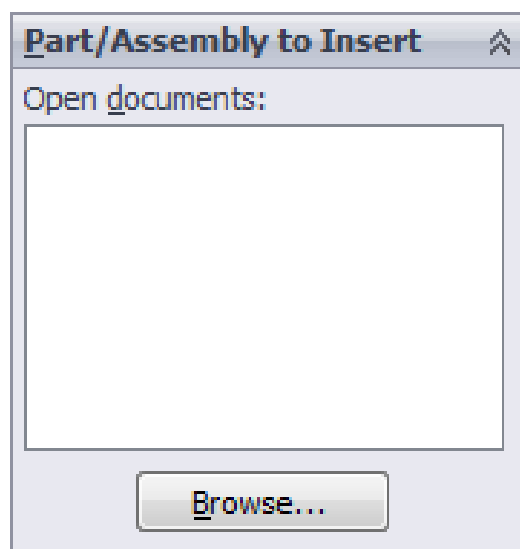


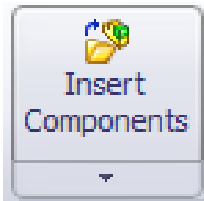
- **Mirror** copies the selected features or all features, mirroring them about the selected plane or face.



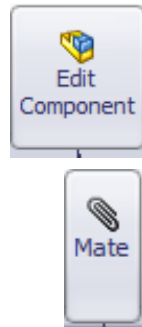
- **Helix and Spiral** adds a helix or spiral curve for a sketched circle.

ASSEMBLY COMMANDS





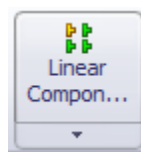
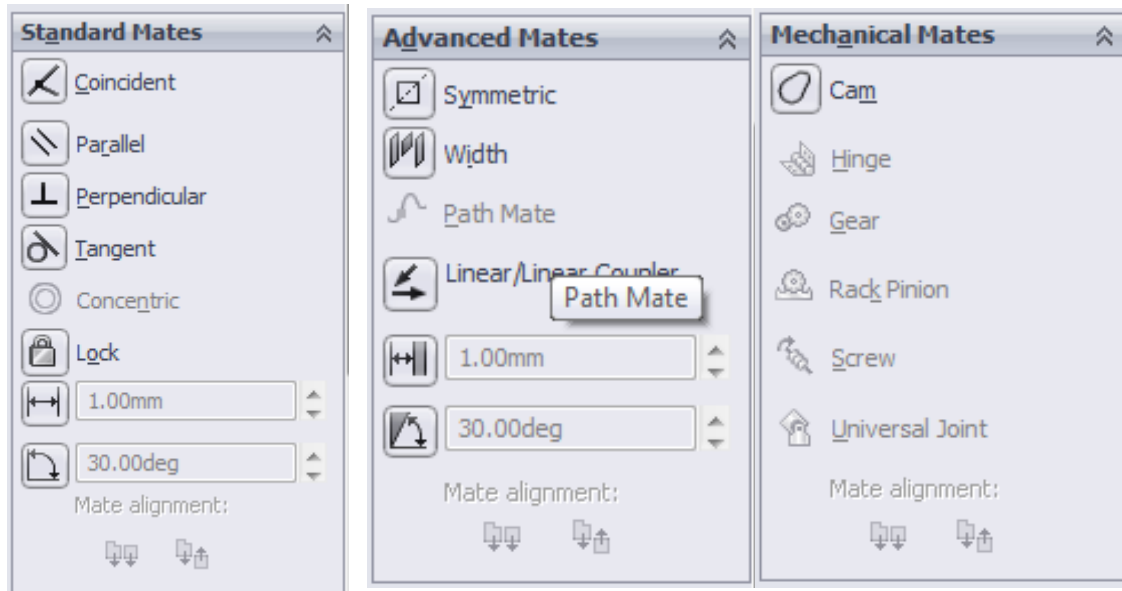
- **Insert Components** adds an existing components (or) part (or) sub-assembly to the assembl



- **Edit Component** toggles between editing a part (or) a sub-assembly & the main assembly.
- **Mate** positions two components relative to each another.

Types of Mates:

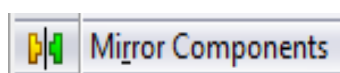
- 1) Standard Mates
- 2) Advanced Mates
- 3) Mechanical Mates



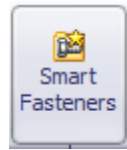
- **Linear Component Pattern** pattern all components in one (or) two directions.



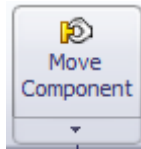
- **Circular Component Pattern** patterns components around the Axis.



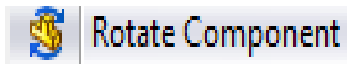
➤ **Mirror Components** mirrors sub-assemblies & parts.



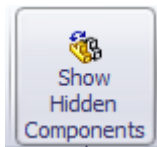
➤ **SmartFasteners** adds fasteners to the assembly using the solid works Tool box Library of Standard Hardware.



➤ **Move Component** moves a component within the degree of freedom defined by its mates.



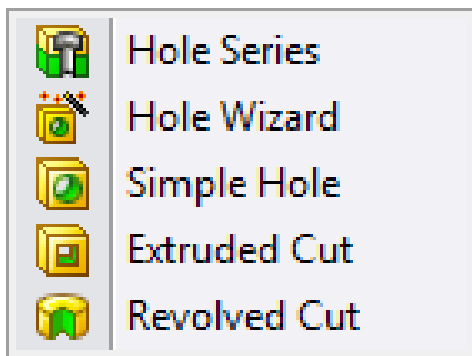
➤ **Rotate Component** rotates the component within the degree of freedom defined by its mates.



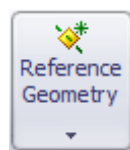
➤ **Show Hidden Components** can hide and unhide the required component on user demand.



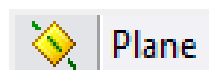
➤ **Assembly Features** creates various assembly features.



- Hole Series
- Hole Wizard
- Simple Hole
- Extruded Cut
- Revolved Cut



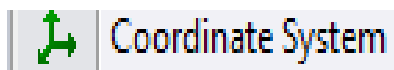
➤ **Reference Geometry** it commands reference geometry.



• **Plane** – Add a reference plane.



• **Axis** – Add a reference axis.



• **Coordinate system** – defines a coordinate system for any part of assembly.



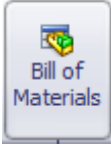
• **Point** – Add a reference point.

 **Mate Reference**

- **Mate reference** – Specifies entities to use as references for automatic mating using smart mates



- **New Motion Study** inserts new motion study.



- **Bills of Materials** add a bill of materials used in designing.



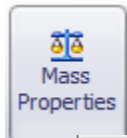
- **Exploded Views** separates the components into an exploded view.



- **Instant 3D** enables dragging of handles, dimensions & sketches of dynamically modify features.



- **Evaluate Section Properties** it evaluates section properties for multiple faces and sketches that lies in parallel plane.



- **Evaluate Mass Properties** it calculates the mass properties of the model. **Ex:-**

```

Mass properties of Part4 ( Part Configuration - Default )
Output coordinate System: -- default --
Density = 0.00 grams per cubic millimeter
Mass = 750.85 grams
Volume = 750847.79 cubic millimeters
Surface area = 47152.60 millimeters^2

Center of mass: ( millimeters )
X = 0.00
Y = 0.00
Z = 35.00

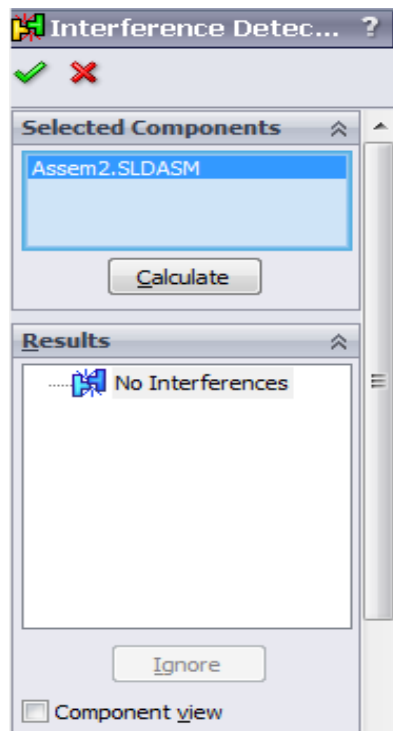
Principal axes of inertia and principal moments of inertia: ( grams * square millimeters )
Taken at the center of mass.
Ix = (1.00, 0.00, 0.00)   Px = 947504.51
Iy = (0.00, 1.00, 0.00)   Py = 947504.51
Iz = (0.00, 0.00, 1.00)   Pz = 1281816.65

Moments of inertia: ( grams * square millimeters )
Taken at the center of mass and aligned with the output coordinate system.
Lxx = 947504.51         Lxy = 0.00             Lxz = 0.00
Lyx = 0.00             Lyy = 947504.51       Lyz = 0.00
Lzx = 0.00             Lzy = 0.00             Lzz = 1281816.65

Moments of inertia: ( grams * square millimeters )
Taken at the output coordinate system.
Ixx = 1867293.05       Ixy = 0.00             Ixz = 0.00
Iyx = 0.00             Iyy = 1867293.05       Iyz = 0.00
Izx = 0.00             Izy = 0.00             Izz = 1281816.65
    
```



➤ **Evaluate Interference Detection** it detects any interference between components after completion of assembly through mating.



EXPERIMENT NO-1

(UNIVERSAL COUPLING)

AIM

- To model the parts of the universal coupling & Do the assembly of the parts using SW-2008 software.

PARTS

- There are 3 main Parts:

1) Centre block

2) Fork

3) Pin & Collar.

PROCEDURE

Part Modelling

- Select the **SOLID EDGE** icon→New→ Part.
- To model **Centre Block**, first select the required plane.
 - Draw the 2D sketch of the fork use the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as centre block
- To model **Fork**→ new→Part.
 - Draw the 2D sketch of the fork use the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Fork.
- To model **Pin**→new→Part.
 - Draw the 2D sketch of the fork use the **SKETCH** tools with required dimensions, then → exit sketch.

- Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
- Save the part as Pin.
- To model **Collar**→new→Part.
 - Draw the 2D sketch of the fork use the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Collar.
- To model **Key**→new→Part.
 - Draw the 2D sketch of the fork use the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Key.

Assembly

- To assemble →new→Assembly.
 - Insert the main component first, by clicking→Browse.
 - Place the main component in the area, this part is fixed and cannot be moved further.
 - Insert the remaining components for assembly by the same procedure as above, these components can moved anywhere.
 - Use the **MATE** tool to assemble all the parts such as(standard mates, mechanical mates, advanced mates).

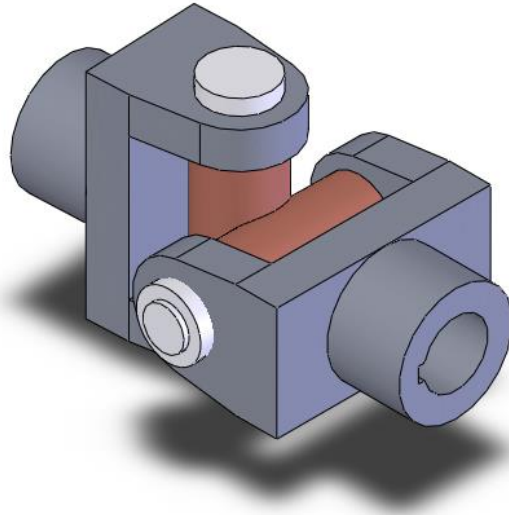
Precautions

- When modelling a component make sure it is not under defined or over defined.
- Give exact dimensions while drawing.
- While doing assemblies provide the appropriate relation between the mating parts.
- Check whether all the parts are mating perfectly without any interference.
- Save the current work on regular basis so that the data is not lost.

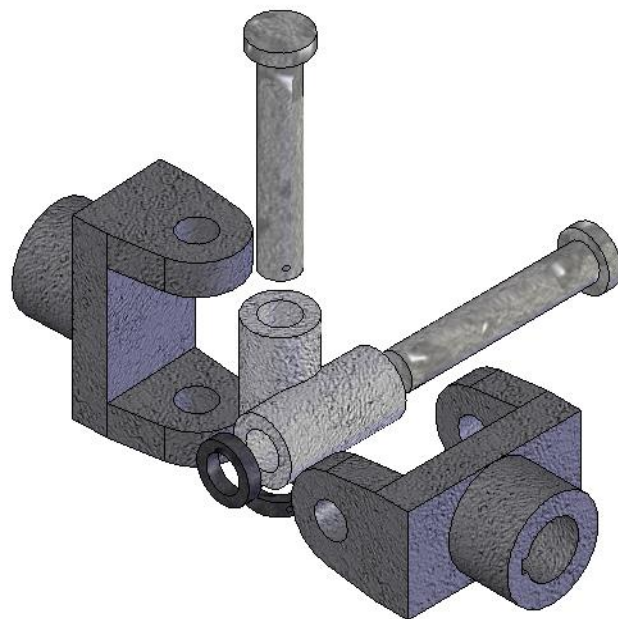
APPLICATION

- It is used to connect the two shafts whose axis is not aligned .
- It is used to connect ENGINE &DIFFRENTIAL.
- It is used in Agitators
- It is used in Balancing Machines
- It is used in Blowers and Fans
- It is used in Compressors
- It is used in Conveyors
- It is used in Cooling Tower Fans
- It is used in Cranes and Hoists
- It is used in Crushers
- It is used in Farming Equipment
- It is used in Generators
- It is used in Glass Manufacturing
- It is used in Lumber Mills
- It is used in Marine Propulsion
- It is used in Mining Equipment

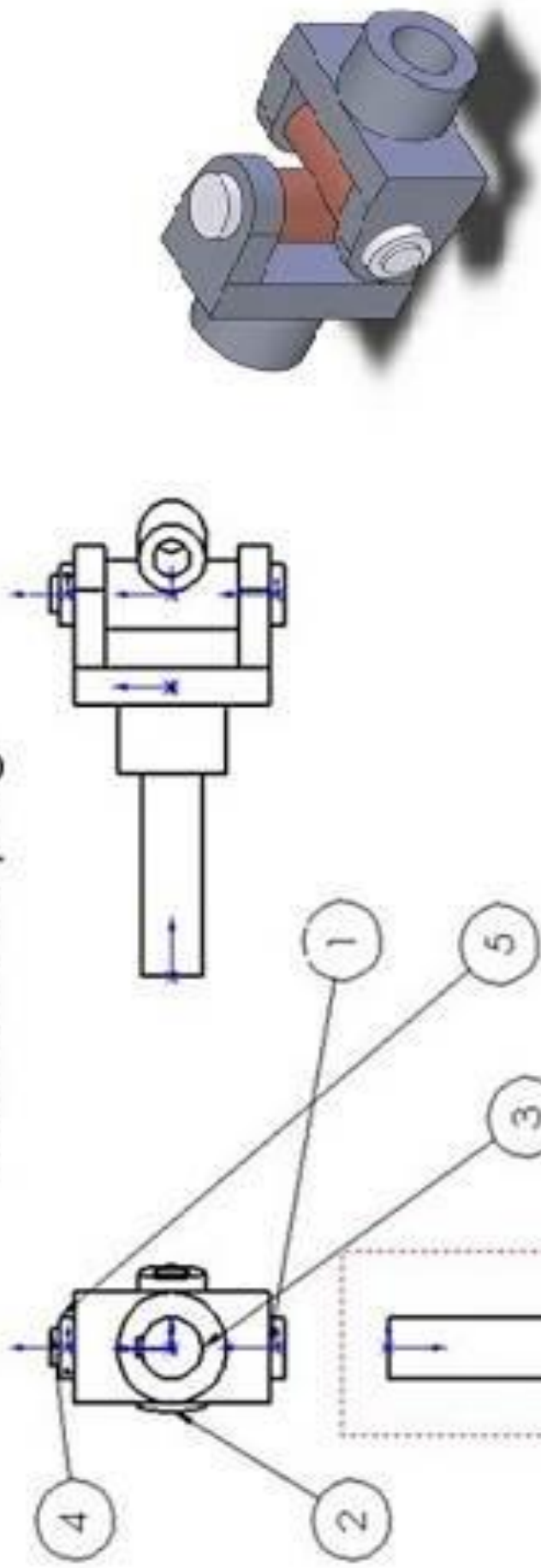
Assembly of Universal Coupling



Exploded View



Universal coupling



Part no.	Name	Material	QTY.
1	fork	FS	1
2	Central block	FS	1
3	shaft	MS	1
4	Pin	MS	1
5	collar	MS	1

Mass properties of Universal Coupling

Mass properties of Universal Coupling (Assembly Configuration - Default)

Output coordinate System: -- default --

Mass = 18351.37 grams

Volume = 2645567.83 cubic millimeters

Surface area = 325853.46 millimeters²

Center of mass: (millimeters)

$$X = -18.98$$

$$Y = 5.00$$

$$Z = 55.94$$

Principal axes of inertia and principal moments of inertia: (grams * square millimeters)

Taken at the center of mass.

$$I_x = (1.00, -0.08, -0.02) \quad P_x = 54023729.47$$

$$I_y = (0.05, 0.78, -0.62) \quad P_y = 130372267.11$$

$$I_z = (0.06, 0.62, 0.78) \quad P_z = 130443344.22$$

Moments of inertia: (grams * square millimeters)

Taken at the center of mass and aligned with the output coordinate system.

$$L_{xx} = 54495896.18 \quad L_{xy} = -5759553.06 \quad L_{xz} = -1635827.56$$

$$L_{yx} = -5759553.06 \quad L_{yy} = 129962683.44 \quad L_{yz} = 89412.89$$

$$L_{zx} = -1635827.56 \quad L_{zy} = 89412.89 \quad L_{zz} = 130380761.18$$

Moments of inertia: (grams * square millimeters)

Taken at the output coordinate system.

$$I_{xx} = 112390027.55 \quad I_{xy} = -7502399.65 \quad I_{xz} = -21120001.40$$

$$I_{yx} = -7502399.65 \quad I_{yy} = 194007103.11 \quad I_{yz} = 5226899.07$$

$$I_{zx} = -21120001.40 \quad I_{zy} = 5226899.07 \quad I_{zz} = 137450139.03$$

EXPERIMENT NO-2

(ECCENTRIC)

AIM

- To model the parts of the Eccentric & Do the assembly of the parts using SW-2008 software.

PARTS

- There are 6 main Parts:

- 1) Straps
- 2) Sheave
- 3) Packing strip
- 4) Strap bolt
- 5) Stud with nut
- 6) Eccentric rod.

PROCEDURE

Part Modelling

- Select the **SOLID EDGE** icon → New → Part.
- To model **Straps**, first select the required plane.
 - Draw the 2D sketch of the Straps use the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Strap
- To model **Sheave** → new → Part.
 - Draw the 2D sketch of the Sheave use the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Sheave.
- To model **Packing strip** → new → Part.
 - Draw the 2D sketch of the Packing strip uses the **SKETCH** tools with required dimensions, then → exit sketch.

- Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Packing strip.
- To model **Strap Bolt**→new→Part.
- Draw the 2D sketch of the Strap bolt uses the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Strap bolt.
- To model **Studs** →new→Part.
- Draw the 2D sketch of the Studs use the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Studs.
- To model **Nuts** →new→Part.
- Draw the 2D sketch of the Nuts use the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Nuts.
- To model **Eccentric rod**→new→Part.
- Draw the 2D sketch of the fork use the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Eccentric rod.

Assembly

- To assemble →new→Assembly.
- Insert the main component first, by clicking→Browse.

- Place the main component in the area, this part is fixed and cannot be moved further.
- Insert the remaining components for assembly by the same procedure as above, these components can be moved anywhere.
- Use the **MATE** tool to assemble all the parts such as (standard mates, mechanical mates, advanced mates) etc,...

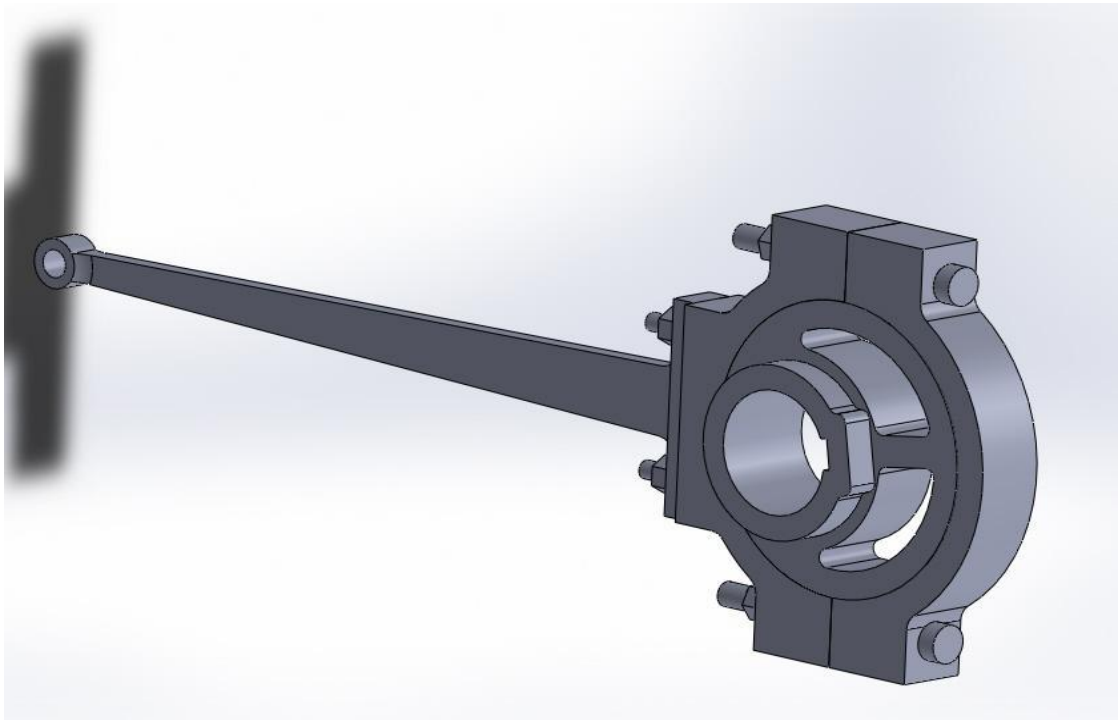
➤ **Precautions:**

- When modelling a component make sure it is not under defined or over defined.
- Give exact dimensions while drawing.
- While doing assemblies provide the appropriate relation between the mating parts.
- Check whether all the parts are mating perfectly without any interference.
- Save the current work on regular basis so that the data is not lost.

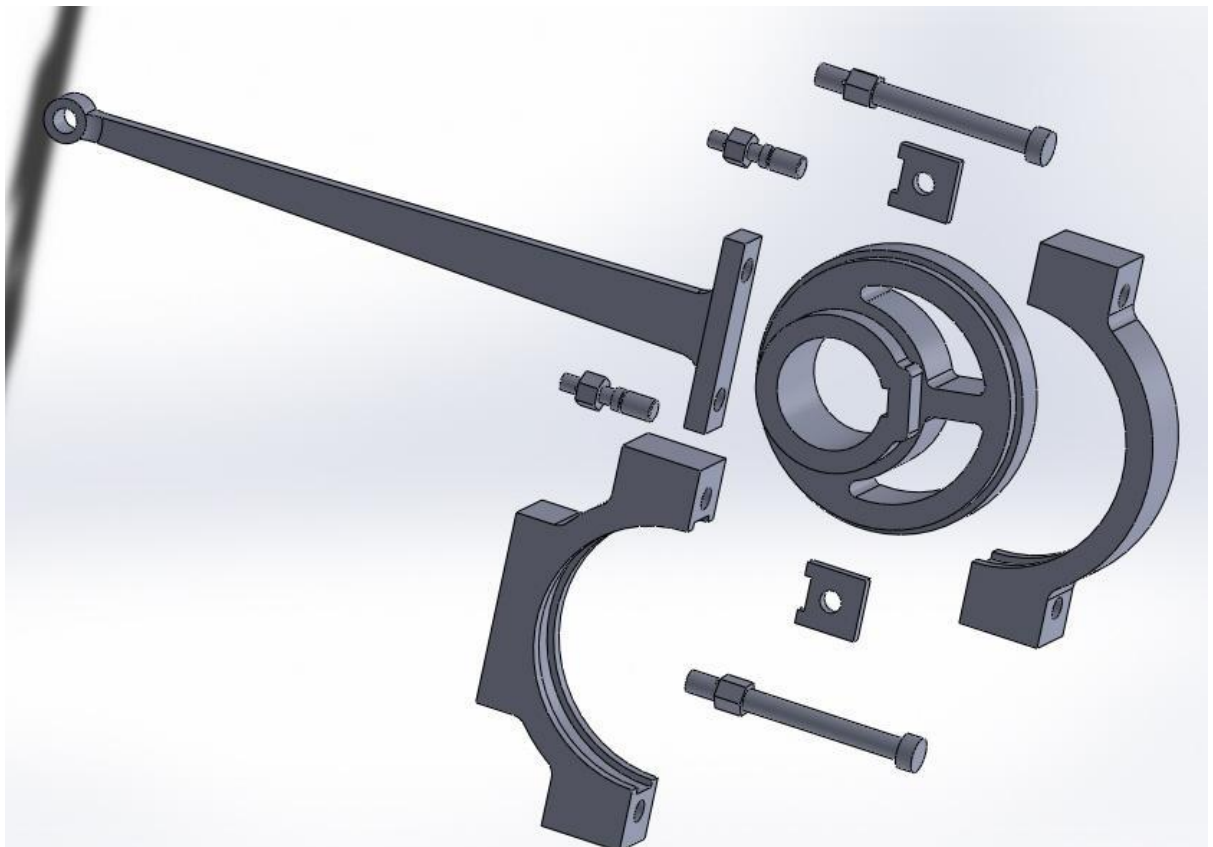
APPLICATIONS

- Eccentrics are used for operating steam valves, small pump plungers, shaking screens, etc.
- It is used to provide a short reciprocating motion, actuated by the rotation of a shaft.

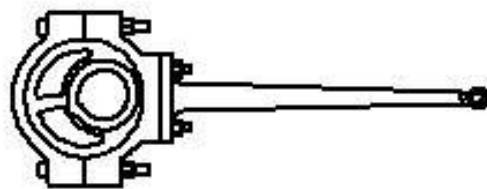
Assembly of Eccentric



Exploded View



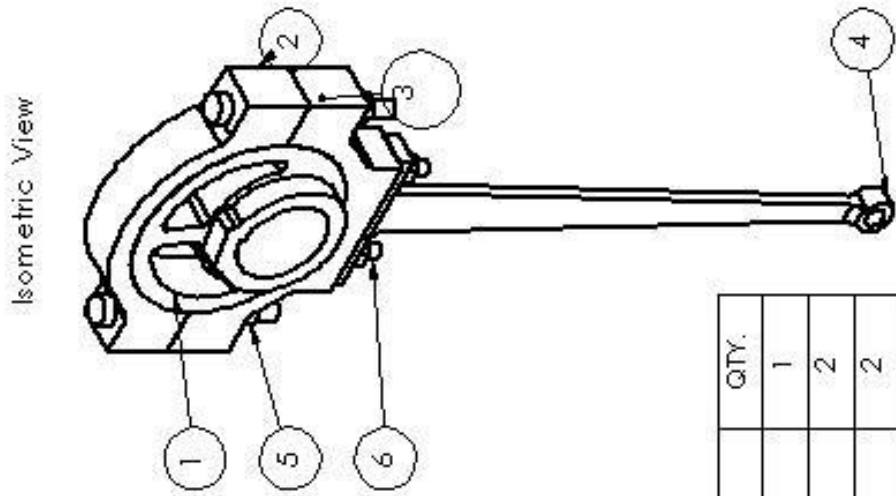
Eccentric



Front View



Side View



Isometric View



Top View

ITEM NO.	PART NUMBER	MATERIAL	QTY.
1	Sheave	Cast Iron	1
2	Straps	Cast Iron	2
3	Packing Strip	Leather	2
4	Eccentric rod	Forged Steel	1
5	Strap bolt	Mild Steel	2
6	Stud with nut	Mild Steel	2

Mass Properties of Eccentric

Mass properties of Eccentric

Configuration: Default

Coordinate system: -- default --

Mass = 2210.31 grams

Volume = 2210307.88 cubic millimeters

Surface area = 349998.31 square millimeters

Center of mass: (millimeters)

X = 10.41

Y = -24.56

Z = -12.52

Principal axes of inertia and principal moments of inertia: (grams * square millimeters)

Taken at the center of mass.

I_x = (0.00, 1.00, 0.00) P_x = 11979372.50

I_y = (-1.00, 0.00, 0.00) P_y = 35114816.78

I_z = (0.00, 0.00, 1.00) P_z = 46256317.53

Moments of inertia: (grams * square millimeters)

Taken at the center of mass and aligned with the output coordinate system.

L_{xx} = 35114804.15 L_{xy} = 26513.06 L_{xz} = 14150.25

L_{yx} = 26513.06 L_{yy} = 11979565.56 L_{yz} = 74656.74

L_{zx} = 14150.25 L_{zy} = 74656.74 L_{zz} =

46256137.10

Moments of inertia: (grams * square millimeters)

Taken at the output coordinate system.

I_{xx} = 36794511.18 I_{xy} = -538524.08 I_{xz} = -273777.31

I_{yx} = -538524.08 I_{yy} = 12565244.86 I_{yz} = 754149.62

I_{zx} = -273777.31 I_{zy} = 754149.62 I_{zz} = 47829021.25

EXPERIMENT NO-3

(STUFFING BOX)

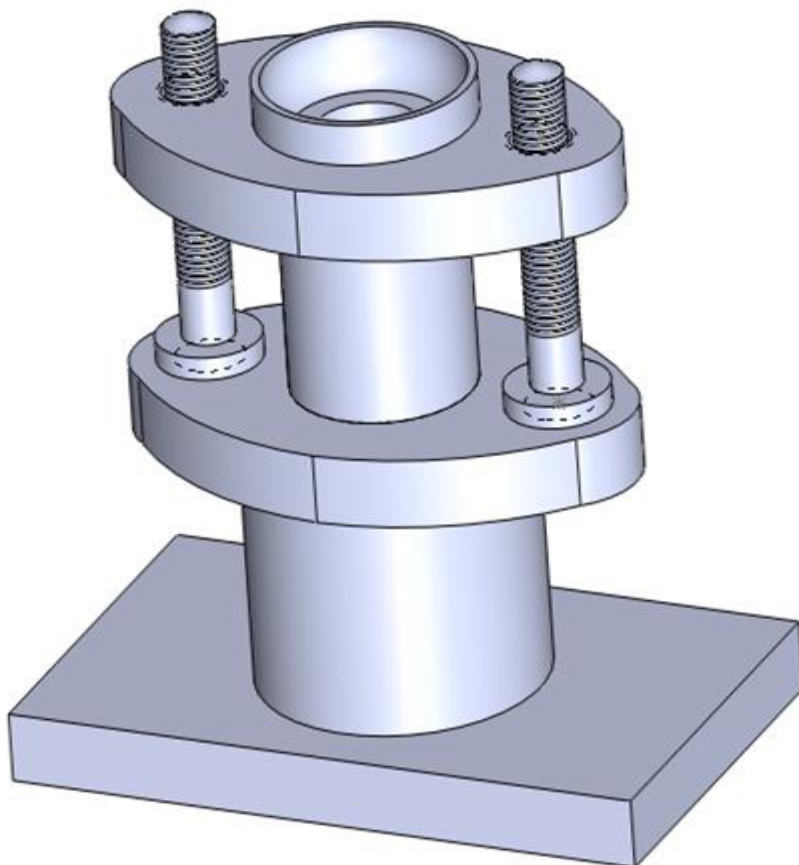
AIM

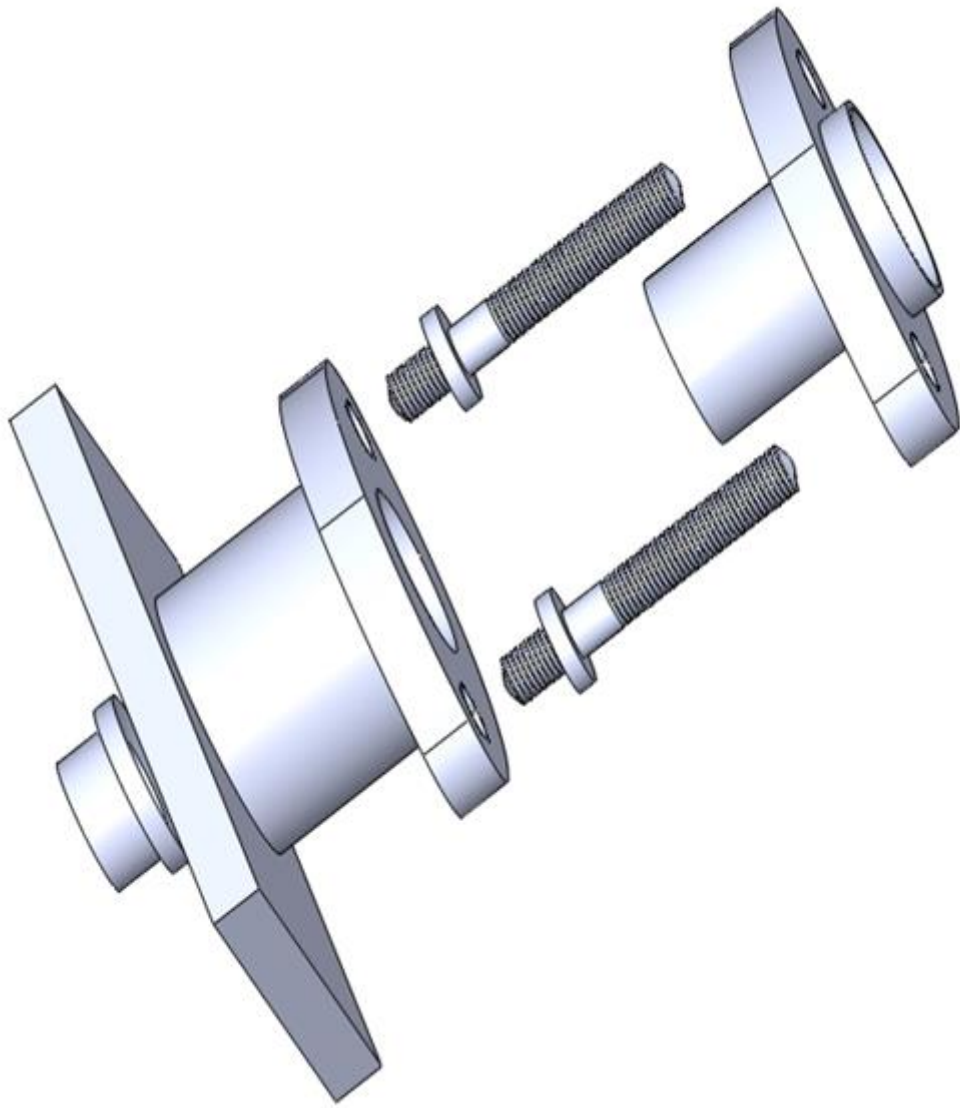
- To model the parts of the Stuffing box & do the assembly of the

➤PARTS

- There are 6 main Parts:

- 1) Straps
- 2) Sheave
- 3) Packing strip
- 4) Strap bolt
- 5) Stud with nut
- 6) Eccentric rod.





MASS PROPERTIES

Mass properties of ass (Assembly Configuration - Default)

Output coordinate System: -- default --

Density = 0.00 grams per cubic millimeter

Mass = 532.82 grams

Volume = 532816.89 cubic millimeters

Surface area = 111922.78 millimeters²

Center of mass: (millimeters)

$$X = 8.82$$

$$Y = 5.78$$

$$Z = -24.59$$

Principal axes of inertia and principal moments of inertia: (grams * square millimeters)

Taken at the center of mass.

$$I_x = (0.00, 0.00, 1.00) \quad P_x = 868521.65$$

$$I_y = (1.00, 0.00, -0.00) \quad P_y = 1497780.98$$

$$I_z = (0.00, 1.00, 0.00) \quad P_z = 1865894.70$$

Moments of inertia: (grams * square millimeters)

Taken at the center of mass and aligned with the output coordinate system.

$$L_{xx} = 1497780.98 \quad L_{xy} = 0.13 \quad L_{xz} = 2.10$$

$$L_{yx} = 0.13 \quad L_{yy} = 1865894.70 \quad L_{yz} = -0.23$$

$$L_{zx} = 2.10 \quad L_{zy} = -0.23 \quad L_{zz} = 868521.65$$

Moments of inertia: (grams * square millimeters)

Taken at the output coordinate system.

$$I_{xx} = 1837757.36 \quad I_{xy} = 27182.30 \quad I_{xz} = -115566.76$$

$$I_{yx} = 27182.30 \quad I_{yy} = 2229508.16 \quad I_{yz} = -75772.04$$

$$I_{zx} = -115566.76 \quad I_{zy} = -75772.04 \quad I_{zz} = 927802.28$$

EXPERIMENT NO-4

(STEAM ENGINE CROSS HEAD)

AIM

- To model the parts of the Steam engine Connecting Rod & Do the assembly of the parts using SW-2008 software.

PARTS

- There are 6 main Parts:

- 1) Connecting rod
- 2) Brasses
- 3) Strap
- 4) Jib
- 5) Cotter
- 6) Set Screw

PROCEDURE

Part Modelling

- Select the **SOLID EDGE** icon → New → Part.
- To model **Connecting rod**, first select the required plane.
 - Draw the 2D sketch of the Connecting rod using the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Connecting rod.
- To model **Brasses** → new → Part.
 - Draw the 2D sketch of the Brasses using the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Brasses.
- To model **Strap** → new → Part.
 - Draw the 2D sketch of the Strap using the **SKETCH** tools with required dimensions, then → exit sketch.

- Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Strap.
- To model **Jib**→new→Part.
- Draw the 2D sketch of the Jib using the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Jib.
- To model **Cotter**→new→Part.
- Draw the 2D sketch of the Cotter using the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Cotter.
- To model **Set Screw**→new→Part.
- Draw the 2D sketch of the Set Screw using the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Set Screw.

Assembly

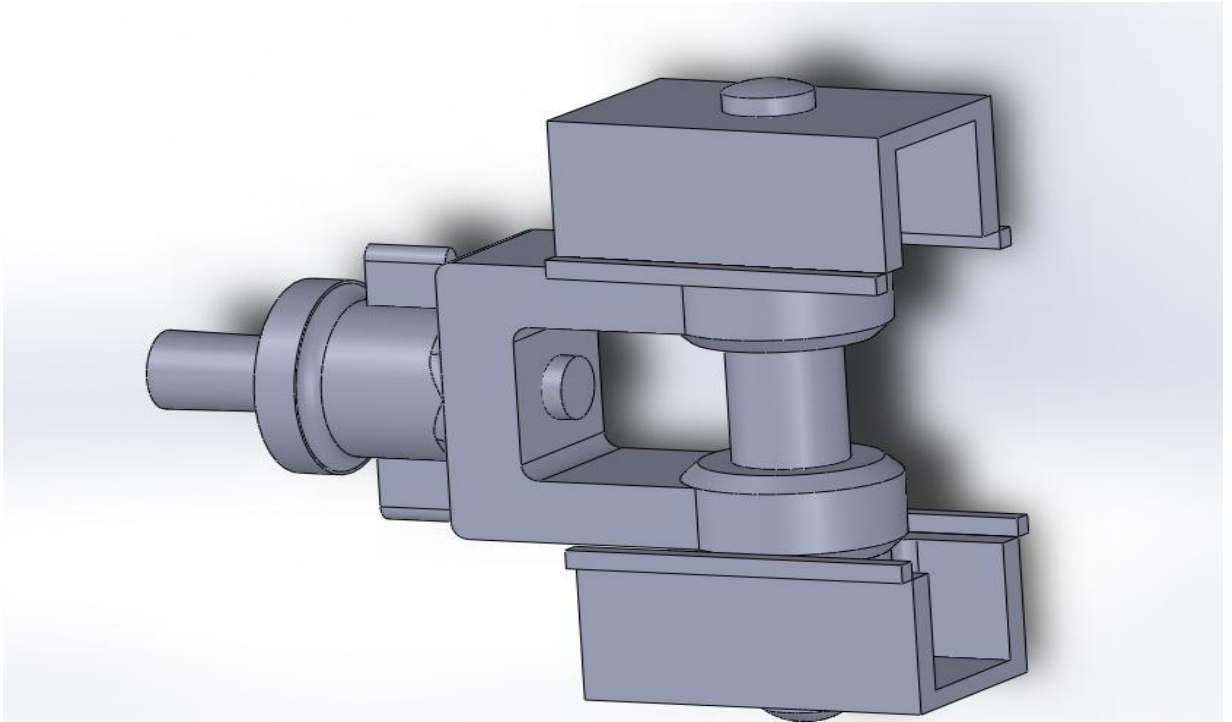
- To assemble →new→Assembly.
- Insert the main component first, by clicking→Browse.
 - Place the main component in the area, this part is fixed and cannot be moved further.
 - Insert the remaining components for assembly by the same procedure as above, these components can moved anywhere.
 - Use the **MATE** tool to assemble all the parts such as(standard mates, mechanical mates, advanced mates).

2D Engineering Drawing

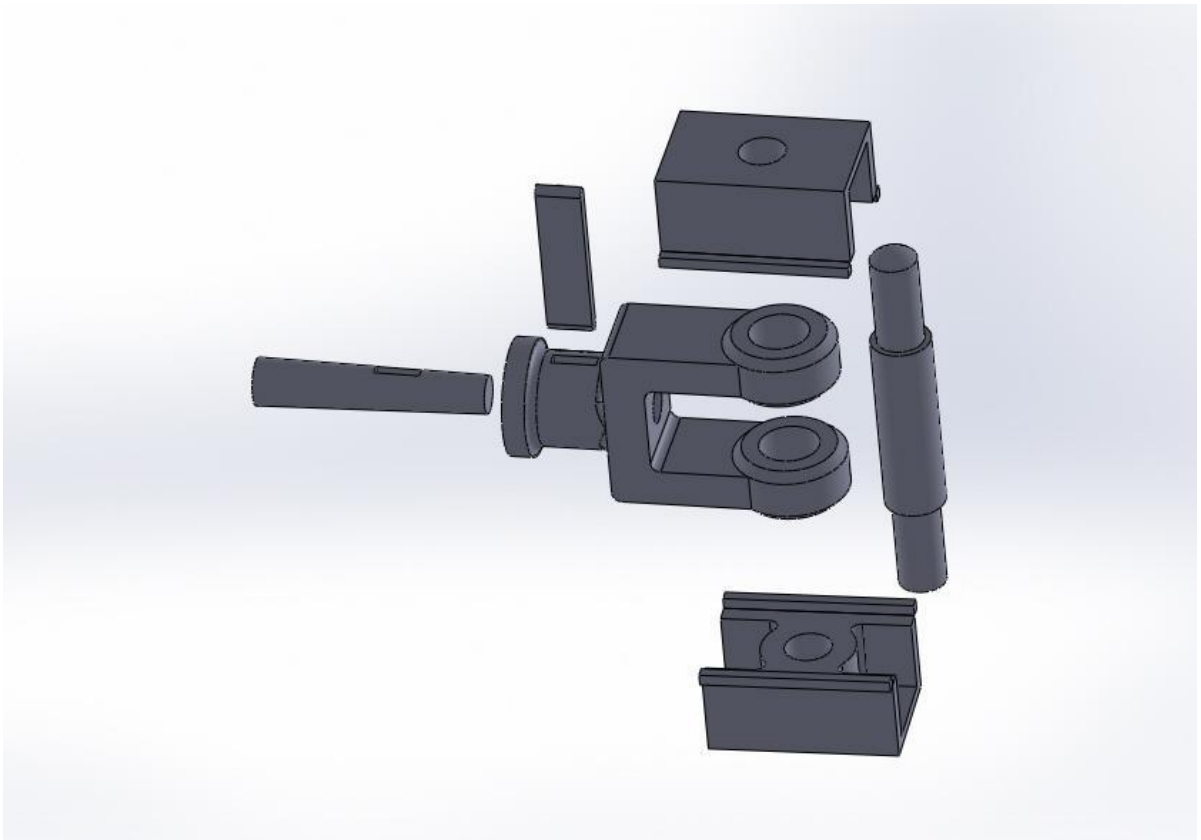
- To start Drawing module →new→Drawing.
 - Select the sheet format(A1,A2,A3,A4)
 - Insert the drawing→Browse.
 - select→**Standard 3view** option.
 - Place the views (top ,front,side) in the required place.
- To insert Bills of Material, select→**Insert** option from the menu bar.
 - Goto→Tables→ Bills of Materials.
 - Place the table in the required position.
 - Edit the columns and rows according to the no. Of components.
 - Save the drawing.

APPLICATIONS

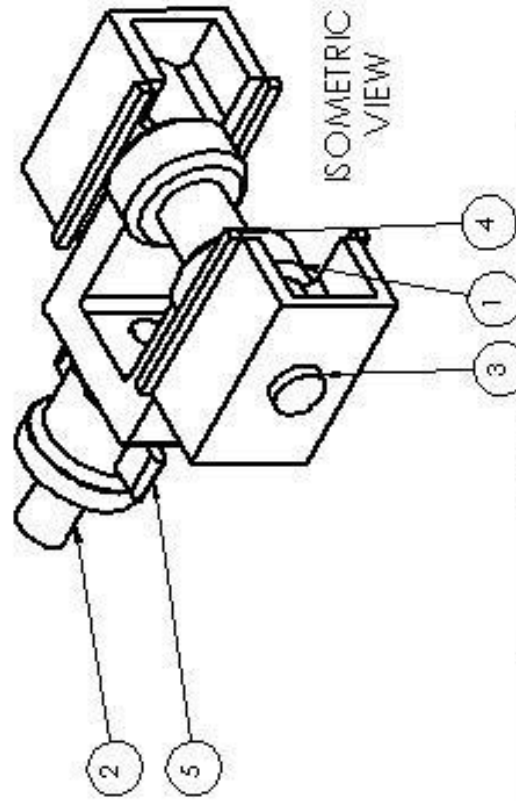
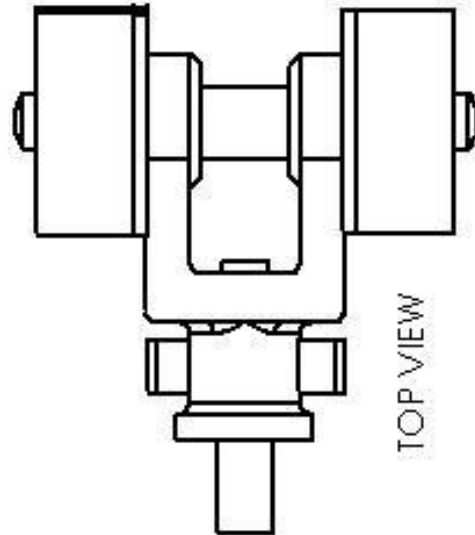
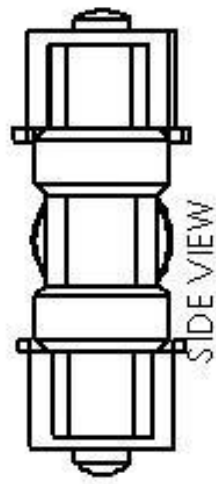
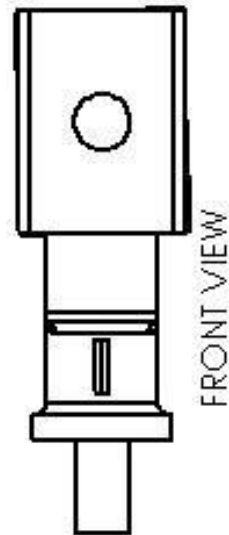
- Connecting rod in a steam engine connects the crosshead at one end (small end) and the crank at the other end (big end).
- The cross-section of the connecting rod can be square/circular in shape.
- The steel arm which connects the piston rod with the crank on the driving wheel or driving axle.
- It is used to convert the forward and aft motion of the piston into the rotating motion of the axle.
- It is designed in a tapered form and has a 'little end', where it is connected to the crosshead on the piston rod, and a 'big end' where it is connected to the crank arm.
- The tapering is to allow for the greater stresses experienced at the crank end.



Exploded View



STEAM ENGINE CROSS HEAD



ITEM NO.	PART NUMBER	MATERIAL	QTY.
1	Block	CAST STEEL	1
2	Piston Rod	MILD STEEL	1
3	Gudgeon pin	MILD STEEL	1
4	Slideblock	CAST IRON	2
5	Cotter	MILD STEEL	1

MASS PROPERTIES

Mass properties of ass (Assembly Configuration - Default)

Output coordinate System: -- default --

Density = 0.00 grams per cubic millimeter

Mass = 971.80 grams

Volume = 971804.31 cubic millimeters

Surface area = 179836.33 millimeters²

Center of mass: (millimeters)

$$X = -19.34$$

$$Y = 4.27$$

$$Z = -24.68$$

Principal axes of inertia and principal moments of inertia: (grams * square millimeters)

Taken at the center of mass.

$$I_x = (0.00, 1.00, 0.00) \quad P_x = 2998003.49$$

$$I_y = (0.00, 0.00, 1.00) \quad P_y = 3159190.45$$

$$I_z = (1.00, 0.00, -0.00) \quad P_z = 5537493.06$$

Moments of inertia: (grams * square millimeters)

Taken at the center of mass and aligned with the output coordinate system.

$$L_{xx} = 5537492.16 \quad L_{xy} = 0.00 \quad L_{xz} = 1464.17$$

$$L_{yx} = 0.00 \quad L_{yy} = 2998003.49 \quad L_{yz} = -0.00$$

$$L_{zx} = 1464.17 \quad L_{zy} = -0.00 \quad L_{zz} = 3159191.36$$

Moments of inertia: (grams * square millimeters)

Taken at the output coordinate system.

$$I_{xx} = 6147375.24 \quad I_{xy} = -80323.11 \quad I_{xz} = 465307.72$$

$$I_{yx} = -80323.11 \quad I_{yy} = 3953482.91 \quad I_{yz} = -102537.73$$

$$I_{zx} = 465307.72 \quad I_{zy} = -102537.73 \quad I_{zz} = 3540300.33$$

EXPERIMENT NO-5

(KNUCKLE JOINT)

AIM

- To model the parts of the knuckle joint & Do the assembly of the parts using SW-2008 software.

PARTS

- There are 4 main Parts:

1) Fork End 2) Eye End 3) Pin 4) Collar

PROCEDURE

Part Modelling

- Switch on the computer and open the solid works 2008 software.
- Model all the given parts separately by selecting a new sheet made for drawing & modeling parts.
- And to model a part select a plane onto which the model has to be prepared.
- Using the sketch tools draw the required part(2D)
- Then by using the feature tools convert the 2D drawing into 3D part.
- Also the part can further be modified by other editing options available.
- Save all the modeled parts after each completion.

Assembly

- Now open a new sheet made for assembling all the model parts.
- Insert components from the location where you have saved the modeled parts.
- Fix the main component to the sheet origin.
- Now insert other components and place randomly on the sheet.
- Using the mate option, mate all the parts assembly in a systematic manner.
- Provide the Exploded view, parts of Assembly, Sectional & Detail view for better understanding
- Give the mass properties of the final assembly.

- Save the file after completion.

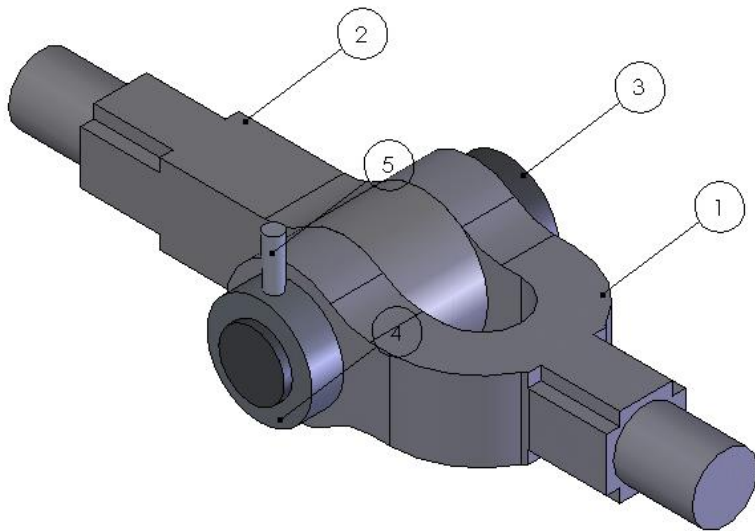
Precautions:

- When modeling a component make sure it is not under defined or over defined.
- Give exact dimensions while drawing.
- While doing assemblies provide the appropriate relation between the mating parts.
- Check whether all the parts are mating perfectly without any interference.
- Save the current work on regular basis so that the data is not lost.

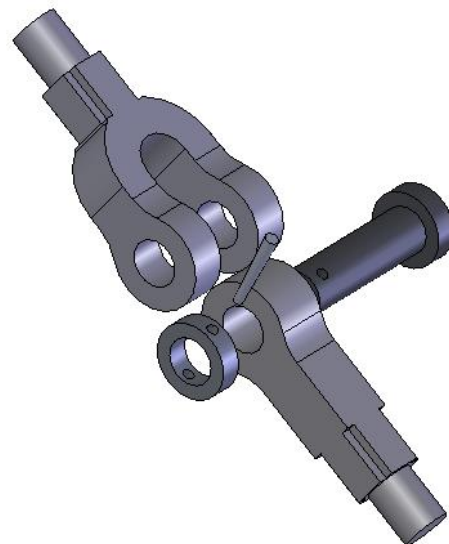
Applications:

A knuckle joint is used to connect two rods which are under the action of tensile loads and if the joint is guided, the rods may support a compressive load. Its use may be found in the link of a cycle chain, tie rod, joint for roof truss, valve rod joint with eccentric rod, pump rod joint, tension link in bridge structure and lever and rod connections of various types.

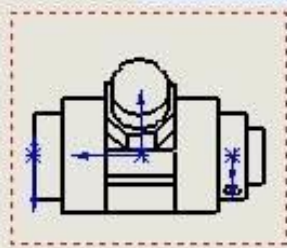
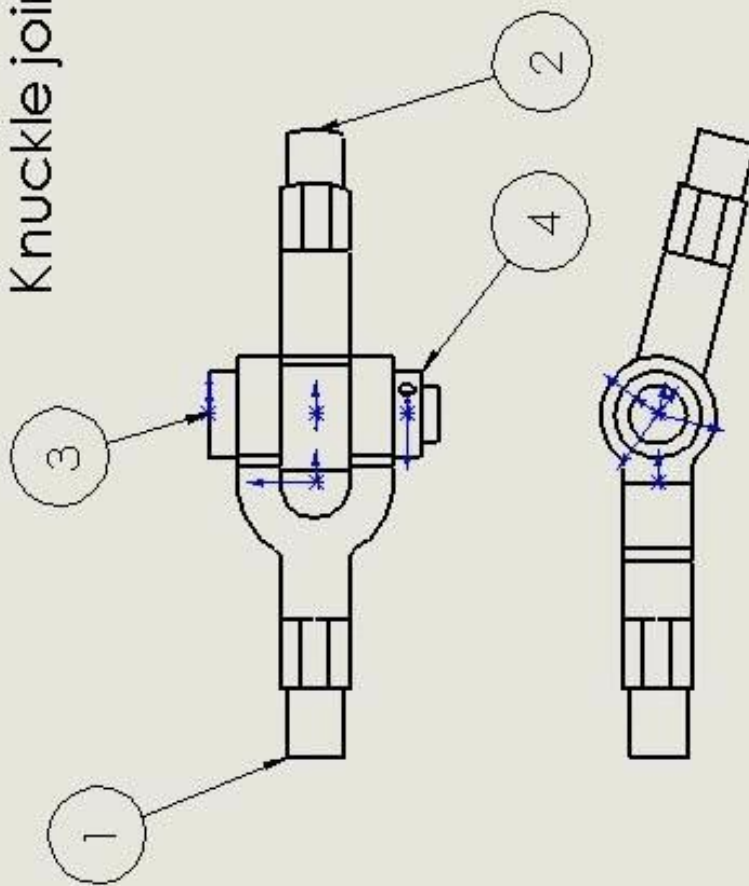
Assembly of Knuckle Joint



Exploded view



Knuckle joint



Part no.	Name	Material	QTY.
1	Fork end	FS	1
2	Eye end	FS	1
3	Pin	MS	1
4	Collar	MS	1

Mass properties of Knuckle Joint

Mass properties of knuckle joint (Assembly Configuration - Default)

Output coordinate System: -- default --

Mass = 347.72 grams

Volume = 347719.90 cubic millimeters

Surface area = 67283.32 millimeters²

Center of mass: (millimeters)

$$X = 9.15$$

$$Y = 0.05$$

$$Z = 33.56$$

Principal axes of inertia and principal moments of inertia: (grams * square millimeters)

Taken at the center of mass.

$$I_x = (1.00, 0.00, 0.01) \quad P_x = 208476.81$$

$$I_y = (0.01, -0.00, -1.00) \quad P_y = 845749.97$$

$$I_z = (-0.00, 1.00, -0.00) \quad P_z = 979769.84$$

Moments of inertia: (grams * square millimeters)

Taken at the center of mass and aligned with the output coordinate system.

$$L_{xx} = 208504.53 \quad L_{xy} = 7.48 \quad L_{xz} = 4202.55$$

$$L_{yx} = 7.48 \quad L_{yy} = 979769.83 \quad L_{yz} = 35.27$$

$$L_{zx} = 4202.55 \quad L_{zy} = 35.27 \quad L_{zz} = 845722.27$$

Moments of inertia: (grams * square millimeters)

Taken at the output coordinate system.

$$I_{xx} = 600152.49 \quad I_{xy} = 172.99 \quad I_{xz} = 110994.19$$

$$I_{yx} = 172.99 \quad I_{yy} = 1400536.07 \quad I_{yz} = 642.26$$

$$I_{zx} = 110994.19 \quad I_{zy} = 642.26 \quad I_{zz} = 874842.42$$

EXPERIMENT NO-6

(PLUMMER BLOCK)

AIM

- To model the parts of the Steam engine Connecting Rod & Do the assembly of the parts using SW-2008 software.

PARTS

- There are 6 main Parts:

- 2) Connecting rod
- 2) Brasses
- 3) Strap
- 4) Jib
- 5) Cotter.
- 6) Set Screw

PROCEDURE

Part Modelling

- Switch on the computer and open the solid works 2008 software.
- Model all the given parts separately by selecting a new sheet made for drawing & modeling parts.
- And to model a part select a plane onto which the model has to be prepared.
- Using the sketch tools draw the required part (2D)
- Then by using the feature tools convert the 2D drawing into 3D part.
- Also the part can further be modified by other editing options available.
- Save all the modelled parts after each completion.

Assembly

- Now open a new sheet made for assembling all the model parts.
- Insert components from the location where you have saved the modeled parts.
- Fix the main component to the sheet origin.
- Now insert other components and place randomly on the sheet.
- Using the mate option, mate all the parts assembly in a systematic manner.

- Provide the Exploded view, parts of Assembly, Sectional & Detail view for better understanding
- Give the mass properties of the final assembly.
- Save the file after completion.

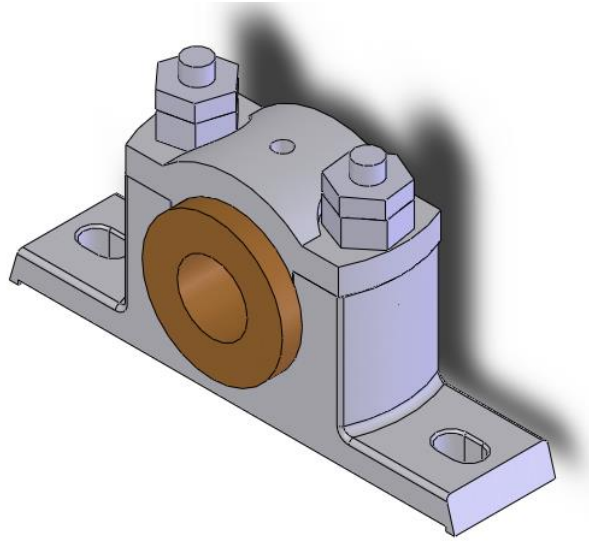
Precautions:

- When modeling a component make sure it is not under defined or over defined.
- Give exact dimensions while drawing.
- While doing assemblies provide the appropriate relation between the mating parts.
- Check whether all the parts are mating perfectly without any interference.
- Save the current work on regular basis so that the data is not lost.

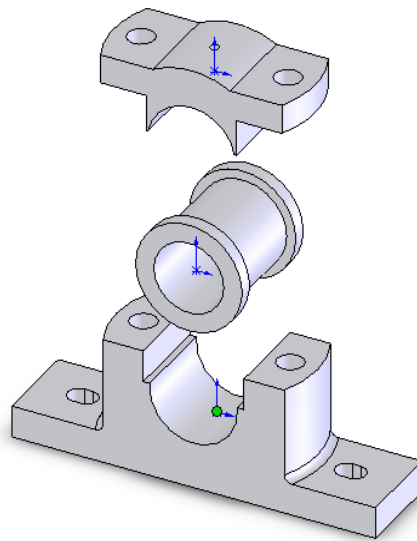
Applications:

A plumber block or split bearing is used for shaft running at high speeds and carrying loads. This type of bearing can be placed anywhere along the shaft length. It is used for long shafts requiring intermediate support. The plumber block is an independent bearing, adopted for bolting to a support on a masonry wall, a steel girder or even on a large machine base

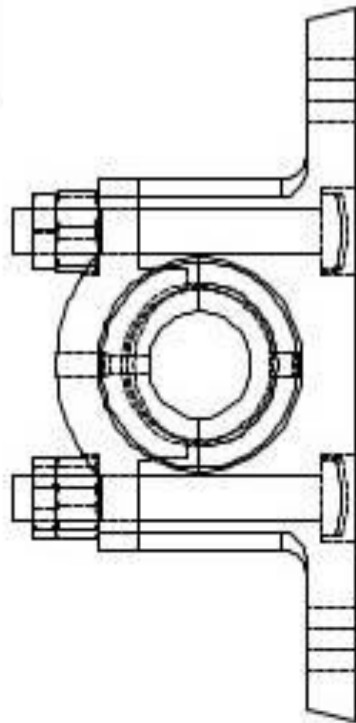
Assembly Of Plummer Block



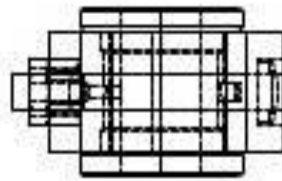
Exploded View



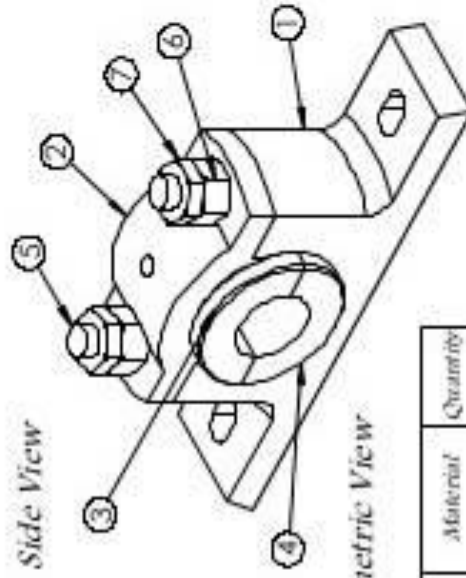
Plummer Block



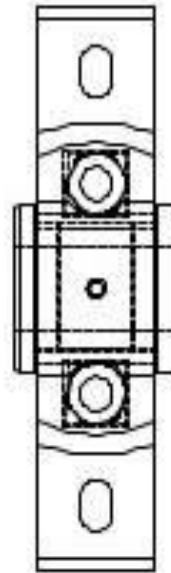
Front View



Side View



Isometric View



Top View

Item number	Title	Material	Quantity
1	Body	Cast Iron	1
2	Cap	Cast Iron	1
3	Upper Brass	Gun Metal	1
4	Lower Brass	Gun Metal	1
5	SQ HD Bolt	Mild Steel	2
6	Nut	Mild Steel	2
7	Locknut	Mild Steel	2

Mass Properties Of Plummer Block

Mass properties of plumber block (Assembly Configuration - Default)

Output coordinate System: -- default --

Mass = 3427.03 grams

Volume = 451689.41 cubic millimeters

Surface area = 110347.95 millimeters²

Center of mass: (millimeters)

$$X = 0.00$$

$$Y = 43.25$$

$$Z = 22.82$$

Principal axes of inertia and principal moments of inertia: (grams * square millimeters)

Taken at the center of mass.

$$I_x = (1.00, 0.00, 0.00) \quad P_x = 3266510.43$$

$$I_y = (0.00, 1.00, -0.00) \quad P_y = 5499831.66$$

$$I_z = (0.00, 0.00, 1.00) \quad P_z = 7123514.51$$

Moments of inertia: (grams * square millimeters)

Taken at the center of mass and aligned with the output coordinate system.

$$L_{xx} = 3266510.43 \quad L_{xy} = 2.11 \quad L_{xz} = 1.56$$

$$L_{yx} = 2.11 \quad L_{yy} = 5499835.22 \quad L_{yz} = -2403.89$$

$$L_{zx} = 1.56 \quad L_{zy} = -2403.89 \quad L_{zz} = 7123510.95$$

Moments of inertia: (grams * square millimeters)

Taken at the output coordinate system.

$$I_{xx} = 11461864.14 \quad I_{xy} = 3.91 \quad I_{xz} = 2.51$$

$$I_{yx} = 3.91 \quad I_{yy} = 7284958.25 \quad I_{yz} = 3380353.92$$

$$I_{zx} = 2.51 \quad I_{zy} = 3380353.92 \quad I_{zz} = 13533741.63$$

EXPERIMENT NO-7

(FOOT-STEP BEARING)

AIM

- To model the parts of the Foot step Bearing & Do the assembly of the parts using SW-2008 software.

PARTS

- There are 5 main Parts:

1) Body 2) Bush 3) Disc 4) Shaft 5) Pin.

PROCEDURE

Part Modelling

- Select the **SOLID EDGE** icon → New → Part.
- To model **Body**, first select the required plane.
 - Draw the 2D sketch of the Body using the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Body.
- To model **Bush** → new → Part.
 - Draw the 2D sketch of the Bush using the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Bush.
- To model **Disc** → new → Part.
 - Draw the 2D sketch of the Disc using the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.

- Save the part as Disc.
- To model **Shaft**→new→Part.
 - Draw the 2D sketch of the Shaft using the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Shaft.
- To model **Pin**→new→Part.
 - Draw the 2D sketch of the Pin using the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Pin.

Assembly

- To assemble →new→Assembly.
 - Insert the main component first, by clicking→Browse.
 - Place the main component in the area, this part is fixed and cannot be moved further.
 - Insert the remaining components for assembly by the same procedure as above, these components can moved anywhere.
 - Use the **MATE** tool to assemble all the parts such as(standard mates, mechanical mates, advanced mates).

2D Engineering Drawing

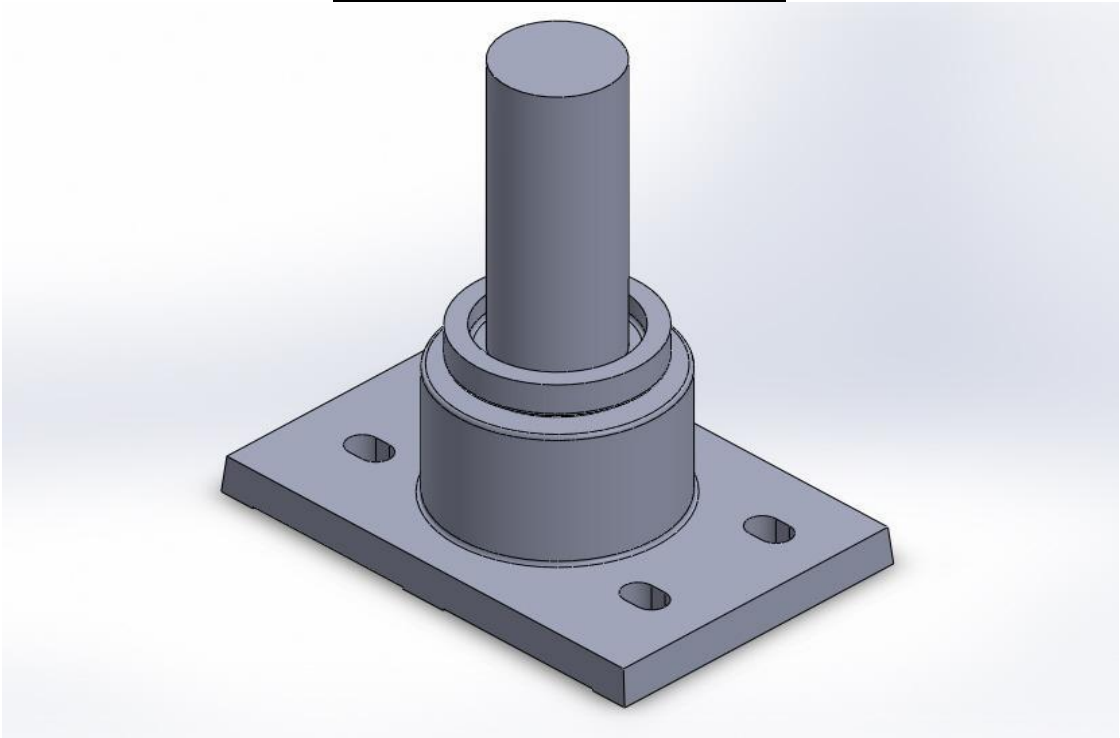
- To start Drawing module →new→Drawing.
 - Select the sheet format(A1,A2,A3,A4)
 - Insert the drawing→Browse.
 - select→**Standard 3view** option.
 - Place the views (top ,front,side) in the required place.
- To insert Bills of Material, select→**Insert** option from the menu bar.

- Goto→Tables→ Bills of Materials.
- Place the table in the required position.
- Edit the columns and rows according to the no. Of components.
- Save the drawing.

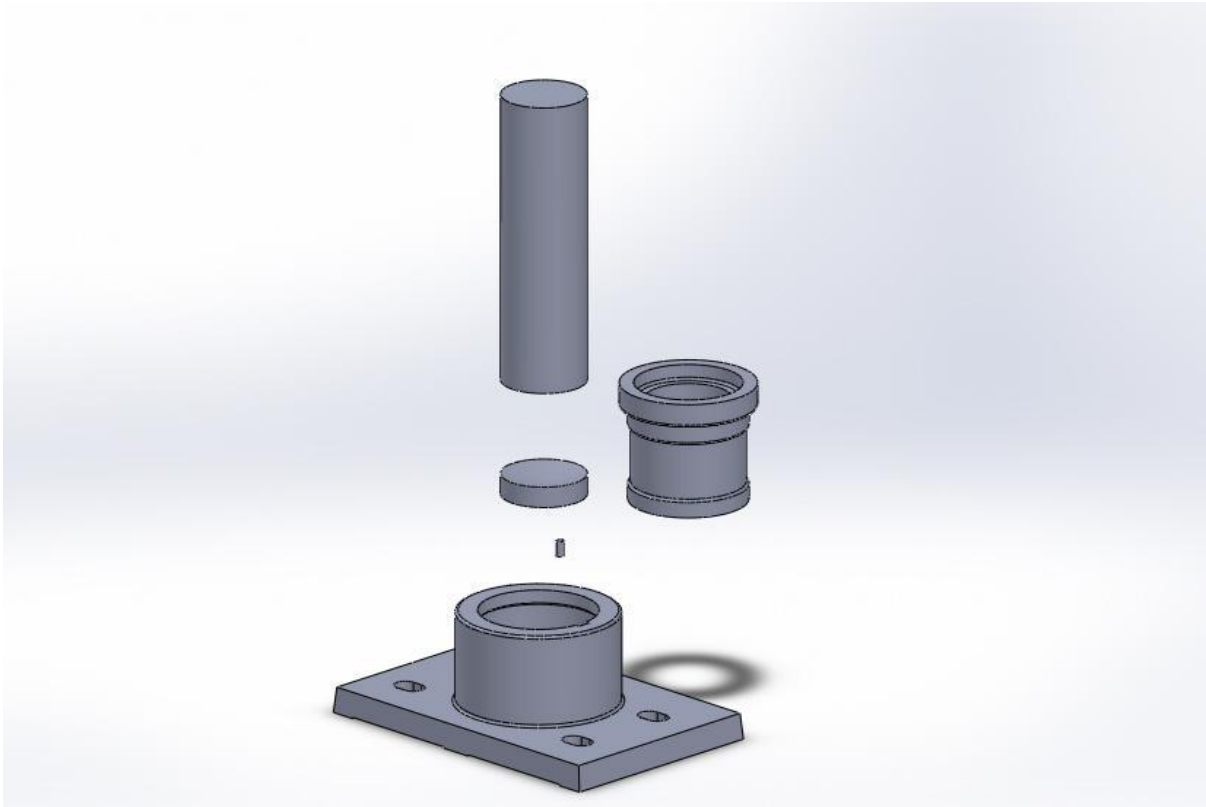
APPLICATIONS

- Footstep bearings are used to support the lower end of the vertical shafts.
- It is fitted at the bottom for carrying the total load.
- These bearings are used in the places where, the shaft end terminates in the bearing.

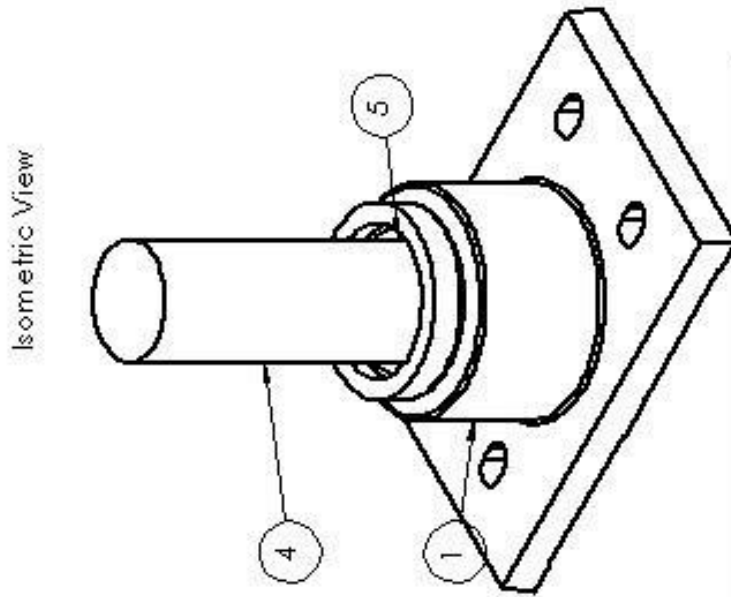
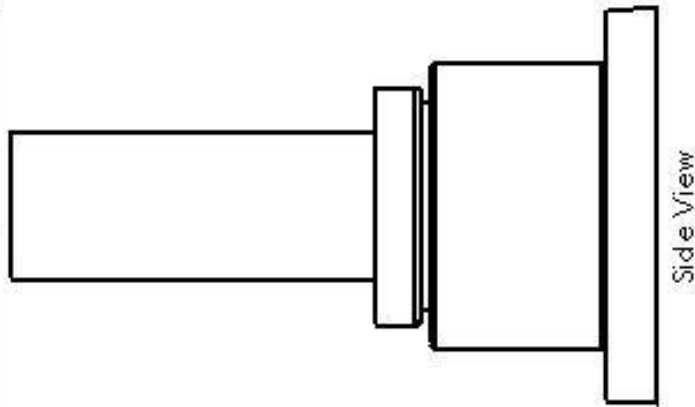
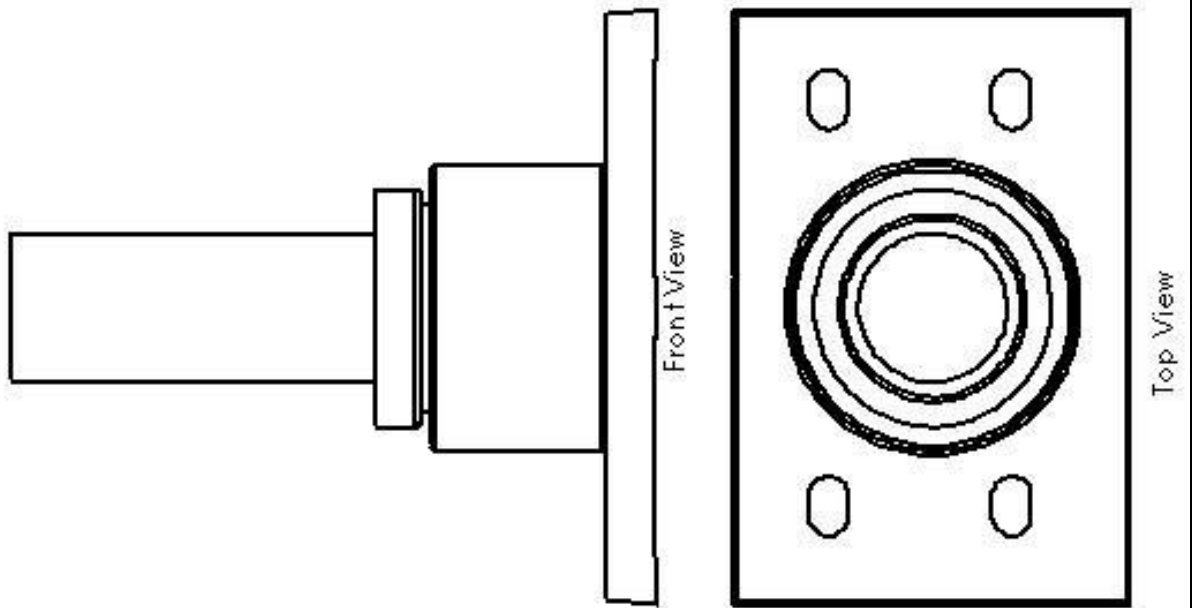
FOOT-STEP BEARING



Exploded View



Foot Step Bearing



ITEM NO.	PART NUMBER	MATERIAL	QTY.
1	Body	Cast Iron	1
2	pin	Mild Steel	1
3	Disc	P Bronze	1
4	Shaft	Mild Steel	1
5	Bush	Brass	1

MASS PROPERTIES

Mass properties of ass (Assembly Configuration - Default)

Output coordinate System: -- default --

Density = 0.00 grams per cubic millimeter

Mass = 1516.85 grams

Volume = 1516853.74 cubic millimeters

Surface area = 221774.96 millimeters²

Center of mass: (millimeters)

$$X = 45.43$$

$$Y = -19.46$$

$$Z = 3.79$$

Principal axes of inertia and principal moments of inertia: (grams * square millimeters)

Taken at the center of mass.

$$I_x = (1.00, 0.00, 0.00) \quad P_x = 4017920.98$$

$$I_y = (0.00, 0.00, -1.00) \quad P_y = 5757305.56$$

$$I_z = (-0.00, 1.00, 0.00) \quad P_z = 6097322.75$$

Moments of inertia: (grams * square millimeters)

Taken at the center of mass and aligned with the output coordinate system.

$$L_{xx} = 4017921.10 \quad L_{xy} = 508.83 \quad L_{xz} = 0.00$$

$$L_{yx} = 508.83 \quad L_{yy} = 6097322.63 \quad L_{yz} = 0.00$$

$$L_{zx} = 0.00 \quad L_{zy} = 0.00 \quad L_{zz} = 5757305.56$$

Moments of inertia: (grams * square millimeters)

Taken at the output coordinate system.

$$I_{xx} = 4613969.35 \quad I_{xy} = -1340208.76 \quad I_{xz} = 260976.22$$

$$I_{yx} = -1340208.76 \quad I_{yy} = 9249084.60 \quad I_{yz} = -111787.60$$

$$I_{zx} = 260976.22 \quad I_{zy} = -111787.60 \quad I_{zz} = 9461595.94$$

EXPERIMENT NO-8

(DRILL JIG)

AIM

- To model the parts of the Drill Jig& Do the assembly of the parts using SW-2008 software.

PARTS

- There are 9 main Parts:

- 1) Base Plate
- 2) Stem
- 3) Jig Plate
- 4) Screw
- 5) Stud
- 6) Nut M20
- 7) Bush (Case Hardened)
- 8) Latch Washer
- 9) Screw.

PROCEDURE

Part Modelling

- Select the **SOLID EDGE** icon→New→Part.
- To model **Base Plate**, first select the required plane.
 - Draw the 2D sketch of the Base Plate using the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Base Plate.
- To model **Stem**→ new→Part.
 - Draw the 2D sketch of the Stem using the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Stem.
- To model **Jig Plate**→new→Part.

- Draw the 2D sketch of the Jig Plate using the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Jig Plate.
- To model **Screw**→new→Part.
- Draw the 2D sketch of the Screw using the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Screw.
- To model **Stud**→new→Part.
- Draw the 2D sketch of the Stud using the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Stud.
- To model **Nut M20**→new→Part.
- Draw the 2D sketch of the Nut M20 using the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Nut M20.
- To model **Bush**→new→Part.
- Draw the 2D sketch of the Bush using the **SKETCH** tools with required dimensions, then → exit sketch.
 - Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
 - Save the part as Bush.
- To model **Latch Washer**→new→Part.

- Draw the 2D sketch of the Latch Washer using the **SKETCH** tools with required dimensions, then → exit sketch.
- Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
- Save the part as Latch Washer.

➤ To model **Screw**→new→Part.

- Draw the 2D sketch of the Screw using the **SKETCH** tools with required dimensions, then → exit sketch.
- Use the **FEATURES** tools to convert the 2D sketch to solid model such as Extrude, Revolve, extrude cut etc.
- Save the part as Screw.

Assembly

➤ To assemble →new→Assembly.

- Insert the main component first, by clicking→Browse.
- Place the main component in the area, this part is fixed and cannot be moved further.
- Insert the remaining components for assembly by the same procedure as above, these components can moved anywhere.
- Use the **MATE** tool to assemble all the parts such as(standard mates, mechanical mates, advanced mates).

2D Engineering Drawing

➤ To start Drawing module →new→Drawing.

- Select the sheet format(A1,A2,A3,A4)
- Insert the drawing→Browse.
- select→**Standard 3view** option.
- Place the views (top ,front,side) in the required place.

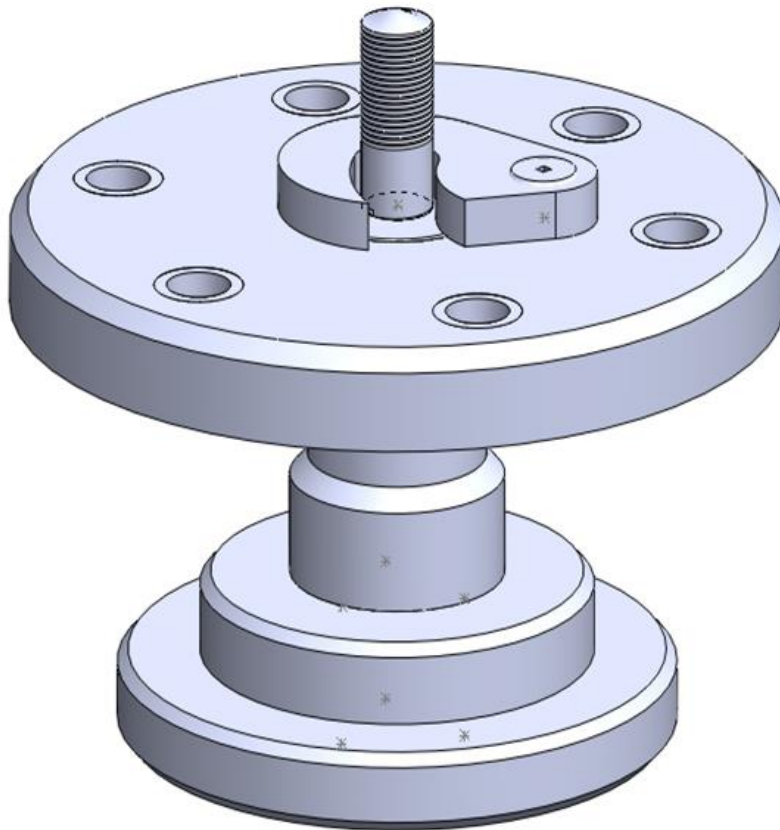
➤ To insert Bills of Material, select→**Insert** option from the menu bar.

- Goto→Tables→ Bills of Materials.
- Place the table in the required position.

- Edit the columns and rows according to the no. Of components.
- Save the drawing.

APPLICATIONS

- A jig is a work holding and tool guiding device which may be used for drilling, reaming, boring and similar operations in mass production.
- The design allows for quick loading and unloading of work pieces.
- A jig's primary purpose is to provide repeatability, accuracy, and interchangeability in the manufacturing of products.
- The most common type of jig is the drill jig, which guides the drill bit for creating holes at desired locations.



MASS PROPERTIES

Mass properties of ass (Assembly Configuration - Default)

Output coordinate System: -- default --

Density = 0.00 grams per cubic millimeter

Mass = 1633.73 grams

Volume = 1633729.84 cubic millimeters

Surface area = 221130.65 millimeters²

Center of mass: (millimeters)

$$X = 34.28$$

$$Y = 47.67$$

$$Z = -8.92$$

Principal axes of inertia and principal moments of inertia: (grams * square millimeters)

Taken at the center of mass.

$$I_x = (0.03, 1.00, -0.01) \quad P_x = 6517397.33$$

$$I_y = (-0.99, 0.03, 0.15) \quad P_y = 7357380.22$$

$$I_z = (0.15, 0.01, 0.99) \quad P_z = 7377214.13$$

Moments of inertia: (grams * square millimeters)

Taken at the center of mass and aligned with the output coordinate system.

$$L_{xx} = 7357219.53 \quad L_{xy} = 23104.16 \quad L_{xz} = -3287.59$$

$$L_{yx} = 23104.16 \quad L_{yy} = 6518133.99 \quad L_{yz} = -9227.34$$

$$L_{zx} = -3287.59 \quad L_{zy} = -9227.34 \quad L_{zz} = 7376638.17$$

Moments of inertia: (grams * square millimeters)

Taken at the output coordinate system.

$$I_{xx} = 11200276.41 \quad I_{xy} = 2692883.67 \quad I_{xz} = -502767.92$$

$$I_{yx} = 2692883.67 \quad I_{yy} = 8567716.16 \quad I_{yz} = -703897.83$$

$$I_{zx} = -502767.92 \quad I_{zy} = -703897.83 \quad I_{zz} = 13009349.96$$

Manual Part Programming

G Codes (Preparatory Functions)

Modal and Non-modal G Codes.

G Codes - General Notes.

G00 (Rapid Positioning / Traverse).

G01 (Linear Interpolation).

G02 / G03 (Circular Interpolation).

G04 (Dwell).

G20 / G21 (Inch/Metric Data Input).

G28 (Reference Point Return).

G40 / G41 / G42 (Tool Nose Radius Compensation).

G50 (Multiple Command Functions).

G70 (Finishing Cycle).

G71 (Stock Removal in X Axis).

G72 (Stock Removal in Facing).

G73 (Pattern Repeating).

G74 (End Face Peck Drilling Cycle).

G75 (Outer / Internal Dia. Drilling & Grooving Cycle).

G76 (Multiple Thread Cutting Cycle).

G81 (Deep Hole Drilling Cycle).

G90 (Outer / Internal Dia. Cutting Cycle).

G92 (Thread Cutting Cycle).

G94 (End/Taper Face Turning Cycle).

G96 (Constant Surface Speed Control).

G97 (Spindle Speed in Rev/Minute).

G98 (Per Minute Feed).

G99 (Per Revolution Feed).

M Codes (Miscellaneous Function)

M00 (Program Stop).

M01 (Optional Stop).

M02 (End of Program).

M03 (Spindle Forward).

M04 (Spindle Reverse).

M05 (Spindle Stop).

M06 (Automatic Tool Change).

M08 (Coolant On).

M09 (Coolant Off).

M10 (Chuck Open).

M11 (Chuck Close).

M13 (Spindle Forward and Coolant On).

M14 (Spindle Reverse and Coolant On).

M25 (Tailstock Quill Extend).

M26 (Tailstock Quill Retract).

M30 (Program Stop and Reset).

M40 (Parts Catcher Extend).

M41 (Parts Catcher Retract).

M62 / M63 / M64 / M65 / M66 / M76 / M77
(Auxiliary Output Functions).

M98 (Sub Program Call).

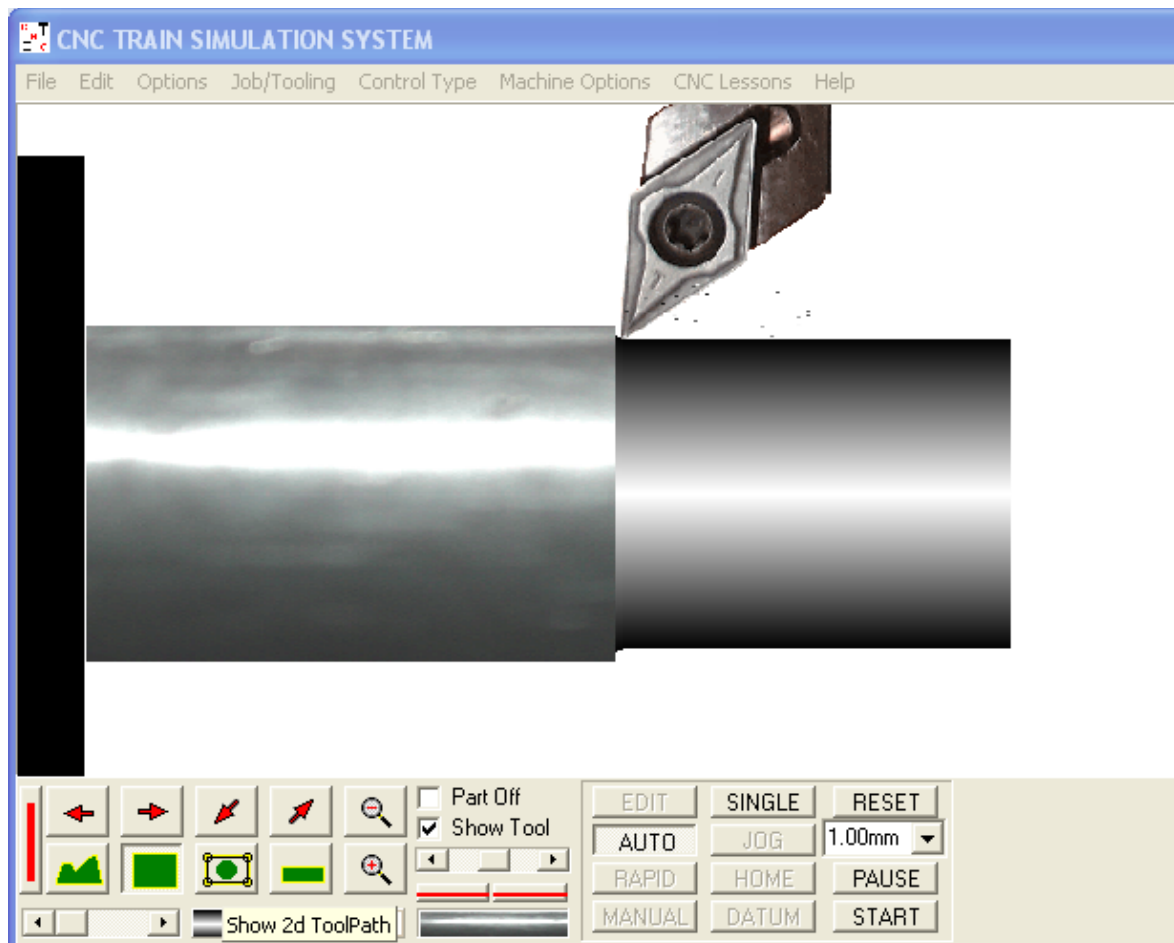
M99 (Sub Program End and Return).

FACING OPERATION



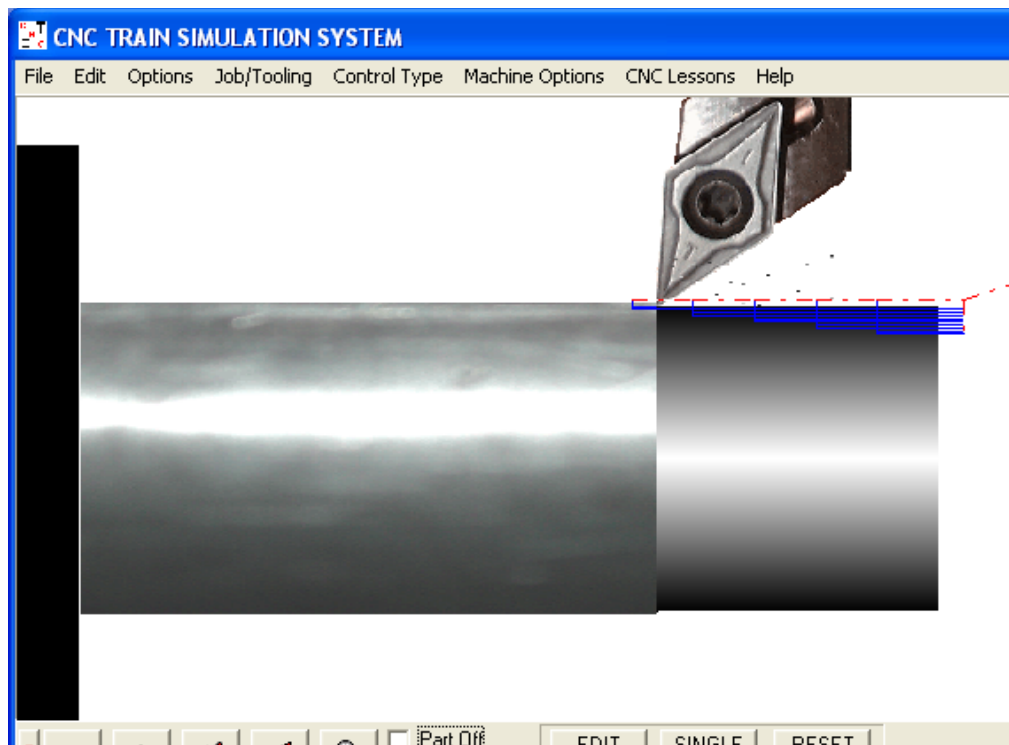
```
G21 G98  
G28 U0 W0  
M06 T08  
M03 S1200  
G00 X26 Z2  
G01 Z-0.5 F60  
X0  
Z1  
G00 X26  
G01 Z-1  
X0  
Z1  
G00 X26  
G01 Z-1.5  
X0  
Z1  
G00 X26  
G01 Z-2  
X0  
Z1  
G00 X26  
G28 U0 W0  
M05  
M30
```

TURNING OPERATION



```
G21 G98  
G28 U0 W0  
M06 T08  
M03 S1200  
G00 X26 Z2  
G90 X24.7 Z-30 F60  
X24.3  
X24  
X23.7  
X23.3  
X23  
X22.7  
X22.4  
X22  
G28 U0 W0  
M05  
M30
```

STEP TURNING



G21 G98

G28 U0 W0

M06 T08

M03 S1300

G00 X26 Z2

G90 X24.9 Z-25 F60

X24.4

X23.9 Z-20

X23.4

X22.9 Z-15

X22.4

X21.9 Z-10

X21.4

X20.9 Z-5

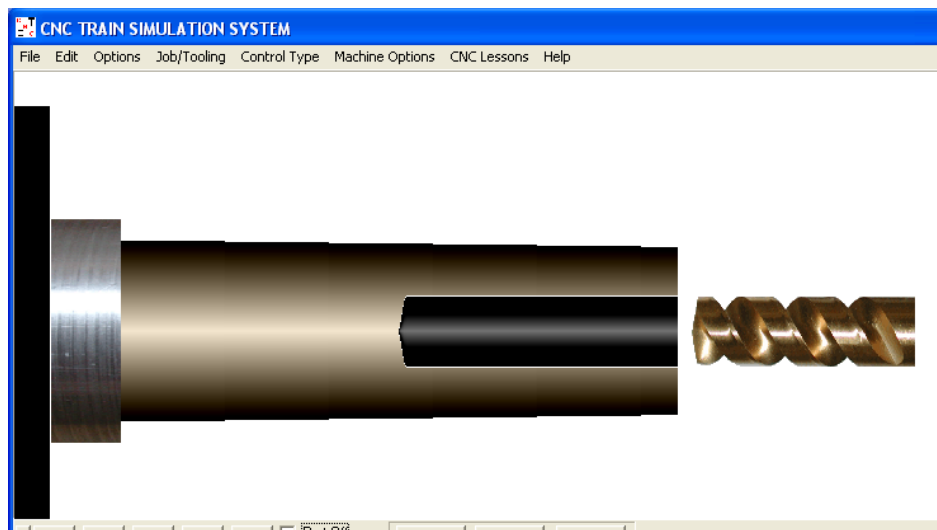
X20.4

G28 U0 W0

M05

M30

TAPER TURNING AND DRILLING



O2002

M06 T01

G00 Z2. X32. M03 S3445

G90 Z-80. X28. R-3.

X26.

G28 U0. W0.

M06 T02

G00 Z2. X0.

M03 S2345

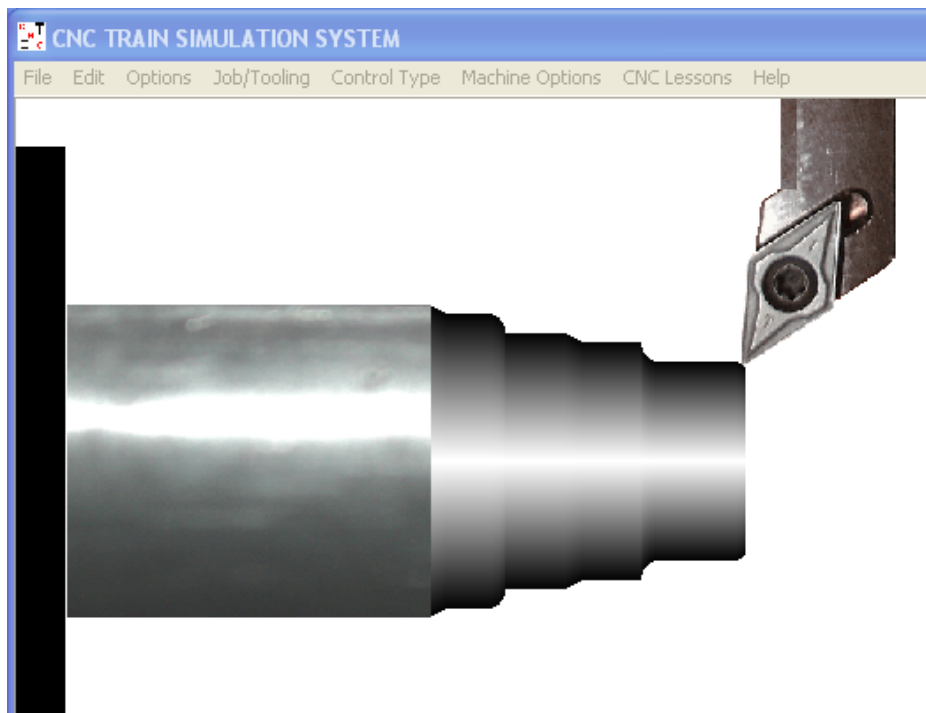
G74 R1.5

G74 Z-40. Q1000 F130.

M05

M30

CONTOURING



G21 G98

G28 U0 W0

M06 T08

M03 S1300

G00 X33 Z2

G71 U0.5 R1

G71 P08 Q20 U0.1 W0.1 F60

N08 G00 X19

G01 Z0

G01 X20 Z-1

G01 Z-9

G02 X24 Z-11 R2

G01 Z-17

G01 X26 Z-19

G01 Z-25

G03 X30 Z-27 R2

G01 Z-31

N20 G01 X32 Z-33

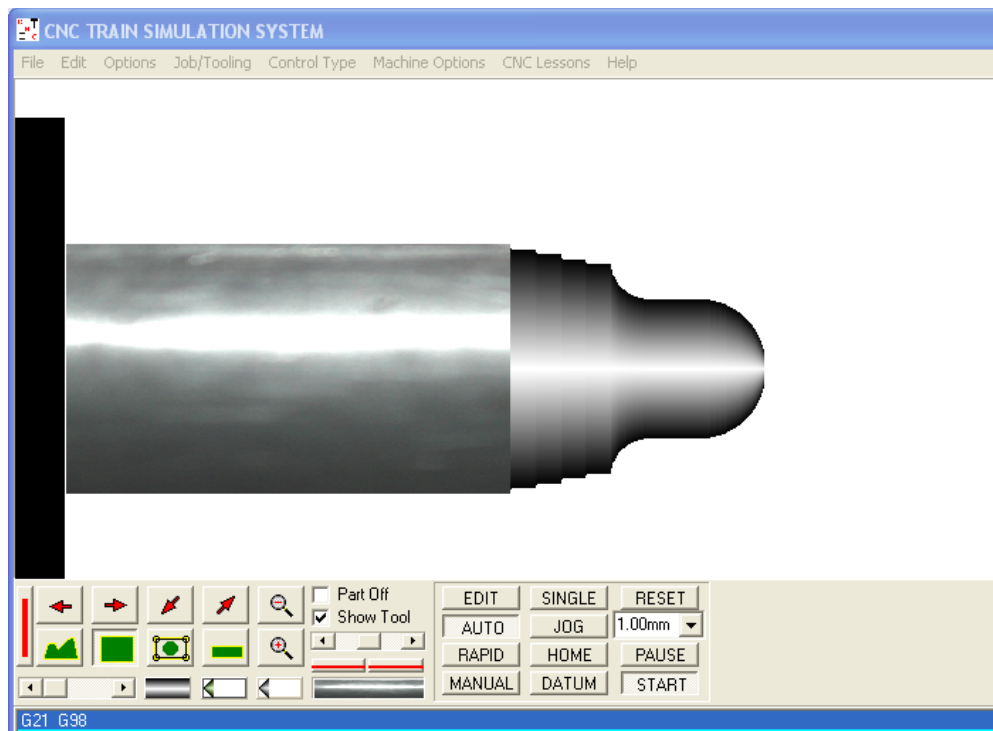
G70 P08 Q20

G28 U0 W0

M05

M30

CONTOURING



G21 G98

G28 U0 W0

M06 T08

M03 S1300

G00 X26 Z2

G71 U0.5 R1

G71 P10 Q20 U0.1 W0.1 F60

N10 G00 X0

G01 Z0

G03 X14 Z-7 R7

G01 Z-12

G02 X21 Z-15.5 R3.5

G01 Z-18

G01 X22

G01 Z-20.5

G01 X23

G01 Z-23

G01 X24

G01 Z-25.5

N20 X25

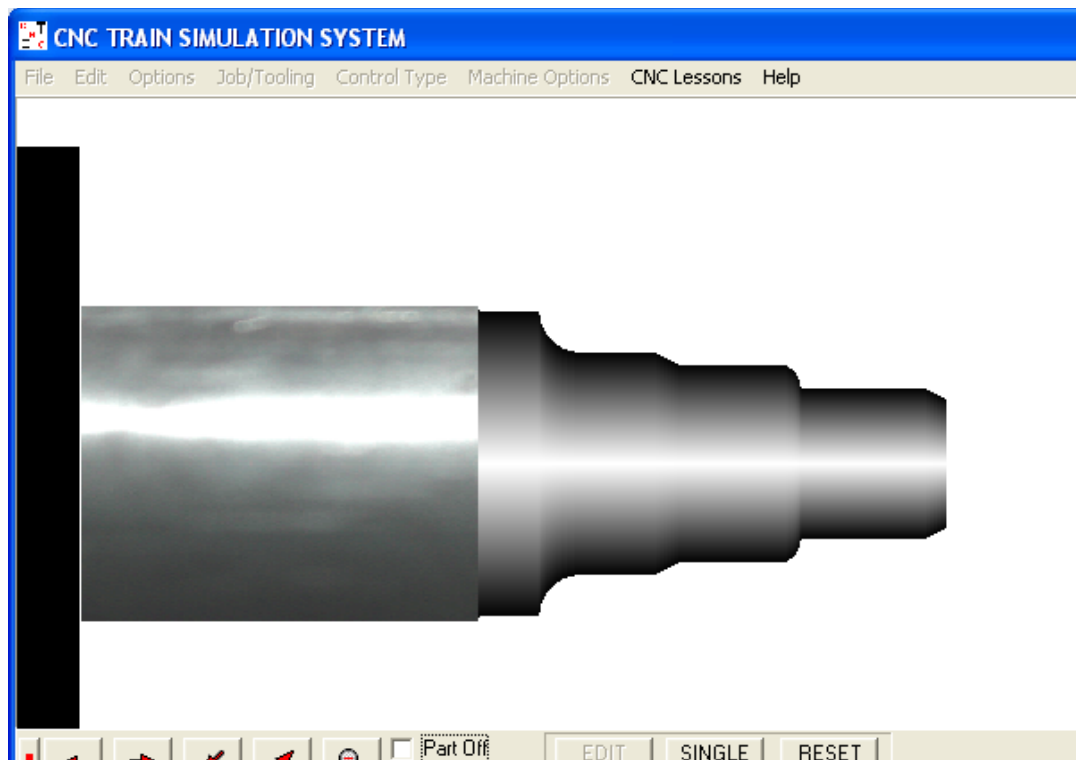
G70 P10 Q20 F40

G28 U0 W0

M05

M30

CONTOURING



G21 G98

G28 U0 W0

M06 T08

M03 S1300

G00 X26.4 Z2

G71 U0.5 R1

G71 P10 Q20 U0.1 W0.1 F60

N10 G00 X10

G01 Z0

G01 X12 Z-2

G01 Z-12

G03 X16 Z-14 R2

G01 Z-22

G01 X18 Z-24

G01 X18 Z-30

G02 X24 Z-33 R3

G01 X24.5

G01 Z-38

N20 X25.4

G70 P10 Q20 F40

G28 U0 W0

M05

M30

THREADING



G21 G98

G28 U0 W0

M06 T08

M03 S1300

G00 X26 Z2

G71 U0.5 R1

G71 P10 Q20 U0.1 W0.1 F60

N10 G00 X9

G01 Z0

G01 X10 Z-1

G01 Z-11

G02 X16 Z-14 R3

G01 Z-20

G03 X22 Z-23 R3

G01 Z-29

N20 X25 Z-33

G70 P10 Q20 F40

G28 U0 W0

M06 T06

M03 S400

G00 X10.2 Z2

G76 P031560 Q60 R0.02

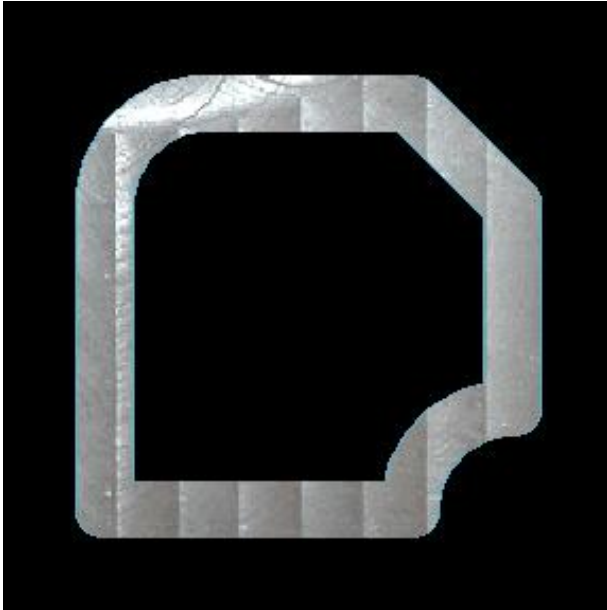
G76 X8.77 Z-10 P613 Q60 F1

G28 U0 W0

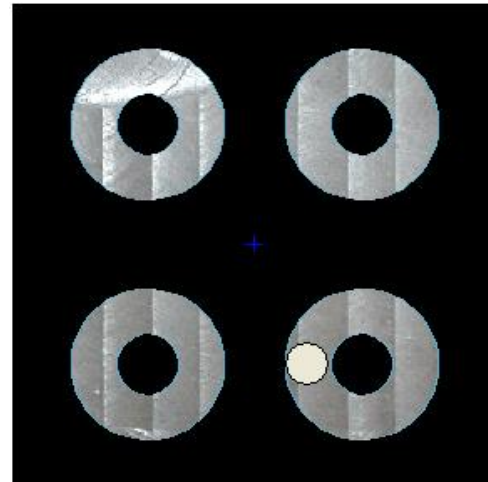
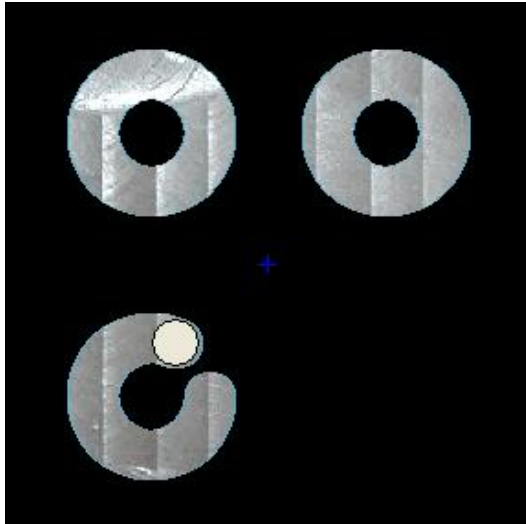
M05

M30

CONTOURING



```
G21 G94
G91 G28 Z0
G28 X0 Y0
M06 T01
M03 S2000
G90 G00 X15 Y15
G00 Z5
G01 Z-0.5 F60
G01 X60 Y15 F100
G02 X75 Y30 R15
G01 X75 Y60
G01 X60 Y75
G01 X30 Y75
G03 X15 Y60 R15
G01 X15 Y15
G00 Z10
G91 G28 Z0
G28 X0 Y0
M05
M30
```



```

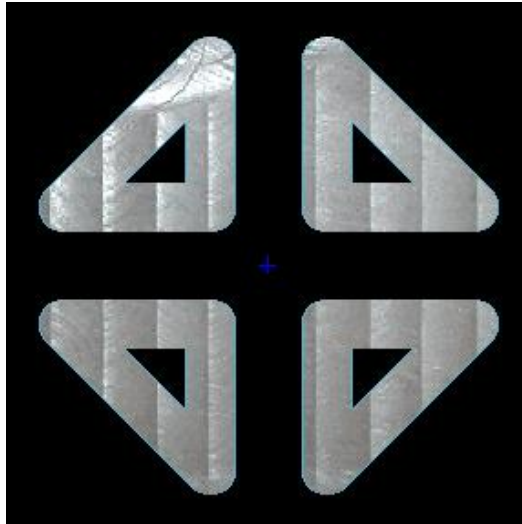
G21 G94
G91 G28 Z0
G28 X0 Y0
M06 T01
M03 S2000
G90 G00 X0 Y0
Z5
M98 P0012015
M70
M98 P0012015
M71
M98 P0012015
M80
M98 P0012015
M81
G91 G28 Z0
G28 X0 Y0
M05
M30

```

```

O2015
G90 G00 X10 Y22.5
G01 Z-0.5 F60
G02 X30 Y22.5 R10
G02 X10 Y22.5 R10
G00 Z5
M99

```



G21 G94
G91 G28 Z0
G28 X0 Y0
M06 T01
M03 S2000
G90 G00 X0 Y0
Z5
M98 P0012015
M70
M98 P0012015
M71
M98 P0012015
M80
M98 P0012015
M81
G91 G28 Z0
G28 X0 Y0
M05
M30

O2015
G90 G00 X10 Y10
G01 Z-0.5 F60
G01 X35 Y10 F80
G01 X10 Y35
G01 X10 Y10
G00 Z5
M99