MALLA REDDY ENGINEERING COLLEGE

(Autonomous)

Department of Mechanical Engineering

CAD/CAM LAB Manual

INTRODUCTION

- CAD /CAM stands for Computer Aided Design and Computer Aided Manufacturing. It is the technology concerned with the use of computers to perform design and manufacturing functions.
- CAD can be defined as the use of computers systems to aid in the creation, modification, analysis, or optimization of a design.
- CAM is the use of computer systems to plan, manage and control the operations of manufacturing plant through either direct or indirect computer interface with the plant's production resources.

CAD software is used to increase the productivity of the designer, improve the quality of design, improve communications through documentation, and to create a database for manufacturing. CAD output is often in the form of electronic files for print, machining, or other manufacturing operations.

CAD (computer-aided design) software is used by architects, engineers, drafters, artists, and others to create precision drawings or technical illustrations. CAD software can be used to create two-dimensional (2-D) drawings or three-dimensional (3-D) models.

CAD is an important industrial art extensively used in many applications, including automotive, shipbuilding, and aerospace industries, industrial and architectural design, prosthetics, and many more

Computer Aided Drafting is a process of preparing a drawing of an object on the screen of a computer. There are various types of drawings in different fields of engineering and sciences. In the fields of mechanical or aeronautical engineering, the drawings of machine components and the layouts of them are prepared. In the field of civil engineering, plans and layouts of the buildings are prepared. In the field of electrical engineering, the layouts of power distribution system are prepared. In all fields of engineering use of computer is made for drawing and drafting.

BENEFITS OF CAD:

The implementation of the CAD system provides variety of benefits to the industries in design and production as given below:

1. Increases Productivity
2. Easier to Read
3. Higher Quality Designs
4. Customer modifications in drawing are easier
5. Reuse and Easily Change Designs
6. Improved accuracy of drawing
7. Simplified Sharing
8. Colours can be used to customize the product
9. Production of orthographic projections with dimensions and tolerances
10. Documenting the Design
11. Preparation of assembly or sub assembly drawings

12. Preparation of part list
13. Machining and tolerance symbols at the required surfaces
14. Printing can be done to any scale

Computer-aided design (CAD) is software that product developers use to create digital 2D drawings and 3D models. These designs are typically used to show objects that will later be manufactured and delivered to customers.

CAD SOFTWARES

1. AUTOCAD
2. PTC CREO (Pro/ENGINEER)
3. CATIA
4. NX (UG NX OR UNIGRAPHICS)5. PAINT
5. On shape
6. Autodesk INVENTOR
7. SOLID WORKS
8. HYPERMESH
9. Autodesk Fusion 360

INTRODUCTION TO PTC CREO

PTC (Parametric Technology Corporation) is a computer software and services company founded in 1985 and headquartered in Boston, Massachusetts. The company began initially developing parametric, associative feature-based, solid computer-aided design (CAD) modeling software in 1988, including an Internet-based product for product lifecycle management (PLM) in 1998.

CREO(Formerly known as Pro/ENGINEER) is a product design software created by PTC for 2D/3D CAD, parametric, direct modelling. It also features simulation tools for analyzing the product’s performance including structural, vibration, and thermal analysis.

Creo is a powerful, integrated family of product design software. It’s used by thousands of leading manufacturers across the globe. It is a PTC product – the originators of parametric CAD technology. The way Creo works is that it is made up of individual apps, including:

- Creo Parametric
- Creo Simulate
- Creo Direct
- Creo Layout
- Creo Options Modeler

Each Creo app serves a different purpose in the product development process. This means that Creo takes you through every stage, including concept design work, design and analysis. Then it also enables you to communicate effectively with downstream partners, for instance manufacturing and technical publications.

Creo Parametric is the essential tool for 3D CAD. It is state-of-the-art software, which promotes best practices in design and maintains your industry standards. Answer your pressing design challenges with Creo Parametric, with its fully-fledged powerful yet flexible 3D CAD abilities. Use it to accommodate multi-CAD data, electromechanical design and make alterations late in the design process.

The PTC Creo suite includes product design and engineering software solutions. Creo runs on Microsoft Windows and provides apps for 3D CAD parametric feature solid modeling, 3D direct modeling, 2D orthographic views, Finite Element Analysis and simulation, schematic design, technical illustrations, and viewing and visualization.

Creo is a family or suite of Computer-aided design (CAD) apps supporting product design for discrete manufacturers and is developed by PTC. The suite consists of apps, each delivering a distinct set of capabilities for a user role within product development.

Creo runs on Microsoft Windows and provides apps for 3D CAD parametric featuresolid modeling, 3D direct modeling, 2D orthographic views, Finite Element Analysis and simulation, schematic design, technical illustrations, and viewing and visualization.

Release history

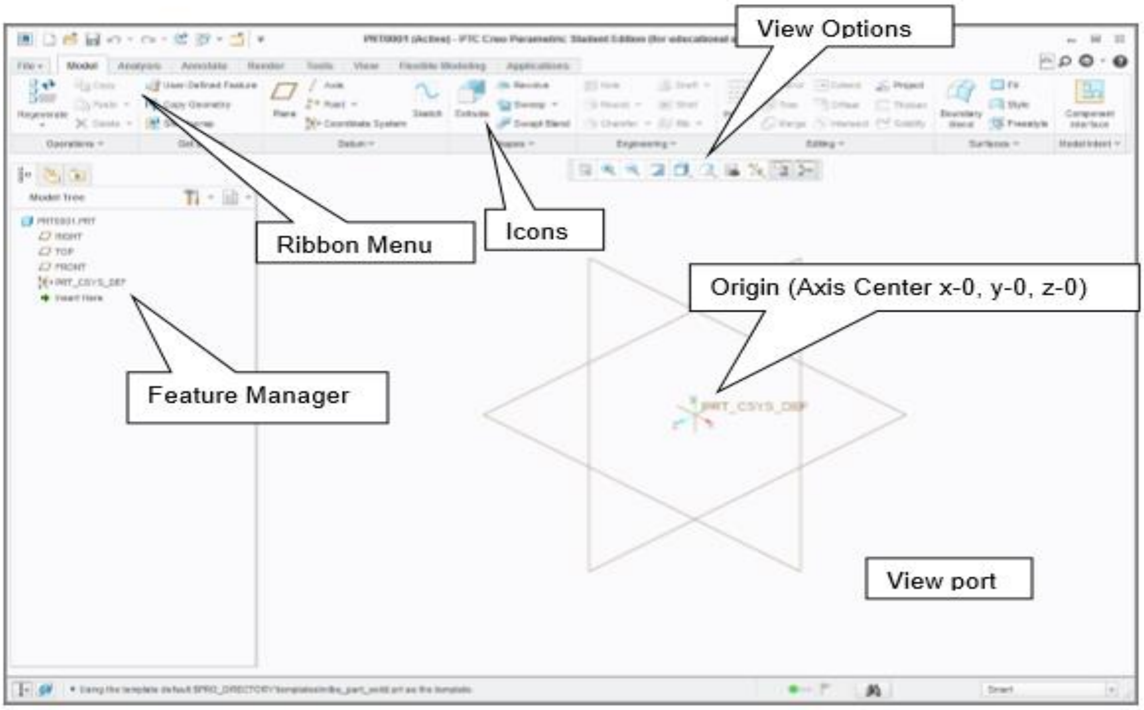
Version	Release date
Creo 1.0	6 January 2011
Creo 2.0	27 March 2012
Creo 3.0	17 March 2014
Creo 4.0	15 December 2016 [5]
Creo 5.0	19 March 2018 [6]
Creo 6.0	19 March 2019

Protrusion features:

Protrusion is the method of adding a solid material of the method. Pro-E provides the following basic methods of adding material to a method.

- Extrude: creates a solid feature by extruding a section named to the section.
- Revolve: creates a solid surface by revolving a section about an axis.
- Sweep: creates a solid feature by sweeping a section about a path.
- Blend: creates a solid feature by blending various cross sections at various levels.

PTC Creo Parametric 3.0 Interface



Mouse Buttons

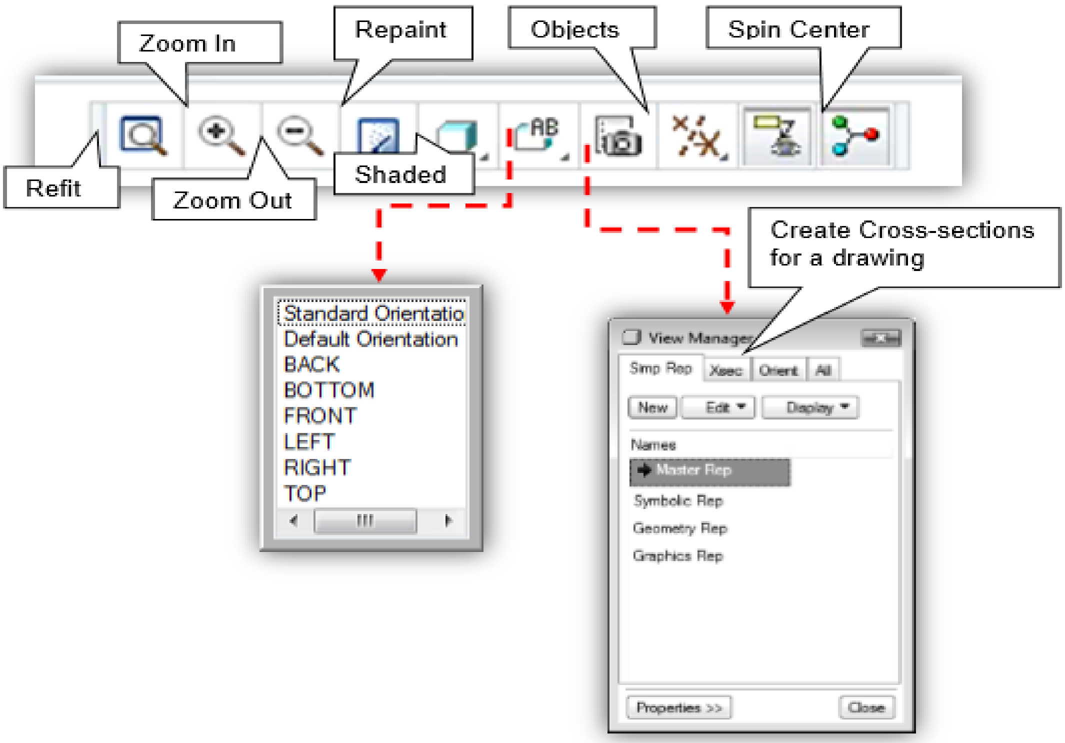
Left Button - Most commonly used for selecting objects on the screen or sketching.

Right Button – Used for activating pop-up menu items, typically used when editing. (Note: you must hold the down button for 2 seconds)

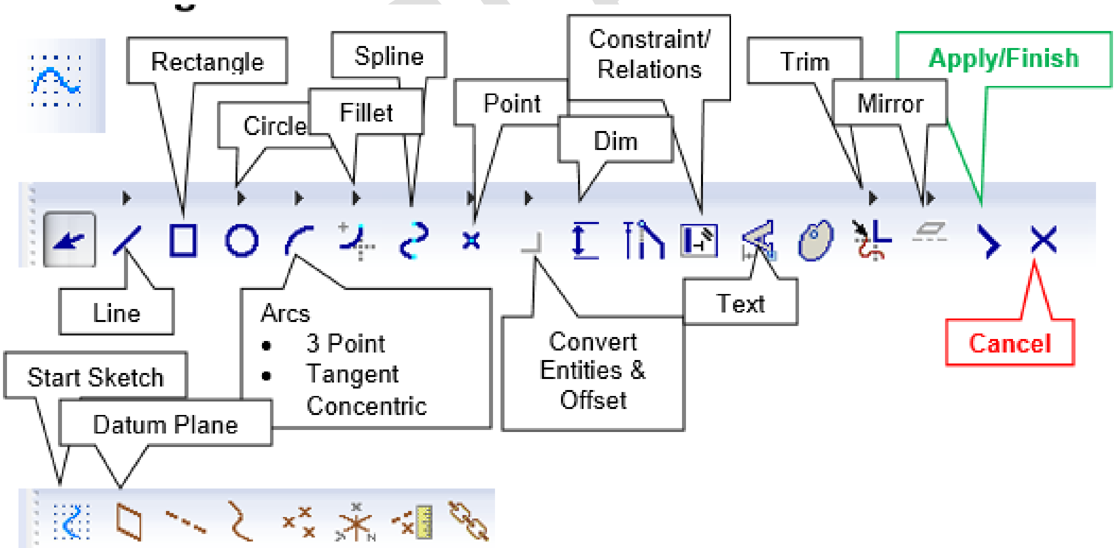
Center Button – (option) Used for model rotation, dimensioning, zoom when holding Ctrl key, and pan when holding Shift key. It also cancels commands and line chains.

Center Scroll Wheel – (option) same as Center Button when depressed, only it activates Zoom feature when scrolling wheel.

View options:



Sketching



NOTE: If you do not see all of these icons on your interface you can customize the toolbars to bring them up. Right mouse button click on the top grey frame of the window and locate the “customize” option.

There are 3 primary file types in Creo, which include…

1. Part (.prt) Single part or volume.
2. Assembly (.asm) Multiple parts in one file assembled.
3. Drawing (.drw) The 2D layout containing views, dimensions, and annotations.

List of Experiments

CAD/CAM Lab

List of Experiment

I. CAD

1. Generation of 3D Model through Extrude Feature.
2. Generation of 3D Model through Extrude, Round & Mirror Features.
3. Generation of 3D Model through Extrude, Hole, Pattern &Rib Features.
4. Generation of 3D Model through Revolve Feature.
5. Feature based and Boolean based 3D Modeling.
6. Modeling and Assembly of Flange Coupling.

II. CAM

7. Step Turning operation using CNC Lathe Machine.
8. Multiple Turning operation using CNC Lathe Machine.
9. Thread cutting operation using CNC Lathe Machine.
10. Drilling Cycle using CNC Milling Machine.
11. Contour Cycle using CNC Milling Machine.
12. Pocketing Cycle using CNC Milling Machine.

EXERCISE:1 **3D Part Modeling – 1**

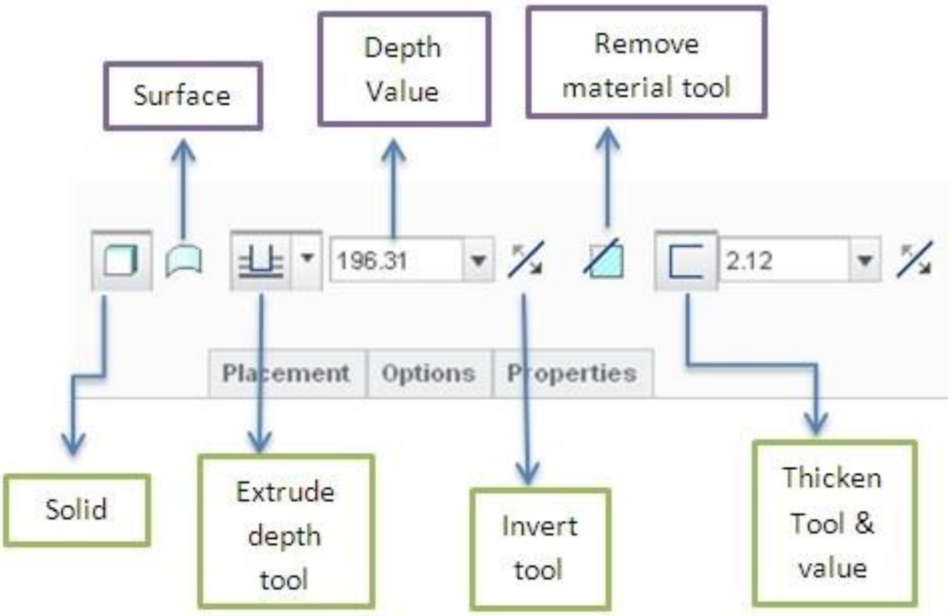
AIM:
To model the given object by using the Extrude features as per the given dimensions.

Software required: PTC Creo Parametric 3.0

Commands used:
2D Commands: line, circle , arc, trim,
3D Commands: Extrude,Mirror, Round

Description of Extrusion Feature:

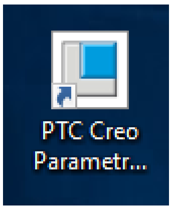
An extrude feature is based on a two-dimensional sketch. It linearly extrudes a sketch perpendicular to the sketching plane to create or remove material. You can either select the sketch first or then start the Extrude tool, or you can start the Extrude tool and then select the sketch.



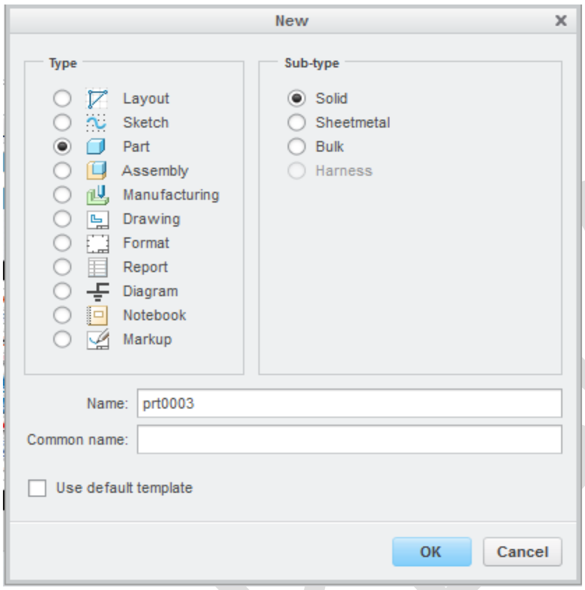
- Solid: This option is selected by default to make solid extrude.
- Surface: This can be used to extrude the sketch as surface.
- Extrude depth tool: is used to control extrude by specifying some constraints.
- Depth value: used to specify the dimension of depth. Some extrude types do not need this.
- Invert tool: used to change the direction of extrude opposite to the reference direction.
- Remove material: this tool is used to remove the material while extruding.
- Thicken tool: is used to extrude as thick sheet. Thickness value can be adjust by entering the value in box (just right to the thicken tool). The invert tool next to the thicken tool is used to specify the direction of thickness by three ways.

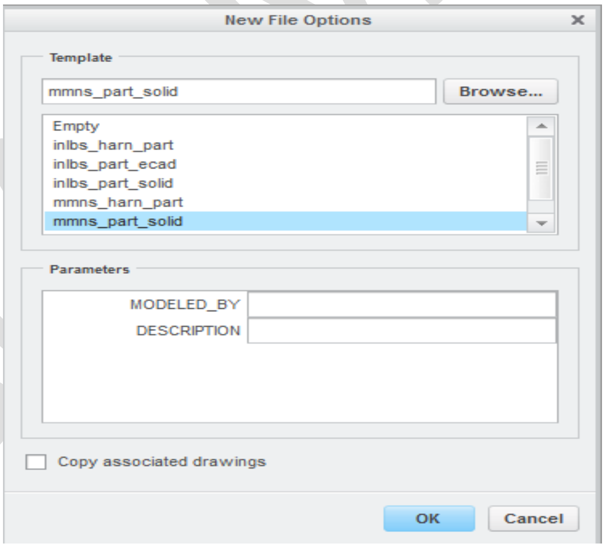
PROCEDURE:

STEP 1. Open PTC Creo Parametric 3.0 by either using the desktop icon or using the program menu

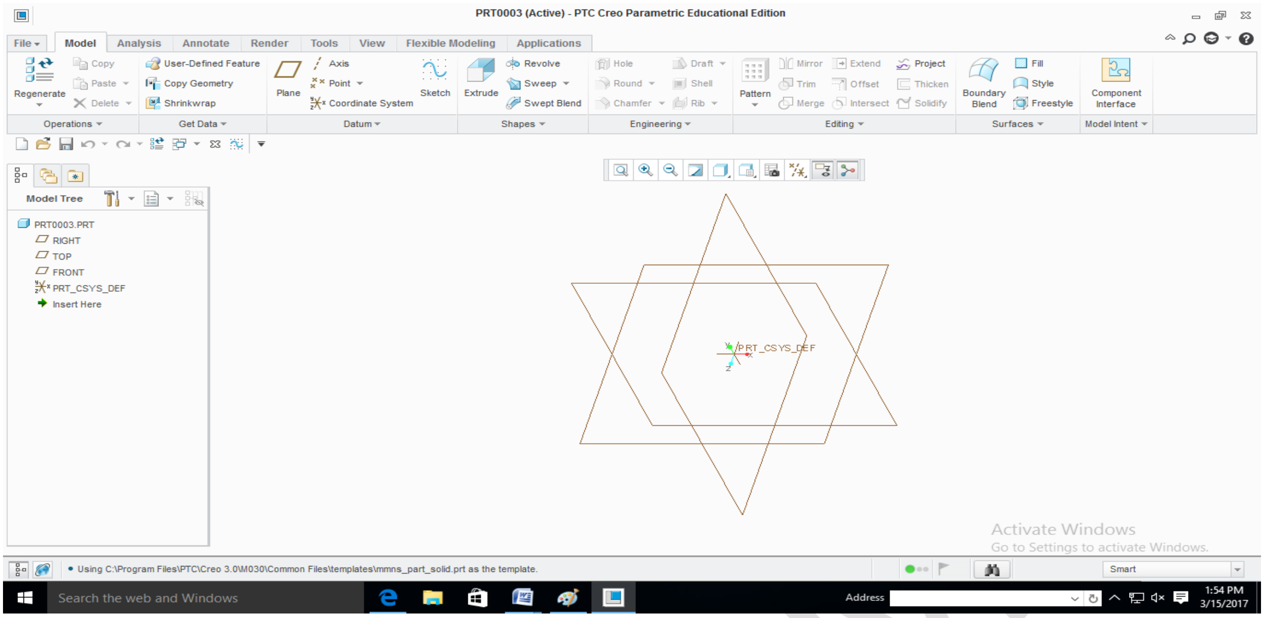


STEP 2. Next select New From Menu bar, make sure the type is set to part and sub type as a Solid. Change the name to whatever you want to name your part and Click OK and also select Template as mmns_part_solid and Click on ok



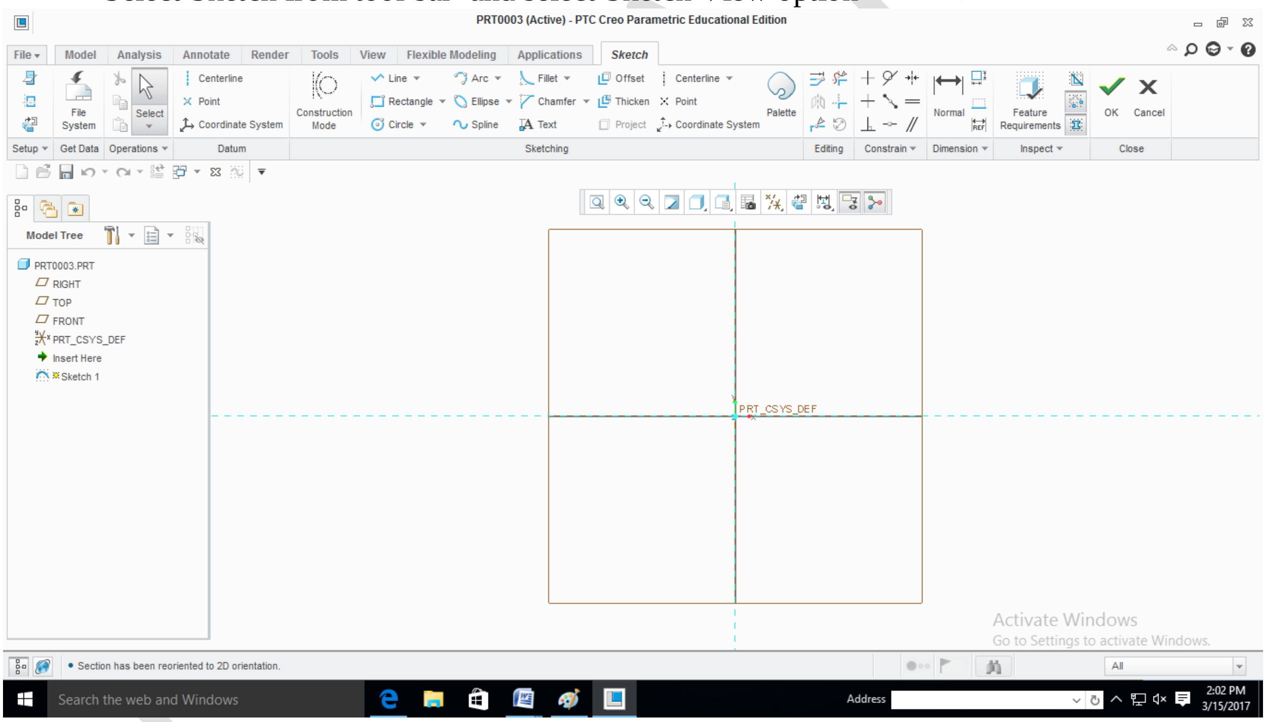


STEP 3.Your screen should now look like this.



STEP 4. Next select the required Sketch plane (Top)

Select Sketch from tool bar and select Sketch View option



Step 5: Sketch **a 2Dprofile** of the model as shown in **Figure.1 as per the dimensions and** Click on ok

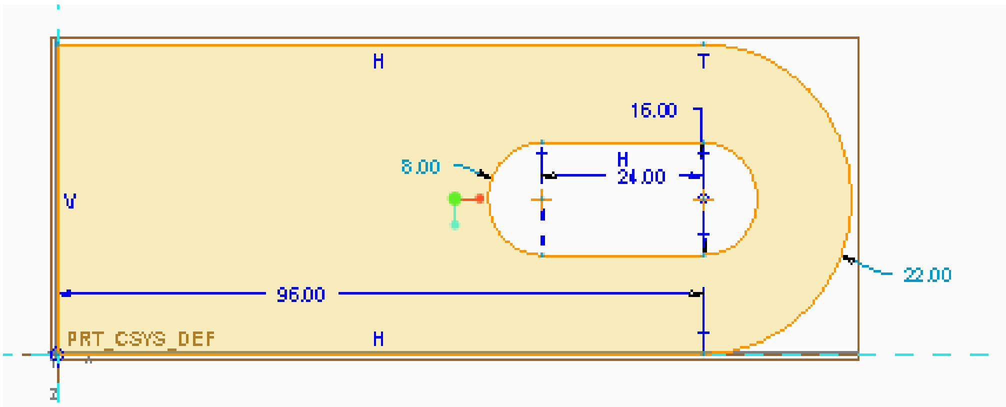
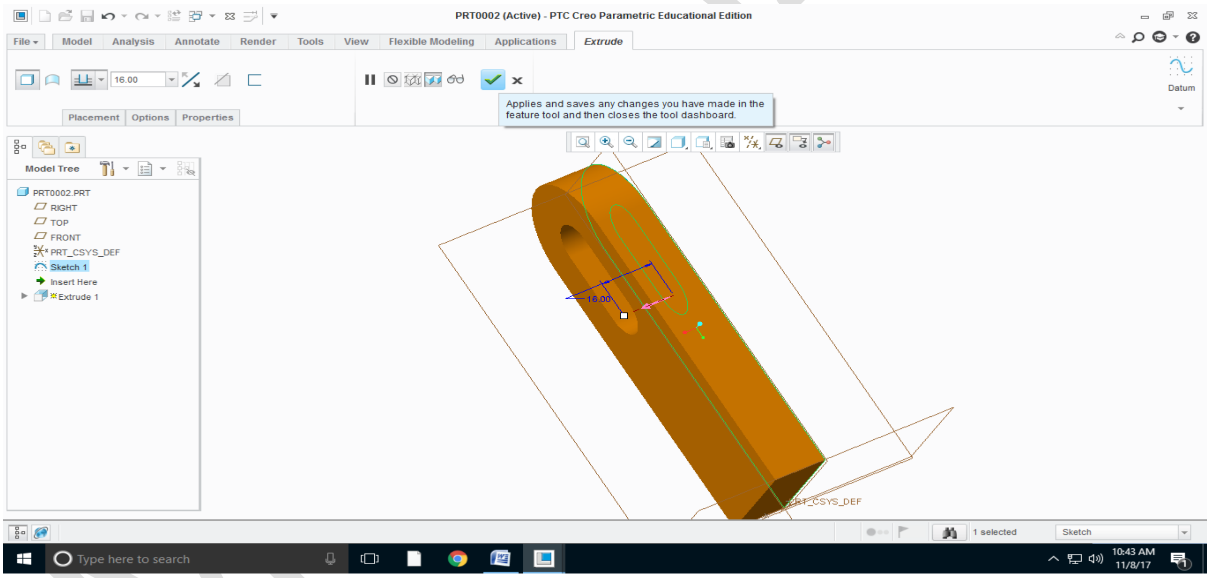


Figure.1

Step6: Extrude the sketch perpendicular to sketch plane. And Specify width and direction of an Extrusion and Click on ok.



Step7: Select Extrude ⟶ Required surface ⟶ Sketch View option ⟶ Specify references ⟶ Draw the required sketch as shown in **Figure.2** Click on ok

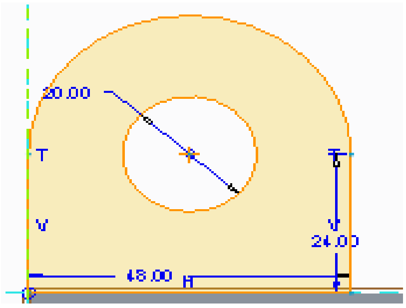
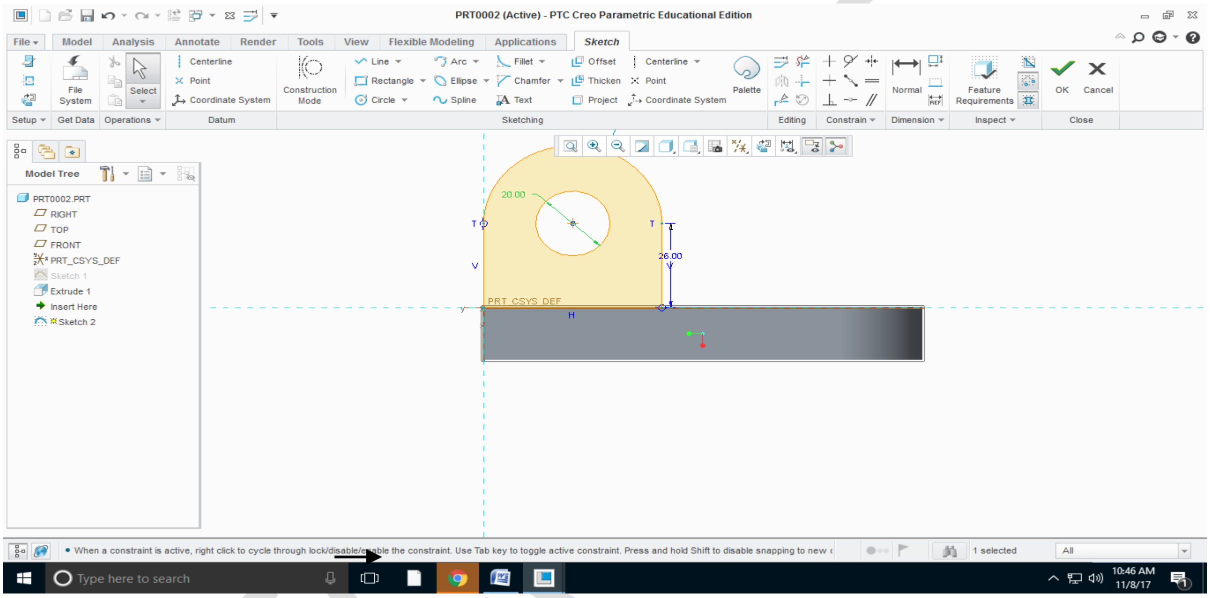
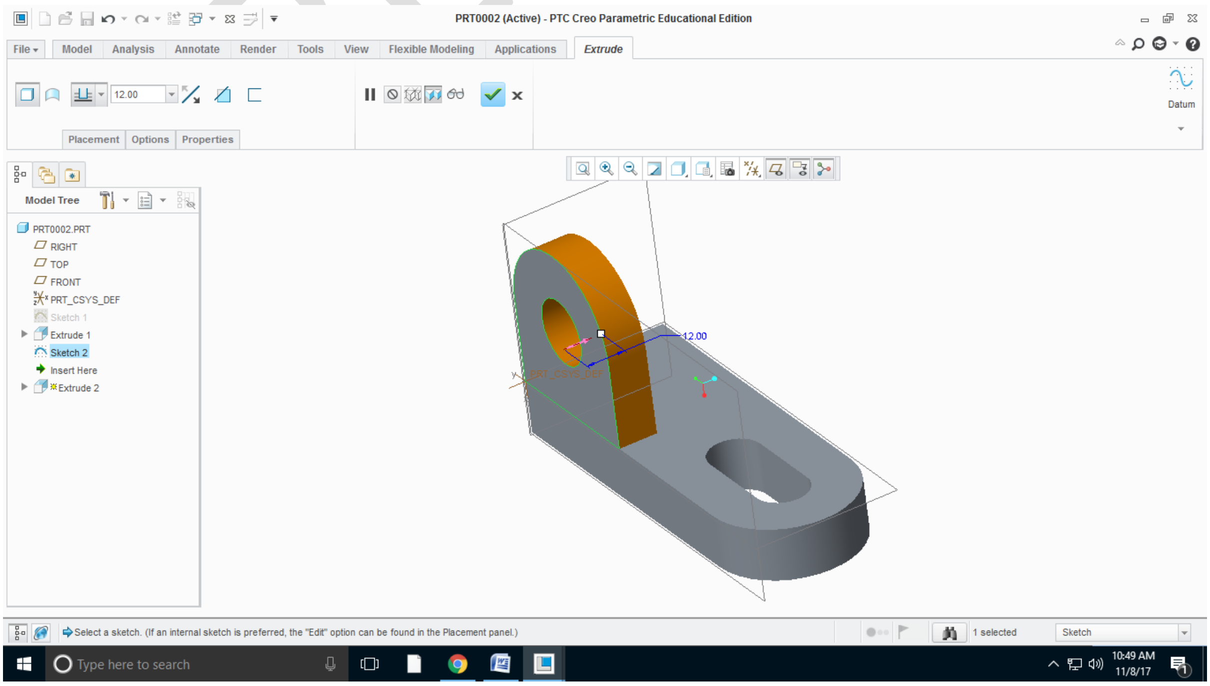


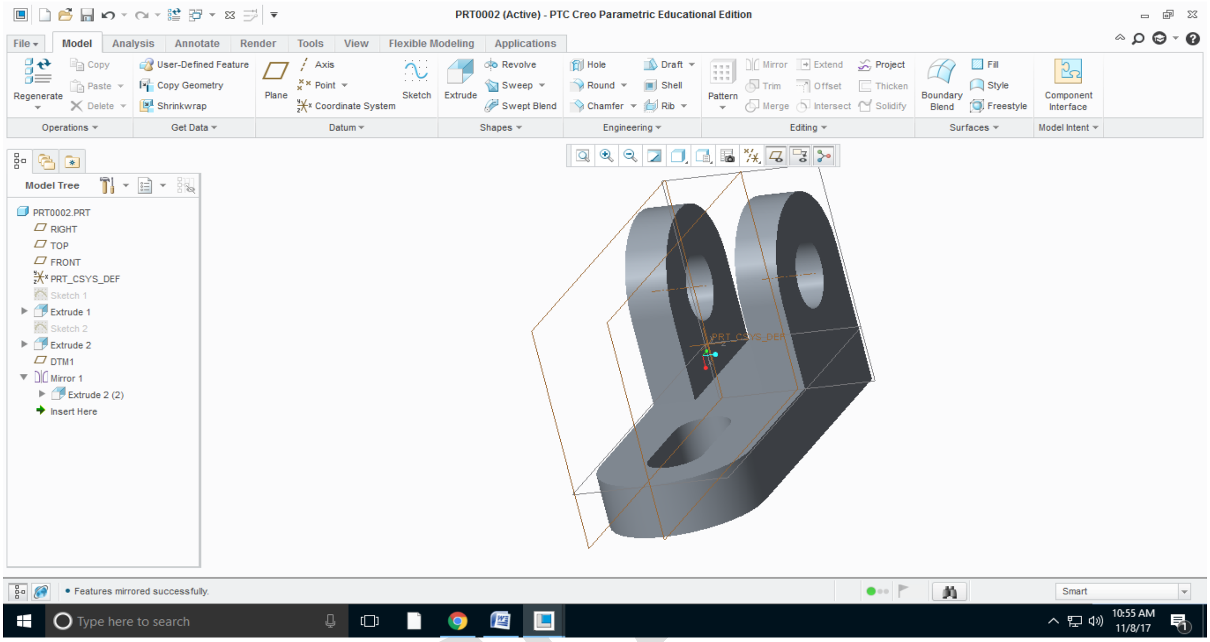
Figure.2



Step8: Extrude the sketch and specify the width and direction of extrusion Click on ok



Step9: Select the datum plane at the middle of the sketch and select the part which is to be duplicated then select the mirror and then select the plane to complete the object Click on ok



Step10: Round the edges of a part to a specified radius by using Round Command Click on ok

RESULT:

Thus the given model is done by using PTC Creo Parametric 3.0

EXERCISE:2 **3D Part Modeling – 2**

AIM:

To model the given object by using the PTC Creo Parametric 3.0 as per the given dimensions.

Software required:PTC Creo Parametric 3.0

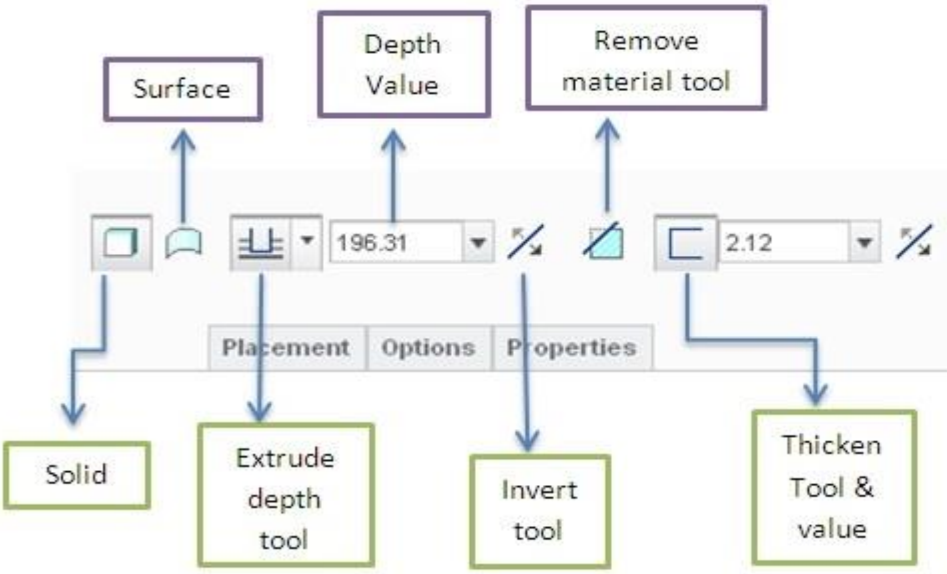
Commands used:

2D Commands: line, circle ,arc, trim

3D Commands: Extrude, Hole, Rib

Description of Extrusion Feature:

An extrude feature is based on a two-dimensional sketch. It linearly extrudes a sketch perpendicular to the sketching plane to create or remove material. You can either select the sketch first or then start the Extrude tool, or you can start the Extrude tool and then select the sketch.



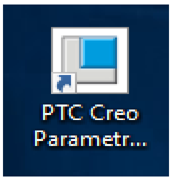
- **Solid**: This option is selected by default to make solid extrude.
- **Surface:** This can be used to extrude the sketch as surface.
- **Extrude depth tool:** is used to control extrude by specifying some constraints.
- **Depth value:** used to specify the dimension of depth. Some extrude types do not need this.
- **Invert tool:** used to change the direction of extrude opposite to the reference direction.
- **Remove material:** this tool is used to remove the material while extruding.
- **Thicken tool:** is used to extrude as thick sheet. Thickness value can be adjust by entering the value in box (just right to the thicken tool). The invert tool next to the thicken tool is used to specify the direction of thickness by three ways.

Rib:

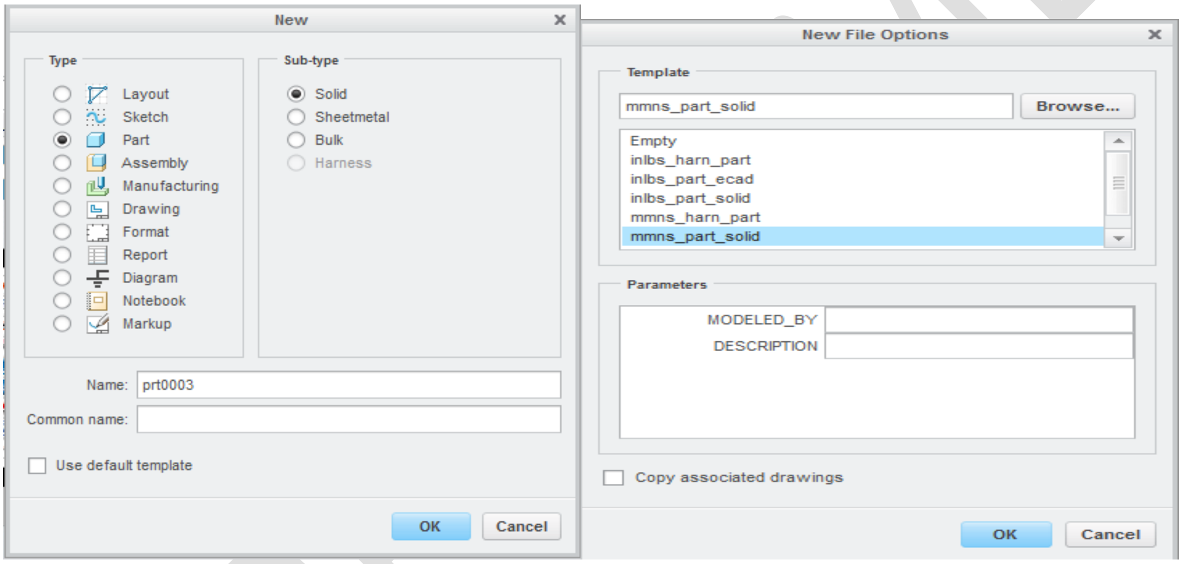
A rib is a special type of protrusion designed to create a thin wall or web to support two surface.

PROCEDURE:

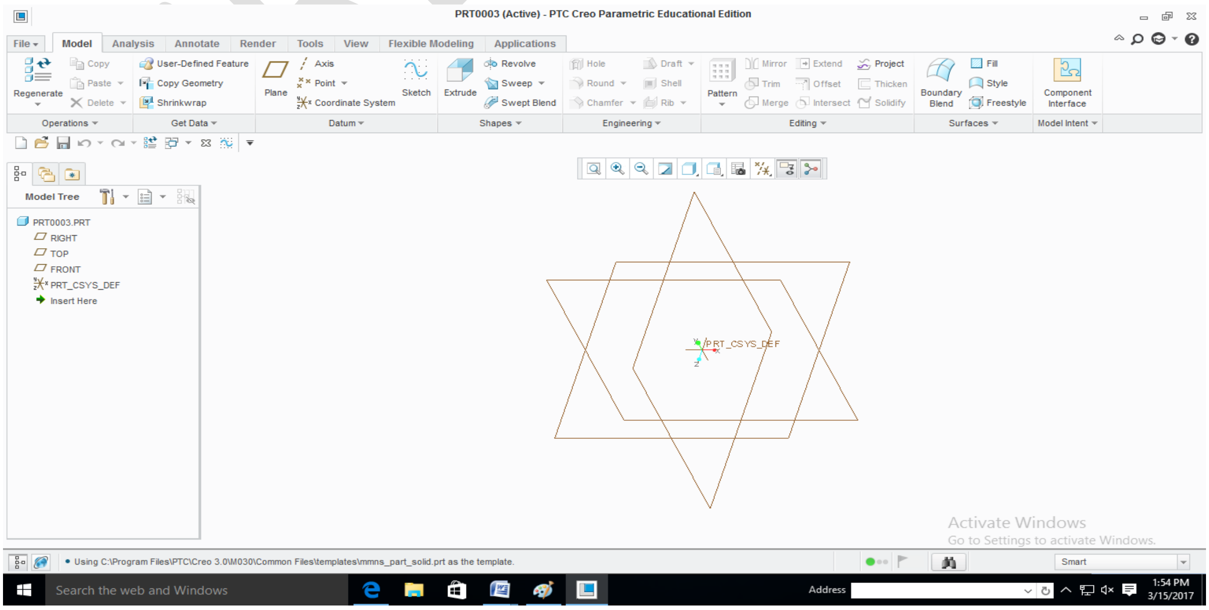
STEP 1. . Open PTC Creo Parametric 3.0 by either using the desktop icon or using the program menu



STEP 2. Next select New From Menu bar, make sure the type is set to part and sub type as a Solid. Change the name to whatever you want to name your part and Click OK And also select Template as mmns_part_solid and Click on ok

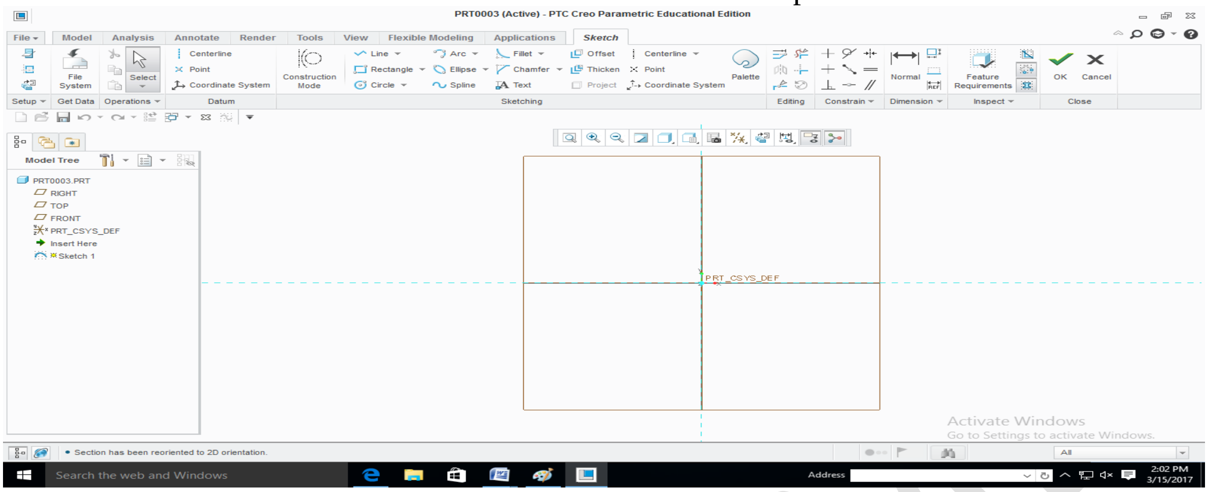


STEP 3.Your screen should now look like this.



STEP 4. Next select the required Sketch plane (Front)

Select Sketch from tool bar and select Sketch View option



Step 5: Select line command and Draw the required Sketch and Edit the Dimensions as shown in Figure.1 **and** Click on ok

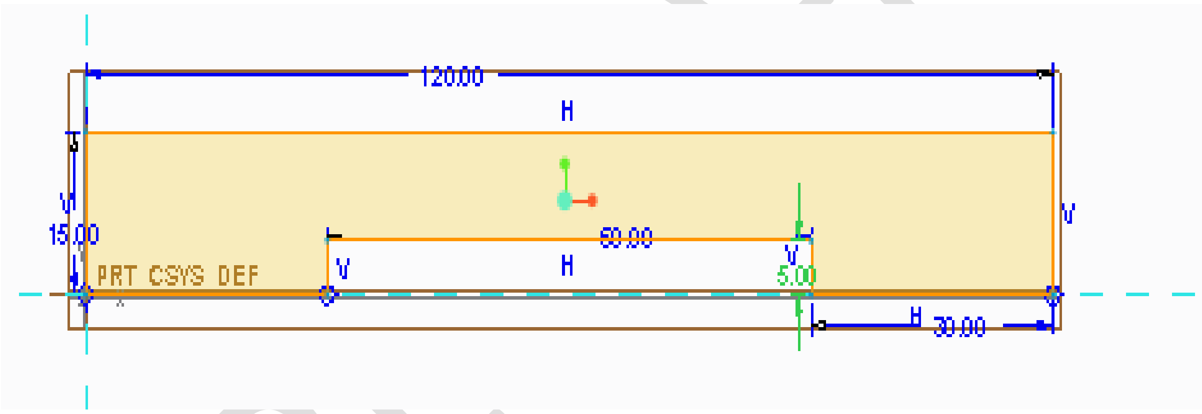
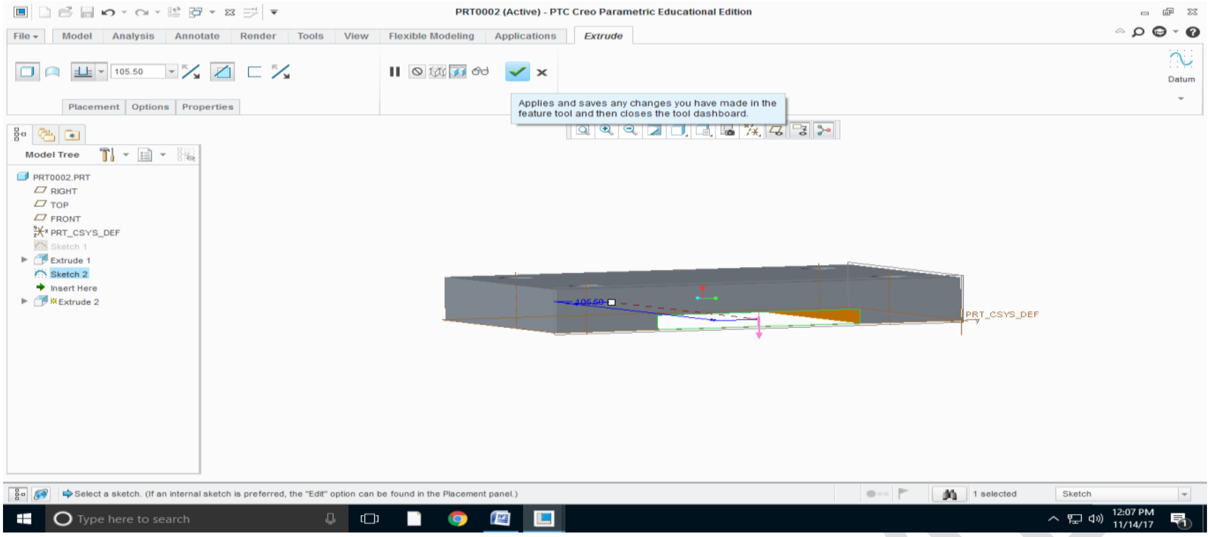


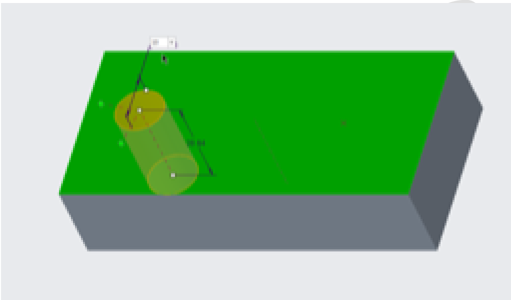
Figure.1

Step6: Extrude the sketch perpendicular to sketch plane. and Specify width and direction of an Extrusion and Click on ok



Step7: Using the Hole Tool

- Select Hole from the Engineering group.
- Specify the hole type as Simple.
- Select the surface to place the hole on and edit the hole diameter.
- Specify the hole depth.
- Add offset references using the dashboard or drag handles in the graphics window.
- Click Complete Feature from the dashboard.



Step8: Create the datum plane at the distance 40 mm from one end of the surface along the length of the sketch and with this as a reference plane draw the required shape of the object and Click on ok

Step9: Select Extrude → select plane (Created Datum plane) → Sketch View option Specify references → Draw the required sketch as shown in **Figure.2** → **Click on ok**

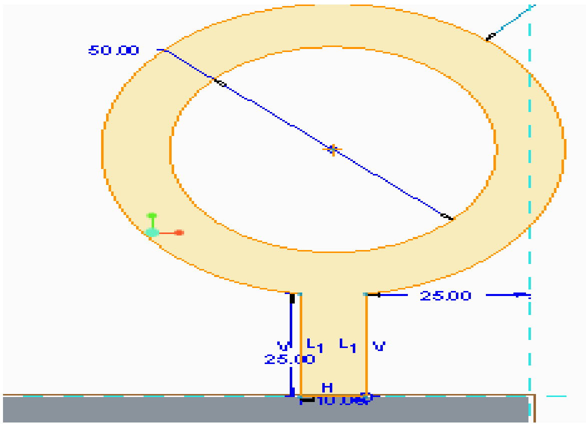
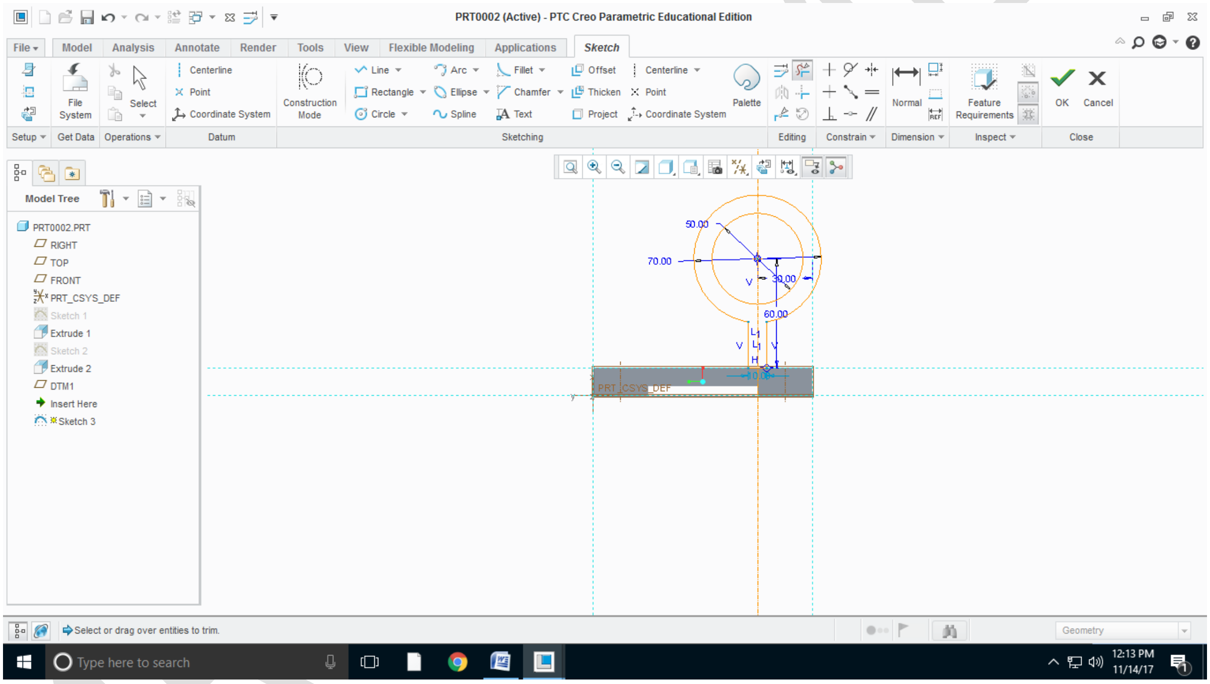
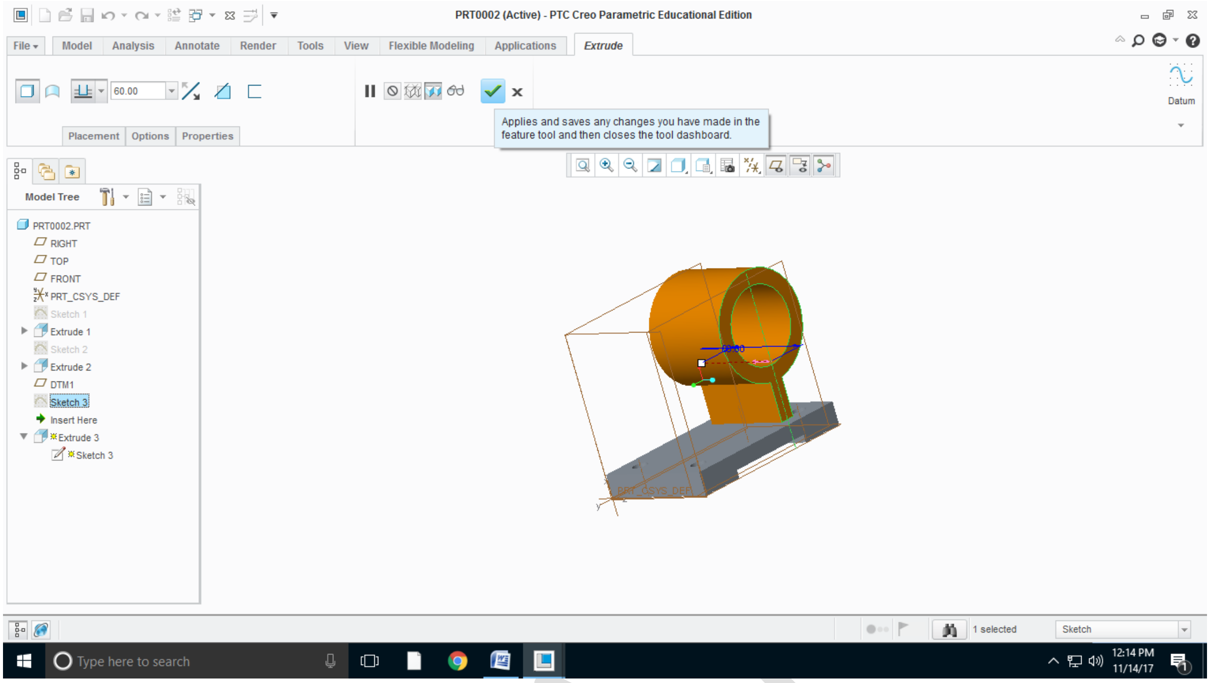


Figure.2



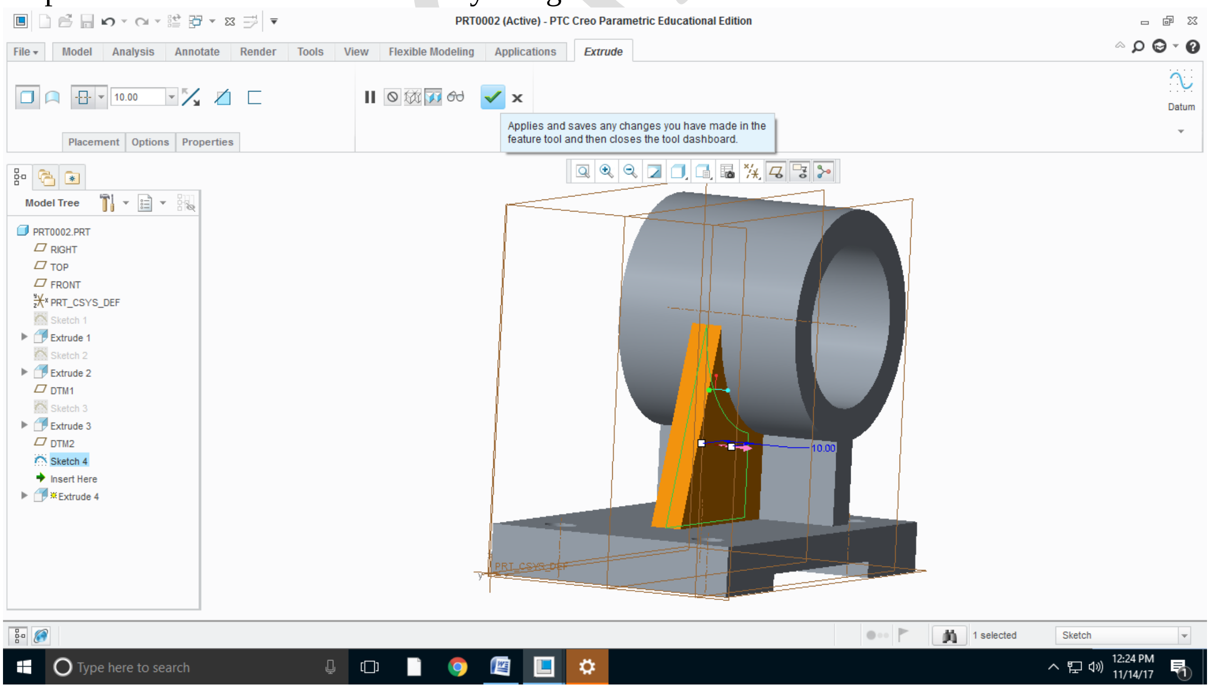
Step9: Extrude the sketch by using Symmetry option and specify the width and direction of extrusion and Click on ok



Step 10: Create an datum plane at specified distance and select Rib Command from Model

Step 11: Specify width of Rib to be Extruded and Click on ok

Step 12: Thus the model is created by using Rib Command



RESULT:

Thus the given model is created by using PTC Creo Parametric 3.0

EXERCISE:3 **3D Part Modeling – 3**

AIM:

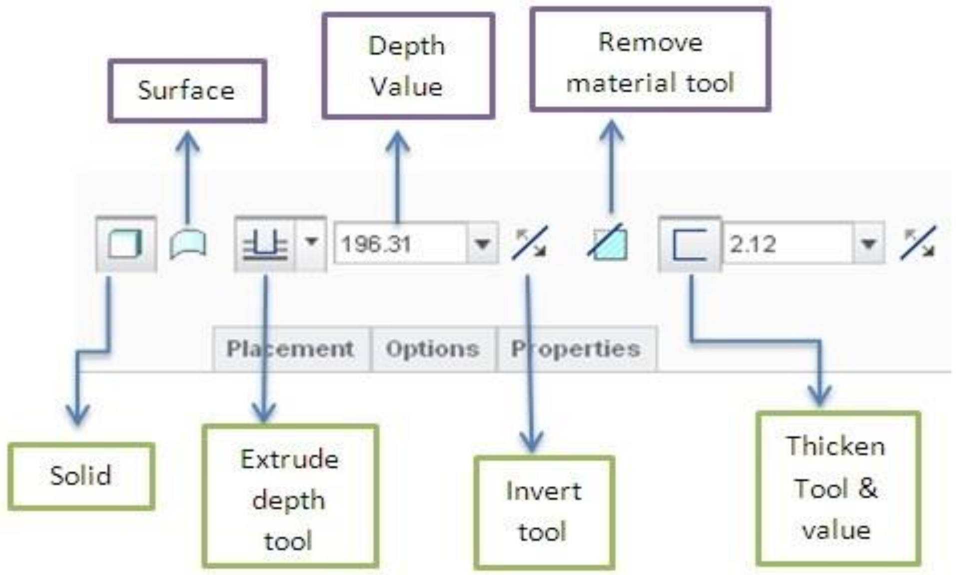
To model the given object by using the PTC Creo Parametric 3.0 as per the given dimensions.

Software required:PTC Creo Parametric 3.0

Commands used:
2D Commands: line, circle ,arc, trim
3D Commands: Extrude, Hole, Rib

Description of Extrusion Feature:

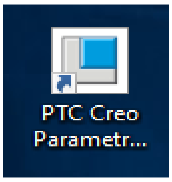
An extrude feature is based on a two-dimensional sketch. It linearly extrudes a sketch perpendicular to the sketching plane to create or remove material. You can either select the sketch first or then start the Extrude tool, or you can start the Extrude tool and then select the sketch.



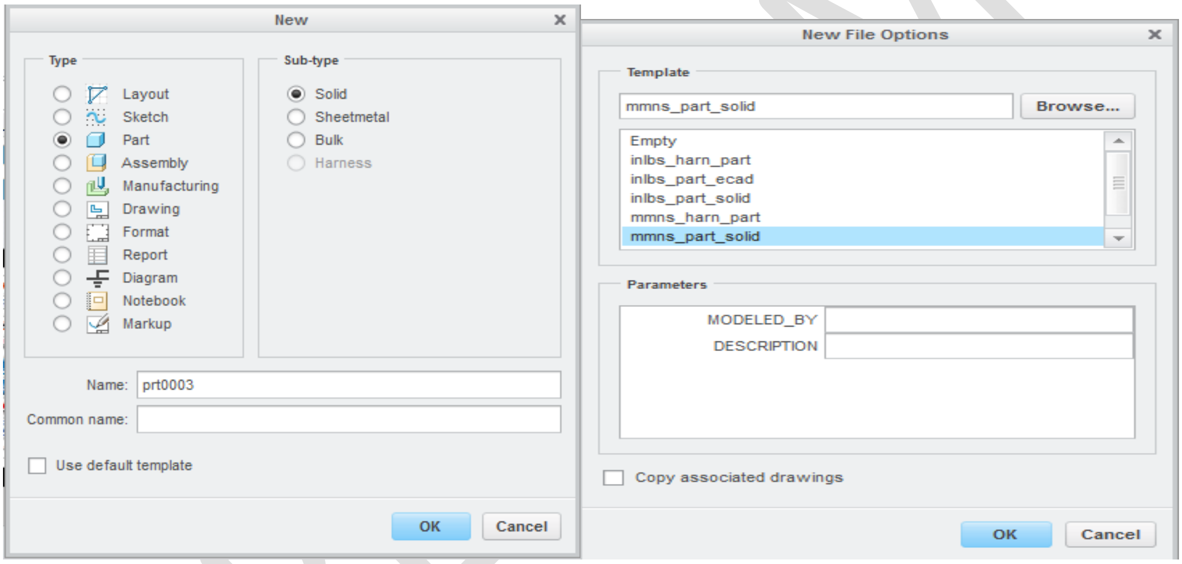
- **Solid**: This option is selected by default to make solid extrude.
- **Surface:** This can be used to extrude the sketch as surface.
- **Extrude depth tool:** is used to control extrude by specifying some constraints.
- **Depth value:** used to specify the dimension of depth. Some extrude types do not need this.
- **Invert tool:** used to change the direction of extrude opposite to the reference direction.
- **Remove material:** this tool is used to remove the material while extruding.
- **Thicken tool:** is used to extrude as thick sheet. Thickness value can be adjust by entering the value in box (just right to the thicken tool). The invert tool next to the thicken tool is used to specify the direction of thickness by three ways.

PROCEDURE:

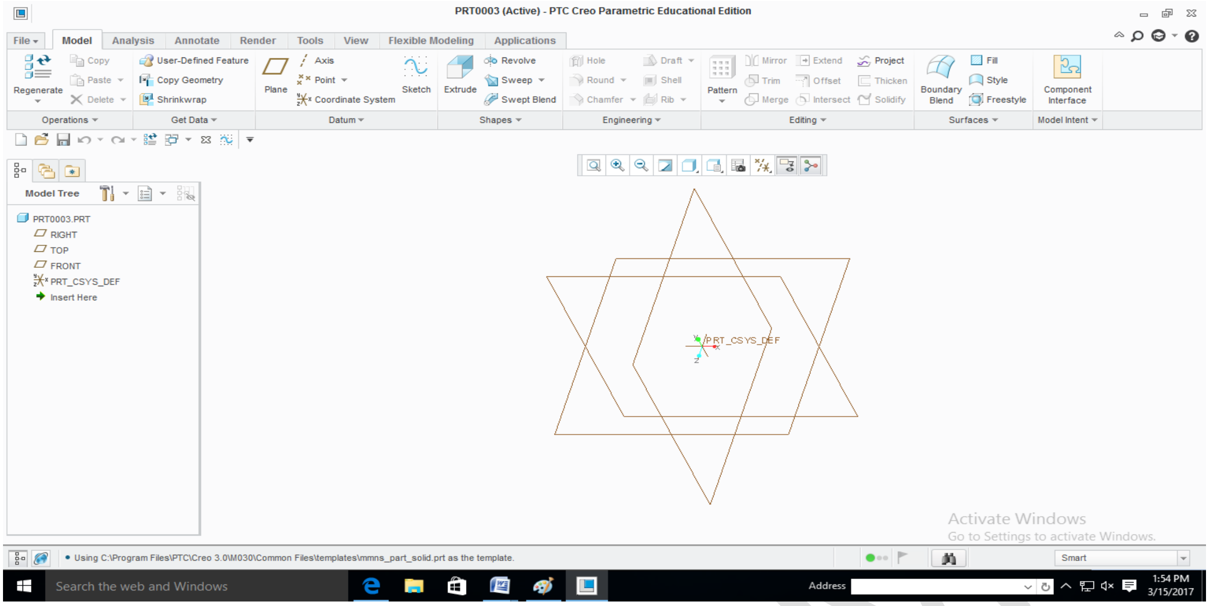
STEP 1. . Open PTC Creo Parametric 3.0 by either using the desktop icon or using the program menu



STEP 2. Next select New From Menu bar, make sure the type is set to part and sub type as a Solid. Change the name to whatever you want to name your part and Click OK And also select Template as mmns_part_solid and Click on ok

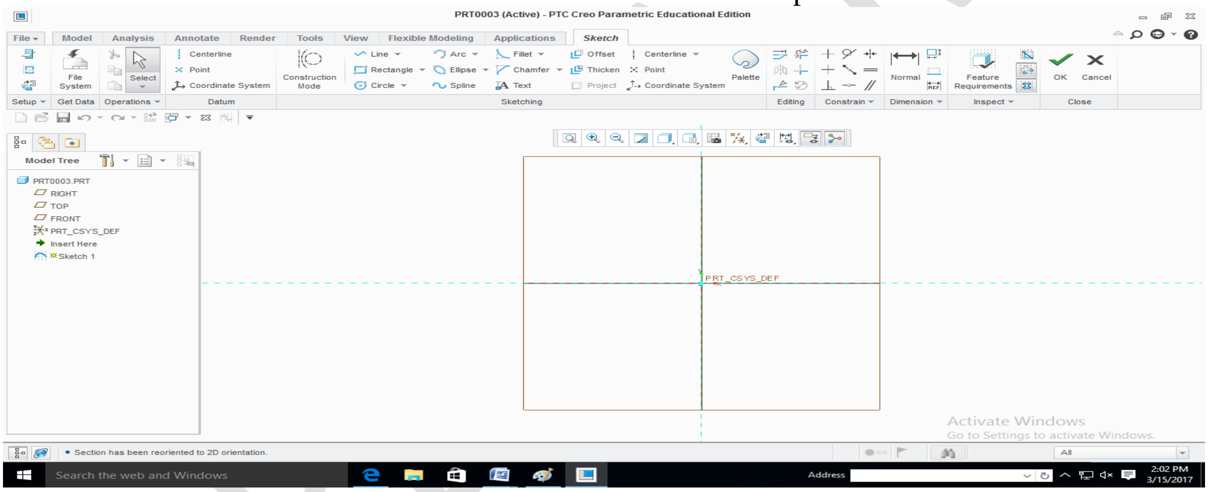


STEP 3.Your screen should now look like this.

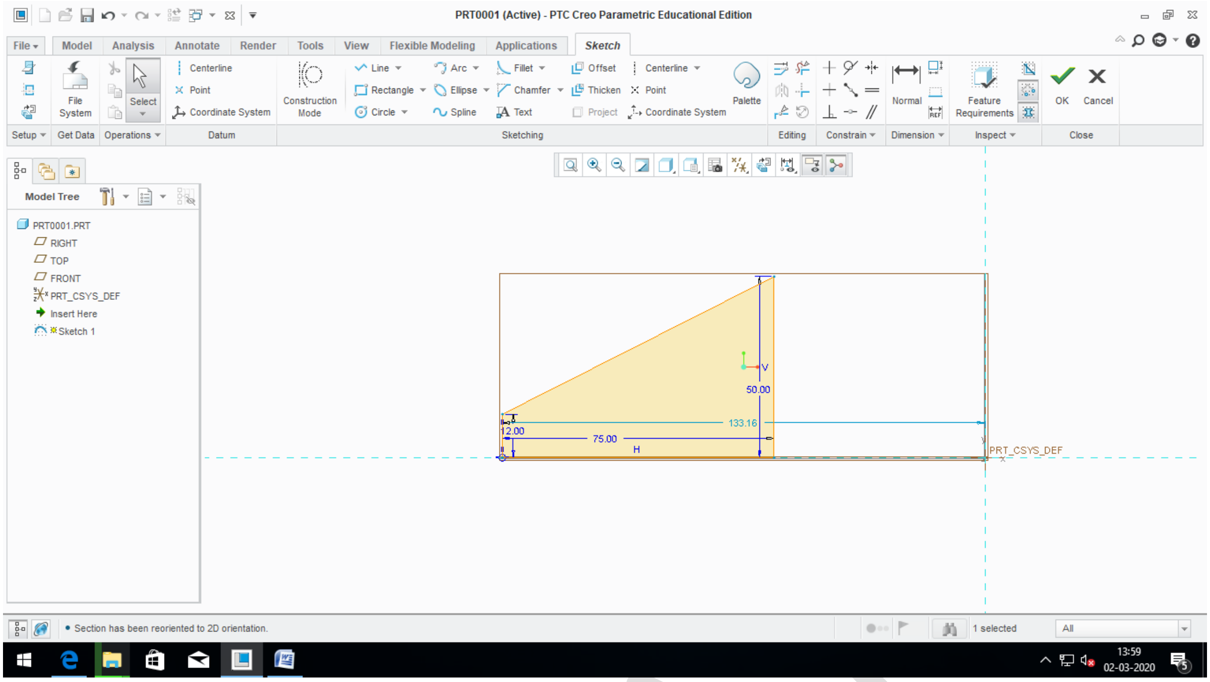


STEP 4. Next select the required Sketch plane (Front)

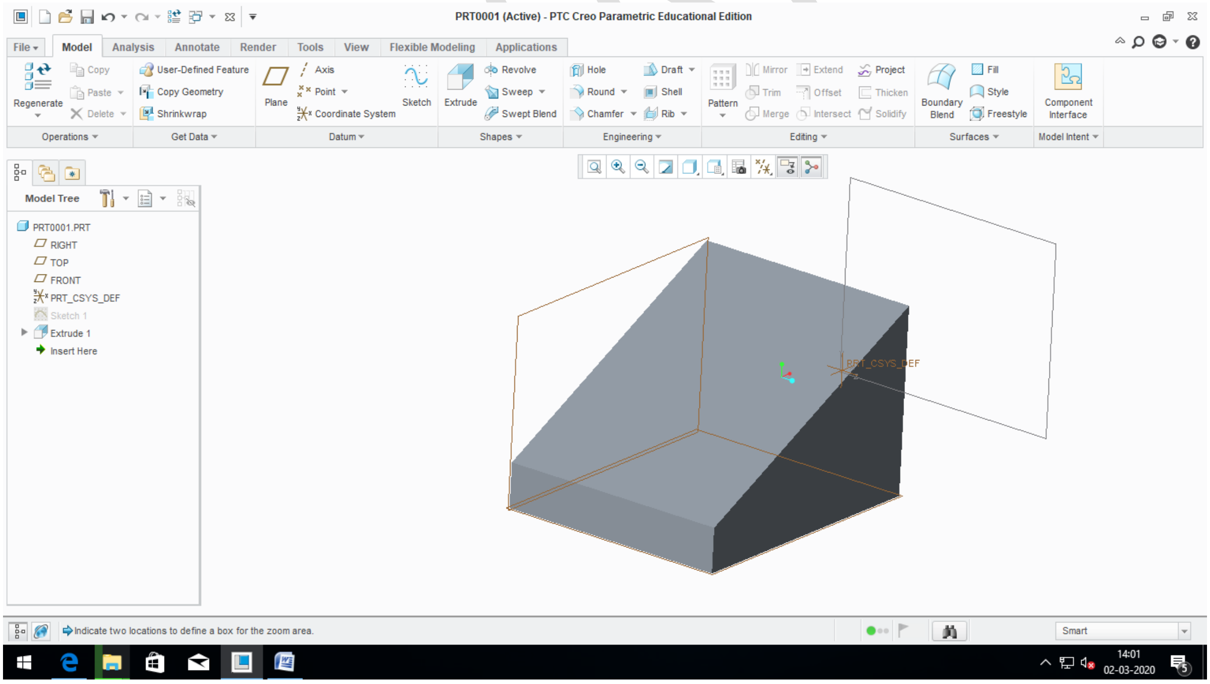
Select Sketch from tool bar and select Sketch View option



Step 5: Select line command and Draw the required Sketch and Edit the Dimensions as shown in Figure.1 **and** Click on ok

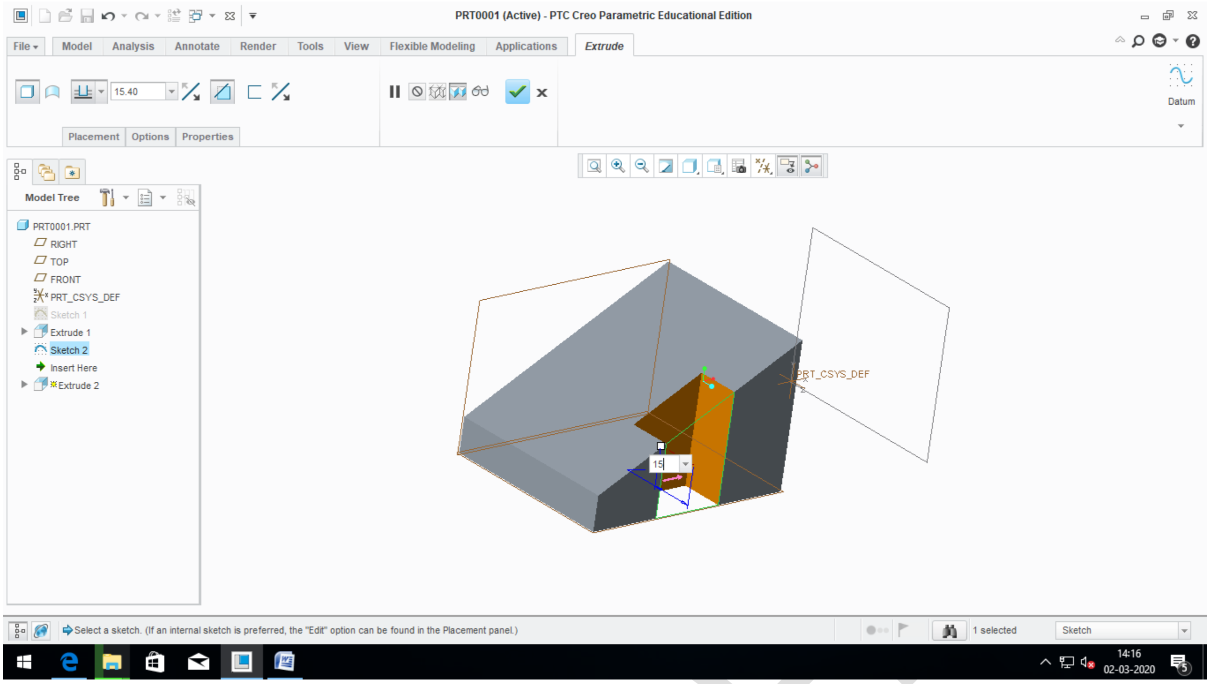


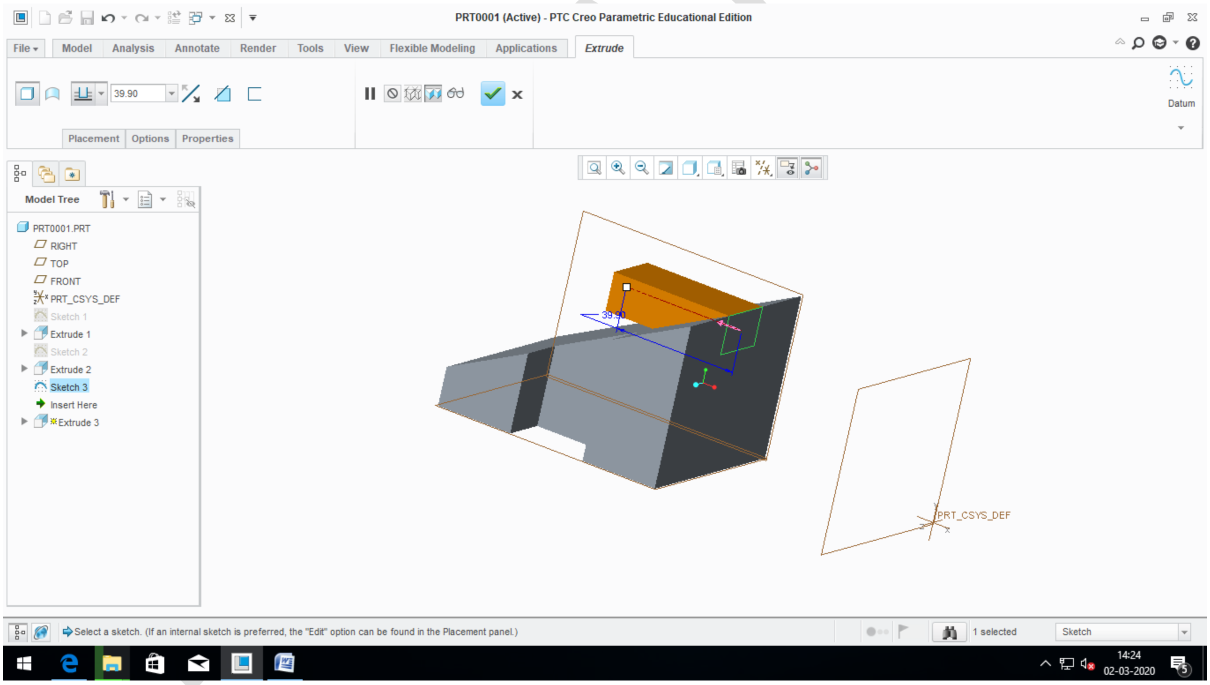
Step6: Extrude the sketch perpendicular to sketch plane. and Specify width and direction of an Extrusion and Click on ok

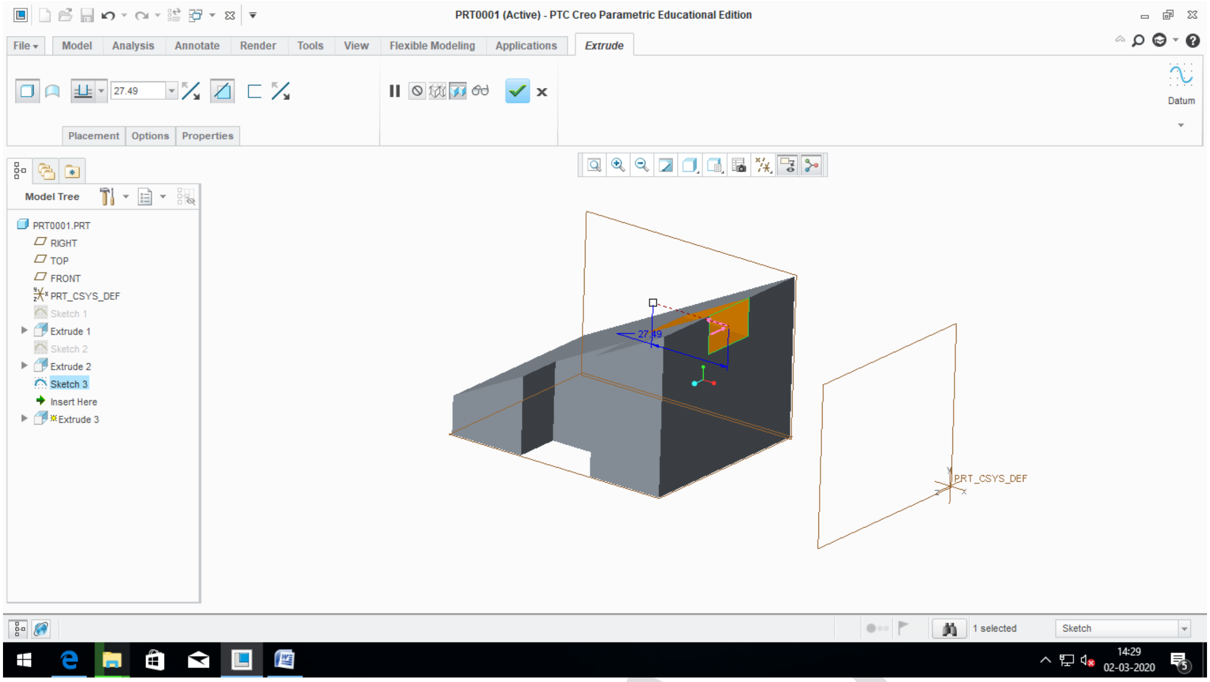


Step7: Using the Material remove Tool

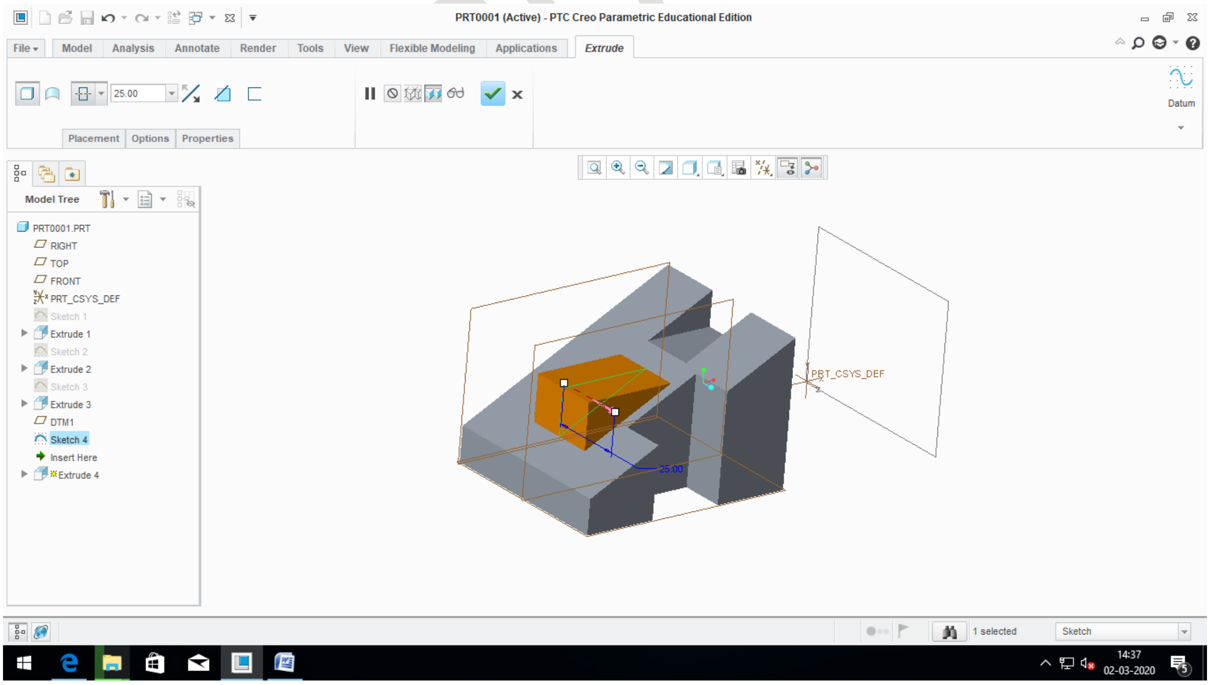
- Select shape from the Engineering group.
- Select the surface to place the shape on and edit the sahpe dimensions.
- Specify the shape depth.
- Add offset references using the dashboard or drag handles in the graphics window.
- Click Complete Feature from the dashboard.


PRT0001 (Active) - PTC Creo Parametric Educational Edition
File
Model
Analysis
Annotate
Render
Tools
View
Flexible Modeling
Applications
Extrude
15.40
Placement
Options
Properties
Datum
Model Tree
PRT0001.PRT
RIGHT
TOP
FRONT
PRT_CSYS_DEF
Sketch 1
Extrude 1
Sketch 2
Insert Here
Extrude 2
Select a sketch. (If an internal sketch is preferred, the "Edit" option can be found in the Placement panel.)
1 selected
Sketch
14:16
02-03-2020


PRT0001 (Active) - PTC Creo Parametric Educational Edition
File
Model
Analysis
Annotate
Render
Tools
View
Flexible Modeling
Applications
Extrude
39.90
Placement
Options
Properties
Datum
Model Tree
PRT0001.PRT
RIGHT
TOP
FRONT
PRT_CSYS_DEF
Sketch 1
Extrude 1
Sketch 2
Extrude 2
Sketch 3
Insert Here
Extrude 3
Select a sketch. (If an internal sketch is preferred, the "Edit" option can be found in the Placement panel.)
1 selected
Sketch
14:24
02-03-2020



Step8: Extrude the sketch by using Symmetry option and specify the width and direction of extrusion and Click on ok



RESULT:

Thus the given model is created by using PTC Creo Parametric 3.0

EXERCISE:4 **3D Part Modeling – 4**

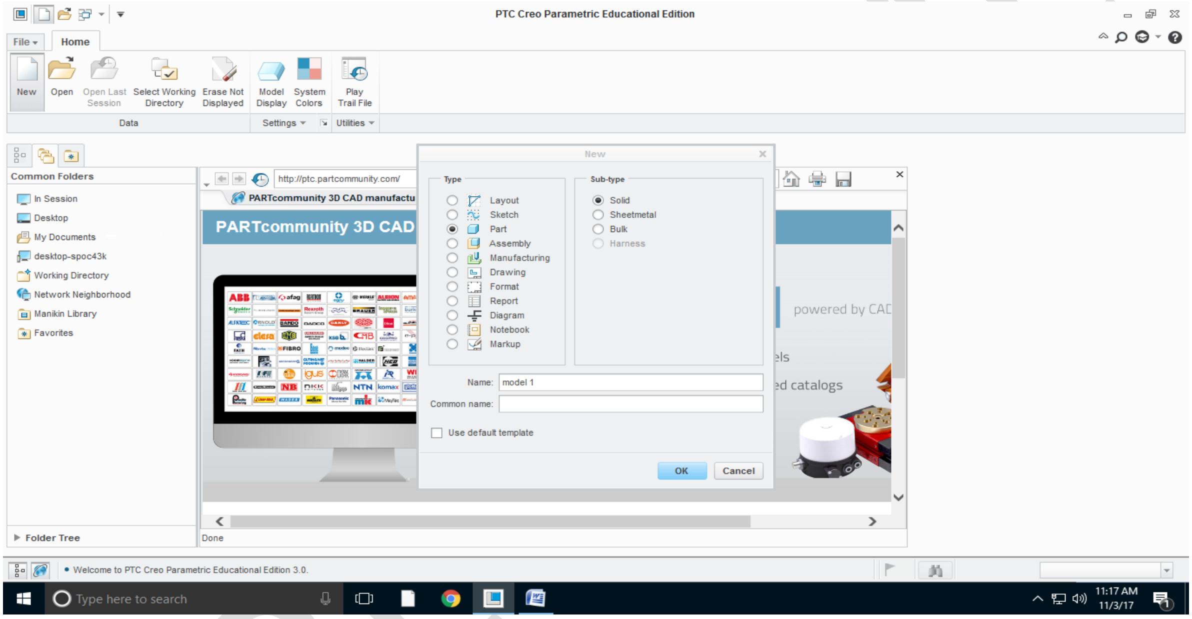
AIM:

To model the given object using the extrude feature as per the given dimensions.

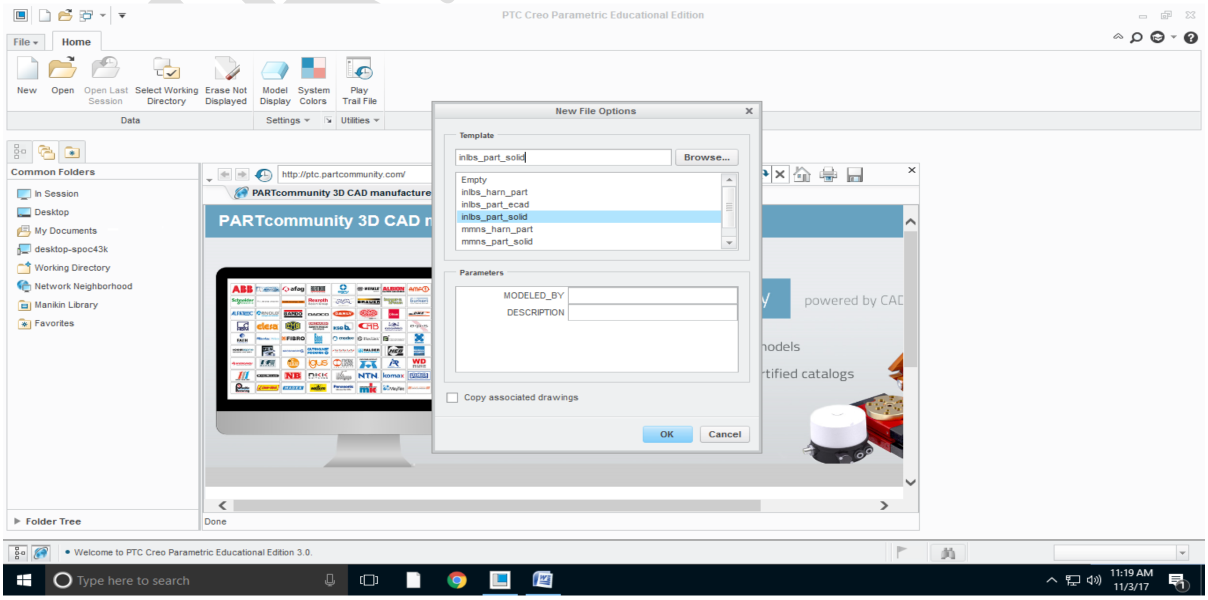
PROCEDURE:

Step1:

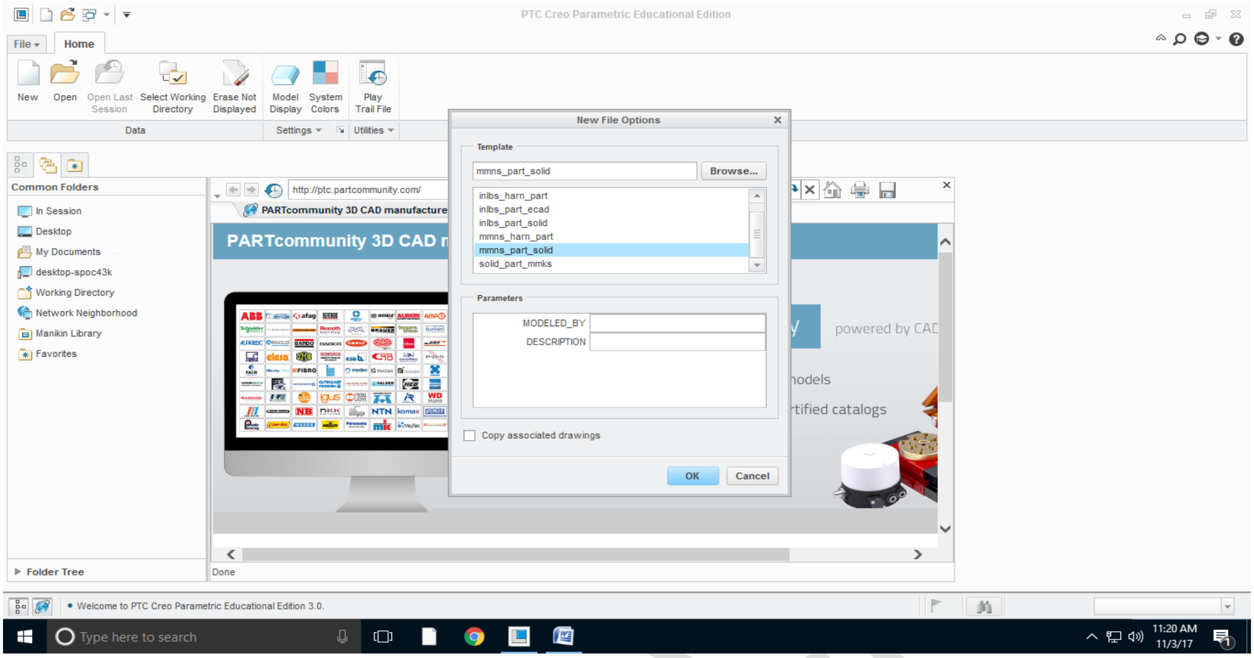
Double click on creo parametric and then open new file



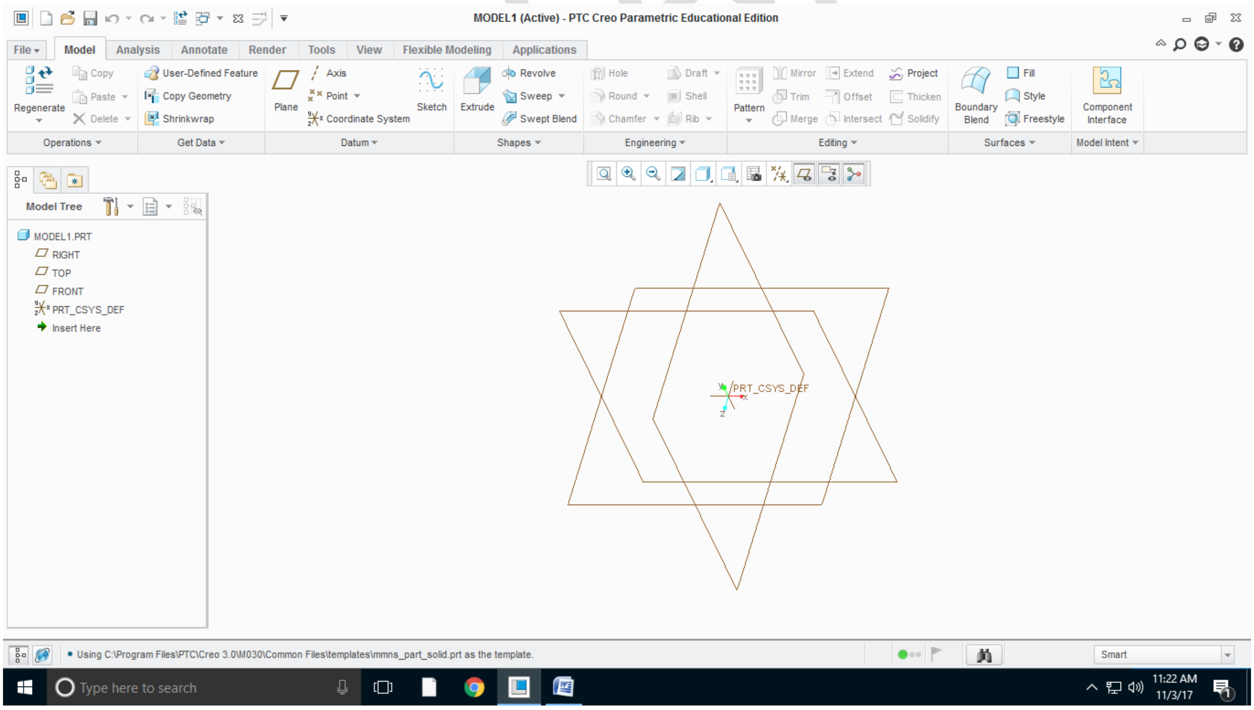
Step 2: select ok



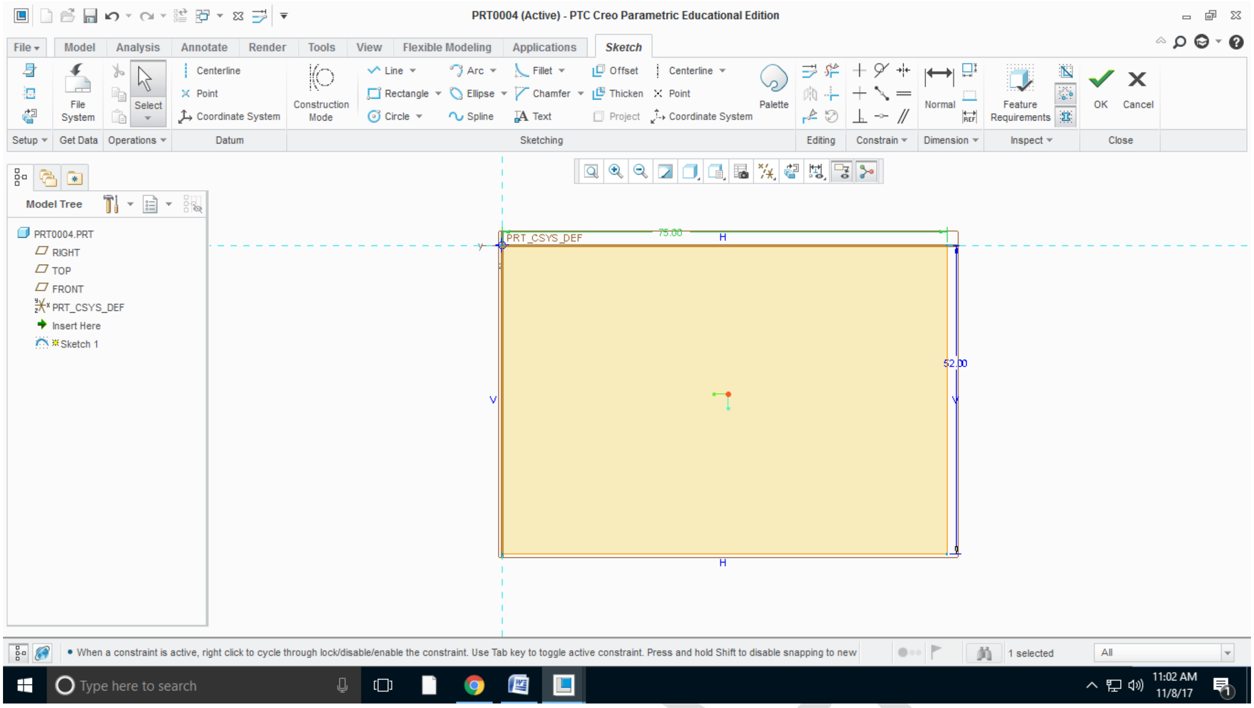
Step3:select mmns_part_solid and then select OK



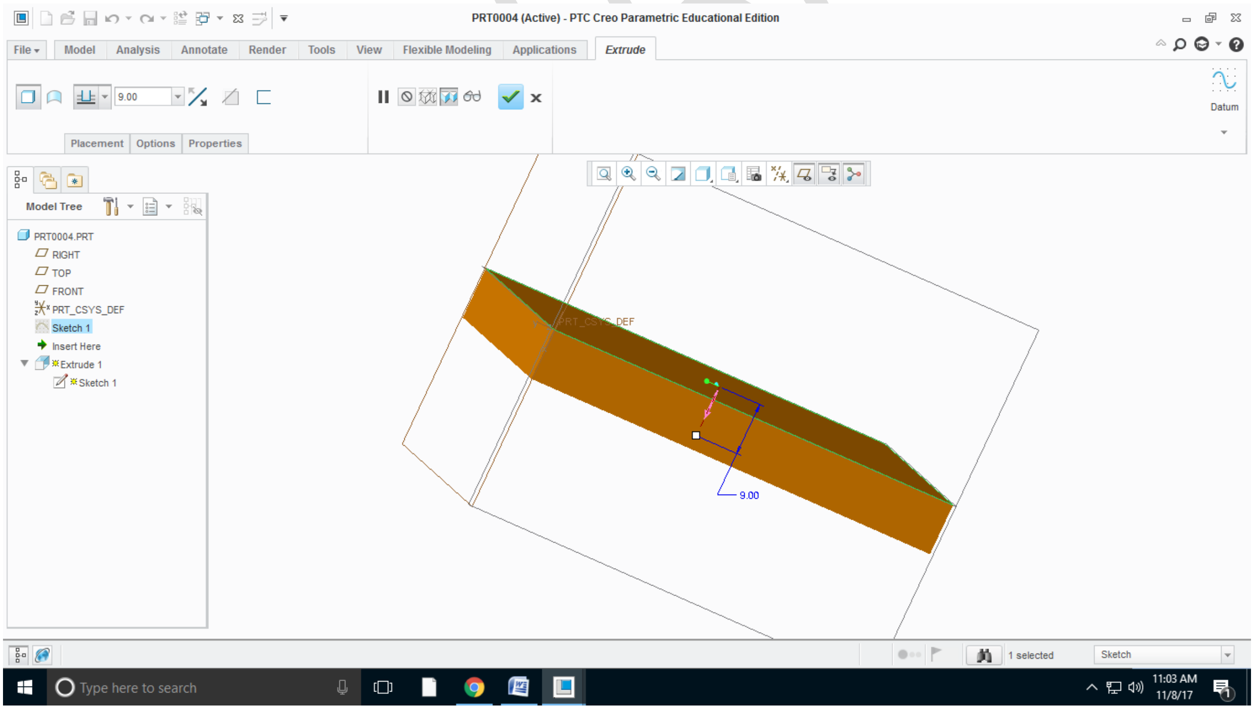
Step4:select a sketch plane



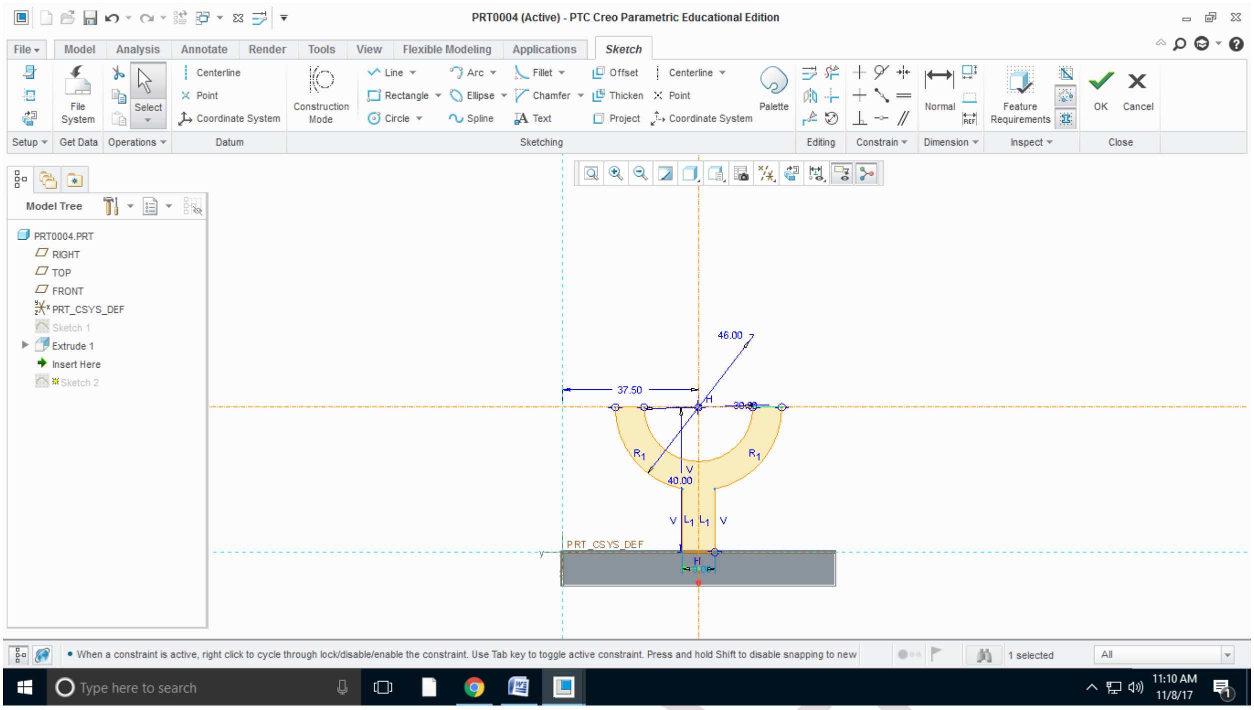
Step5:sketch a 2D profile of the model



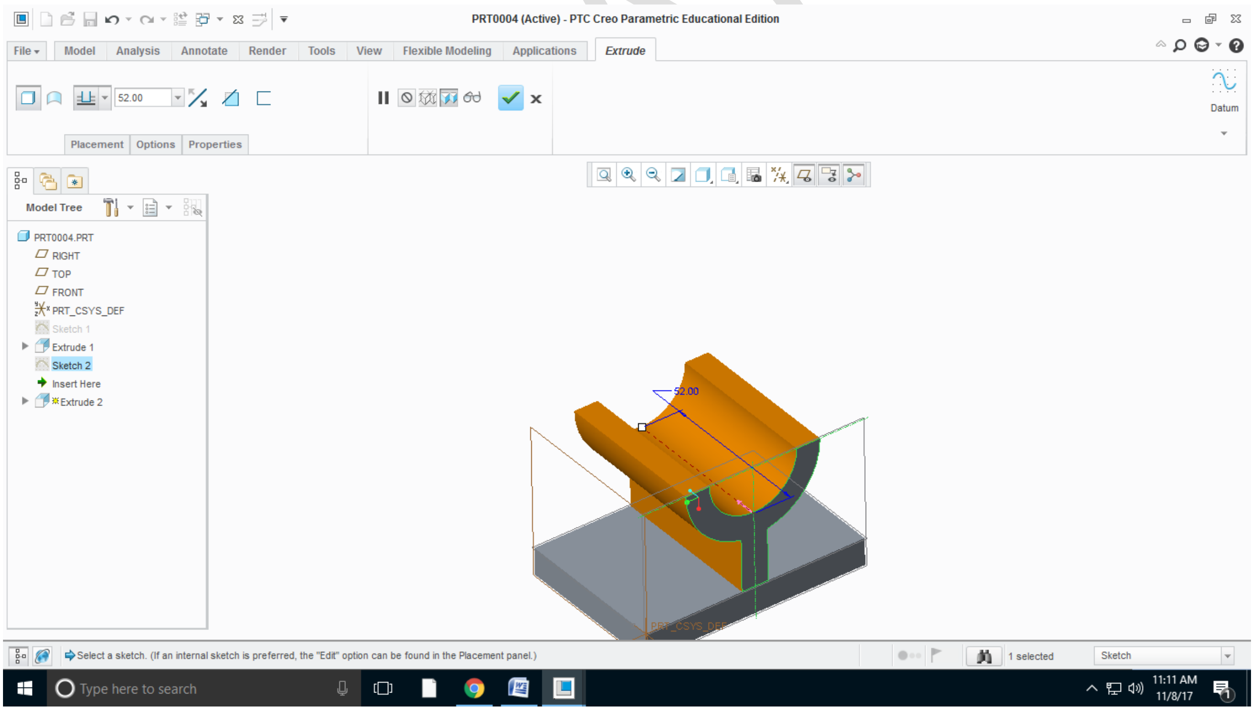
Step6: extrude the sketch perpendicular to the sketch plane and specify width and direction of extrusion



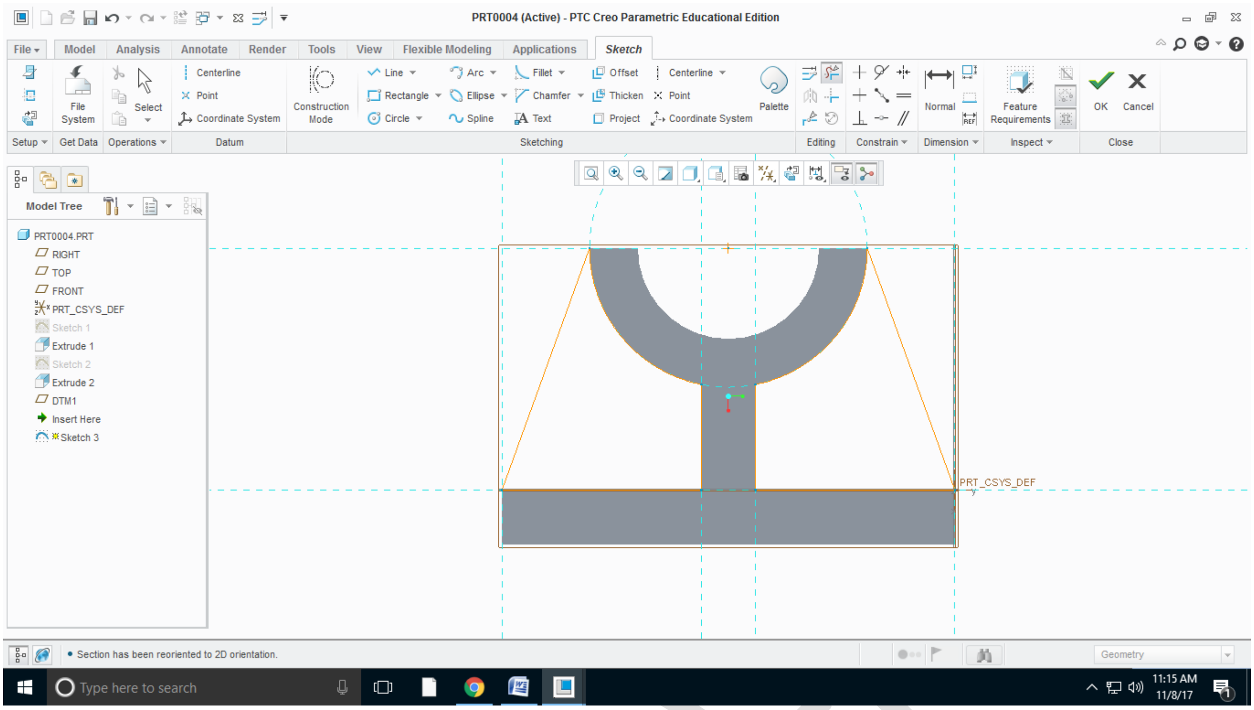
Step7: select the required surface to add the material of the part and also draw the required sketch.

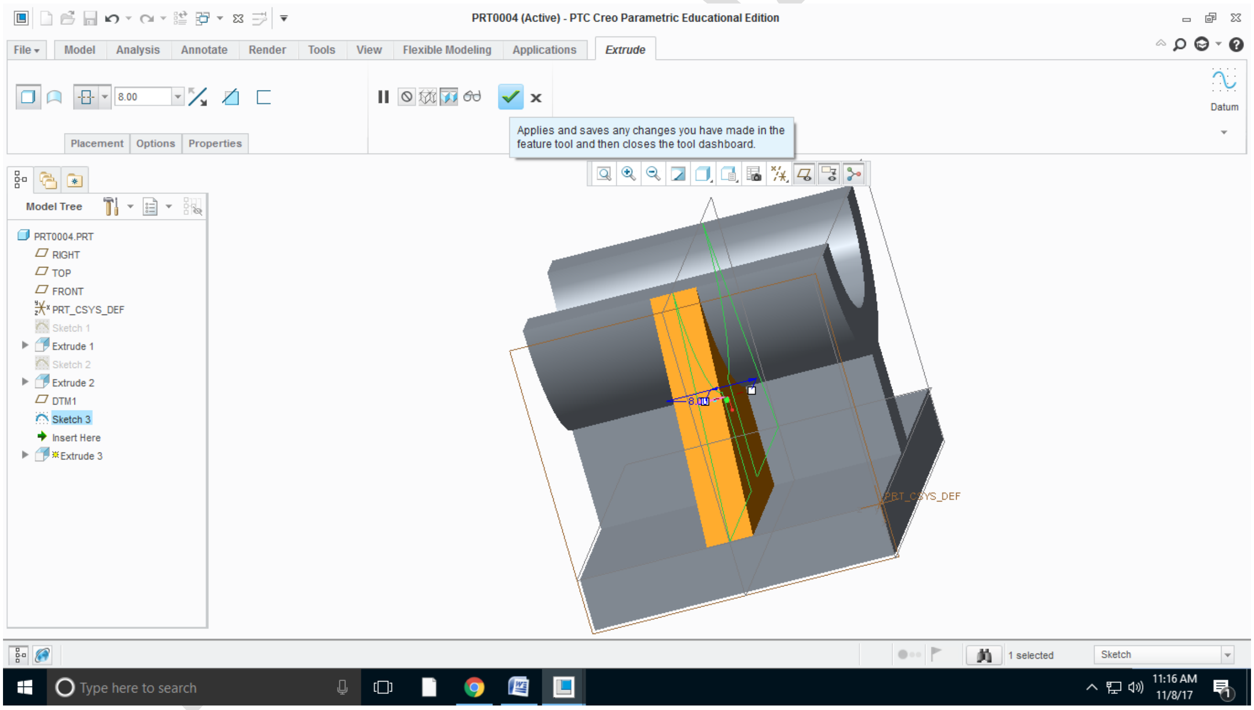


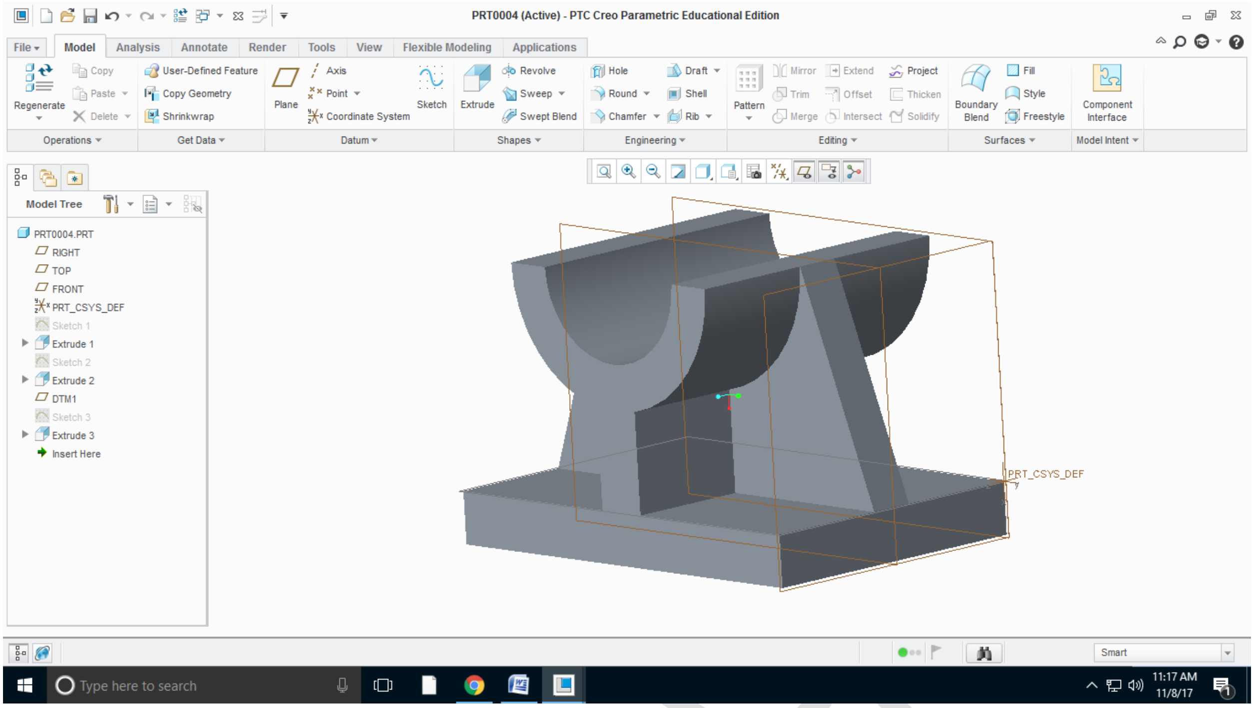
Step8: extrude the sketch and specify the width and direction of extrusion



Step9:select the datum plane at the middle of the sketch and with reference to that plane draw the required shape of the object and then extrude it with require dimensions.


PRT0004 (Active) - PTC Creo Parametric Educational Edition
File
Model
Analysis
Annotate
Render
Tools
View
Flexible Modeling
Applications
Sketch
Centerline
Point
Coordinate System
Construction Mode
Line
Rectangle
Circle
Arc
Ellipse
Spline
Fillet
Chamfer
Text
Offset
Thicken
Project
Palette
Normal
Feature Requirements
OK
Cancel
Setup
Get Data
Operations
Datum
Sketching
Editing
Constrain
Dimension
Inspect
Close
Model Tree
PRT0004.PRT
RIGHT
TOP
FRONT
PRT_CSYS_DEF
Sketch 1
Extrude 1
Sketch 2
Extrude 2
DTM1
Insert Here
Sketch 3
PRT_CSYS_DEF
Section has been reoriented to 2D orientation.
Geometry
Type here to search
11:15 AM
11/8/17


PRT0004 (Active) - PTC Creo Parametric Educational Edition
File
Model
Analysis
Annotate
Render
Tools
View
Flexible Modeling
Applications
Extrude
8.00
Placement
Options
Properties
Datum
Applies and saves any changes you have made in the feature tool and then closes the tool dashboard.
Model Tree
PRT0004.PRT
RIGHT
TOP
FRONT
PRT_CSYS_DEF
Sketch 1
Extrude 1
Sketch 2
Extrude 2
DTM1
Sketch 3
Insert Here
Extrude 3
PRT_CSYS_DEF
1 selected
Sketch
Type here to search
11:16 AM
11/8/17



RESULT:

Thus the given model is created by using PTC Creo Parametric 3.0

EXERCISE:5 **3D Part Modeling – 5**

AIM:

To model the given object using the Extrusion feature as per the dimensions given.

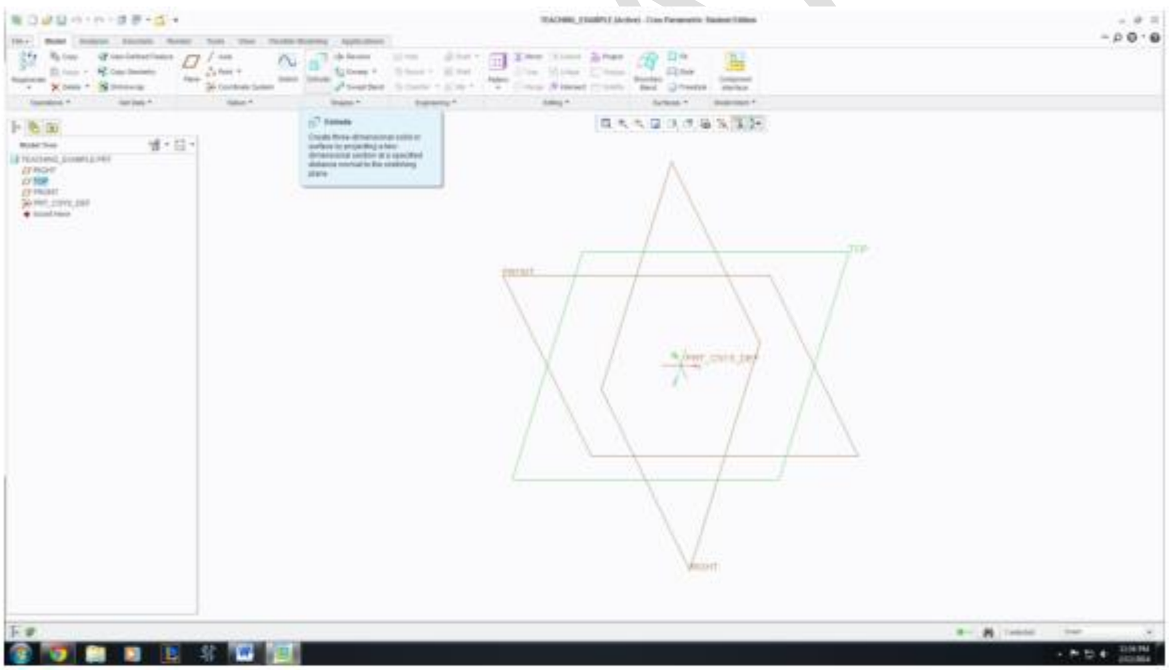
Description of Extrusion Feature:

Base Feature:

- **The first feature that is created.**
- **The foundation of the part.**
- **The base feature geometry for the box is an extrusion.**
- **The extrusion is named** *Extrude1*.

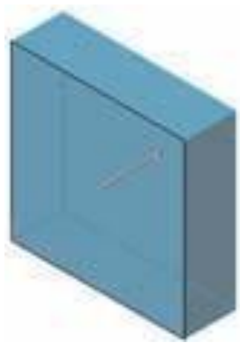
To Create an Extruded Base Feature:

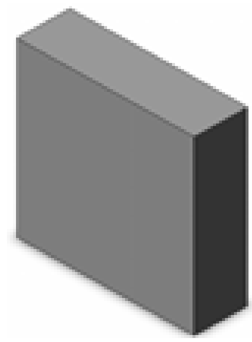
1. Select a sketch plane.



PTC CREO Work Plane display

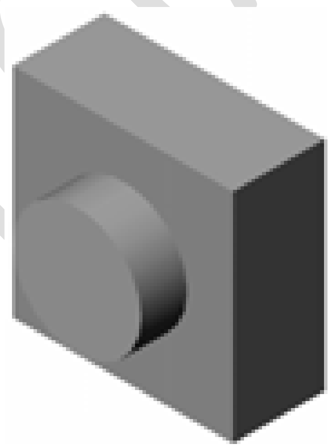
1. **Sketch a 2D profile of the model**
2. **Extrude the sketch perpendicular to sketch plane.**





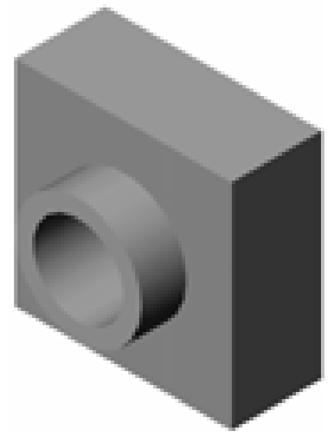
Extruded Boss Feature:

- It Adds material to the part and requires a sketch.



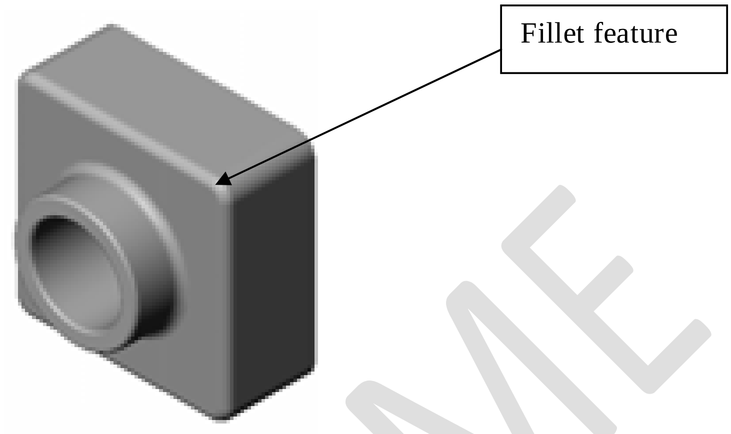
Extruded Cut Feature:

- It Removes material from the part and also it requires a sketch.



Fillet Feature:

❖ Rounds the edges or faces of a part to a specified radius.



Procedure:

1. Select a sketch plane.(Front, top or Side)
2. Sketch a 2D profile of the model.
3. Dimension the model using Smart Dimension icon.
4. Check the sketch is fully defined.
5. Extrude the sketch perpendicular to sketch plane.
6. Use extruded cut feature to cut the solid as given in the drawing.

Result:

Thus the given model is extruded.

EXERCISE:6 **3D Part Modeling – 6**

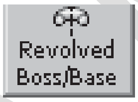
Ex: 2 **Exercise on Revolve**

AIM:

To model the given object using the Revolve feature as per the dimensions given.

Description of Revolve Feature:

Command Manager: Features > Revolved Boss/Base Menu: Insert > Boss/Base > Revolve

Toolbar: Features > Revolved Boss/Base. 

Using this tool, the sketch is revolved about the revolution axis. The revolution axis could be an axis, an entity of the sketch, or an edge of another feature to create the revolved feature. Note that whether you use a centerline or an edge to revolve the sketch, the sketch should be drawn on one side of the centerline or the edge.

After drawing the sketch, as you choose this tool, you will notice that the sketching environment is closed and the part modeling environment is invoked. Similar to extruding the sketches, the resulting feature can be a solid feature or a thin feature, depending on the sketch and the options selected to be revolved. If the sketch is closed, it can be converted into a solid feature or a thin feature. However, if the sketch is open, it can be converted only into a thin feature.

Now a window with planes will open. now we have to draw a sketch to revolve. Here we start Select front plane and using sketch option draw a sketch as in the fig. Don’t worry about dimensions. after drawing sketch draw a center line over Y-axis.

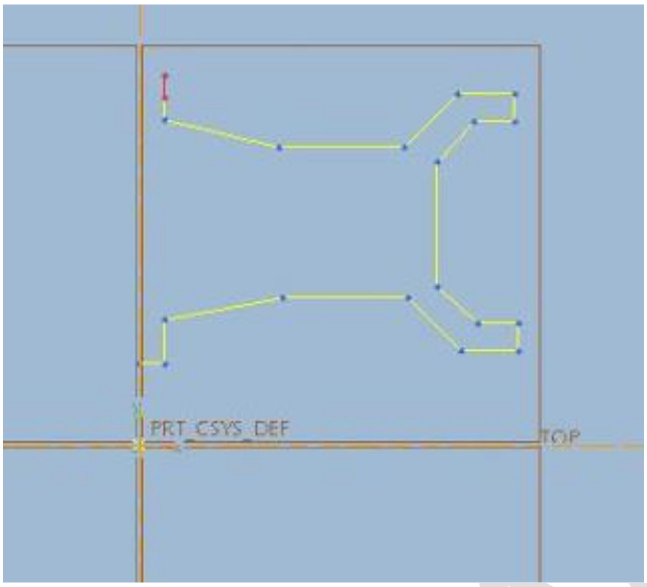
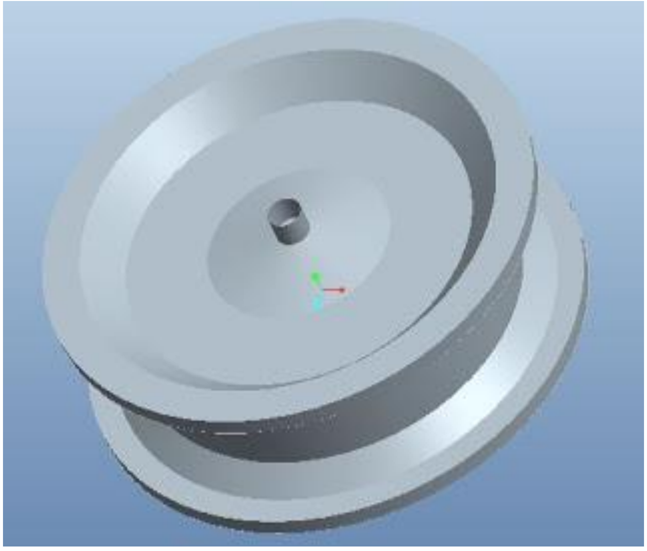


Fig: Sketch of piston to be revolve

will notice that the view is automatically changed to a 3D view, and the **Revolve Property Manager** is displayed,after sketching click at done button. in new window click at revolve button

some command line will open and program will ask you to select a reference. here the reference is a line about which our sketch has to move around. in this practice we will use that center line that we have draw in sketching.after selecting that line as reference program will show you a demo of your sketch as a revolved part. if you want to see how your part will look you may click at "glasses" button if your part is well according to you then click at done revolving.

now your part will look like this.



note : you may draw sketch as you like but remember the conditions to revolve that is a reference line.

Fig: 8 Revolve Property Manager

Fig:9 Feature Created after revolving to 360 ^{0}

Procedure:

1. Select a sketch plane.(Front, top or Side)
2. Sketch a 2D profile of the model.

3. Dimension the model using Smart Dimension icon.
4. Check the sketch is fully defined.
5. Revolve thesketch.

Result:

Thus the given model is drawn using revolve feature.

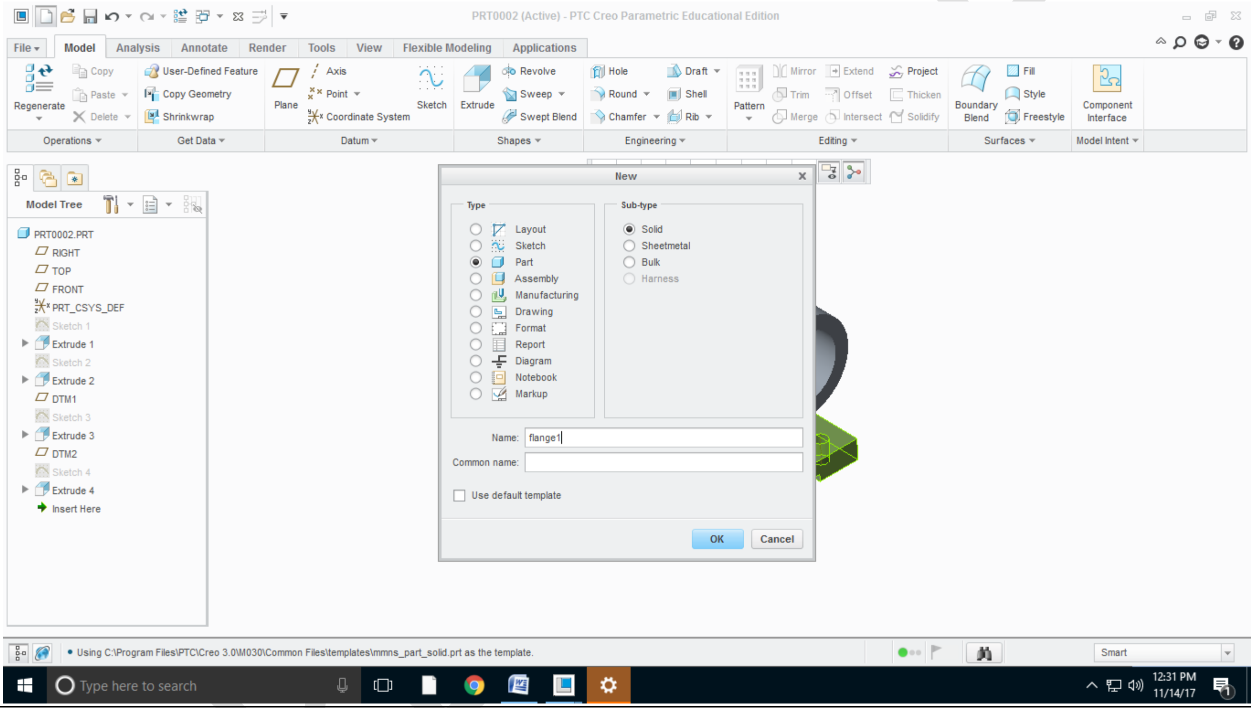
EXERCISE:7 **Part Modeling & Assembly**

<u>AIM:</u> To model the given flange coupling and assemble them by using PTC Creo Parametric 3.0

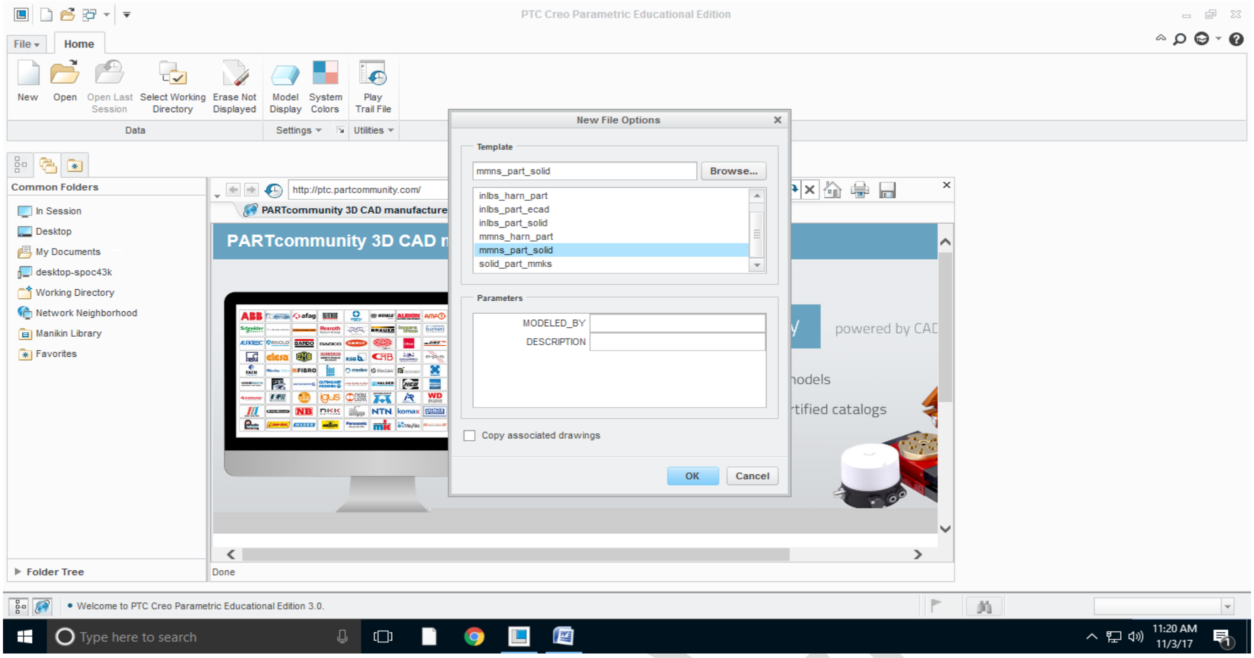
<u>PROCEDURE:</u>

<u>Step1:</u>

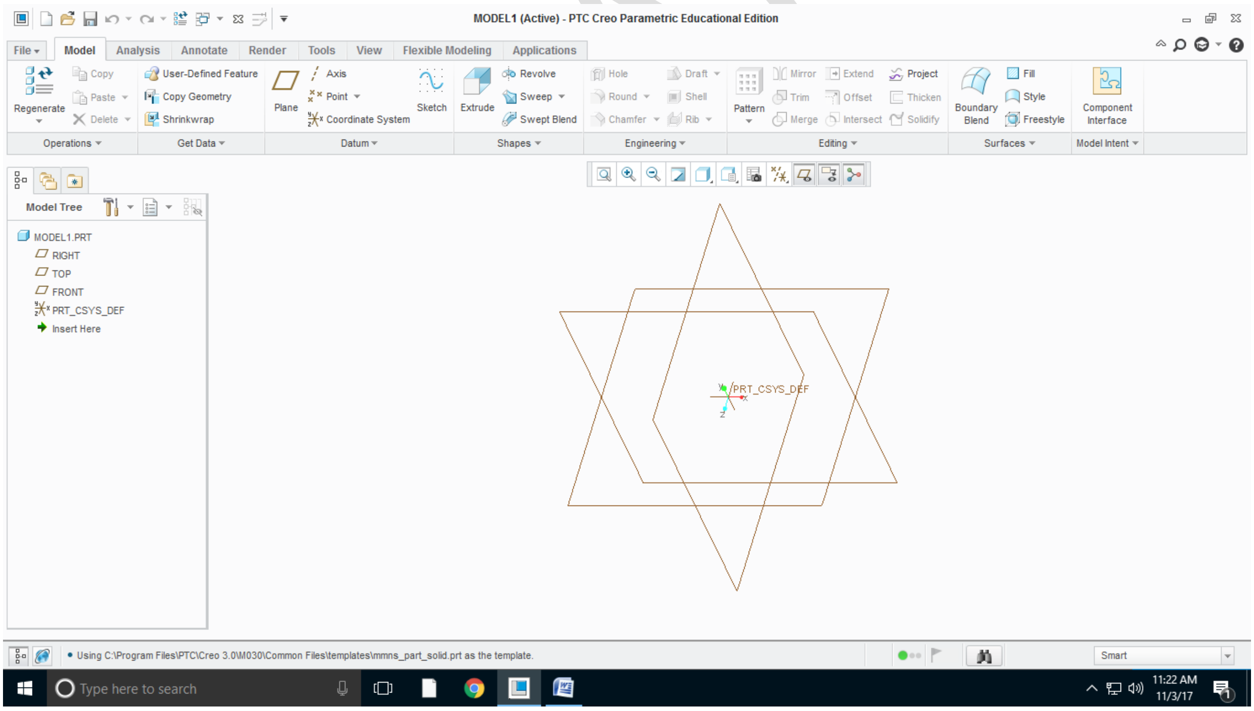
Double click on creo parametric and then open new file



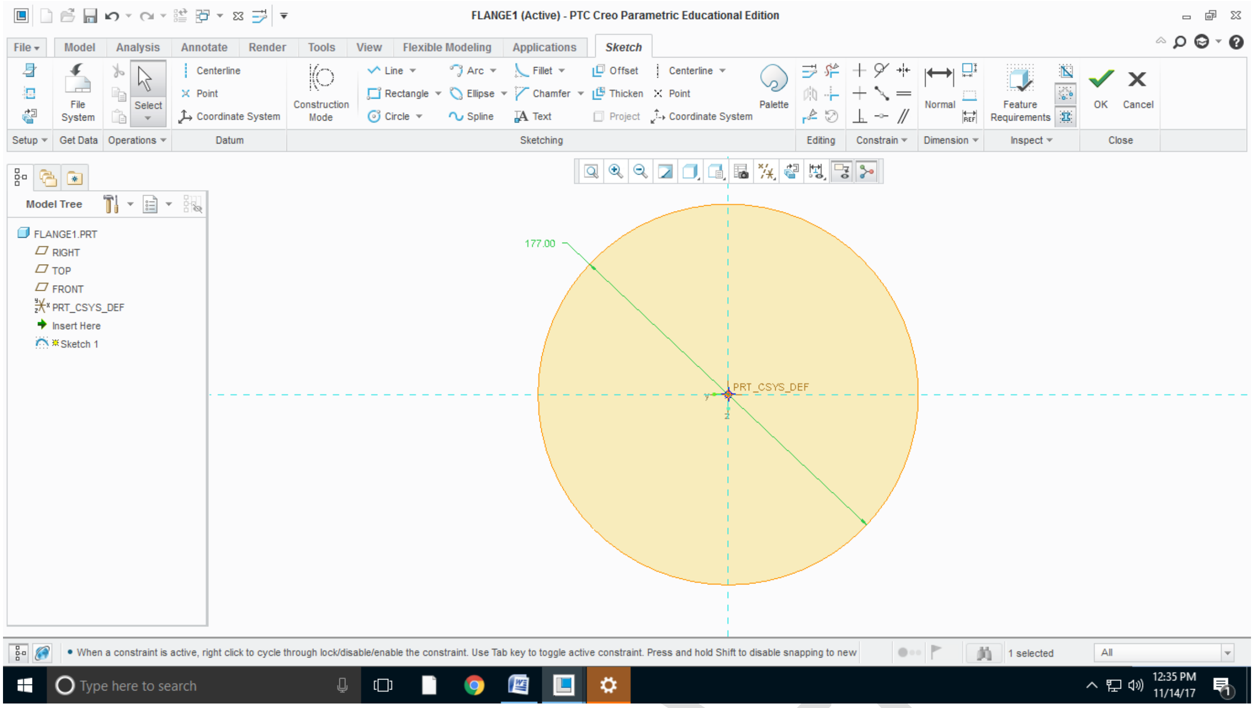
Step 2: select ok and select mmns_part_solid and then select OK



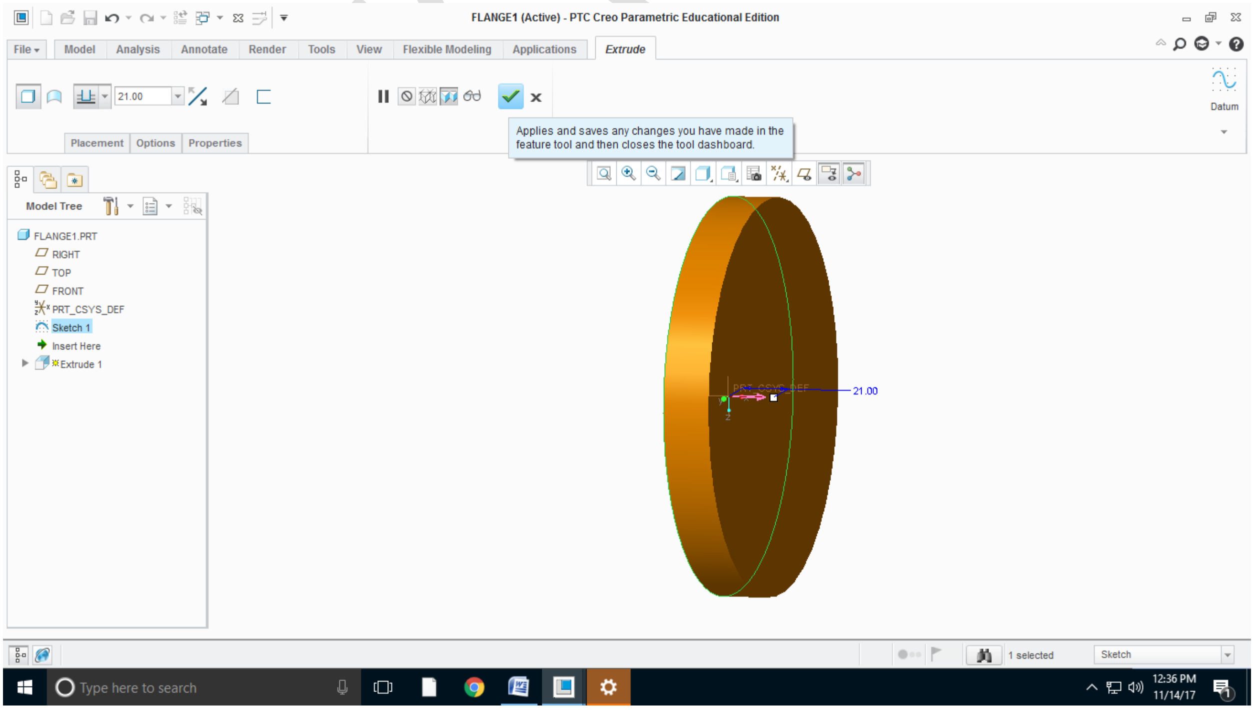
Step4:select a sketch plane



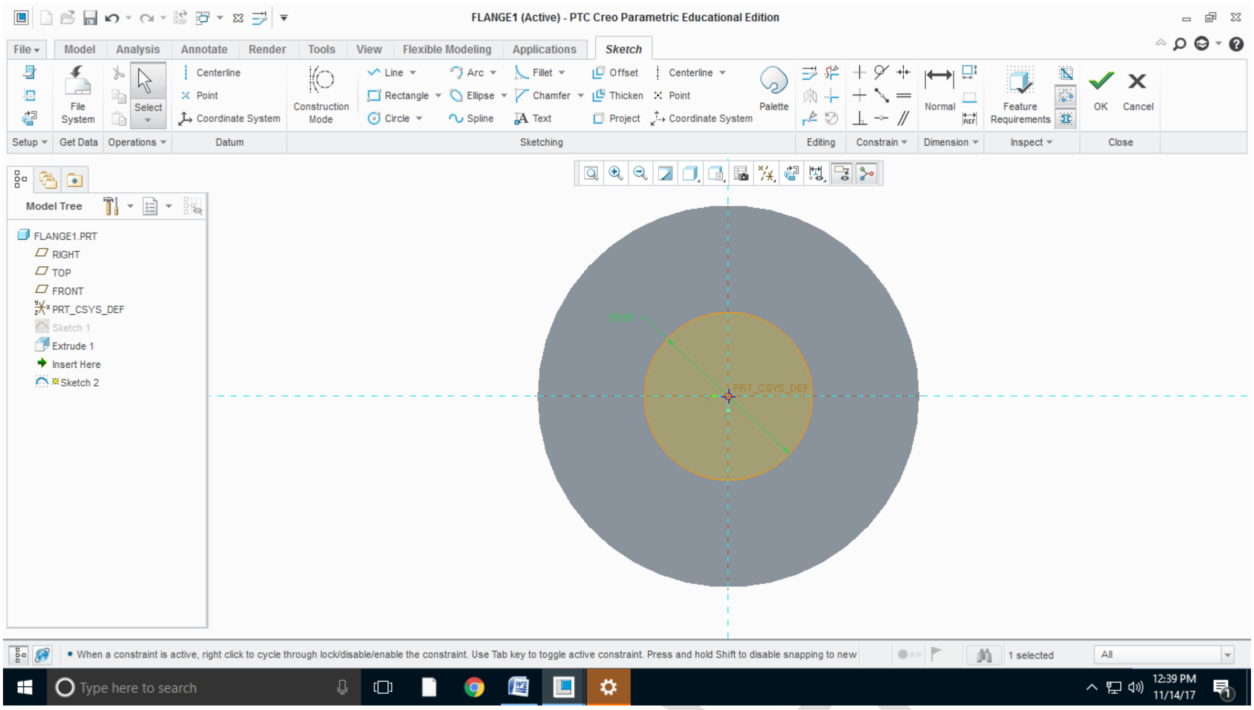
Step5:sketch a 2D profile of the model

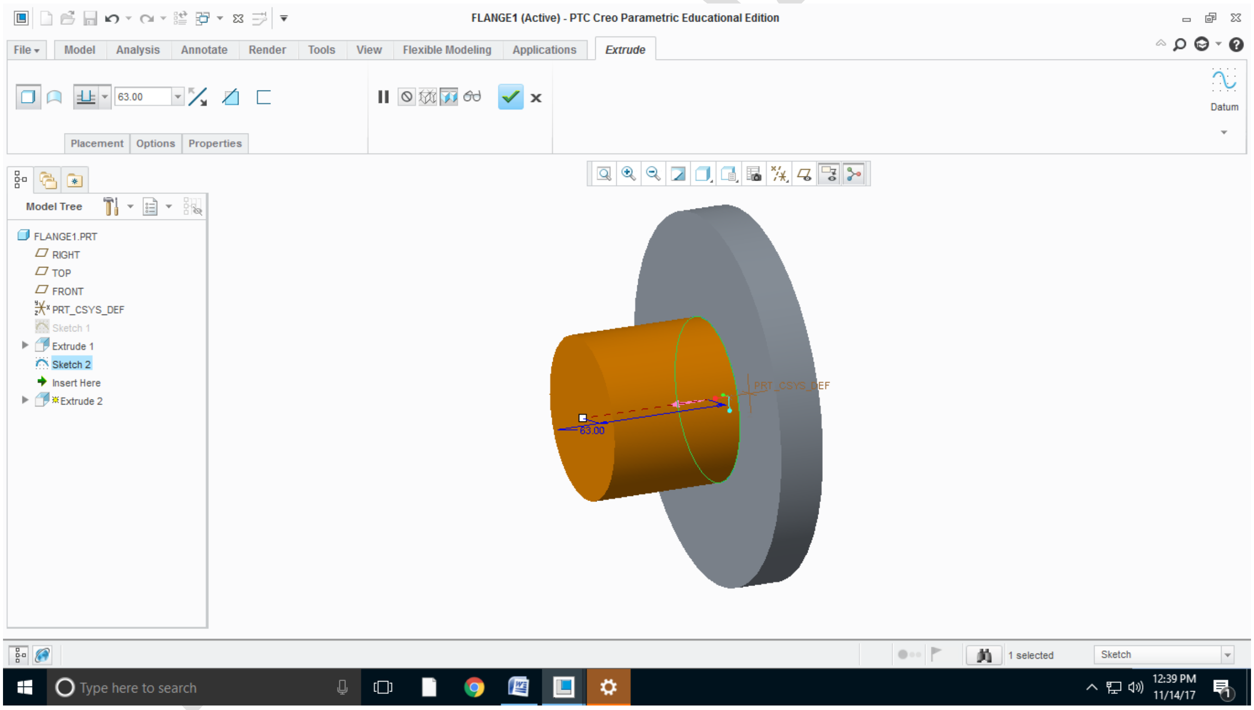


Step6: extrude the sketch perpendicular to the sketch plane and specify width and direction of extrusion

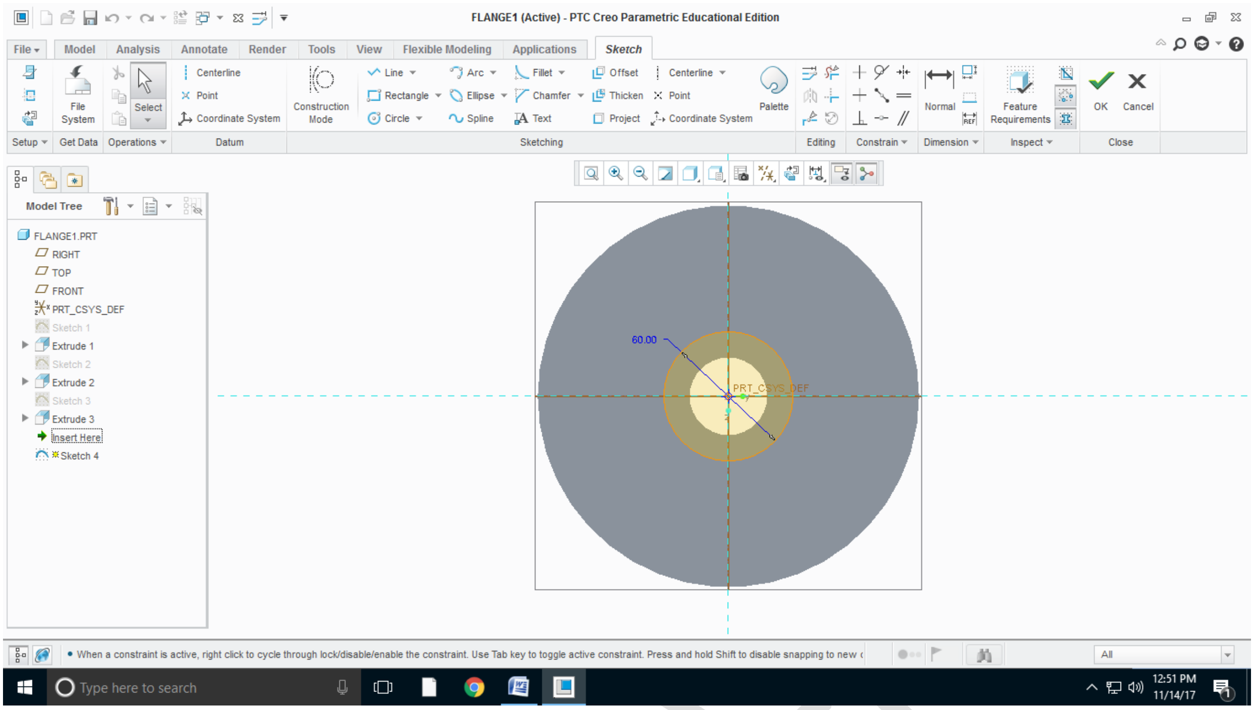


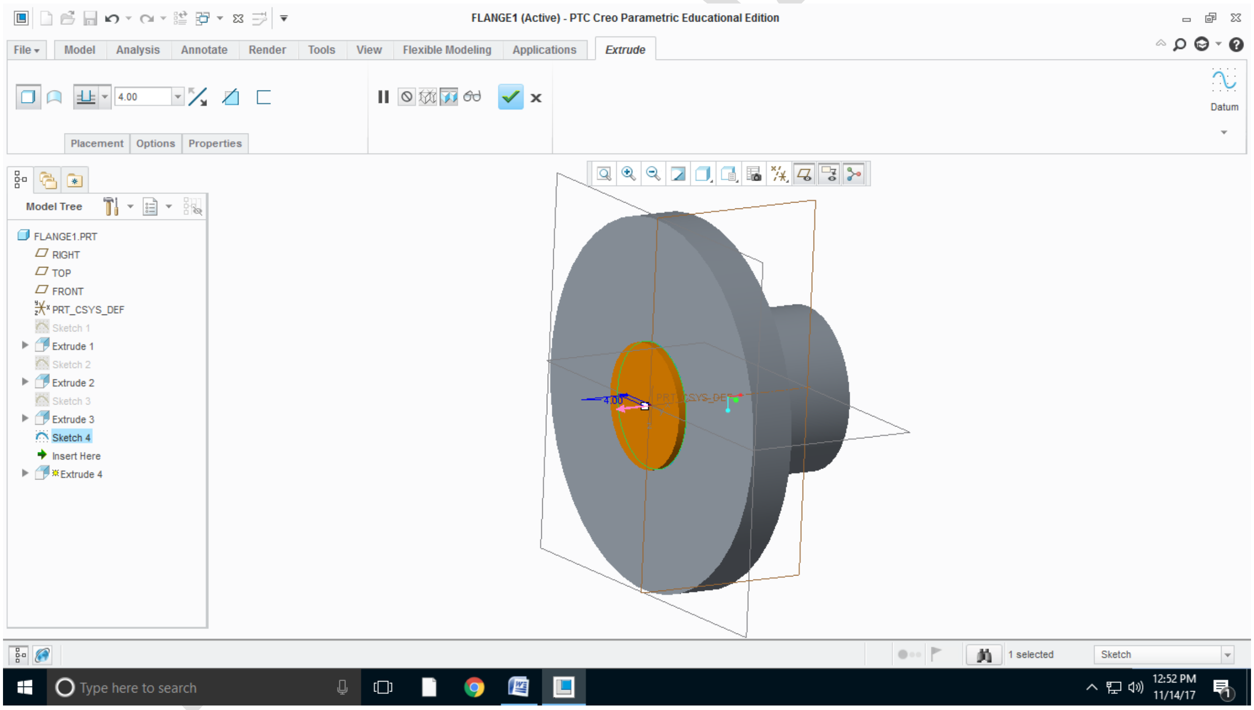
Step7:select the required surface to add the hub and then draw the hub in 2d sketch plane and extrude it by giving the required dimensions



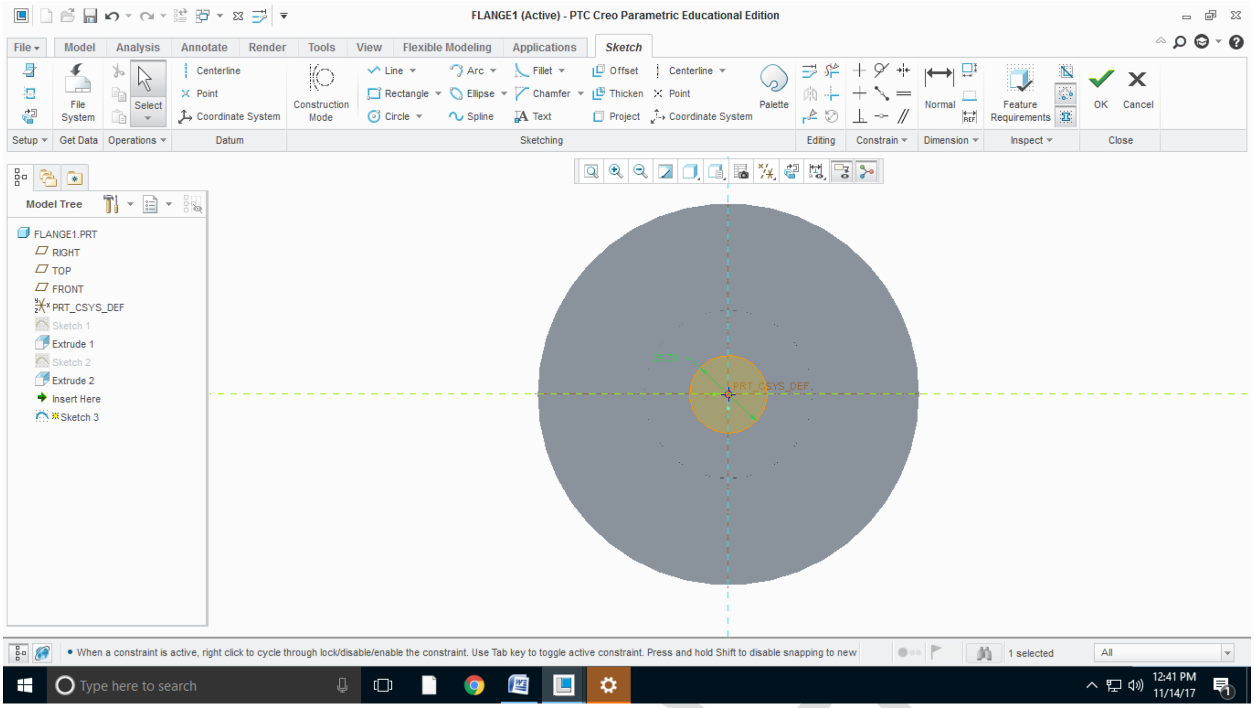


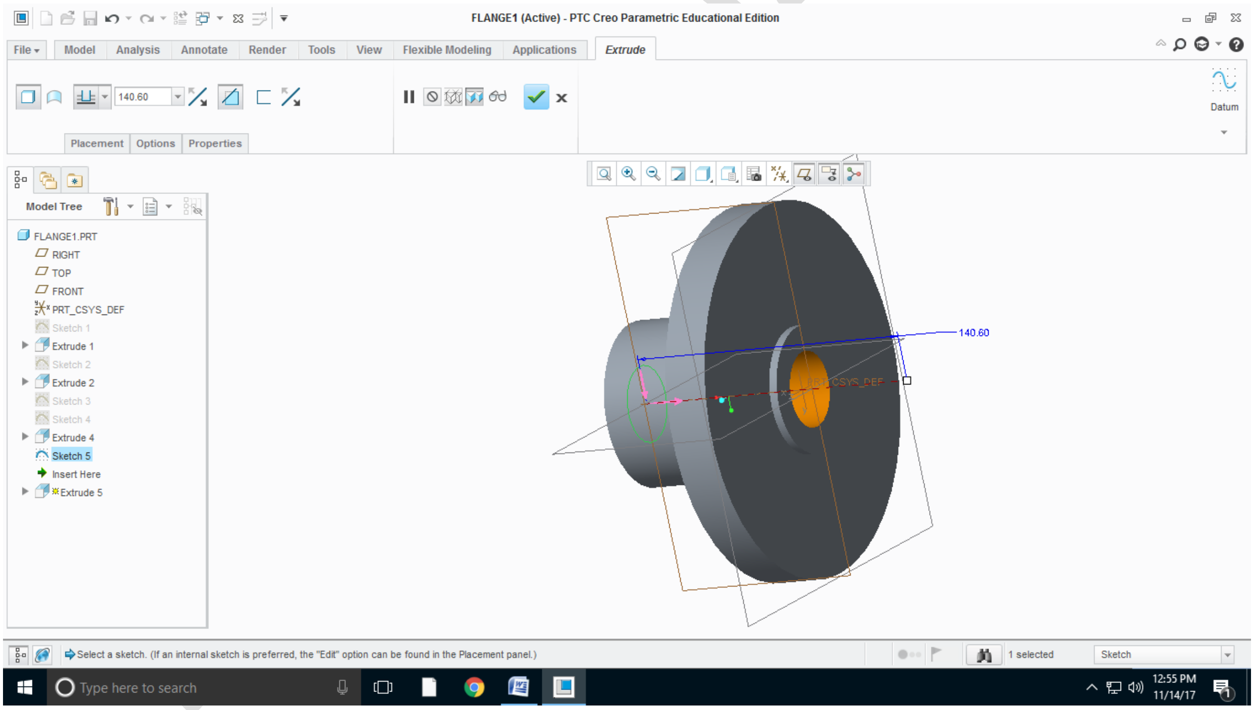
Step 8:select the required surface to add the material and then draw the hub in 2d sketch plane and extrude it by giving the required dimensions

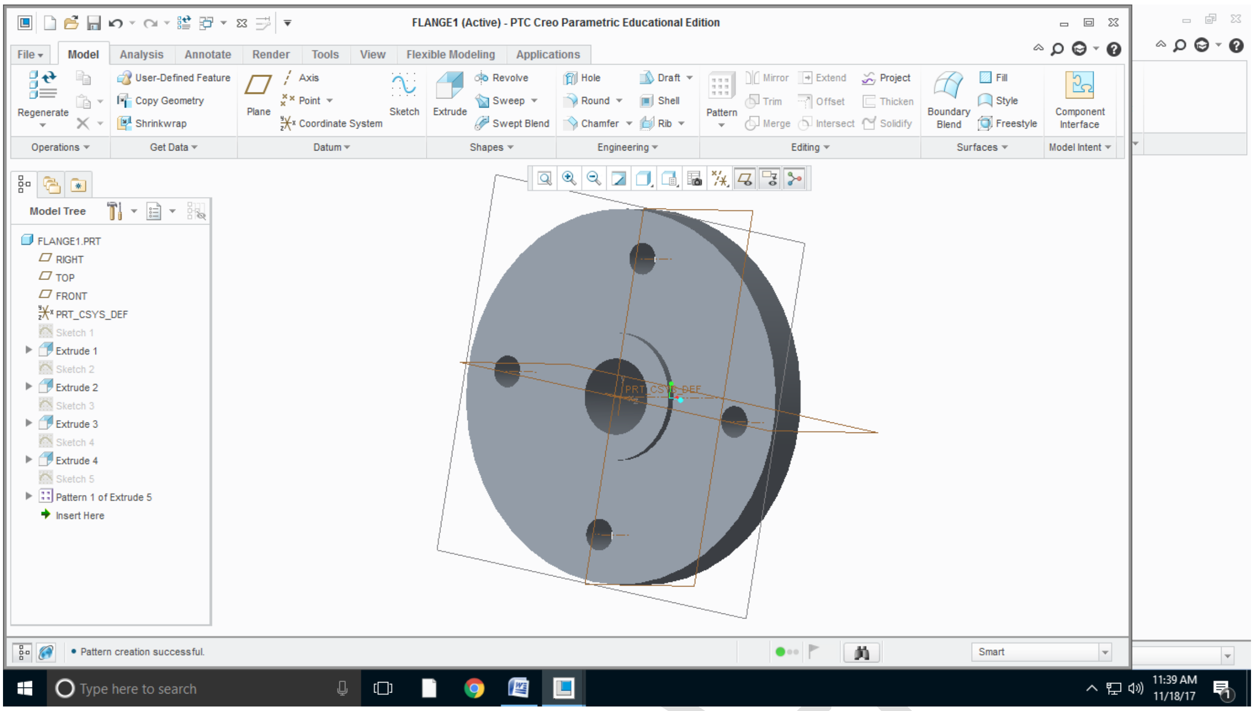




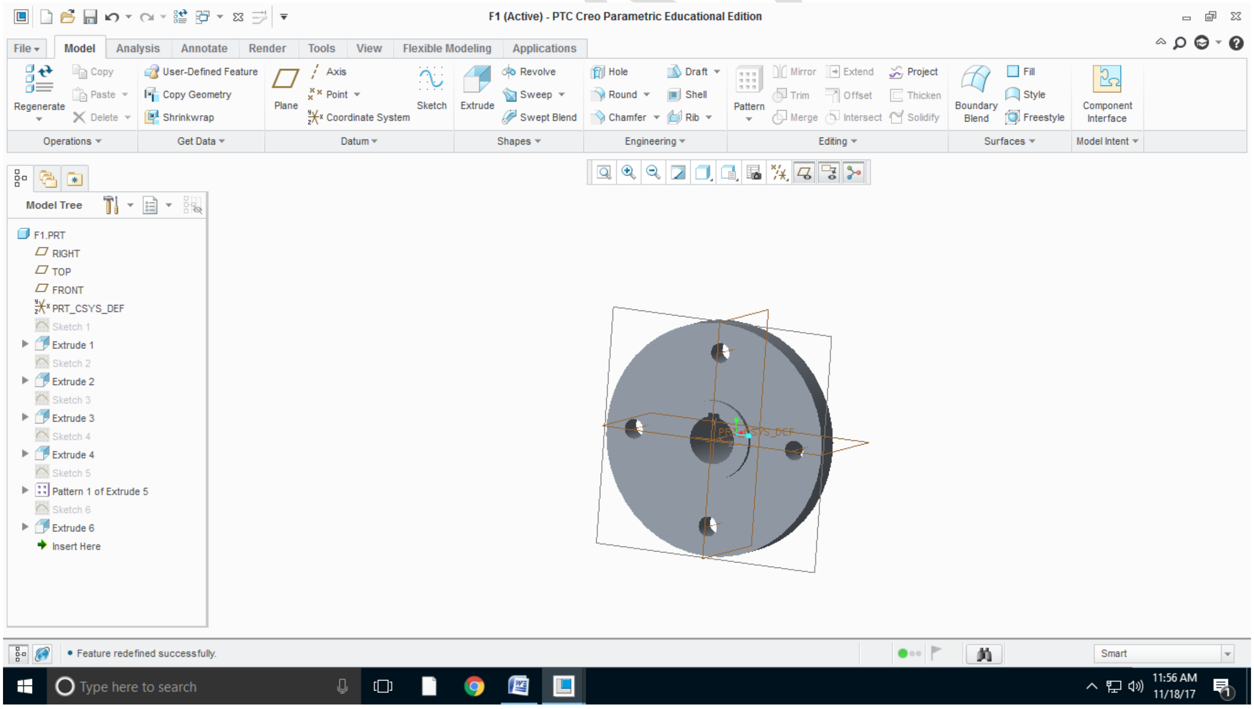
Step9:select the required surface to remove the material in the hub and then draw the shaft in 2d sketch plane and extrude it by giving the required dimensions



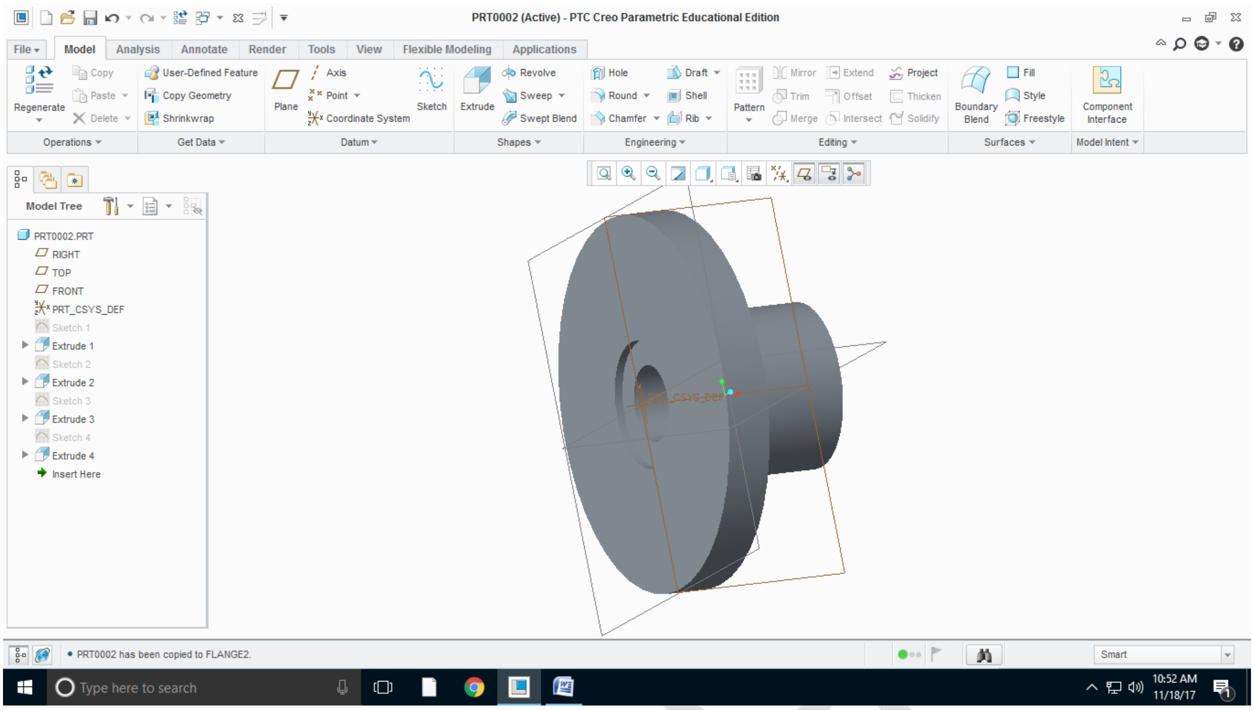


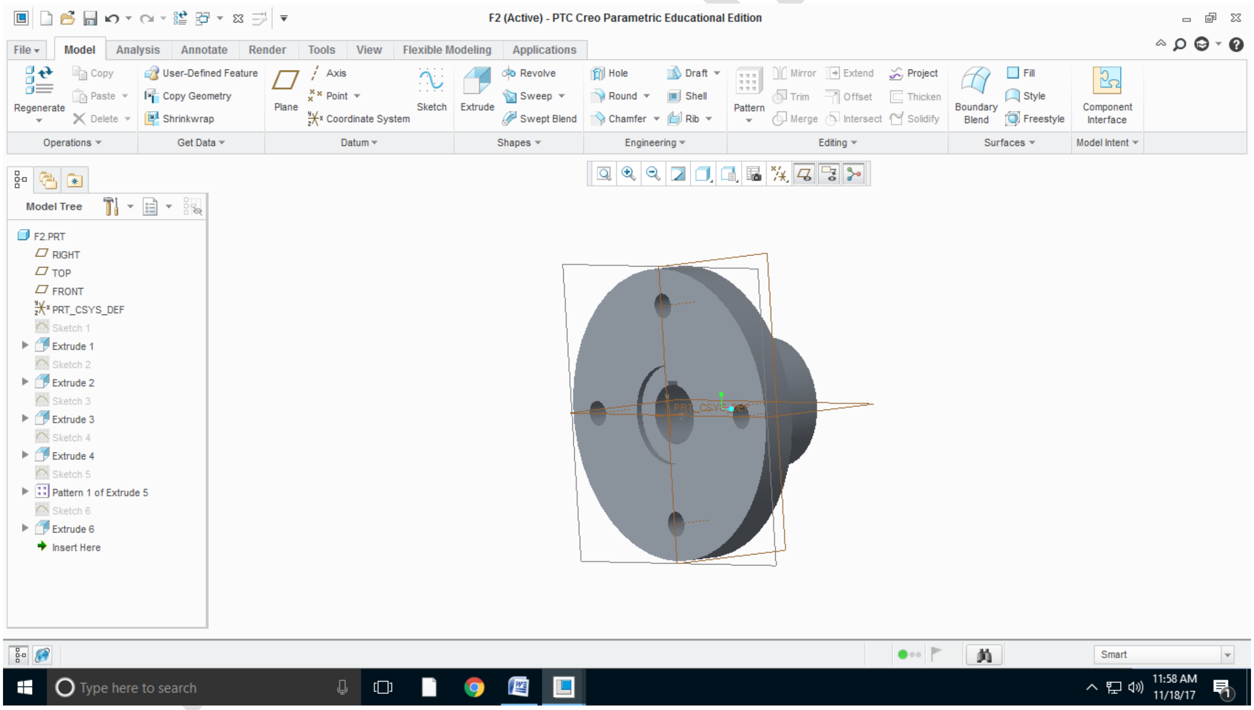


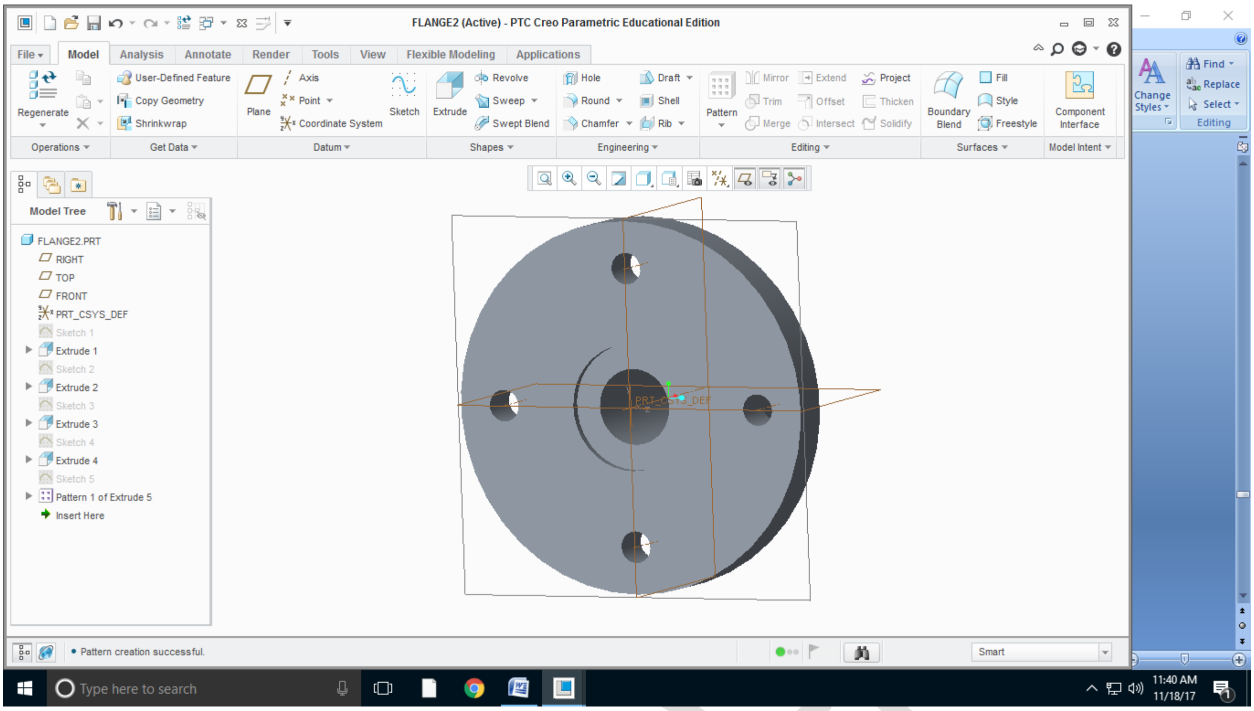
Create key way in the hub



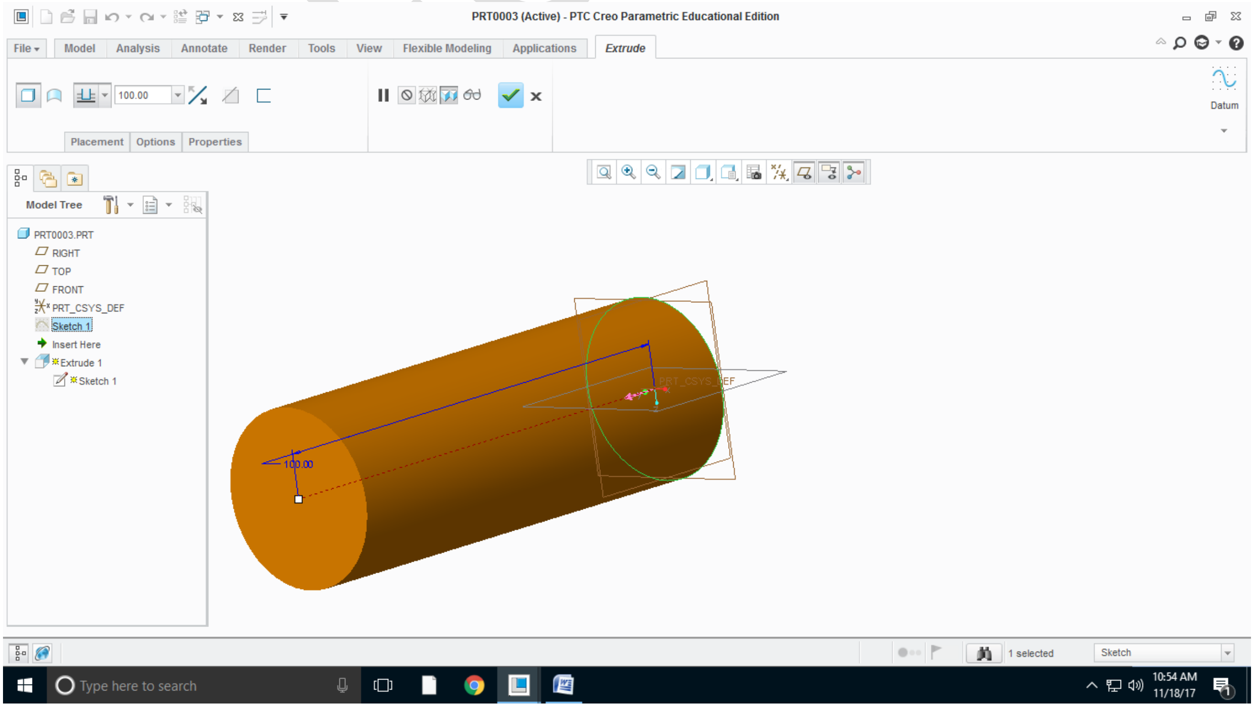
Step10:similarly create another flange and save it as another file

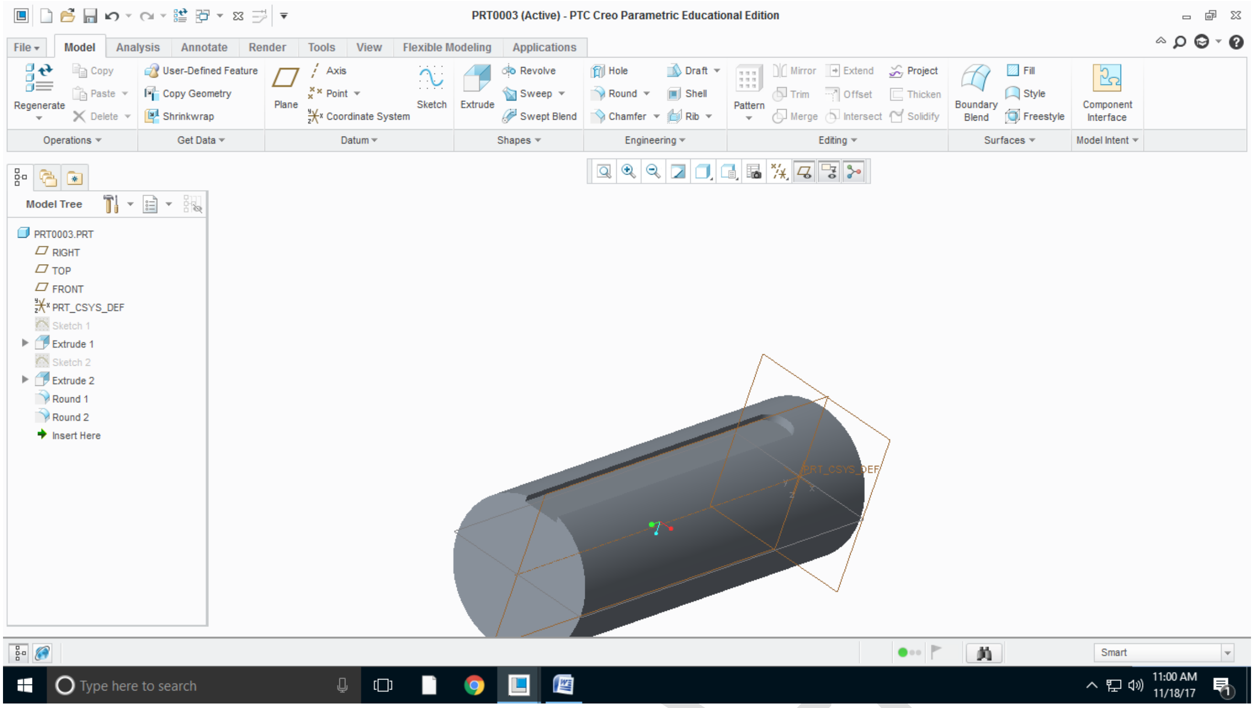

PRT0002 (Active) - PTC Creo Parametric Educational Edition
File Model Analysis Annotate Render Tools View Flexible Modeling Applications
Model Tree
PRT0002.PRT
RIGHT
TOP
FRONT
PRT_CSYS_DEF
Sketch 1
Extrude 1
Sketch 2
Extrude 2
Sketch 3
Extrude 3
Sketch 4
Extrude 4
Insert Here
PRT0002 has been copied to FLANGE2.


F2 (Active) - PTC Creo Parametric Educational Edition
File Model Analysis Annotate Render Tools View Flexible Modeling Applications
Model Tree
F2.PRT
RIGHT
TOP
FRONT
PRT_CSYS_DEF
Sketch 1
Extrude 1
Sketch 2
Extrude 2
Sketch 3
Extrude 3
Sketch 4
Extrude 4
Sketch 5
Pattern 1 of Extrude 5
Sketch 6
Extrude 6
Insert Here

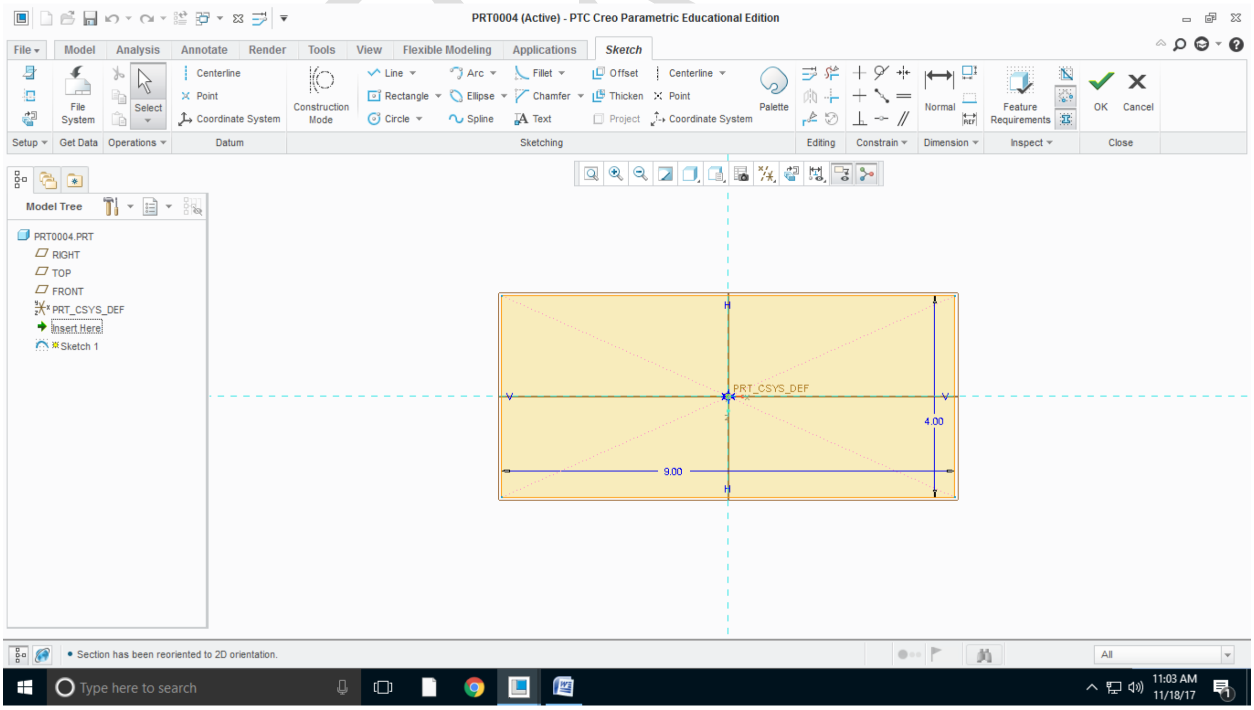


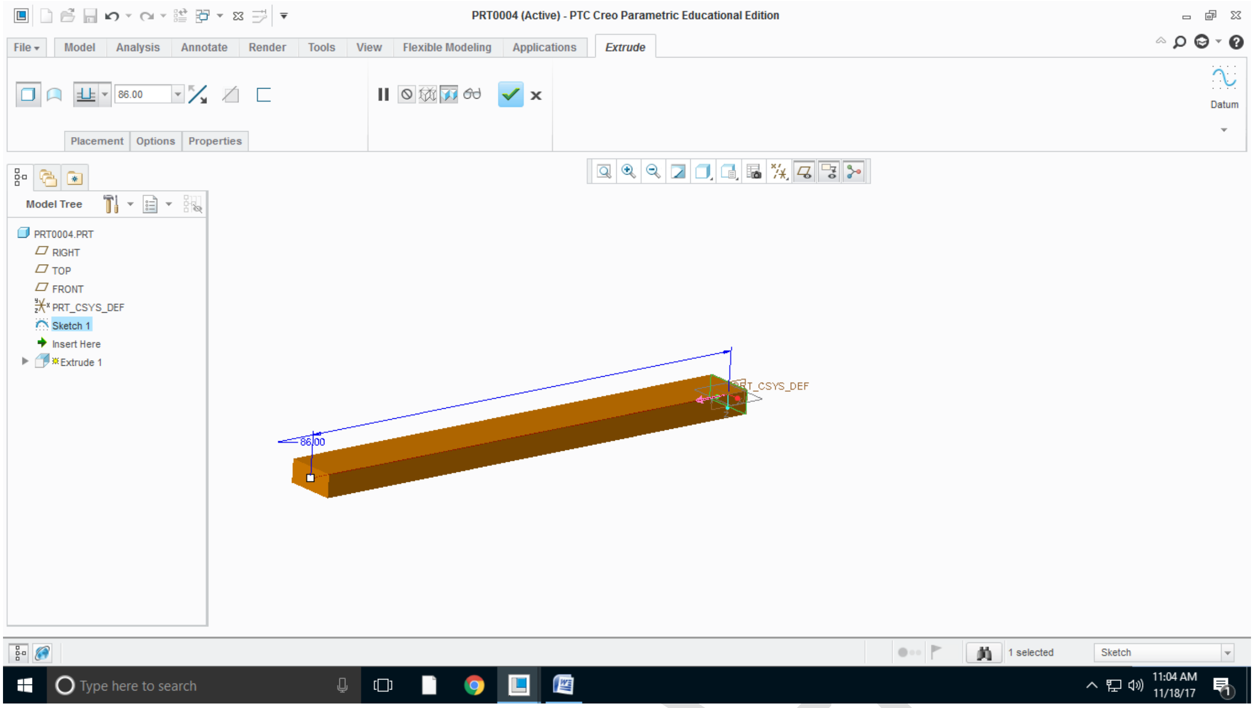
Step11:create a shaft and then select required sirface of the shaft to create a keyway on the shaft and save it

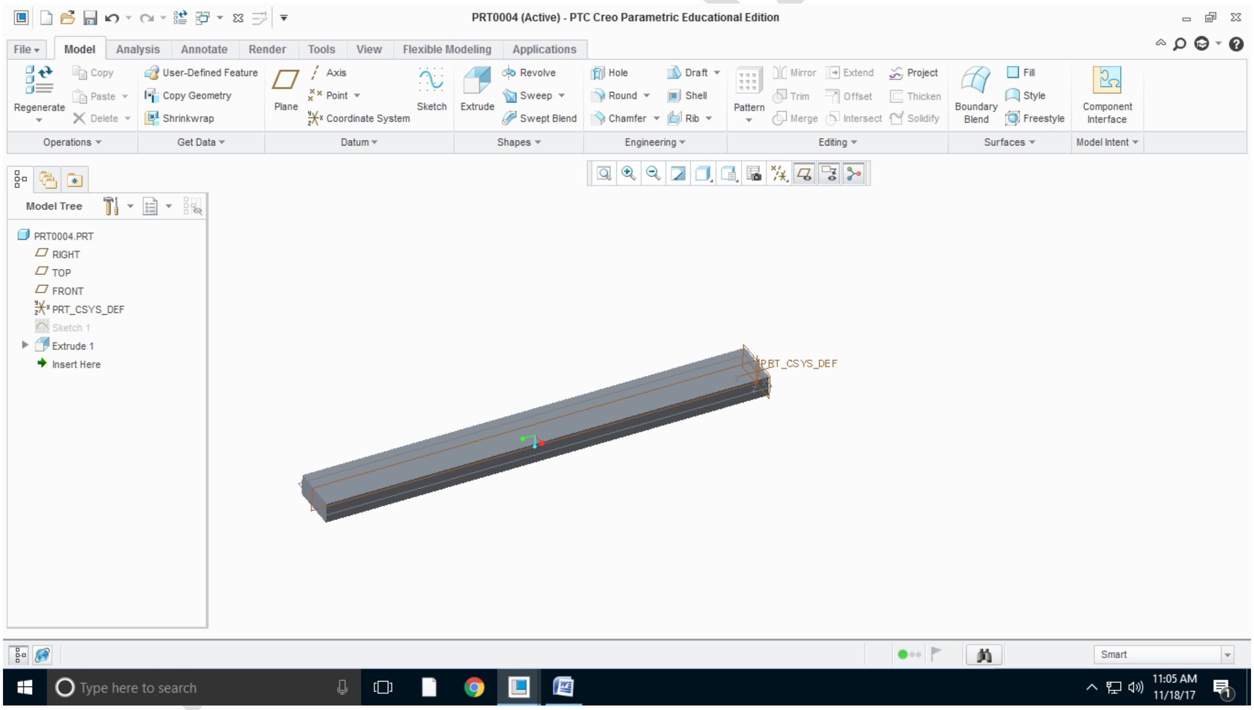




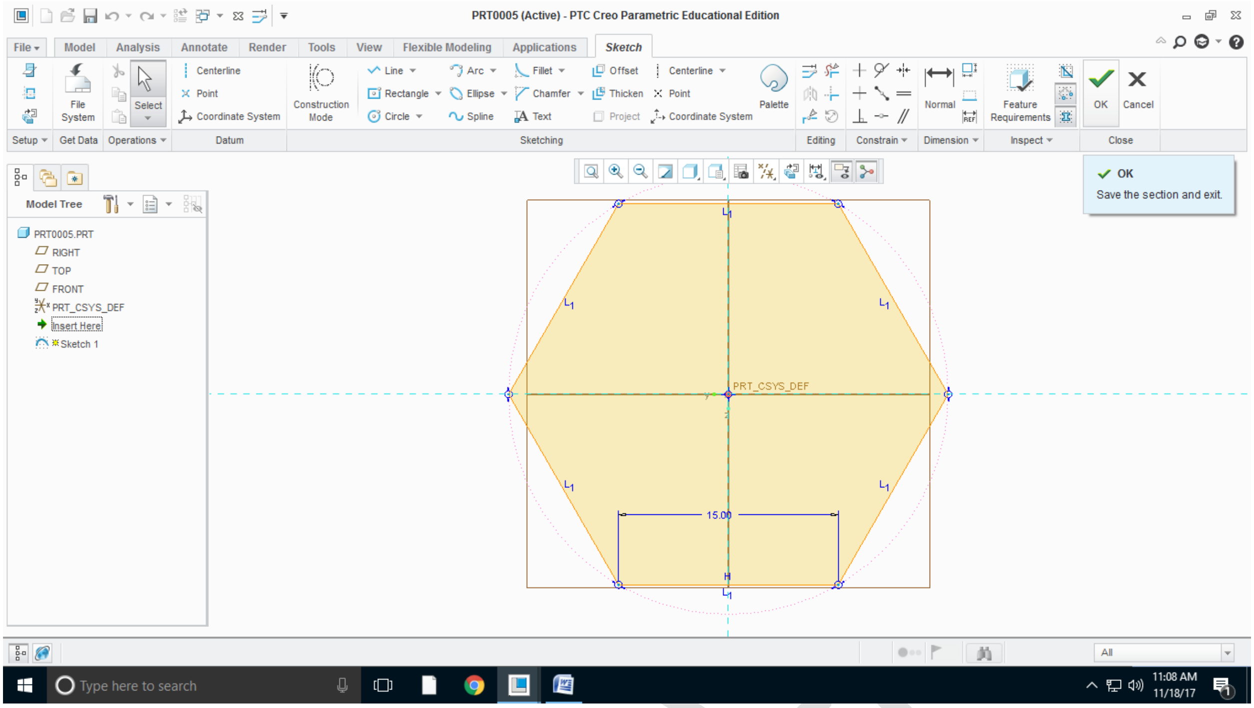
Step12:create a keyway and save it

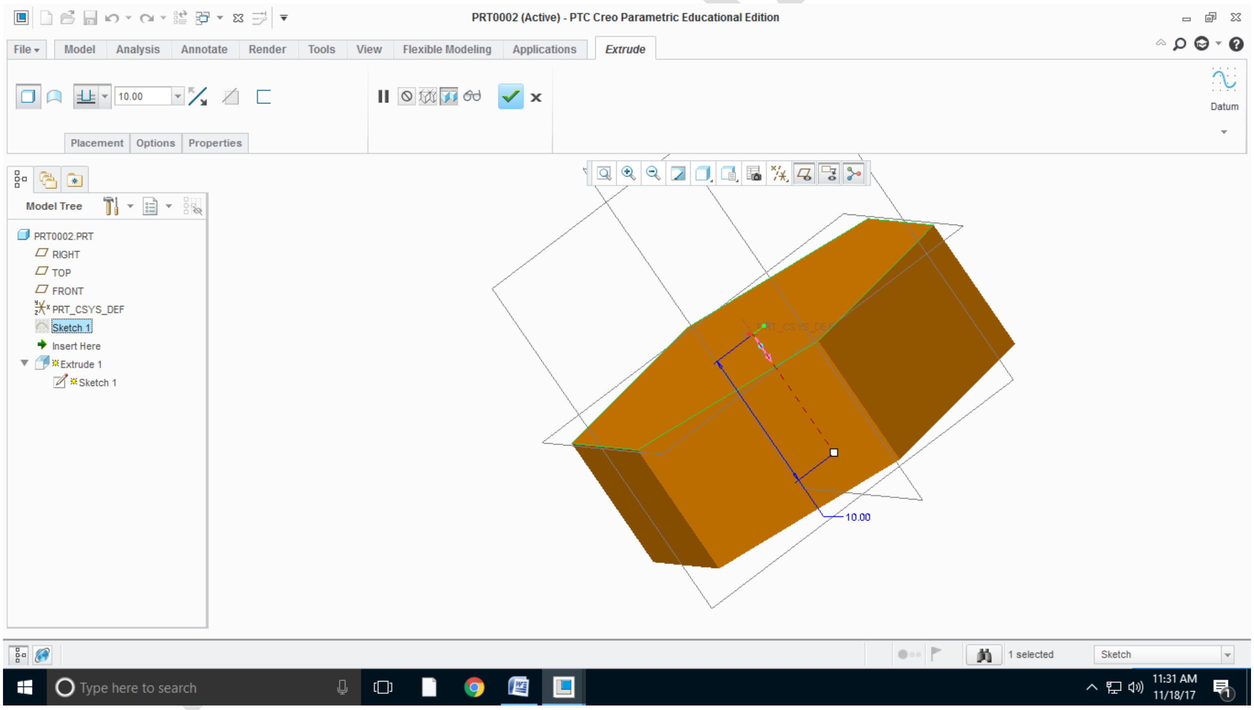




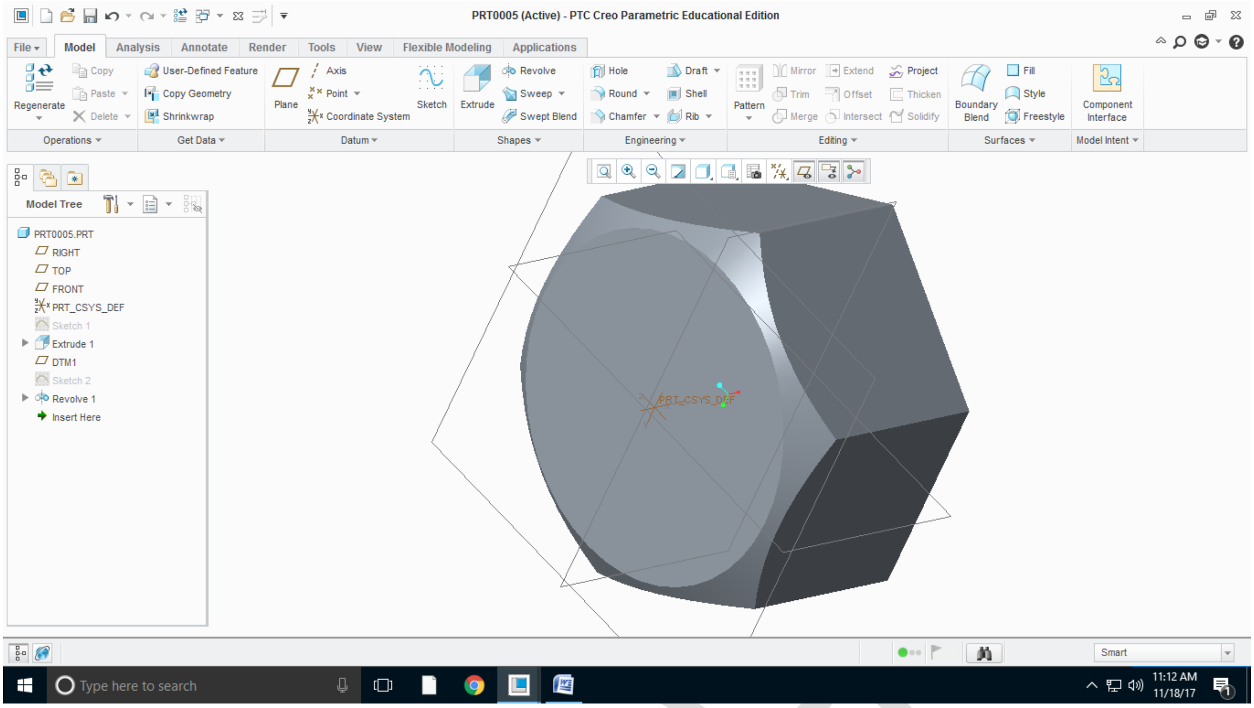


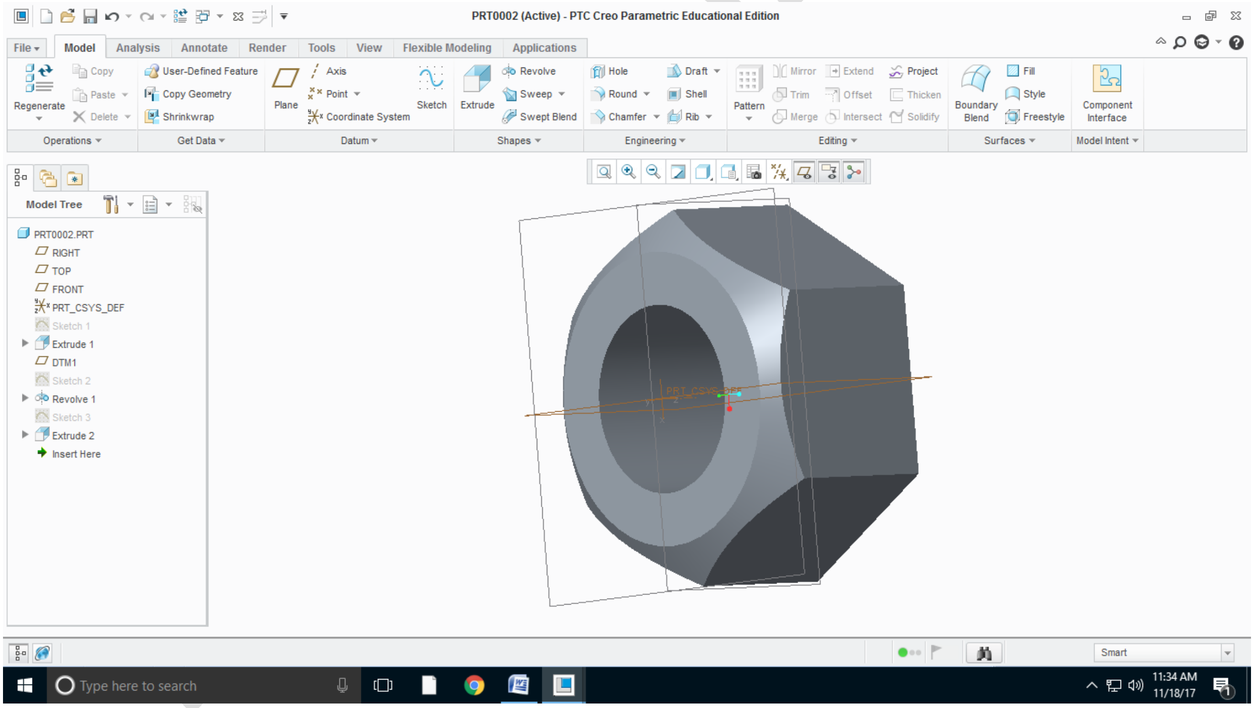
Step13:create a bolt and save it



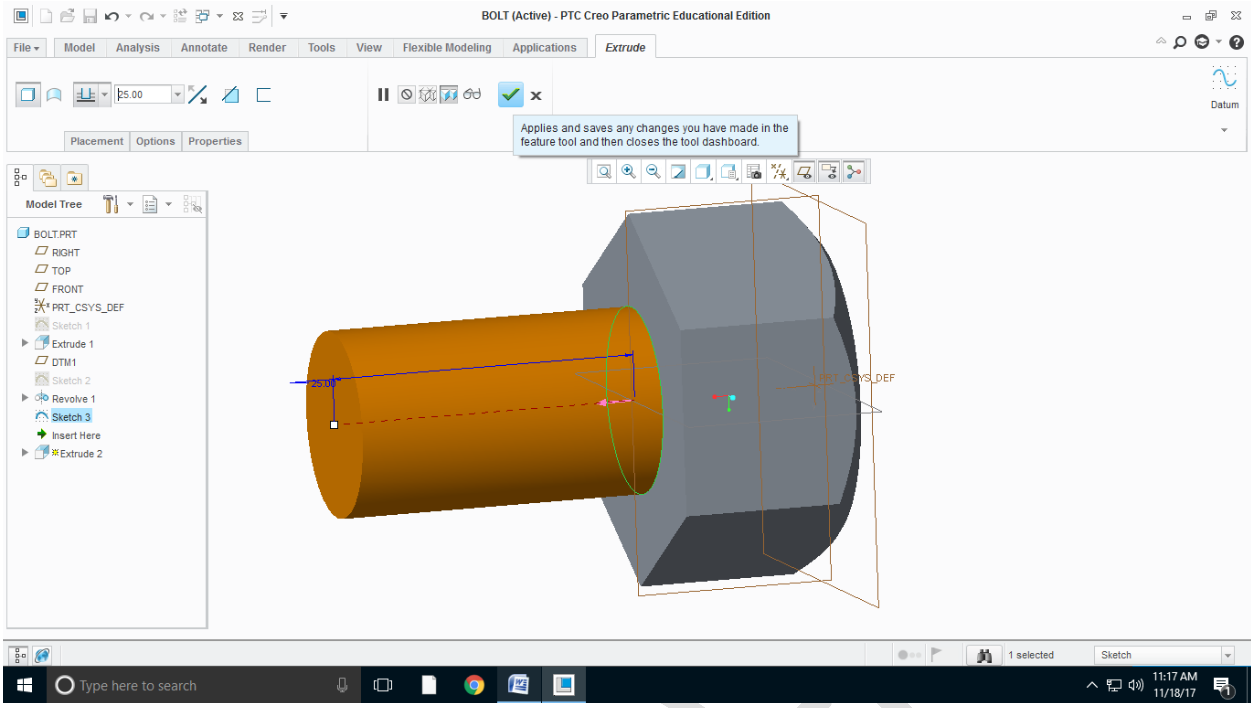


Step13:Select a datum plane as shown in figure and with reference to that draw the required shape to smooth the bolt surface using revolve option nad save it

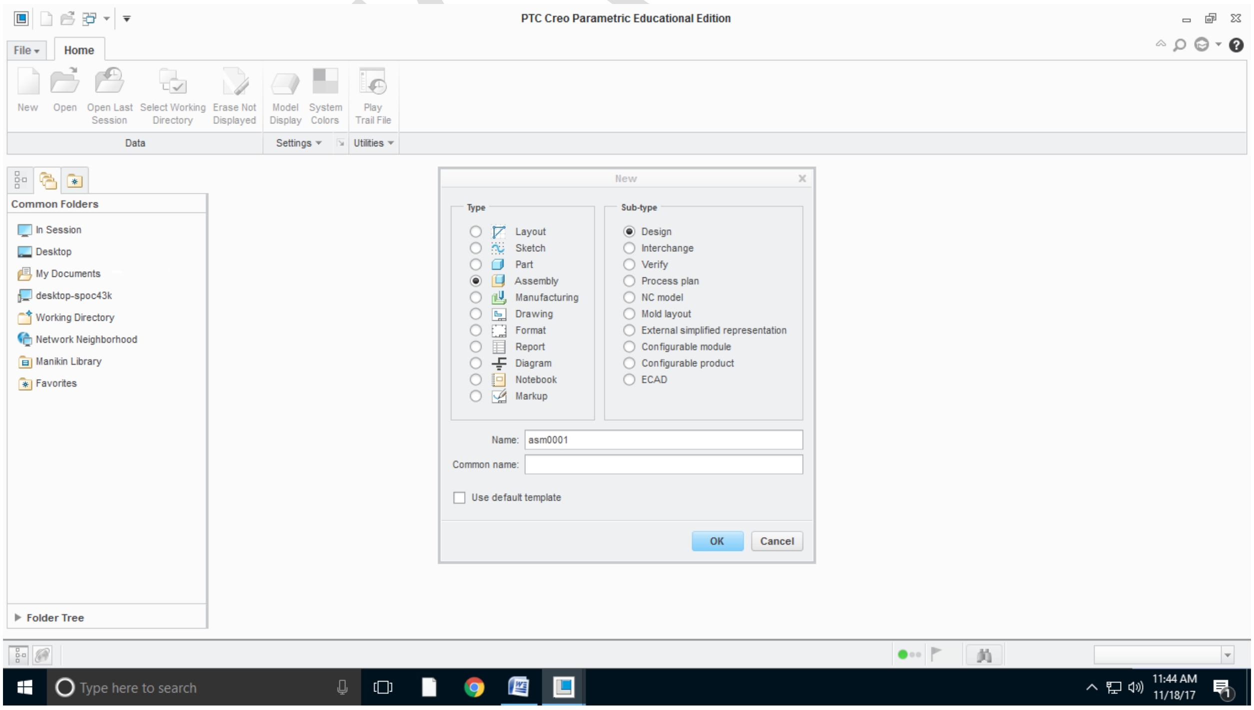




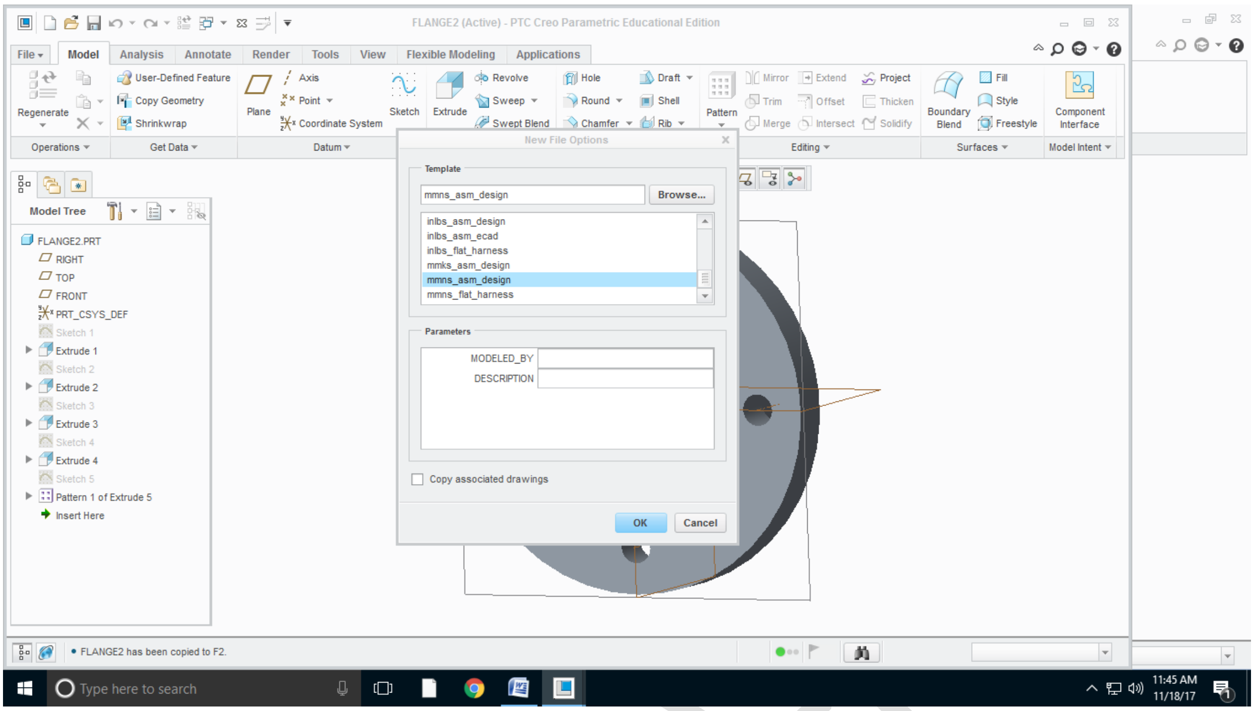
Step 14:create a nut and save it



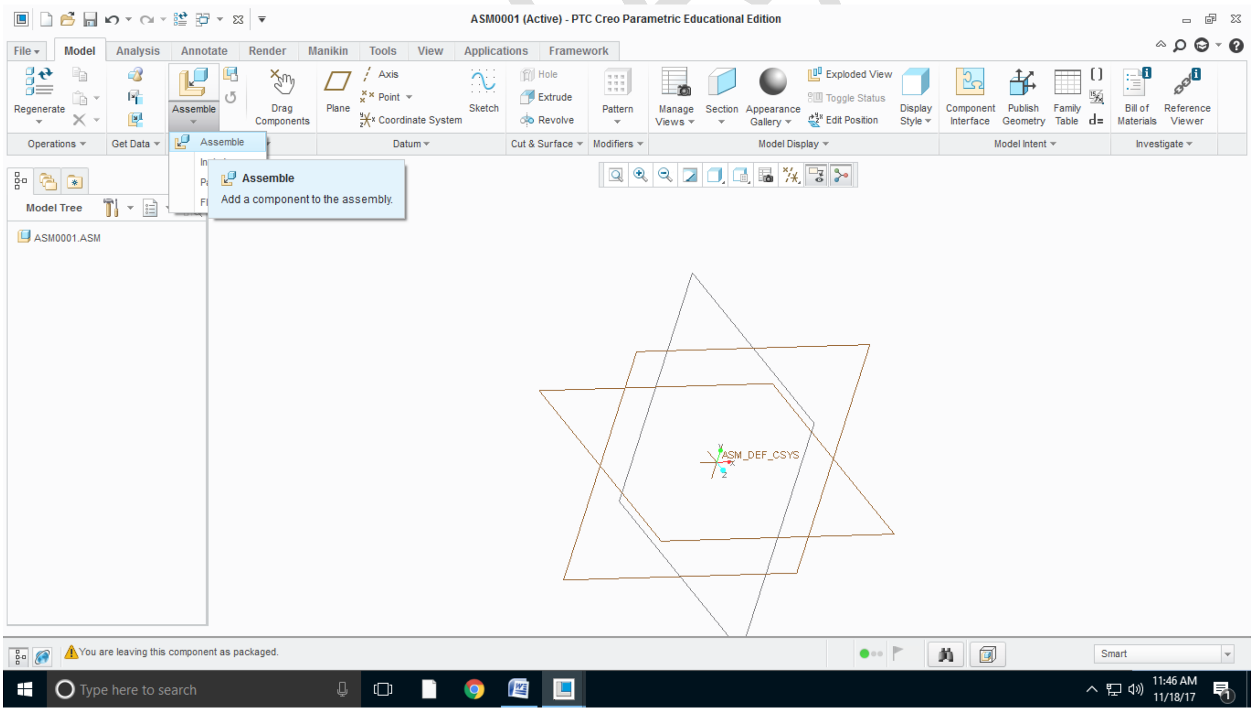
Step15:now open creo parametric and select assembly in type design in sub type and give the name of the assembly and then select ok.



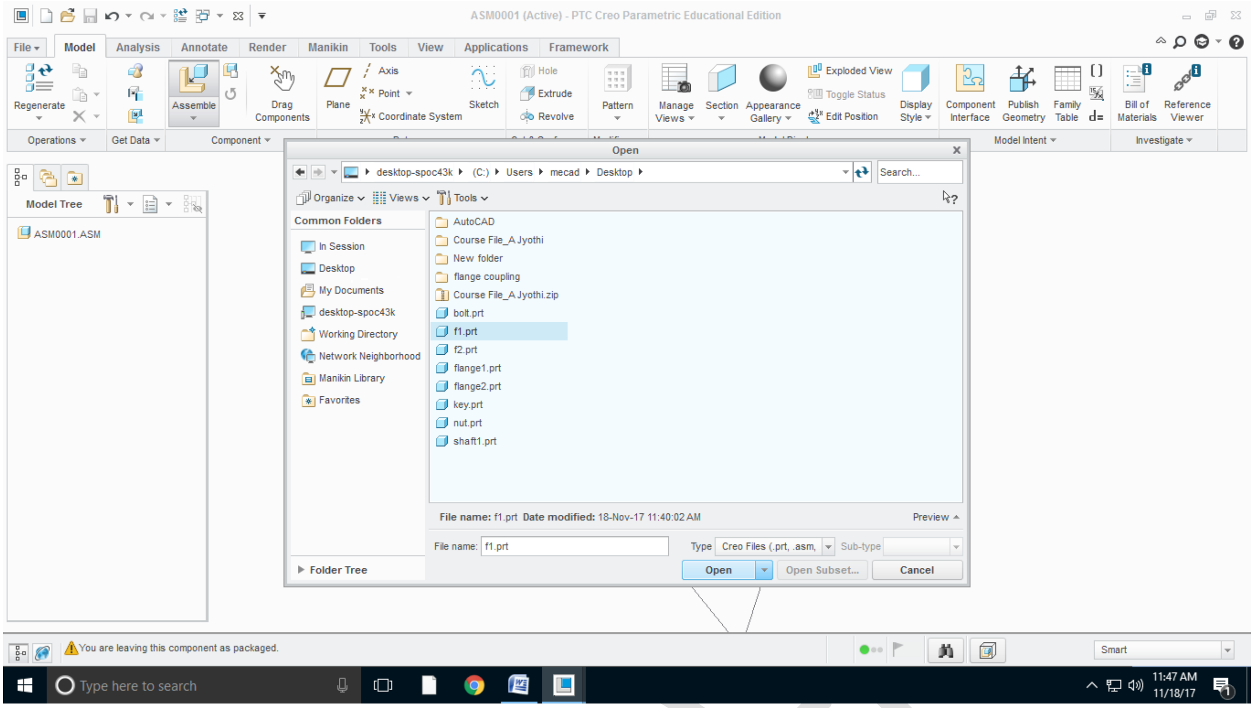
Step16:select mmns-ass-design and then ok

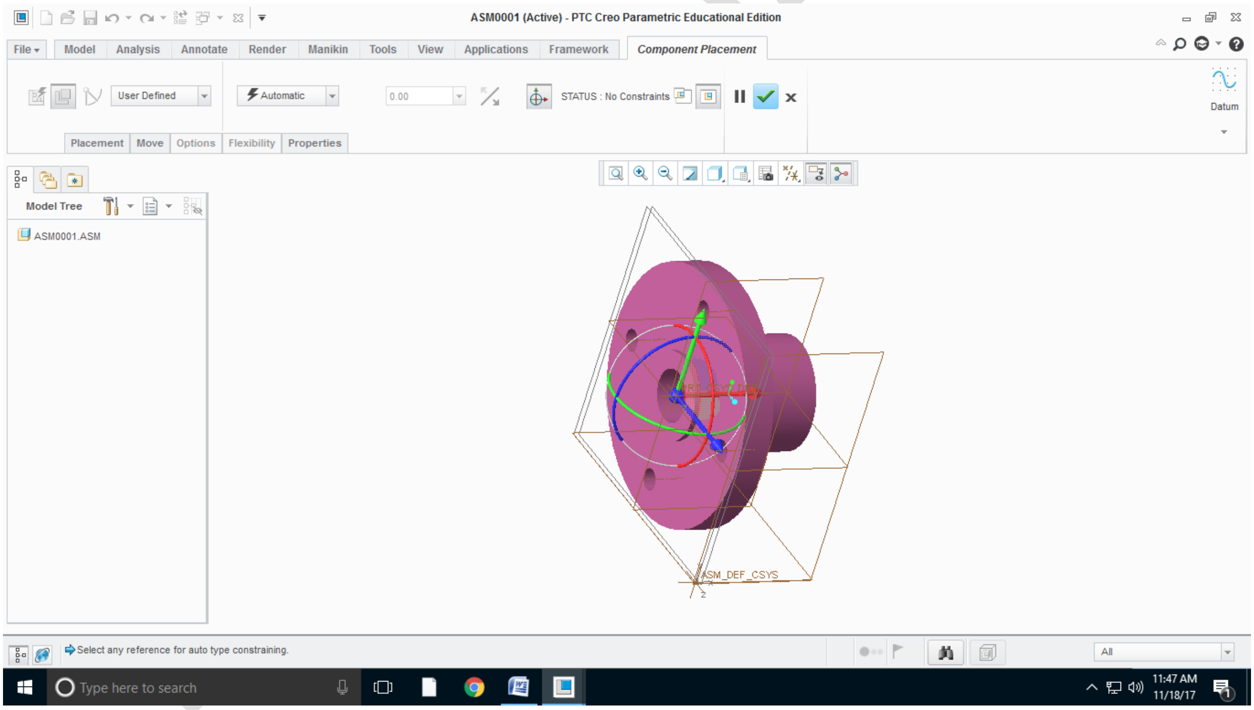


Step17:import the required part to do assembling

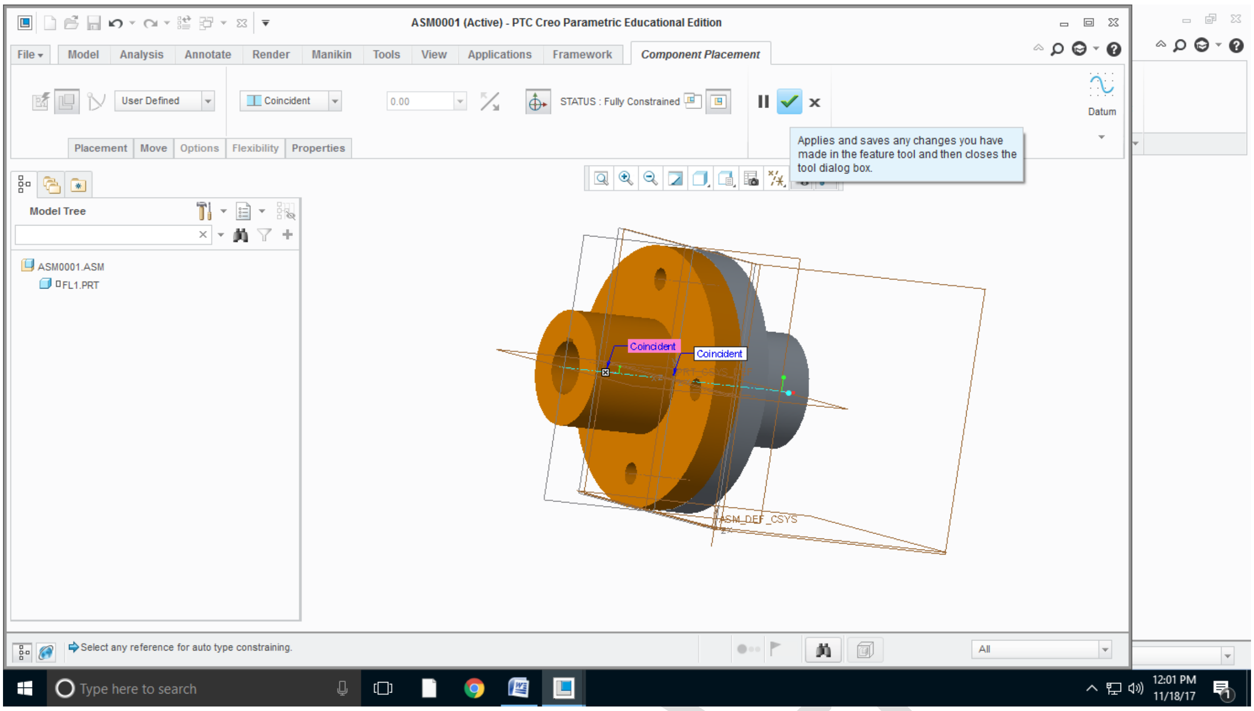


Step18:Select the required file from the saved files and then open

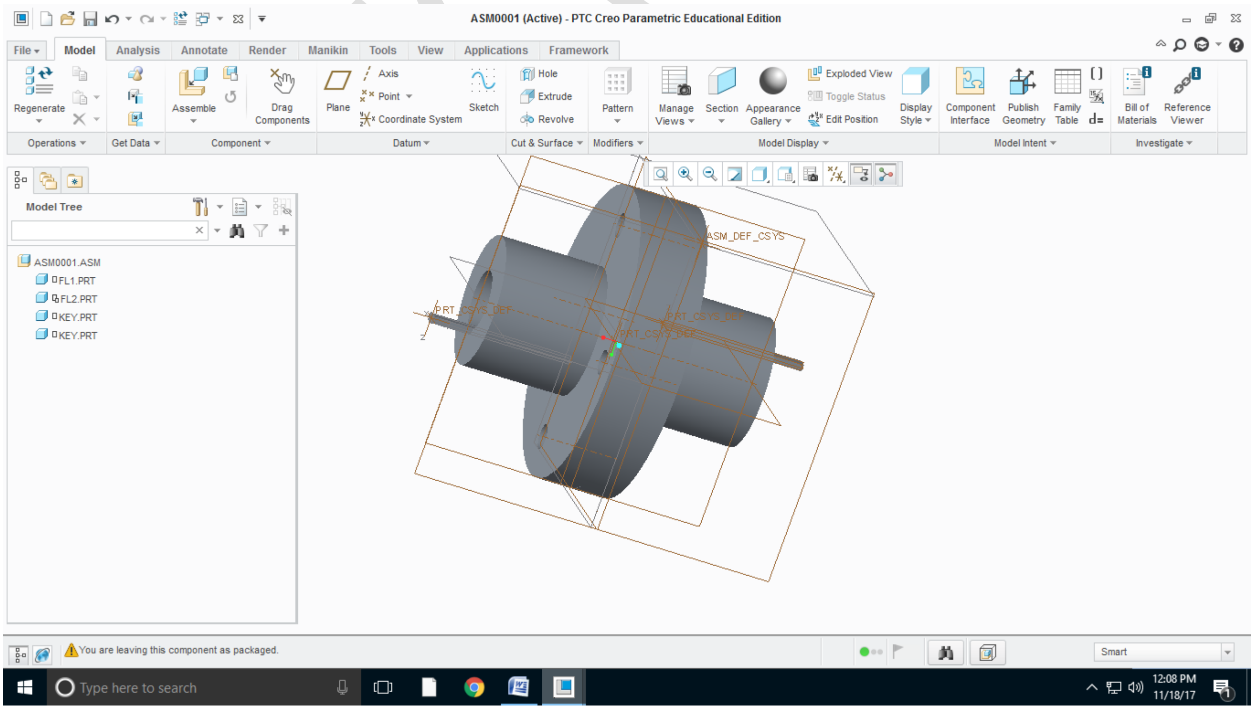


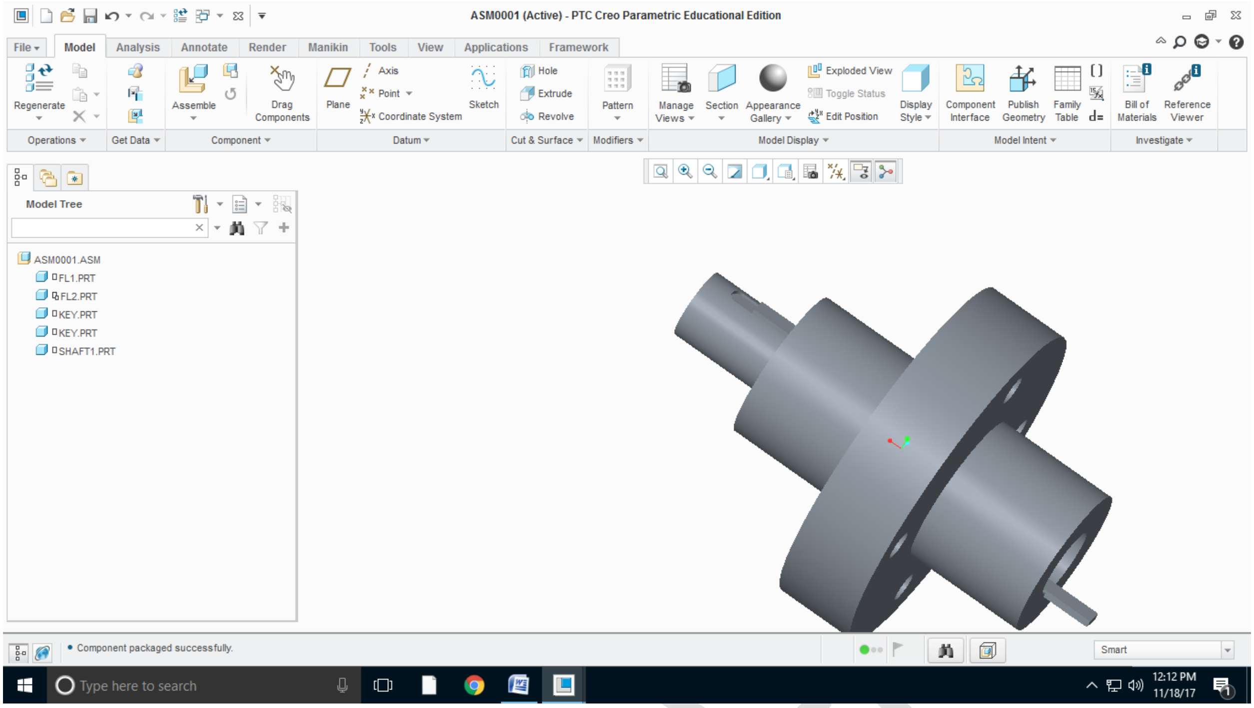


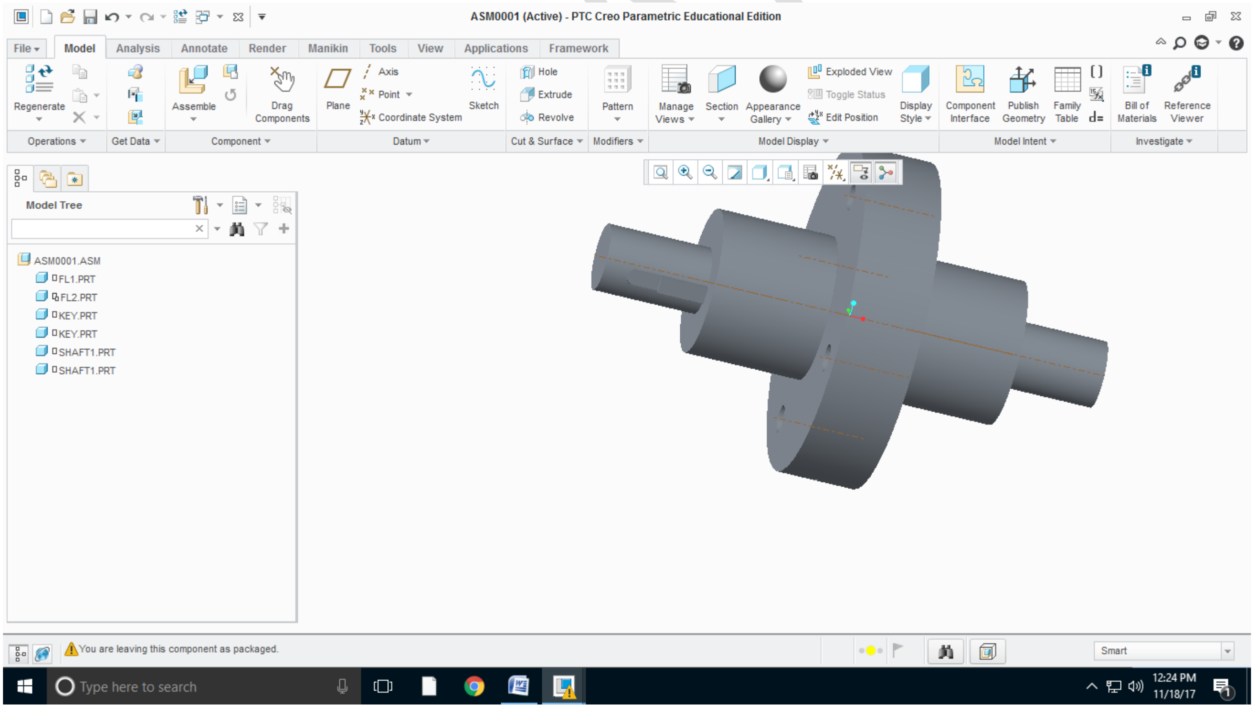
Step 19:Similarly get the 2nd part to do the assembling and select the coinside option and select the parts and axis to be coinsided.

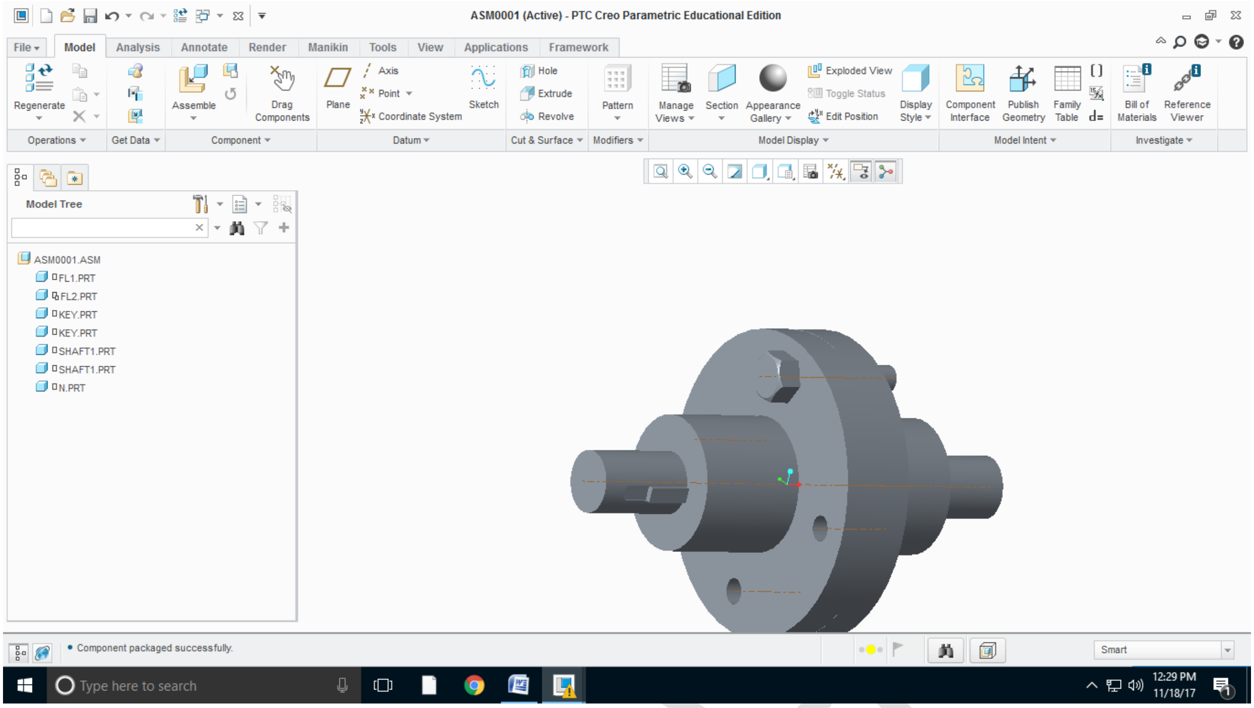


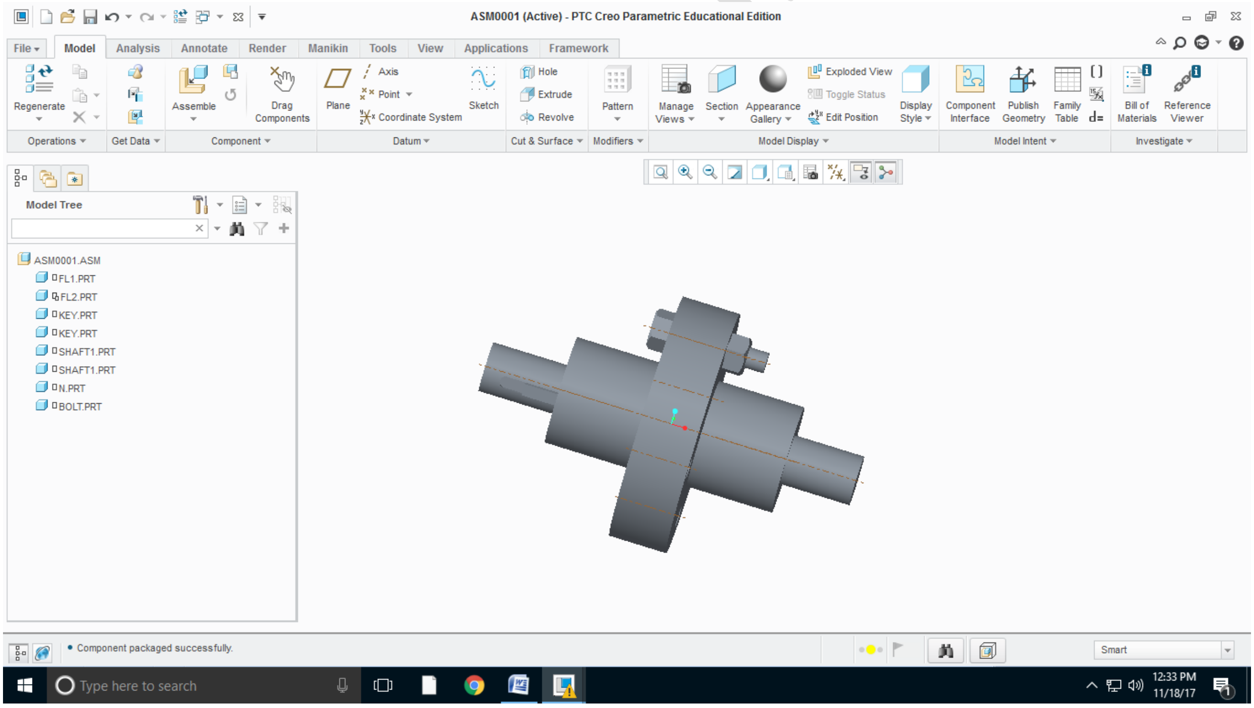
Step 20: similarly do the assembling of remaing parts to get complete of the flange coupling assembly








ASM0001 (Active) - PTC Creo Parametric Educational Edition
File Model Analysis Annotate Render Manikin Tools View Applications Framework
Regenerate Assemble Drag Components Plane Axis Point Coordinate System Sketch Hole Extrude Revolve Pattern Manage Views Section Appearance Gallery Exploded View Toggle Status Edit Position Display Style Component Interface Publish Geometry Family Table Bill of Materials Reference Viewer
Operations Get Data Component Datum Cut & Surface Modifiers Model Display Model Intent Investigate
Model Tree
ASM0001.ASM
FL1.PRT
FL2.PRT
KEY.PRT
KEY.PRT
SHAFT1.PRT
SHAFT1.PRT
N.PRT
• Component packaged successfully.
Smart
Type here to search
12:29 PM
11/18/17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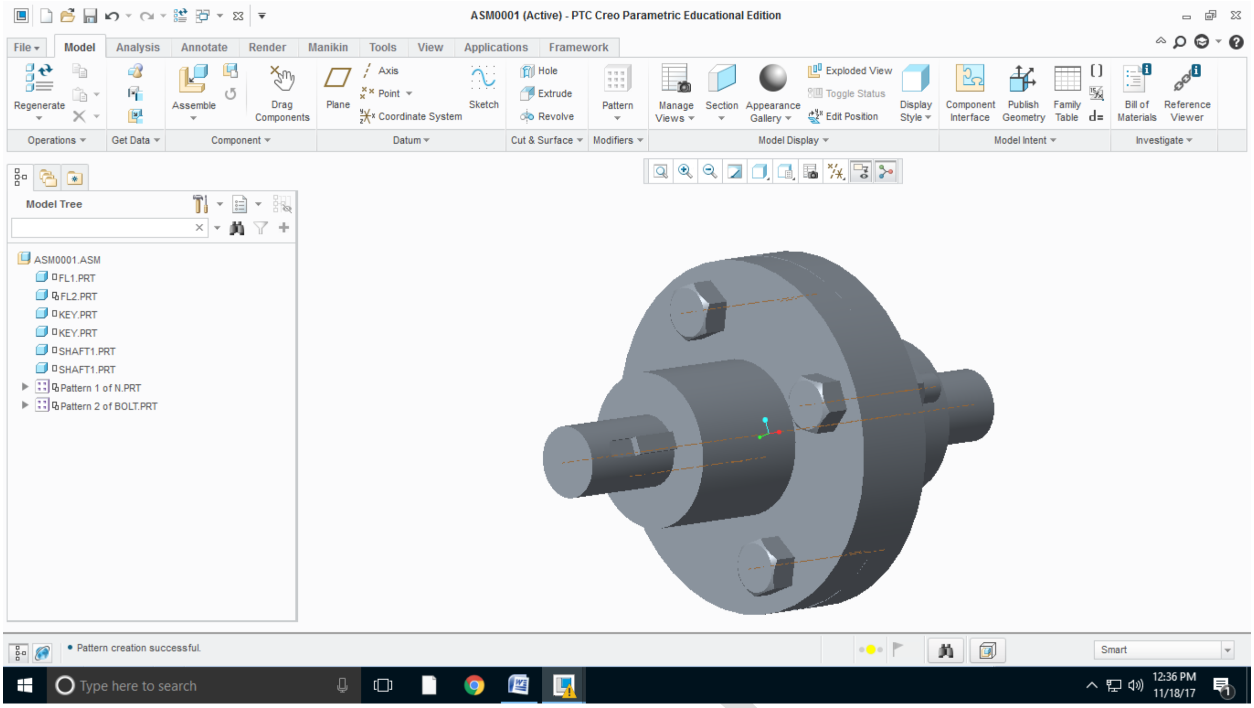


RESULT:

Thus the given model is created and assembled by using PTC Creo Parametric 3.0

COMPUTER-AIDED MANUFACTURING

Computer-Aided Manufacturing (CAM) is the use of computer-based software tools that assist engineers and machinists in manufacturing or prototyping product components. Its primary purpose is to create a faster production process and components with more precise dimensions and material consistency, which in some cases, uses only the required amount of raw material (thus minimizing waste), while simultaneously reducing energy consumption. CAM is a programming tool that makes it possible to manufacture physical models using computer-aided design (CAD) programs. CAM creates real life versions of components designed within a software package

NC Technology

Numerical Control (NC) is a software-based machine tool control technique developed at Massachusetts Institute of Technology (MIT) in early 1960s. It has now evolved into a mature technology known as Computer Numerical Control (CNC). Although major applications of CNC even today continue to be in machining, it finds applications in other processes such as sheet metal working, non-traditional machining and inspection. Robots and Rapid Prototyping machines are also CNC controlled. In fact, any process that can be visualized as a sequence of motions and switching functions can be controlled by CNC. These motions and switching functions are input in the form of alphanumeric instructions. CNC is the basis of flexible automation which helps industries cut down time-to-market and enables launch of even low volume products. Unlimited muscle power, unmanned operation, independent axes coordinated through software, simplified generic tooling even for the most complex jobs and accurate construction are some of the salient features of CNC.

CNC Machining

Automats and Special Purpose Machines (SPMs) require special cams/ templates and clutch settings for each part. Manufacture of these cams/ templates is costly and slow. Furthermore, changing over from one part to the other on these machines also consumes considerable time. The high cost and long time of these hard automated machines to produce parts can be justified only in mass production. With the advent of fast, rigid and accurate CNC machines and sophisticated CAM packages, generation of NC programs and change over from one product to the other are easy and fast as it does not require any mechanical change. These in conjunction with advanced cutting tools have made High Speed Cutting (HSC) of hard materials a reality. Therefore, CNC machining has become a standard means to produce dies and molds; tool makers today require EDM only for producing inaccessible regions, sharp corners, tiny features and desired surface quality. Intricate aerospace parts are realized through 5 axis CNC machining. Internet technology in a global village enables designing in one place, NC programming and verification in another place and actual machining in yet another place.

G&M Codes are the generic name for a control language for CNC machines. It is a way for you to tell the machine to move to various points at a desired speed , feed , control the spindle speed , turn on and off various coolants, and all sorts of other things.

PREPARATORY FUNCTIONS

G code is Pre-set function associated with the movement of machine axes and associated geometry.

List of G-codes:

Code	Description	Milling(M)	Turning(T)
G00	Rapid positioning	M	T
G01	Linear interpolation	M	T
G02	Circular interpolation, clockwise	M	T
G03	Circular interpolation, counterclockwise	M	T
G04	Dwell	M	T
G05P 10000	High-precision contour control (HPCC)	M	
G05.1 Q1.	AI Advanced Preview Control	M	
G06.1	Non-uniform rational B-spline (NURBS) Machining	M	

G07	Imaginary axis designation	M	
G09	Exact stop check, non-modal	M	T
G10	Programmable data input	M	T
G11	Data write cancel	M	T
G12	Full-circle interpolation, clockwise	M	
G13	Full-circle interpolation, counterclockwise	M	
G17	XY plane selection	M	
G18	ZX plane selection	M	T
G19	YZ plane selection	M	
G20	Programming in inches	M	T
G21	Programming in millimeters (mm)	M	T
G28	Return to home position (machine zero, aka machine reference point)	M	T
G30	Return to secondary home position (machine zero, aka machine reference point)	M	T
G31	Skip function (used for probes and tool length measurement systems)	M	

G32	Single-point threading, longhand style (if not using a cycle, e.g., G76)		T
G33	Constant-pitch threading	M	
G33	Single-point threading, longhand style (if not using a cycle, e.g., G76)		T
G34	Variable-pitch threading	M	
G40	Tool radius compensation off	M	T
G41	Tool radius compensation left	M	T
G42	Tool radius compensation right	M	T
G43	Tool height offset compensation negative	M	
G44	Tool height offset compensation positive	M	
G45	Axis offset single increase	M	
G46	Axis offset single decrease	M	
G47	Axis offset double increase	M	
G48	Axis offset double decrease	M	
G49	Tool length offset compensation cancel	M	
G50	Define the maximum spindle speed		T
G50	Scaling function cancel	M	
G50	Position register (programming of vector from part zero to tool tip)		T

G52	Local coordinate system (LCS)	M	
G53	Machine coordinate system	M	T
G54 to G59	Work coordinate systems (WCSs)	M	T
G61	Exact stop check, modal	M	T
G62	Automatic corner override	M	T
G64	Default cutting mode (cancel exact stop check mode)	M	T
G68	Rotate coordinate system.	M	
G69	Turn off coordinate system rotation.	M	
G70	Fixed cycle, multiple repetitive cycle, for finishing (including contours)		T
G71	Fixed cycle, multiple repetitive cycle, for roughing (Z-axis emphasis)		T
G72	Fixed cycle, multiple repetitive cycle, for roughing (X-axis emphasis)		T
G73	Fixed cycle, multiple repetitive cycle, for roughing, with pattern repetition		T
G73	Peck drilling cycle for milling – high-speed (NO full retraction from pecks)	M	
G74	Peck drilling cycle for turning		T
G74	Tapping cycle for milling, lefthand thread, M04 spindle direction	M	
G75	Peck grooving cycle for turning		T
G76	Fine boring cycle for milling	M	
G76	Threading cycle for turning, multiple repetitive cycle		T

G80	Cancel canned cycle	M	T
G81	Simple drilling cycle	M	
G82	Drilling cycle with dwell	M	
G83	Peck drilling cycle (full retraction from pecks)	M	
G84	Tapping cycle,righthand thread,M03 spindle direction	M	
G85	boring cycle, feed in/feed out	M	
G86	boring cycle, feed in/spindle stop/rapid out	M	
G87	boring cycle, backboring	M	
G88	boring cycle, feed in/spindle stop/manual operation	M	
G89	boring cycle, feed in/dwell/feed out	M	
G90	Absolute programming	M	T (B)
G90	Fixed cycle, simple cycle, for roughing (Z-axis emphasis)		T (A)
G91	Incremental programming	M	T (B)
G92	Position register (programming of vector from part zero to tool tip)	M	T (B)
G92	Threading cycle, simple cycle		T (A)
G94	Feedrate per minute	M	T (B)
G94	Fixed cycle, simple cycle, for roughing (X-axis emphasis)		T (A)
G95	Feedrate per revolution	M	T (B)
G96	Constant surface speed (CSS)		T
G97	Constant spindle speed	M	T

G98	Return to initial Z level in canned cycle	M	
G98	Feedrate per minute (group type A)		T (A)
G99	Feedrate per revolution (group type A)		T (A)

M code is actually operate some controls on the machine tool and thus affect the running of the machine.

List of M-codes- MISCELLANEOUS FUNCTIONS

Code	Description	Milling (M)	Turning (T)
M00	Compulsory stop	M	T
M01	Optional stop	M	T
M02	End of program	M	T
M03	Spindle on (clockwise rotation)	M	T
M04	Spindle on (counterclockwise rotation)	M	T
M05	Spindle stop	M	T
M06	Automatic tool change (ATC)	M	T (some-times)
M07	Coolant on (mist)	M	T
M08	Coolant on (flood)	M	T

M09	Coolant off	M	T
M10	Pallet clamp on	M	
M11	Pallet clamp off	M	
M13	Spindle on (clockwise rotation) and coolant on (flood)	M	
M19	Spindle orientation	M	T
M21	Mirror, X-axis	M	
M21	Tailstock forward		T
M22	Mirror, Y-axis	M	
M22	Tailstock backward		T
M23	Mirror OFF	M	
M23	Thread gradual pullout ON		T
M24	Thread gradual pullout OFF		T
M30	End of program, with return to program top	M	T
M41	Gear select – gear 1		T
M42	Gear select – gear 2		T
M43	Gear select – gear 3		T
M44	Gear select – gear 4		T
M48	Feedrate override allowed	M	T
M49	Feedrate override NOT allowed	M	T

M52	Unload Last tool from spindle	M	T
M60	Automatic pallet change (APC)	M	
M98	Subprogram call	M	T
M99	Subprogram end	M	T

ADDRESSES -
N- refers to the block number.
G- refers to the G code (Preparatory function).
X- refers to the absolute/incremental distance travelled by the slide tool in the X axis direction.
Y- refers to the absolute/incremental distance travelled by the slide tool in the Y axis direction.
Z- refers to the absolute/incremental distance travelled by the slide tool in the Z axis direction.
F -refers to the feed rate.
M -refers to the M code (Miscellaneous function).
S -refers to the spindle speed.
T- refers to the tooling management.

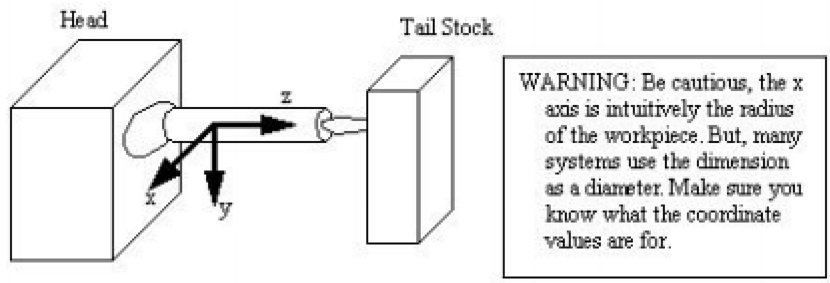
Introduction to CNC Milling

CNC LATHE

CNC lathes are rapidly replacing the older production lathes (multispindle, etc) due to their ease of setting and operation. They are designed to use modern carbide tooling and fully utilize modern processes. The part may be designed and the tool paths programmed by the CAD/CAM process, and the resulting file uploaded to the machine, and once set and trailed the machine will continue to turn out parts under the occasional supervision of an operator. The machine is controlled electronically via a computer menu style interface; the program may be modified and displayed at the machine, along with a simulated view of the process. The setter/operator needs a high level of skill to perform the process, however the knowledge base is broader compared to the older production machines where intimate knowledge of each machine was considered essential. These machines are often set and operated by the same person, where the operator will supervise a small number of machines (cell).

CNC PROGRAMMING

The coordinates are almost exclusively Cartesian and the origin is on the work piece. For a lathe, the in feed/radial axis is the x-axis, the carriage/length axis is the z-axis. There is no need for a y-axis because the tool moves in a plane through the rotational center of the work. Coordinates on the work piece shown below are relative to the work



CNC lathe / CNC XLTURN center

PROCEDURE FOR OPERATING CNC XL- Turn

- Computer on
- Wait for few minutes for display
- Open CNC trainer software
- Select file –new
- Enter the program
- Save the program(save as)-ok

FOR SIMULATION→AUTO→RESET→CYCLE START

- Stabilizer on
- Control unit switch on
- Release emergency valve
- Go to menu bar→machine option→ machine link→ok

- Select ‘HOME’, click on ‘x’, click on ‘z’→ok
- Select ‘RAPID’, move the tool towards work piece in z-ve direction and x-ve direction to touch the work piece. Before touching the work piece with tool, Switch the on the spindle manually by using M03.
- For slow movement of the tool, select ‘jog’ mode
- Touch the work piece –select datum-click on ‘x’→specify billet diameter
- Move the tool in the ‘x’+ve direction and ‘z+ve ‘ direction and touch the workpiece at the face of the billet →select datum→ click on ‘ z→’ok
- Now select ‘HOME’, click on X and Z→ok

FOR PROGRAM EXECUTION: AUTO→RESET→CYCLE START

Note:

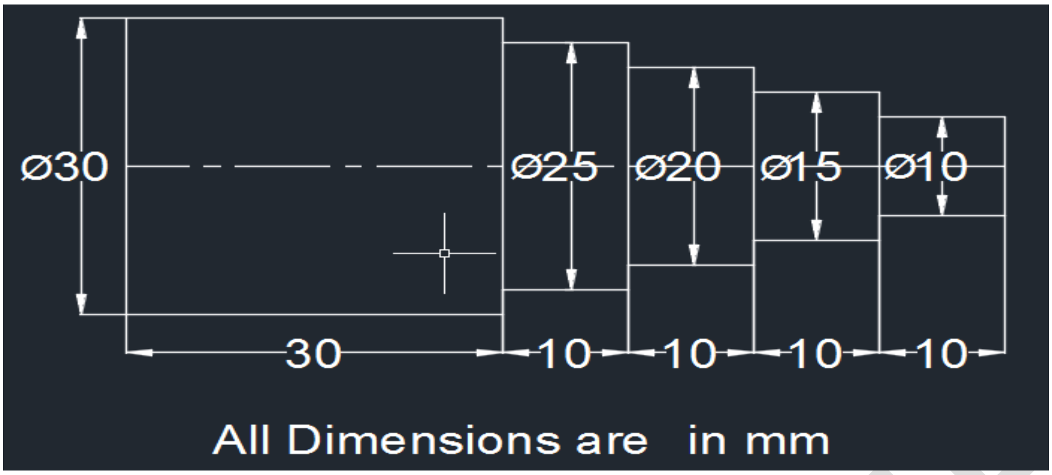
To stop machine in between, press Emergency stop or ‘Reset’.

EXPT:7 STEP TURNING OPERATION

AIM: To write the manual part program for the given component as per given dimensions, execute the same in CNC Lathe and manufacture the Component.

MATERIAL REQUIRED: Aluminum round bar of 30 mm diameter and 70 mm length.

PART DRAWING:



PARTPROGRAM:

```
[BILLET X30 Z60
G21 G98
G28 U0W0
M06 T01
M03 S1000
G00 X31Z1
G90 X30 Z-40 F60
        X29.5
        X29
        X28.5
        X28
        X27.5
        X27
        X26.5
        X26
        X25.5
        X25
G90 X24.5 Z-30F60
        X24
        X23.5
        X23
        X22.5
        X22
        X21.5
        X21
        X20.5
        X20
G90 X19.5 Z-20 F60
        X19
        X18.5
        X18
        X17.5
        X17
```

```
        X16.5
        X16
        X15.5
        X15
G90 X14.5 Z-10 F60
        X14
        X13.5
        X13
        X12.5
        X12
        X11.5
        X11
        X10.5
        X10
G28 U0W0
M05
M30;
```

PROCEDURE:

1. Write the Part Program using G codes and M codes for the component.

2. Using the Simulation Software FANUC run the part program and make corrections if any.

3. Place the workpiece in the Chuck and fix it with Chuck Key.

4. Set the tool home position.

5. Move the tool towards the work piece manually to specify the tool offset values.

6. Again Set the tool home position.

7. Execute the program to get the required shape of the work piece by Selecting
   Auto → reset → cycle start.

8. Remove the finished work piece from the chuck.

RESULT:

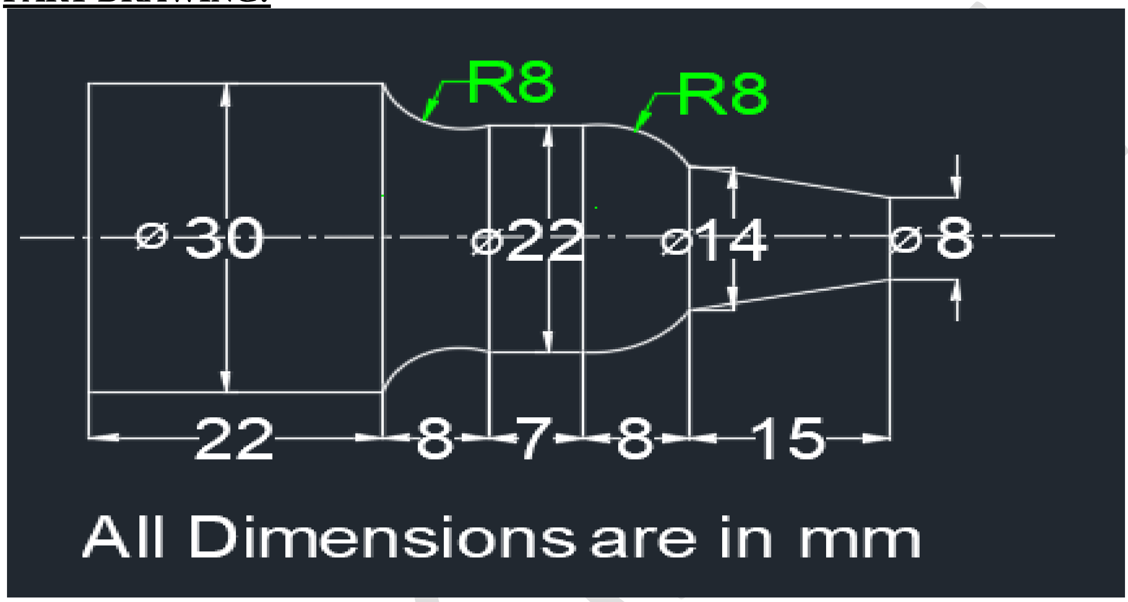
The part program for the given component is written and the Component is machined as per the dimensions.

EXPT:8 MULTIPLE TURNING CYCLE

AIM: To write the manual part program for the given component as per given dimensions, execute the same in CNC Lathe and manufacture the Component.

MATERIAL REQUIRED: Aluminum round bar of 30 mm diameter and 60 mm length..

PART DRAWING:



PARTPROGRAM:

```
[BILLET X30 Z60
G21 G98
G28 U0 W0
M06 T01
M03 S1200
G00 X32 Z1
G71 U0.25 R0.5
G71 P10 Q50 U0.1 W0.1 F60
N10 G00 X8 Z0
N20 G01  X14 Z-15
N30 G03  X22 Z-23 R8
N40 G01  X22  Z-30
N50 G02  X30  Z-38 R8
G70 P10 Q50
G28 U0 W0
M05
M30
```

PROCEDURE:

1. Write the Part Program using G codes and M codes for the component.

2. Using the Simulation Software FANUC run the part program and make corrections if any.

3. Place the workpiece in the Chuck and fix it with Chuck Key.

4. Set the tool home position.

5. Move the tool towards the work piece manually to specify the tool offset values.

6. Again Set the tool home position.

7. Execute the program to get the required shape of the work piece by Selecting
   Auto → reset → cycle start.

8. Remove the finished work piece from the chuck.

RESULT:

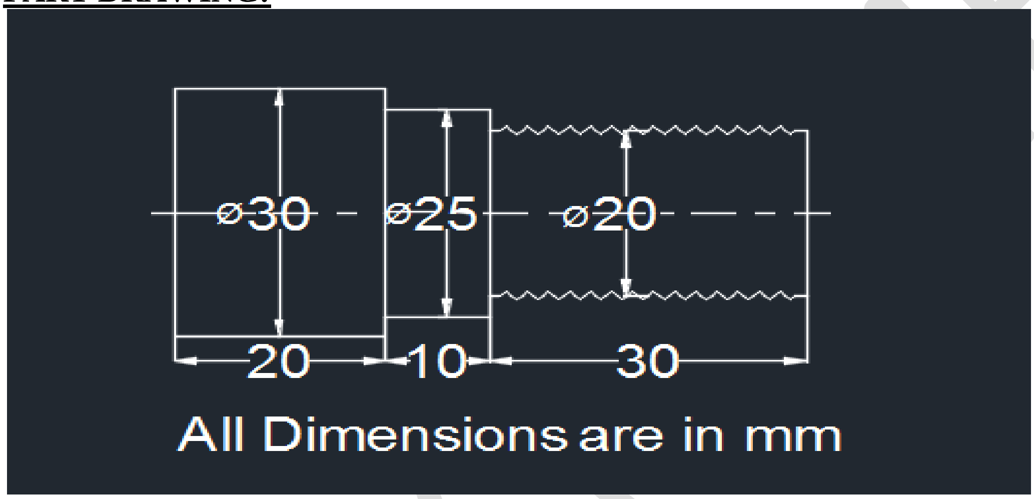
The part program for the given component is written and the Component is machined as per the dimensions.

EXPT: 9 THREADING CYCLE

AIM: To write the manual part program for the given component as per given dimensions, execute the same in CNC Lathe and manufacture the Component.

MATERIAL REQUIRED: Aluminum round bar of 30 mm diameter and 60 mm length.

PART DRAWING:



PARTPROGRAM:

```
[BILLET X30 Z60
G21 G98
G28 U0W0
M06 T01
M03 S1000
G00 X31Z1
G90 X30  Z-40  F60
        X29.5
        X29
        X28.5
        X28
        X27.5
        X27
        X26.5
        X26
        X25.5
        X25
G90 X24.5 Z-30F60
        X24
        X23.5
```

```
    X23
    X22.5
    X22
    X21.5
    X21
    X20.5
    X20
G28 V0W0
M05
G21 G98
G28 U0W0
M06T05
M03 S1000
G00 X20 Z0
G76 P031560 Q60 R0.01
G76 X 18.774  Z-30 P613 Q60 F1
G28 U0W0
M05
M30
```

PROCEDURE:

1. Write the Part Program using G codes and M codes for the component.

2. Using the Simulation Software FANUC run the part program and make corrections if any.

3. Place the workpiece in the Chuck and fix it with Chuck Key.

4. Set the tool home position.

5. Move the tool towards the work piece manually to specify the tool offset values.

6. Again Set the tool home position.

7. Execute the program to get the required shape of the work piece by Selecting
   Auto → reset → cycle start.

8. Remove the finished work piece from the chuck.

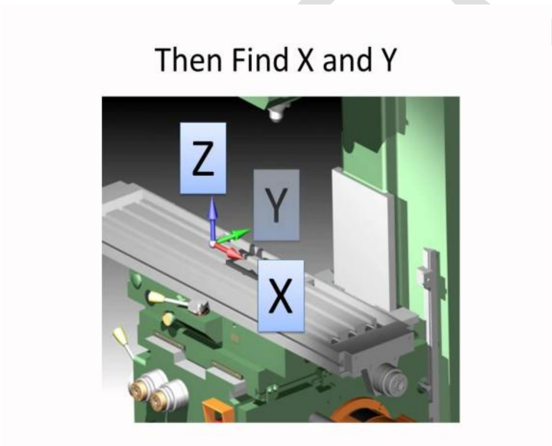
RESULT:

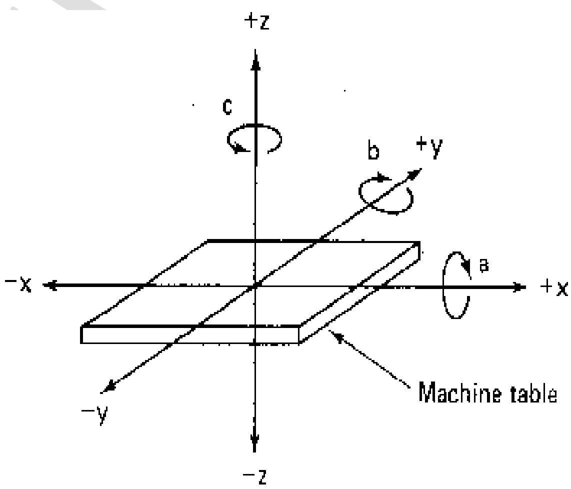
The part program for the given component is written and the Component is machined as per the dimensions.

Introduction to CNC Milling

CNC milling is a specific form of **computer numerical controlled** (**CNC**) machining. **Milling** itself is a machining process similar to both drilling and cutting, and able to achieve many of the operations performed by cutting and drilling machines. Like drilling, **milling** uses a rotating cylindrical cutting tool.

CNC MILLING COORDINAE SYSTEM:





G

PROCEDURE FOR OPERATING CNC XL-MILL MACHINE.

1. Main switch **ON.**
2. Controller **ON.**
3. Green switch **ON.**
4. Wait for few minutes for Display (Beep sound).
5. Release Emergency switch.(In clockwise)
6. Reset.
7. Select Home
8. Then Start
9. Now machine is ready

How to Write/Edit a Program

1. File——Local Disk (D)-----EOB------Program Folder------EOB
2. Edit------New-------Enter folder name------EOB(this saves the program with the given folder name)
3. Now Select the saved program------EOB
4. Press Reset while entering the program.
5. Edit(F1)----New(F1)----Name----EOB
6. In order to select the previous program (prog sec) (F2).

To Change Tool Manually

1. Go to Monitor----MDI-----Edit----M25---Start EOB (Hold tool)
2. To fix the tool M24----Start EOB

Tool Offsets

1. To move the axis always select **JOG** mode.

JOG:

X+ve,Y-ve,Z-ve,

2. To Reduce the spindle feed go to Monitor ------press ⇩

This reduces the feed rate.

3. Should touch the work piece by Z-ve,

For References

1. Select POS(position)---REL(Relative)----press X-ve----CANCEL,

2. Select X+ve,for center setting,
3. For Decimal point adjustment go to MPG-----Press arrow for getting zero
4. Step Mode Press arrows for getting the decimal point to Zero.
5. Select POS-----REL-----Press Y-ve-----CANCEL,
6. Select Y+ve EOB.
7. Same as above.
8. For Saving all, the above go to

Co-Ordinate----EDIT----G56----EDIT-----Press X+ve-----EOB

Press---Y+ve----EOB

Offset saved.

9. For Z, Press EXP(F2)------Height Offset,

CO-Ordinate -----EDIT----EXP----TOOL-OFFSET------5-----Press Z+ve-----EOB

Simulation

1. Select the program-----EDIT-----Select program(PGM SEC)----Reset(save)
2. Monitor-----Path----Middle.

Program execution

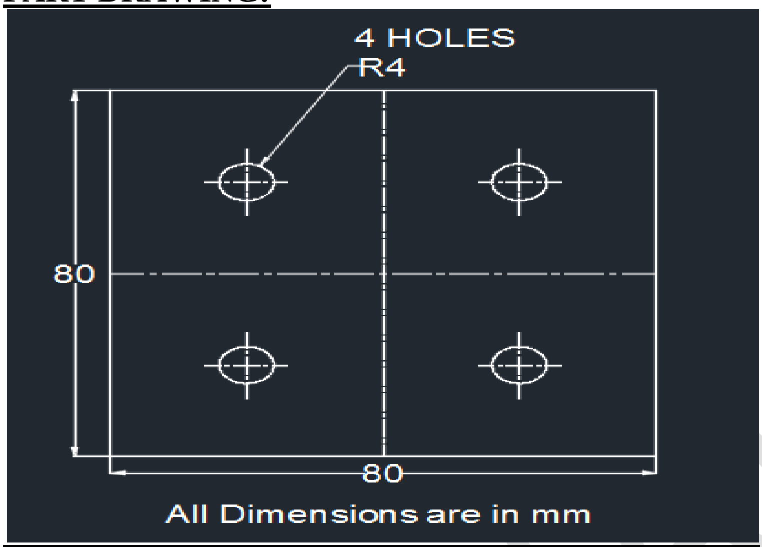
1. PROGRAM------MONITOR----AUTO----RESET----START
2. To Stop in between Press RESET.
3. Again to continue the program START.

EXPT: 10 DRILLING CYCLE

AIM: To write the manual part program for the given component as per given dimensions and execute the same in CNC Mill.

MATERIAL REQUIRED: Aluminum Plate of 80mm×80mm×10mm

PART DRAWING:



PARTPROGRAM:

```
G21 G94
G28 G91 X0 Y0 Z0
M06 T01
M03 S1500
G00 G90 G54 X0 Y0
G00 G43 H1 Z5
G01 Z0 F200
G83 G98 X-20 Y-20 Z-10  Q2  R1 F80
        X-20 Y20
        X20 Y20
        X20 Y-20
G00 G90 G80 Z10
G28 G91 X0 Y0 Z0
M05
```

M30

PROCEDURE:

1. Write the Part Program using G codes and M codes for the component.

2. Using the Simulation Software FANUC run the part program and make corrections if any.

3. Fix the workpiece on the worktable using Allen Keys.

4. Set the home position of the Machine.

5. Move the tool towards the work piece manually to specify the tool offset values.

6. Again set the home position of the Machine.

7. Execute the program to get the required shape of the work piece by Selecting
   Auto → reset → cycle start.

8. Remove the finished work piece from the worktable.

RESULT:

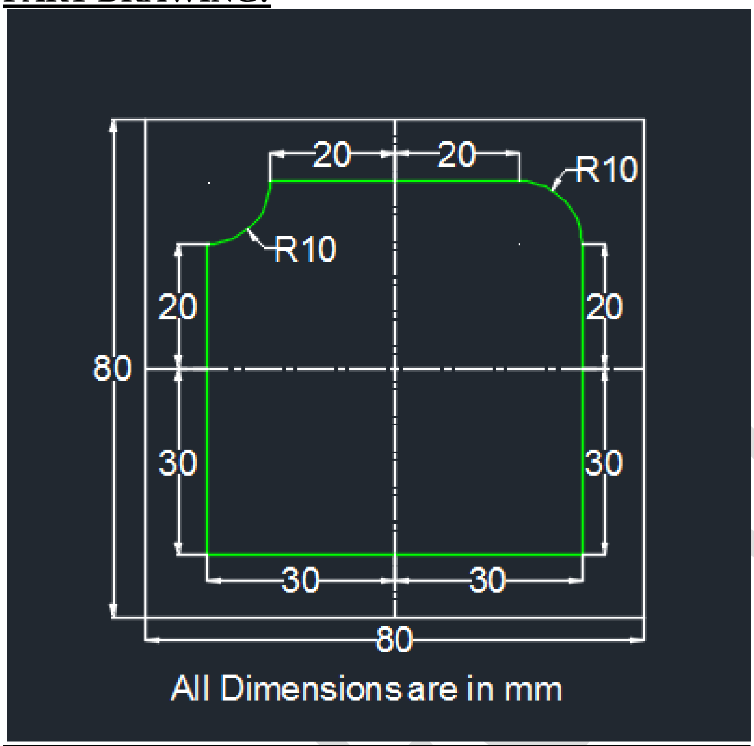
The part program for the given workpiece is written and the Component is Machined as per the given dimensions.

EXPT: 11 CONTOUR CYCLE

AIM: To write the manual part program for the given component as per given dimensions and execute the same in CNC Mill.

MATERIAL REQUIRED: Aluminum Plate of 80mm×80mm×10mm

PART DRAWING:



PARTPROGRAM:

0001:

BILLET 90mm×90mm×10mm

G21G94

G91 G28 X0 Y0 Z0

M06 T04

M03 S1500

G00 G90 G54 X0 Y0

G00 X-30 Y-30

G00 G43 H4 Z5

G01 Z0 F200

M98 P2 L10

G00 G90 Z10

G28G91 X0 Y0 Z0

M05

M30

<u>0002</u>

G01 G91 Z-0.5 F80

G01 X30 Y-30

G01 X30 Y20

G03 X20 Y30 R10

G01 X-20 Y30

G02 X-30 Y20 R10

G01 X-30 Y-30

M99

<u>PROCEDURE:</u>

1. Write the Part Program using G codes and M codes for the component.

2. Using the Simulation Software FANUC run the part program and make corrections if any.

3. Fix the workpiece on the worktable using Allen Keys.

4. Set the home position of the Machine.

5. Move the tool towards the work piece manually to specify the tool offset values.

6. Again set the home position of the Machine.

7. Execute the program to get the required shape of the work piece by Selecting
   Auto → reset → cycle start.

8. Remove the finished work piece from the worktable.

<u>RESULT:</u>

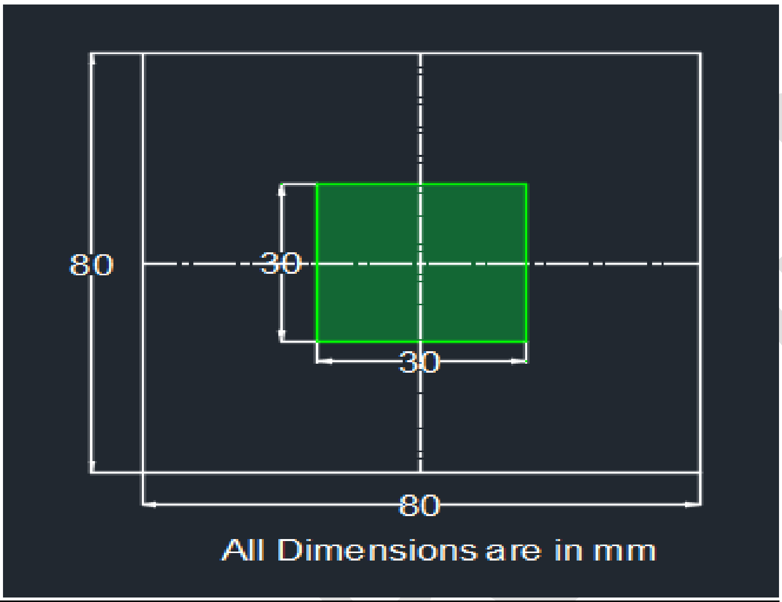
The part program for the given workpiece is written and the Component is Machined as per the given dimensions.

EXPT: 12 POCKETING OPERATION

AIM: To write the manual part program for the given component as per given dimensions and execute the same in CNC Mill.

MATERIAL REQUIRED: Aluminum Plate of 80mm×80mm×10mm

PART DRAWING:



PARTPROGRAM:

0001:

BILLET 90mm×90mm×10mm

G21G94

G91 G28 X0 Y0 Z0

M06 T01

M03 S1500

G00 G90 G54 X0 Y0

G00 X-30 Y-30

G00 G43 H1 Z5

G01 Z0 F200

M98 P2 L10

```
G00 G90 Z10
G28G91 X0 Y0 Z0
M05
M30
0002
G01 G91 Z-0.5 F80
G01 G90 G42 D10 X-5 Y0 F500
        `       Y5
                X5
                Y-5
                Y0
G40
G01 G90 G42 D10 X-10 Y0 F500
        `       Y10
                X10
                Y-10
                X-10
                Y0
G40
G01 G90 G42 D10 X-15 Y0 F500
        `       Y15
                X15
                Y-15
                X-15
                Y0
G40 G00 X0 Y0
M99
```

PROCEDURE:

1. Write the Part Program using G codes and M codes for the component.

2. Using the Simulation Software FANUC run the part program and make corrections if any.

3. Fix the workpiece on the worktable using Allen Keys.

4. Set the home position of the Machine.

5. Move the tool towards the work piece manually to specify the tool offset values.

6. Again set the home position of the Machine.

7. Execute the program to get the required shape of the work piece by Selecting
   Auto → reset → cycle start.

8. Remove the finished work piece from the worktable.

RESULT:

The part program for the given workpiece is written and the Component is Machined as per the given dimensions.